INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN ENGINEERING, VOL. 22, 363–375 (1986)

INVERSE DESIGN OF MULTIHOLED INTERNALLY COOLED TURBINE BLADES

STEPHEN R. KENNON† AND GEORGE S. DULIKRAVICH‡

Texas Institute for Computational Mechanics, Department of Aerospace Engineering and Engineering Mechanics, The University of Texas at Austin, Texas, U.S.A.

SUMMARY

A methodology is described for the inverse design and/or analysis of coolant flow passage shapes in multiholed internally cooled turbine blades. The user of this technique may specify the temperature (or heat flux) distribution along the aerofoil outer surface. In addition, the temperature on the surface of each of interior coolant flow passages (holes) may be specified. The numerical solution of the outer hot gas flow field determines the remaining unspecified aerofoil outer surface quantity—surface heat flux if temperature was originally specified or vice versa. The position and shape of each turbine blade coolant hole is then found iteratively by solving the heat conduction problem within the solid portions of the blade. This solution procedure involves satisfying the dual Dirichlet and Neumann specified boundary conditions of temperature and heat flux on the outer boundary of the aerofoil. The inner hole geometry is then modified using an optimization procedure in such a way as to minimize the error in satisfying the specified Dirichlet temperature boundary condition on the surface of each of the evolving interior holes. Results are shown for single-hole and double-hole configurations that have analytic solutions and for a realistic turbine blade design problem.

INTRODUCTION

The need for highly efficient turbomachinery has become clear in the past decade due to rising fuel costs. At the same time, the costs of large-scale testing of new designs have also escalated, making preliminary design using computational methods more attractive.

Due to its inherent complexity, the design of internally cooled turbine aerofoils has traditionally been accomplished using various approximate and empirical techniques. The shapes and locations of internal (coolant) flow passages are determined from one-dimensional analysis and experience gathered from expensive testing. As a consequence, the turbine aerofoil designer has no direct control of the detailed aerofoil temperature distribution.

This paper describes an entirely new concept for the inverse design and analysis of multiholed internally cooled turbine aerofoils. In particular, this work is an extension of a method[1,2] developed by the authors for designing internally cooled turbine aerofoils that have a single coolant flow passage (hole) to the case of more than one hole. This inverse design technique allows the aerofoil designer to specify the desired temperature or heat flux at each point on the turbine aerofoil outer surface, using whatever rational criteria he chooses (thermal stress considerations, coolant flow availability, effects on the outer flow field, etc.). Thus, the use of this technique allows the designer almost complete control over the temperature field within the

† Graduate Research Assistant.
‡ Assistant Professor.

0029-5981/86/040363-13$06.50

solid portions of the hollow aerofoil. Potential savings from using this design method include the possibility of allowing higher turbine inlet temperatures and therefore higher stage efficiency, lower coolant flow rates (implying less bleed-air-induced compressor losses), and an overall increase in the engine's efficiency and reliability (burn-through and thermal stress problems are alleviated). It should be noted that existing internally cooled turbine aerofoil cascades can also be analysed using this technique, and can possibly be redesigned for better performance.

THE GLOBAL INVERSE DESIGN CONCEPT

The first step in this inverse design procedure is the specification of either the temperature or the heat flux distribution on the blade outer surface. The hot gas flow field exterior to the blade can then be calculated numerically using, say, an inviscid flow model[3,4] coupled with an appropriate boundary layer model.[5] Instead of using these viscous/inviscid coupling concepts the Navier–Stokes equations[6] could be used. Regardless of what flow solver is used for the determination of the exterior flow, the end result of the flow field calculation will be the remaining blade outer surface quantity (temperature or heat flux) that was not initially specified. Alternatively, experimental data in the form of a Nusselt number distribution could be used with a specified surface temperature distribution to obtain the surface heat flux distribution. Thus, the dual boundary conditions of temperature and heat flux on the blade outer surface are obtained.

In addition, the designer must specify the number of holes that the blade is to have, an initial guess for the shapes and positions of these holes, and the desired temperature on the surface of each of the holes.

The steady flow of heat within the solid portions of the blade is found by solving Laplace's equation while imposing the dual boundary conditions of temperature and heat flux on the outer surface of the blade. Since both Dirichlet and Neumann conditions are imposed at each point of the blade outer surface, the problem would be ill-posed if we explicitly enforced any thermal boundary conditions on the inner hole boundaries. However, the desired temperatures on the inner holes are achieved implicitly by allowing the shapes and locations of the coolant flow passage holes to iteratively evolve.

