1 Introduction

In order to minimize the weight and size of an aircraft engine, one strives to increase both the efficiency and total pressure ratio of the compressor. To maximize the total pressure ratio, the flow near the outer blade span is transonic, whereas the flow near the hub is usually subsonic. One of the drawbacks of such a transonic compressor rotor is that strong shock waves usually appear near the blade tip and extend over part of the blade surface, which inevitably cause entropy production and thus efficiency deficit. For low-aspect-ratio rotor blades, tip-leakage flow loss and secondary flow loss account for a considerable part of the total flow loss [1–3]. Hereby, although the adiabatic efficiency of most compressor rotors is already over 90% nowadays, it is still quite possible to obtain much gain on performance improvement by using an advanced optimization method.

In recent years, many research papers on design optimization of turbomachinery blade using computational fluid dynamics (CFD) [4–11] have been published. Oyama et al. [4] performed design optimization of NASA Rotor 67 to reduce entropy production. An approximately 2% efficiency gain was achieved by using an evolutionary algorithm. The maximal discrepancies of total pressure ratio and mass flow rate of the optimized blade are about 1% and 0.5%, respectively, compared with those of the reference blade. Moreover, in order to satisfy higher level requirements, multi-point and multi-objective optimizations have been proposed and investigated. Mengistu and Ghaly [8] performed single- and multi-point optimizations based on a multi-point optimization method and improved the performance over the full operating range. Benini [9] investigated multi-objective optimization of NASA Rotor 37 to maximize both compressor efficiency and total pressure ratio. Liu and Liou [10] redesigned the blade of Rotor 67 to maximize total pressure ratio with minimized compressor weight and about 1.8% total pressure ratio increment was achieved.

Due to its robustness and excellent compatibility in design optimization, the evolutionary algorithm has been widely applied to both external and internal optimization designs. However, numerous populations and individuals are required to support the global optimal by using this method, which leads to thousands of flow calculations in each design cycle. In contrast, the design optimization based on the adjoint method can significantly improve the computational efficiency. The adjoint method requires the solution of the governing flow equations and the corresponding adjoint equations each only once to obtain the complete gradient information for each cost function of interest. The computational effort for the solution of the adjoint equations is about the same as that for the solution of the flow equations, regardless of the number of design parameters. For an unconstrained problem and with a simple steepest descent method this gradient information is sufficient for us to update the blade profile in one design cycle. In a constrained optimization problem, an additional solution of the adjoint equations must be computed for each separately added function of constraint in order to obtain complete gradient information with the constraint. A penalty method may be used to avoid such additional computations by merging the constraint functions and the cost function into one single cost function and thus converting the constrained problem into an unconstrained problem. Furthermore, it must be pointed out that a more sophisticated search algorithm other than the steepest descent method may require more than one evaluation of the cost function and its gradient to update the blade profile in one design cycle.

The adjoint method, proposed by Jameson [12,13], has been used in the design optimizations of airfoils, wing, wing-body configurations due to its high-efficiency and accuracy on gradient calculation. In recent years, this high-efficiency optimization method was introduced to the design optimizations of turbomachinery blade. Liu et al. [14,15] introduced this method to the design optimizations of cascade and turbine blade. Luo et al. [16–18] performed design optimizations of turbine blades to reduce shock loss, profile loss, and secondary flow loss through blade profiling, restaggering and endwall contouring by using a continuous adjoint method. Wang et al. [19,20] presented the applications of the adjoint method to the design optimizations of multistages.

In the present study, the continuous adjoint method developed for single-point design optimization of turbine nozzle vanes [16–18] is extended to study multipoint design optimization of transonic compressor rotor blades. Multipoint optimization is achieved by using a combined cost function consisting of weighted performance measures at the operating conditions of interest. In the present study, constraints are incorporated by using penalty functions to convert the problem into an unconstrained problem. Therefore, gradient information of the cost function with respect to the blade shape can be obtained at the cost of about two equivalent flow solutions per operating point independent of the number of design parameters used in representing the blade shape. The method is demonstrated for the redesign of NASA Rotor 67.
blade row. Two design optimization exercises of the blade shape are performed: (1) single-point maximization of the adiabatic efficiency at the operating condition near peak efficiency with the constraints of total pressure ratio and mass flow rate and (2) multipoint maximization of the adiabatic efficiency at three operating conditions: near peak efficiency, near stall, and near choke to improve the overall performance in the full operating range. The single- and multipoint optimization results are compared and presented in detail, and the effects of blade profiling on adiabatic efficiency, total pressure ratio and shock/tip-leakage interaction are analyzed.