The iterative design procedure begins with the specification of the initial blade and hole geometry. Then Laplace's equation is solved with the aforementioned boundary conditions. From this solution, the temperatures at the control points of each of the panels comprising the inner hole geometry are calculated. These calculated temperatures will in general be different from the desired ones specified by the blade designer. Thus, an optimization procedure is used to modify the inner hole geometry to minimize the difference between the calculated and specified temperatures. Throughout the optimization procedure, the dual outer surface thermal boundary conditions are rigidly enforced. When the temperature difference is below a certain tolerance, the optimization process is halted and the heat flux on the surface of each inner hole is calculated.

This heat flux alone then serves as an input for the solution of the coolant fluid flow field. The solution of the coolant flow field using the heat flux boundary condition (but not the temperature boundary condition) will then produce temperature distributions on the blade inner holes that are generally different from the temperatures that were just calculated. Thus, the newly found temperature distributions, or alternatively some combination of the new temperatures and the old ones, are used as the new thermal boundary conditions on the surfaces of each inner hole. These new temperatures are then used with the optimization procedure to find the new shapes and positions of the holes that satisfy the new temperature boundary conditions.

This global iterative process is repeated until a point is reached at which the blade inner surface temperature distributions calculated from the coolant flow field solution match the temperatures calculated from Laplace's equation governing the temperature field inside the blade.

This paper will only address the two-dimensional heat transfer inverse design problem in blade cross-sections, although it must be noted that this is not a limitation of the technique.

COOLANT FLOW PASSAGE SHAPE DETERMINATION

Given the temperature and heat flux distributions on the outer surface, Ω_s, of a given turbine blade cross-section (Figure 1), the problem is to find the correct shapes and positions of the inner holes that satisfy all the specified boundary conditions. The three boundary conditions to be satisfied are: (1) the aerofoil outer surface temperature, T_s, (2) the aerofoil outer surface heat flux q_s, and (3) the temperature on the surface of each inner hole, $Tc_m (1 \leqslant m \leqslant N_h)$. The solid portions of the hollow aerofoil are assumed to be homogeneous and made of a material with a constant coefficient of heat conduction, λ. The heat flow is assumed to be steady so that the temperature field in the material satisfies Laplace's equation

$$\lambda \nabla^2 \phi = 0 \tag{1}$$

where ϕ is the temperature in the blade material.

Panel technique

From potential theory, it is known that a solution to Laplace's equation can be found by superimposing a series of fundamental solutions. This is the principle that panel or singularity distribution methods are based on.

Laplace's equation is solved using a first-order panel method. The reasons for using a panel method, as opposed to a finite difference or finite element method, are that panel methods are inherently very efficient, and they are able to handle changes in the shape of the solution domain boundary without the problem of regeneration of a computational grid. In addition, the method

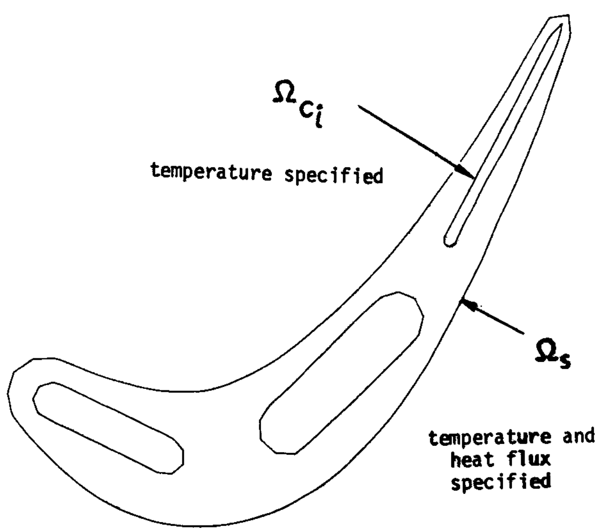

Figure 1. Geometry and boundary conditions[9]

is well understood, is relatively easy to implement and has been widely tested in a variety of problems requiring solution of Laplace's equation.