2 Rotor 67 Flow Analysis Validation

Rotor 67 is a transonic axial-flow compressor blade in the first-stage rotor of a two-stage fan. The rotor has 22 blades and an aspect ratio of 1.56. The design total pressure ratio is 1.63 at a mass flow rate of 33.25 kg/s. The design rotational speed is 16,043 rpm with a tip speed of 429 m/s and a relative Mach number of 1.38 at the inlet tip. The Reynolds number based on the axial chord at the hub is about 1.797 x 10^6. Rotor 67 was developed in the 1970s and was then experimentally investigated by Strazisar et al. [21]. Reference [21] presented the performance of Rotor 67 in detail and thus provided a data base, with which the computational results presented in the present study are compared and validated.

Besides shocks, viscosity, and secondary flow, leakage flow contributes a considerable part to the total flow loss of rotors. Due to its significant influence on the overall performance, many reports on numerical simulation and control of tip-leakage flow have been published [1,2,22–28]. The spanwise distance of tip clearance of Rotor 67 is approximately 1.0 mm. The design optimization presented in the study includes the effects of tip clearance. The flow through Rotor 67 is transonic near the blade tip. The shock/tip-leakage interaction has detrimental effects on the rotor performance, such as aerodynamic losses, blockage, and instabilities. Hereby, the present research concentrates on (1) how to improve the performance as required by using the viscous adjoint method, and (2) how the blade profiling influences the shock/tip-leakage interaction.

In the present study, the flow solver Turb90 developed by CFD Laboratory of University of California, Irvine is introduced to calculate the three-dimensional turbomachinery flow [29,30]. This flow solver solves the Reynolds-averaged Navier–Stokes (RANS) equation on a single block using a central finite-volume method with multigrid acceleration technology and includes several turbulence models. In the present study, Spalart–Allmaras turbulence model [31] is adopted. The governing flow equation with a source term is given as

\[
\frac{\partial \mathbf{u}}{\partial t} + \frac{\partial (\mathbf{u} \cdot \mathbf{U})}{\partial x_j} = -\frac{\partial \mathbf{T}}{\partial x_j} + \mathbf{F} + \mathbf{S}
\]

(1)

where \( \mathbf{u} \) and \( \mathbf{S} \) are grid velocity components and the source term, respectively, and

\[
\begin{align*}
\mathbf{u}_1 &= 0, \quad \mathbf{u}_2 = -\Omega z, \quad \mathbf{u}_3 = 2y \\
\mathbf{S} &= (0, \quad 0, \quad \rho u_2 \Omega, \quad -\rho u_2 \Omega, \quad 0)^T
\end{align*}
\]

An H-grid containing 120, 48, and 44 cells in the axial, pitchwise, and spanwise directions, respectively, is used for flow calculations. Along the axial direction, 24, 64, and 32 cells are distributed in the inlet block, blade block and exit block, respectively. Kirtley et al. [3] and Chima [24] suggest that a simple periodicity clearance model is adequate for capturing the effect of tip clearance flow. Such a model is used in the present study. Only four cells are used in the tip gap. Although four cells might not be enough to accurately capture the detailed leakage flow within the clearance gap, the model is sufficient in providing information on the effect of the tip leakage flow on the main passage flow and also on the shock/tip-leakage interaction.

The flow calculations are performed with different back pressure to obtain the overall performance of Rotor 67, as presented in Fig. 1. In this picture, the mass flow rates of the CFD and experiment are normalized by their corresponding choked values. The total pressure ratio \( \pi \) and adiabatic efficiency \( \eta \) obtained from CFD agree with those of the experiment.

The computed spanwise distributions of total pressure ratio \( \pi \), total temperature ratio \( \theta \), and flow turning angle \( \beta \) near peak efficiency are presented and compared with the experimental results, as shown in Fig. 2. In order to investigate the influence of tip clearance, flow calculation without tip gap is also performed with the same back pressure. In this picture, with the tip gap the flow turning has a significant increment, while the total pressure ratio is significantly decreased near the casing. The tip gap reduces the capability for the blade to impart work to the flow near the blade tip. On the other hand, severe flow blockage appears in the tip gap of transonic compressor rotors due to the shock/tip-leakage interaction [21–25], which leads to an axial momentum deficit and consequently the flow turning increases. Van Zante et al. [2] and Dunham and Meauze [32] confirmed the increase of total temperature in the gap through numerical simulation and experimental measurement and suggested that the total temperature excess was attributed to the increased flow turning of the tip-leakage flow. The computed spanwise distributions with tip clearance agree well with experiment.