The panel technique described here utilizes straight panels with a constant heat source strength distribution to represent the heat flow between the outer contour Ω_s, and the inner holes, Ωc_m, $1 \leqslant m \leqslant N_h$ (Figure 2). The temperature induced at a point z_0 by a panel of strength k is given by

$$\phi(z_0) = k \int \ln|z_0 - z| \, ds \tag{2}$$

where the integration is performed along the panel. The outer (fixed) contour, Ω_s, and the inner (floating) contours, Ωc_m, are discretized with an equal number of straight panels (line segments). That is, the total number of panels on all the inner holes must add up to the number of panels that discretize the outer contour Ω_s

$$\sum_{m=1}^{Nh} (N_m) = N \tag{3}$$

where N_m is the number of panels on the mth inner hole. If we number the hole panels consecutively such that the index j is given by

$$j(m, i) = \sum_{l=1}^{m} (l-1)N_l + i \qquad (1 \leqslant j \leqslant N) \tag{4}$$

where $1 \leqslant i \leqslant N_m$ and $1 \leqslant m \leqslant N_h$, then we denote the strength of the jth inner panel (ith panel on the mth hole) by kc_j (or $kc_{i,m}$). The strength of the jth panel on the outer contour Ω_s is then denoted by ks_j. This gives

$$\phi(z_0) = \sum_{j=1}^{N} \left\{ ks_j \int \ln|z_0 - z| \, ds_j + kc_j \int \ln|z_0 - z| \, ds_j \right\} \tag{5}$$

for the temperature ϕ at any point z_0 in the complex z-plane.

From Fourier's heat conduction law the heat flux in the direction n at any point z_0 is

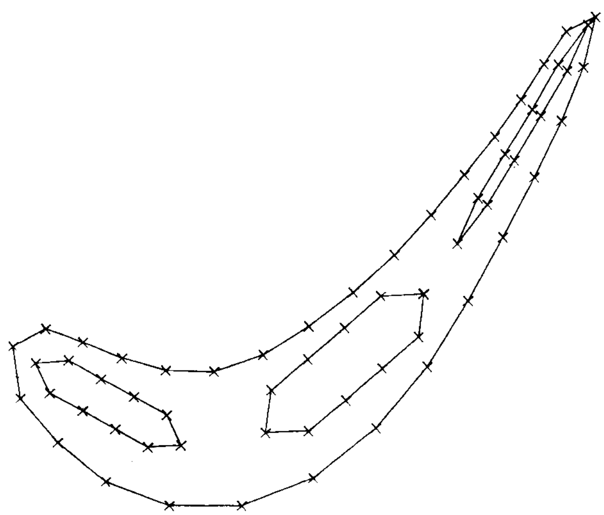

Figure 2. Inner and outer contours discretized with panels

given by

$$-\lambda \frac{\partial}{\partial n} \phi(z_0) = -\lambda \sum_{j=1}^{N} \frac{\partial}{\partial n} \left\{ ks_j \int \ln|z_0 - z| ds_j + kc_j \int \ln|z_0 - z| ds_j \right\} \tag{6}$$

The aerofoil outer surface thermal boundary conditions of temperature and heat flux are enforced at the control points of the panels, zs_j, defined as the average of the panel end points, zs_j^* and zs_{j+1}^* (note that $zs_{N+1}^* = zs_1^*$). Since there are N control points, and we enforce two boundary conditions at each control point, we need a total of $2N$ unknowns to satisfy the required boundary conditions. From equation (3) we see that there are a total of $2N$ panels on the outer and inner contours; thus, there are $2N$ unknowns of the problem, namely the strengths of the source panels ks_j and kc_j, $1 \leqslant j \leqslant N$. Satisfying the $2N$ boundary conditions produces four $N \times N$ influence coefficient matrices that multiply the source strengths as

$$[Is_{ij}]\{ks_j\} + [Ic_{ij}]\{kc_j\} = \{Ts_i\} \tag{7}$$

$$[Js_{ij}]\{ks_j\} + [Jc_{ij}]\{kc_j\} = \{-qs_i/\lambda\} \tag{8}$$

This can also be written as the $2N \times 2N$ partitioned matrix:

$$\begin{bmatrix} Is_{ij} & Ic_{ij} \\ Js_{ij} & Jc_{ij} \end{bmatrix} \begin{Bmatrix} kc_j \\ kc_j \end{Bmatrix} = \begin{Bmatrix} Ts_i \\ -qs_i/\lambda \end{Bmatrix} \tag{9}$$

Here, Is_{ij} and Ic_{ij} denote the influence of the ith outer or inner panel on the temperature at the control point of the jth outer panel, and Js_{ij} and Jc_{ij} denote the influence of the ith outer or inner panel on the heat flux at the control point of the jth outer panel. Solving equation (9) gives the values of ks_j and kc_j which satisfy the dual thermal boundary conditions on the outer contour, Ω_s. The temperature at the control point of each inner hole panel is then calculated using equation (2).