3 The Optimization Problem

It is known that flow losses can be measured by entropy production [33]. Consequently, entropy production per unit mass flow rate is selected as the cost function in the present study. The multipoint design optimization will be studied based on the viscous adjoint method.

In the multipoint design optimization, the blade is optimized for performance at three operating conditions. The optimization results will be compared with that of the single point design optimization near peak efficiency to illustrate the overall performance improvement in the full operating range. Constraint on the mass flow rate and total pressure ratio at each design point is enforced by incorporating penalty functions in the cost function. In this case the design optimization with constraints is converted into a design optimization without constraint. The cost function of multipoint design optimization is defined as

\[
\begin{align*}
\min \quad & \sum_{i=1}^{3} \left( \frac{\pi_i - \pi_{\text{CFD}}}{\pi_{\text{CFD}}} ight)^2 \\
\text{subject to} \quad & \sum_{i=1}^{3} \left( \theta_i - \theta_{\text{CFD}} \right)^2 \\
& \sum_{i=1}^{3} \left( \beta_i - \beta_{\text{CFD}} \right)^2 \\
& \left( \frac{\rho u_2 \Omega}{\rho u_2 \Omega_{\text{CFD}}} - 1 \right) \\& \left( \frac{\rho u_2 \Omega}{\rho u_2 \Omega_{\text{CFD}}} - 1 \right)
\end{align*}
\]
\[ I = \sum_{i=1}^{N} \lambda_i I_i \quad I_i = \sum_{i=1}^{N} \lambda_i \sigma_i - 1 \]  
\[ s_{gen} = \frac{\int_B n_i \rho u_i dB}{\int_B n_i \rho u_i dB} \]  
\[ \delta I = \frac{\partial I}{\partial w} \psi^T \frac{\partial R}{\partial w} \delta w + \frac{\partial I}{\partial F} \psi^T \frac{\partial R}{\partial F} \delta F \]  

where \( N \) is the number of design points, \( \sigma_i \) and \( \bar{\sigma}_i \) are the ratios of mass flow rate and total pressure ratio of the optimized blade to those of the reference blade at the \( i \)th design point, respectively, \( \lambda_i \) are the weights of the cost function corresponding to different design point satisfying \( \sum \lambda_i = 1 \), \( s_{gen} \) denotes the entropy production per unit mass flow rate with the definition

Minimization of cost function described above attempts to obtain an optimized blade profile that maximizes the adiabatic efficiency for the same total pressure ratios and mass flow rates that are representative of the range of operation of the blade row.

Hicks–Henne shape functions [18] are introduced to perturb the reference blade geometry in the present study and 16 shape functions are uniformly distributed on each side of the blade surface (32 total) at each of the selected spanwise sections. Then the total perturbation on each of the blade surfaces can be determined by the sum of the weighted shape functions, as shown in Eq. (4),

\[ \delta p(x) = \sum_{i=1}^{16} a_i b_i(x) \]  

where \( b_i(x) \) are the Hicks–Henne shape functions defined in Ref. [18], \( a_i \) are the design parameters, \( \delta p(x) \) is the local circumferential perturbation.

Figure 3 presents five different blade profiles at the hub, where the profiles named as fourth, eighth, twelfth, and All are the modified profiles with the perturbation from the fourth, eighth, and twelfth shape function and all three of them imposed on the reference suction surface, respectively. It is known from this picture that each of the shape functions can influence the blade profile from the leading edge to the trailing edge, and the maximum perturbation appears at the corresponding control point. With all of the shape functions overlaid, the perturbation of the profile All is more significant, however, it is still smooth enough.

4 The Adjoint Method

The implementation of the adjoint method was described previously [16,18]. It is briefly summarized here, Let \( R(w, F) = 0 \) be the steady Reynolds-averaged Navier–Stokes equations, where the dependence on the blade geometry through boundary conditions are explicitly specified. The variation of cost function \( \delta I \) and the variation of governing flow equation \( \delta R \) consist of two terms, one due to the variation of flow field \( \delta w \) and the other due to the modification of boundaries \( \delta F \). By introducing a set of adjoint variables and regarding the variation of governing flow equation as a constraint \( \delta R = 0 \), the variation of cost function can be written as

\[ \delta I = \frac{\partial I}{\partial w} \psi^T \frac{\partial R}{\partial w} \delta w + \frac{\partial I}{\partial F} \psi^T \frac{\partial R}{\partial F} \delta F \]  

The fundamental key of the adjoint method is to eliminate the contribution of \( \delta w \) to \( \delta I \) through the solution of the adjoint equation in order to calculate the gradient with high efficiency. From Eq. (5), the adjoint equation and the gradient \( G \) are

\[ \frac{\partial I}{\partial w} - \psi^T \frac{\partial R}{\partial w} = 0 \quad G = \frac{\partial I}{\partial F} - \psi^T \frac{\partial R}{\partial F} \]  

Once the flow variables \( w \) and the adjoint variables \( \Psi \) are obtained through solving the governing flow equation and the
corresponding adjoint equation each once, the gradient can then be calculated on obtaining additional derivatives of the cost function and computational grid metrics on the design parameters, which can be evaluated conveniently and with relatively small computational cost by a finite-difference scheme and by using the chain rule for derivatives.

4.1 Adjoint Method for Multipoint Design Optimization.

For the multipoint design optimization, the variations of the $i$th cost function and its corresponding governing flow equation are

$$
\delta l_i = \frac{\partial l_i}{\partial w_i} \delta w_i + \frac{\partial l_i}{\partial F} \delta F
$$

$$
\delta R_i = \frac{\partial R_i}{\partial w_i} \delta w_i + \frac{\partial R_i}{\partial F} \delta F = 0
$$

Introducing an adjoint vector $\Psi_i$, the variation of the $i$th cost function can be specified following Eq. (5). Similarly, the $i$th adjoint equation and the $i$th gradient can be specified following Eq. (6). Finally the gradient for the multipoint design optimization can be calculated by

$$
G = \sum_{i=1}^{N} \lambda_i G_i, \quad G_i = \frac{\partial l_i}{\partial F} - \Psi_i^T \frac{\partial R_i}{\partial F}
$$  (7)

The multipoint design optimization by using the adjoint method can be summarized as follows:

Step 1. Solve the governing flow equation to determine the $i$th flow field $w_i$.

Step 2. Solve the corresponding adjoint equation to obtain the $i$th adjoint field $\Psi_i$.

Step 3. Calculate the $i$th gradient $G_i$ from Eq. (7).

Go to Step 4 if the gradient calculation is completed at all of the selected operating conditions; or else go to Step 1.

Step 4. Calculate the weighted gradient $G$ with respect to $a_i$ in Eq. (4) from Eq. (7).

Step 5. Determine the design parameters $a_i$ by using an appropriate gradient-based optimization method or search algorithm of choice.

In the present study, the simplest steepest descent method is used, in which $a_i$ is determined by

$$
a_i = -G_i/l_i
$$

where $g_i$ is the $i$th gradient with respect to $a_i$ and $l$ is the step length empirically determined. A one-dimensional line search in the gradient direction may be conducted to more precisely determine the step length $l$. However, that requires additional solutions of the flow equations to evaluate the cost function. In a more sophisticated method, Steps 1–4 may need to be repeated to obtain additional gradient information as well as cost function values. Our computational results show the simplest steepest descent method offers an overall cost-effective approach.

Step 6. Determine the perturbation from Eq. (4) to update the blade profile, and then go to Step 1.

4.2 Adjoint Equation and Boundary Conditions for Rotor Optimization.

Source terms due to rotation must be included in the governing flow equations and appropriate boundary conditions in terms of the relative velocity must be imposed for the flow in a rotor blade row. The source term $S$ in the governing flow equation contributes an additional adjoint source term; and meanwhile, the corresponding adjoint boundary conditions are redetermined accordingly. The viscous adjoint equation corresponding to Eq. (1) can be given as

$$
A_{ij}^T \frac{\partial \Psi}{\partial x_j} + B_i^T \Psi + \frac{1}{J} [M^{-1}]^T Y = 0
$$  (8)

where $1/J[M^{-1}]^T Y$ is the viscous adjoint operator as presented in the Ref. [16], and $A_{ij}$ and $B_i$ are the Jacobian matrices and the source matrix with the definitions

$$
A_{ij} = A_j - \dot{u}_i L_j, \quad B_i = \frac{\partial S}{\partial w}
$$  (9)

where $L_j$ denotes the unit matrix.

The determination of the wall boundary conditions of the adjoint equation is quite similar to that presented in the previous publications. The adjoint variables on inviscid wall satisfy

$$
\sum_{j=1}^3 n_j (\psi_{j+1} + \dot{u}_j \psi_3) = 0
$$  (10)

while on the adiabatic wall, the viscous boundary conditions of the adjoint equation are given as

$$
\frac{\partial \psi_3}{\partial n} = 0, \quad \psi_{j+1} + \dot{u}_j \psi_3 = 0, \quad j = 1, 2, 3
$$  (11)

Defining $\dot{u}_j + \dot{u}_3 \psi_3$ as the relative adjoint variables, it is known from Eqs. (10) and (11) that the relative adjoint variables satisfy the same wall boundary conditions as those that the absolute adjoint variables satisfy in the case of a stator blade row described in [16].

On the inlet and outlet boundaries (1) the viscous effects could usually be neglected, and (2) the boundary planes are usually perpendicular to the axial direction, i.e., $n_2 = 0$, $n_3 = 0$, the determination of the inlet and outlet boundary conditions depends on only the Jacobian matrix $A_{ij}$. Since $A_{ij} = A_i$ as shown by Eq. (9), it can be concluded that with a given cost function, the inlet and outlet boundary conditions of the adjoint equation are the same for inviscid and viscous design optimizations of both stator and rotor blades.

In performing the derivations of the adjoint equations of the present study, variations of the viscosity and thermal conductivity including their turbulent contributions are neglected. In other words, the turbulence viscosity and conductivity are assumed to be frozen in the adjoint equations of the Navier–Stokes equations and no adjoint equation(s) for the turbulence model equation(s) are needed. This is acceptable since we assume the variation of the flow field is small within each design cycle. In addition, we expect the flow to be relatively well-behaved since we are seeking an optimized design so that the dependence of the turbulence eddy viscosity and heat diffusivity on the flow field is relatively weak. Notice that both the viscosity and thermal conductivity are updated when the Navier–Stokes equations and the turbulence model equation(s) are solved again with the updated geometry. Therefore, the flow solutions will converge with the correct turbulence parameters once the design reaches an optimum.

5 Results and Discussion

Two different design optimization studies for Rotor 67 based on the above viscous adjoint method are investigated. The shape functions are uniformly distributed on each blade surface at nine different span locations from the hub to the casing and thus there are totally 288 design parameters in the design optimization. The results are presented and compared in the following.

In the design optimization using the adjoint method, the solution of the governing flow equations, the solution of the adjoint equations, and the evaluation of the gradient vector $G$ in Eq. (7), are the three major parts of calculation that cost most of the computer time. The design optimization studies mentioned above are performed on only one core of Intel Xeon E5-2670. Table 1 gives the CPU time required by each one of the three parts, where the CPU times are normalized by that of governing flow equation.
and the adjoint equation and the governing flow equation are iteratively solved with the same time marching steps. The results demonstrate that the computation effort of adjoint equation is almost the same as that of governing flow equation, however, the evaluation of the gradient vector with 288 design parameters costs about 25% of computer time of governing flow equation.

Before the redesign, the accuracy of the gradients obtained by the adjoint method is verified by comparing with those by finite difference method (FDM). Figure 4 shows the gradients at 50% blade span at the first design cycle with the cost function defined as the entropy production per unit mass flow rate, where all the step sizes are selected supposing the axial chord of the blade is unit and all the gradients are normalized by the corresponding $L_2$ norm,

$$
\tilde{g}_i = \frac{g_i}{|G|_2}
$$

The FDM gradients calculated by using different step sizes of perturbation are compared with those obtained by the adjoint method. The accuracy is acceptable.

### 5.1 Single-Point Optimization Near Peak Efficiency

A single-point design optimization at the operating condition near peak efficiency is first investigated. The redesign attempts to seek a blade geometry with improved performance on adiabatic efficiency, maintaining both total pressure ratio and mass flow rate. Many different sets of the coefficients $K_1$ and $K_2$ are selected to perform the design optimization while the values $K_1 = 1.6$ and $K_2 = 0.15$ are used to obtain the results presented here, which may be the optimal values for the particular cases. We find other choices may also give good results.

Table 2 presents the performance of both reference and the optimized blades, where the entropy production is normalized by that of the reference blade. The total pressure ratio and mass flow rate of the optimized blade show little change compared with the references, whereas the adiabatic efficiency has an increment of about 1.10%. The flow turning is slightly decreased because of the increased efficiency at the same mass flow rate.

Table 5 displays the contours of the relative isentropic Mach number on the blade surface. The relative isentropic Mach number is calculated based on the static pressure and the spanwise relative total pressure at the inlet. The picture indicates that on the pressure surface the strong shock wave is located on the front portion, while on the suction surface it is located after the midchord. The redesigned blade weakens the shock due to the decreased Mach number before the shock, and drives the high Mach region to move toward the inner blade span. The Mach contours on the optimized pressure surface show that the new rotor has replaced the
single shock of the reference blade by a double-shock system, with the first wave more inclined upstream than the single one, which is also evident in Fig. 6. Denton and Xu [34], Biollo and Benini [27] pointed out that such a solution can help to reduce the aerodynamic losses associated with the shock, providing a more efficient local flow diffusion.

Figure 6 presents the blade profiles and the corresponding relative isentropic Mach number distributions at three different span locations. At 25% blade span the optimized Mach number distribution is almost a duplicate of the reference with slightly decreased downstream Mach number on the rear portion. At 60% blade span the shock on the pressure surface is significantly weakened; and the optimized loading slightly increases on the front portion, while slightly decreases on the rear portion. Such an optimized Mach number distribution favors maintaining the flow turning. At 95% blade span, there exists an evident double-shock system on the pressure surface of the optimized blade and the shock on the suction surface is also significantly weakened. Besides, the decreased loading at 95% blade span of the optimized blade contributes a decrement to flow turning. The weakened shock of the redesigned blade decreases the section loading and thus the total pressure ratio at both 60% and 95% span locations. Such variations of total pressure ratio and flow turning are also visualized in Fig. 7.

In Fig. 7, both total pressure ratio and total temperature ratio increase on the inner span and decrease on the outer span. The optimized flow turning keeps almost the same as the reference. However, it decreases from 80% span to the blade tip. The adiabatic efficiency is increased along the whole blade span. On the outer span the weakened shock attributes to the decrease of total pressure ratio and the increase of adiabatic efficiency. On the inner span the slightly increased downstream pressure favors the increase of total pressure ratio.

Although the leakage flow within the tip gap may not be captured with sufficient accuracy because of the limited number of grids in the gap, the effects of the tip leakage flow on the blade shocks can be observed before and after optimization. Figure 8 presents the contours of relative Mach number in a cross-channel plane from 80% blade span to the casing at three axial locations to identify the leakage flow, where P.S. and S.S. denote pressure surface and suction surface, respectively. As suggested by Suder and Celestina [1] and Chima [24], the leakage vortex is usually evident just downstream of the leading edge, followed by a region of low-speed flow. Lower relative Mach number means more intensive leakage vortex with higher loss of relative dynamic head. From this picture it is known that the leakage vortex in the 40% cross-channel plane is located near the casing and exhibits a significant inward radial penetration and a circumferential extend of about 50% of blade pitch. Along with the through-flow, the leakage vortex moves toward the pressure surface and induces a greater deficit on the relative Mach number, as presented by the contours in the 80% cross-channel plane. In the 110% cross-channel plane, the minimum relative Mach number increases because the leakage vortex merges with the rotor wake downstream of the trailing edge. Compared to the reference blade, the leakage vortex of the optimized blade is significantly suppressed, except in the 40% cross-channel plane. As shown in Fig. 6, at 95% blade span the pitchwise pressure gradient increases from the leading edge to about 30% chord of the optimized blade, which favors the generation of leakage flow; consequently in the 40% cross-channel plane of the optimized blade the relative Mach number deficit is slightly increased. However, the pitchwise pressure gradient of the optimized blade decreases after about 30% chord, and consequently the leakage flow downstream is significantly suppressed.

For this single point design optimization, improvement of adiabatic efficiency is achieved with strictly enforced constraints of total pressure ratio and mass flow rate. The leakage flow is suppressed due to the decreased pitchwise pressure gradient near the blade tip.

5.2 Multipoint Optimization. The multipoint optimization takes into account the performance of the blade profile at three different operating conditions through the use of different weights. The three operating conditions are chosen to be one near choke, one near peak efficiency, and one near stall. The weights used in Eq. (2) for the three conditions are chosen to be $\lambda_1 = 0.3$, $\lambda_2 = 0.4$, $\lambda_3 = 0.4$. 
and $z_3 = 0.3$, respectively. The given set of the weights constitutes only one test case. Designers can choose different sets of these parameters according to different requirements for industrial application.

Table 3 presents the performance of the blades at the three operating conditions, where Ref and Opt denote the reference and the optimized blades, respectively, and P. E. is the abbreviation of peak efficiency. The total pressure ratios and mass flow rates of the redesigned blade at the three specified operating conditions are very close to their corresponding reference values. Significant gains of adiabatic efficiency are achieved at all of the three operating conditions, with increments of about 1.24%, 0.84%, and 0.56%, respectively, compared with the references. As the total pressure ratios and mass flow rates are strictly enforced, the flow turning at each design point is slightly decreased due to the improved performance.

As shown in the single-point design optimization, the optimized shock plays an important role on the performance improvement. In order to not only present the changes of shock structure but also visualize the leakage vortex patterns, Fig. 9 shows the pressure contours on a blade-to-blade stream surface at the blade tip of both the reference and the optimized blades, where the pressure is normalized by the inlet total pressure. In these pictures, the leakage vortices originating from the suction side can be identified by the clustered pressure contour lines. The leakage flow before the shock forms a well-defined vortex, which moves toward the pressure side and distorts the shock. As the back pressure increases, the shock inside the passage moves upstream and even beyond the leading edge to form a strong bow shock at the operating condition near the stall. The leakage vortex also separates closer to the leading edge and the leakage flow moves toward the pressure side faster.

After the blade redesign the downstream pressure at each operating condition keeps almost unchanged. The shocks at the operating conditions near choke and near peak efficiency are significantly weakened. However, it is found that the shock of the optimized blade is still perpendicular to the casing and the location keeps almost unchanged compared with that of the reference blade. Besides the strength, the location of the shock perpendicular to the casing can significantly influence the flow blockage in the tip gap. References [27,34] have demonstrated that the

<table>
<thead>
<tr>
<th>Blades</th>
<th>$\dot{m}$ (kg/s)</th>
<th>$\pi$</th>
<th>$\eta$ (%)</th>
<th>$\beta$ (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ref_choke</td>
<td>34.94</td>
<td>1.509</td>
<td>88.53</td>
<td>-33.16</td>
</tr>
<tr>
<td>Opt_choke</td>
<td>34.90</td>
<td>1.512</td>
<td>91.64</td>
<td>-33.11</td>
</tr>
<tr>
<td>Ref_P.E.</td>
<td>34.59</td>
<td>1.627</td>
<td>92.64</td>
<td>-35.85</td>
</tr>
<tr>
<td>Opt_P.E.</td>
<td>34.58</td>
<td>1.628</td>
<td>92.86</td>
<td>-38.76</td>
</tr>
<tr>
<td>Ref_stall</td>
<td>32.86</td>
<td>1.690</td>
<td>90.94</td>
<td>-43.72</td>
</tr>
<tr>
<td>Opt_stall</td>
<td>32.88</td>
<td>1.690</td>
<td>91.45</td>
<td>-43.61</td>
</tr>
</tbody>
</table>

Fig. 7 Spanwise distributions of (a) total pressure ratio and adiabatic efficiency and (b) total temperature ratio and flow turning

Fig. 8 Contours of relative Mach number at different axial locations (a) 40% chord, (b) 80% chord, and (c) 110% chord

Table 3 Performance at three different operating conditions
optimized location of such shock perpendicular to the casing through blade sweeping can effectively reduce the flow blockage in the tip gap and consequently increase the stall margin. Future research may seek an optimal distribution of blade sweeping along span to improve near-stall performance.

Figure 10 compares the overall performance of the three different blades: the reference blade, the single-point, and multipoint optimized blades. The single-point design optimization at the near-peak efficiency point provides significant gains of adiabatic efficiency without any loss of total pressure ratio around the 98% normalized mass flow rate and beyond. In fact, it offers noticeable increase in adiabatic efficiency over the whole operating range. However, for the mass flow rates below the 98% mark, the single-point design optimization shows a noticeable decrease in the total pressure ratio compared to the reference blade. The multipoint optimization, on the other hand, is able to achieve higher adiabatic efficiency than that of the single-point design optimization over the whole operating mass flow range below 98% while maintaining the same total pressure ratio of the reference blade. The adiabatic efficiency of the multipoint design is only slightly reduced near the peak-efficiency point compared to the single-point design, but still markedly higher than that of the reference blade.

Experiment [21] suggests that Rotor 67 approaches stall at about 93% normalized mass flow rate. The true stall cannot be captured in the present study. Reasons include (1) the steady-state flow model might not be able to simulate the high unsteady stall flow and consequently capture the real stall phenomena, and (2) the numerical simulation of leakage flow is still an open issue and different turbulence models may support quite different results as shown by Biollo and Benini [27]. Whether the multipoint optimized blade offers a wider stall margin cannot be confirmed by the present steady-state code.

Shock/tip-leakage interaction is regarded as one of the main stall triggers in transonic compressor rotors. Reminded that low relative Mach number means low dynamic head along with severe flow blockage induced by shock/tip-leakage interaction. Figure 11 presents the contours of relative Mach number at the operating conditions near peak efficiency and near stall. At the operating condition near peak efficiency, due to the weakened shock of the optimized blades, the regions of low dynamic head are significantly reduced for the optimized blades. At the operating condition near stall, the leakage flow spreads out and occupies almost the whole blade pitch downstream of the shock. Compared with the reference blade, the single-point optimization makes little change to the shock/tip-leakage interaction. However, it is clear that the multipoint optimization effectively improved the shock/tip-leakage interaction near stall, leading to a reduced region of low dynamic head as shown in Fig. 11. The improved shock/tip-leakage interaction lends evidence for extended stall margin, although as mentioned earlier the present steady code is unable to confirm this directly.
6 Conclusion

A continuous adjoint method based on the Reynolds-averaged Navier–Stokes equation using the Spalart–Allmaras turbulence model is formulated and demonstrated for the aerodynamic design of a compressor rotor blade. Both single and multiple operating-point optimization problems are considered.

The proposed method is used to optimize the shape of the NASA Rotor 67 blade. A single-point optimization at the design operating point is able to raise the adiabatic efficiency of the blade row by 1.10% while maintaining the same mass flow rate and total pressure ratio near the peak efficiency point. The optimization significantly weakens the shock in the blade passage and replaced the strong single shock near the blade tip section by a double shock system. Examination of the tip leakage flow also shows reduced shock/tip-leakage interaction at the design point. This single-point optimization also increased the adiabatic efficiency in a wide operating range away from the design point compared to the reference blade, but suffered from noticeable reduction of total pressure ratio for mass flow rate below 98% range. In order to obtain optimized blade shape good for a broad range of operation, a three-point optimization is performed based on near peak-efficiency, near-choke, and near-stall operating conditions. Although the blade shape obtained by the three-point optimization offers a slightly smaller gain of peak efficiency compared to the single-point design optimization, it offered significantly higher adiabatic efficiency and suffered no decrease in the total pressure ratio over a wide operating range beyond the peak efficiency point. The shock/tip-leakage interaction is slightly improved at near-stall condition for the multipoint optimized blade, which indicates potential extension of stall margin.

Acknowledgment

Luo J. would like to thank the National Natural Science Foundation of China (Grant No. 51206003) and the National Science Foundation for Postdoctoral Scientists of China (Grant No. 2012M510267, 2013T60035) for supporting the research work.

Nomenclature

- \( \mathbf{A}_i \): Jacobian matrices, \( \mathbf{A}_i = \partial f_i \partial \mathbf{w} \)
- \( \mathbf{B} \): boundaries of computational domain
- \( \mathbf{B}_i \): source matrix, \( \mathbf{B}_i = \partial \mathbf{S} \partial \mathbf{w} \)
- \( \mathbf{f}, \mathbf{f}_i \): inviscid and viscous fluxes
- \( \mathbf{n}_i \): unit normal vector in the computational domain
- \( \rho_0, \rho \): references of pressure and density
- \( \mathbf{R} \): flow governing equation
- \( s \): entropy, \( s = c_p \ln(p/\rho_0) - c_p \ln(p/\rho_0) \)
- \( S \): source term
- \( s_{\text{gen}} \): entropy production per unit mass flow rate
- \( \psi \): absolute velocity components
- \( \mathbf{w} \): conservative flow variables, \( \mathbf{w} = (\rho, \mathbf{u}, \mathbf{v}, \rho_0, \mathbf{E})^T \)
- \( \beta \): flow turning
- \( \eta \): adiabatic efficiency
- \( \theta \): total temperature ratio
- \( \lambda \): weight of penalty function
- \( \sigma \): ratio of mass flow rate, \( \sigma = \mathbf{m}/\mathbf{m}_0 \)
- \( \Psi \): adjoint variables, \( \Psi = \{\mathbf{\psi}_1, \mathbf{\psi}_2, \mathbf{\psi}_3, \mathbf{\psi}_4\}^T \)
- \( \Omega \): rotation velocity

References