Inner contour correction

These calculated temperatures will in general be different from the specified temperatures Tc_m, $1 \leqslant m \leqslant N_h$. The next step of the global inverse design procedure is thus the minimization of the temperature difference using an optimization procedure.

Define the temperature difference at the jth inner panel control point by

$$e_j \equiv \phi(\bar{z}c_j) - Tc_j \tag{10}$$

and the global error function E by

$$E(x_n^*, y_n^*) \equiv \sum_{j=1}^{N} (e_j)^2 = e^{\mathrm{T}} e \tag{11}$$

The reason for using the global error function E is that setting the e_j's to zero would give N equations but in $2N$ unknowns, namely the x and y co-ordinates of the end-points of the hole panels. Thus, we use the formulation

$$\begin{gathered} \text{find } (x^*, y^*) \in R^N \text{ such that} \\ E(x^*, y^*) \text{ is minimized} \end{gathered} \tag{12}$$

to render the problem determinate. The gradient of E is denoted by $g(x^*, y^*)$ and is a vector of length $2N$ (N partials with respect to x^* and N partials with respect to y^*). The iterative optimization procedure is given by

$$X^{*n+1} = X^{*n} - \alpha^n d^n \tag{13}$$

where X^* is the vector of length $2N$ formed from the two vectors of length N, x^* and y^* (the co-ordinates of the panel end-points), d is the so-called 'search direction vector' (of length $2N$) and α is the scalar 'line search' parameter. The vector d is determined using either Cauchy's method of steepest descent ($d = g$) or by the Davidon–Fletcher–Powell 'variable metric' method[7] ($d = Hg$; H is an updated approximation to the inverse of the second partial derivative, or Hessian, matrix $\partial^2 E/\partial X_i^* \partial X_j^*$, $1 \leq i \leq 2N$, $1 \leq j \leq 2N$). The line search parameter α is determined using Powell's quadratic search algorithm.[7]

The gradient g is calculated analytically. This calculation involves the inversion of the $2N \times 2N$ matrix of equation (9); however, the LU decomposition of the matrix is stored from the solution for the panel strengths and thus can be used to determine the gradient vector g with negligible extra computation.

The iterative procedure is concluded when the maximum error in satisfying the temperature boundary condition on the inner contour is below a certain tolerance.

RESULTS

A computer program was developed to implement the inverse design procedure. Input to the program includes the outer contour co-ordinates, temperature and heat flux distributions, and the desired inner hole temperatures. In addition, an initial guess for the position and shape of each inner hole must be specified in the input data.

Results for three cases are presented: a simple eccentrical bore pipe, a cylinder with two circular holes, and a realistic turbine aerofoil. The first two test cases have analytic solutions that result from fundamental solutions to Laplace's equation.

The eccentrical bore pipe is analogous to the buried cable problem and has an analytic solution[8] in which the isotherms are non-concentric circles. The case we considered had the following characteristics: outer circle radius = 10, and temperature = 20; inner circle temperature = 10. The initial guess for the inner hole shape was specified as a circle of radius 5 positioned concentrically within the fixed outer circle. The initial guess is shown in Figure 3(a), and the iteration sequence is shown in Figures 3(b–f). The final converged shape is not quite circular due to the fact not enough panels were used in the solution procedure.

The second analytic test case consisted of a unit strength heat source and a unit strength heat sink placed a distance of two units apart. This produces isotherms which are non-concentric circles about the singularities. Choosing a temperature on the first hole of -1 and on the second hole of $+1$ gives two circles of radii = 0·724 with their centres a distance of 2·626 units apart. The outer contour was specified to be a circle of radius 3. The temperature and heat flux distributions on the outer contour were calculated using the analytic solution and formed the input to the computer program. To form the initial guess for this two-hole problem, the circles were perturbed from their analytic positions, as shown in Figure 4(a). The corrections to the positions of the hole panel end-points were averaged during the first part of the iteration. This averaging does not allow the shape of the holes to change, only the position. When the average gradient became small, the averaging was turned off. Thus the iterative procedure moves the holes into the general area of the correct solution without allowing the shapes of the holes to change, and then the iterative procedure finds the correct shape of the holes. The iteration sequence is shown in Figures 4(b–f).

The final test case is a turbine aerofoil with three coolant holes (Figure 5a) which was used

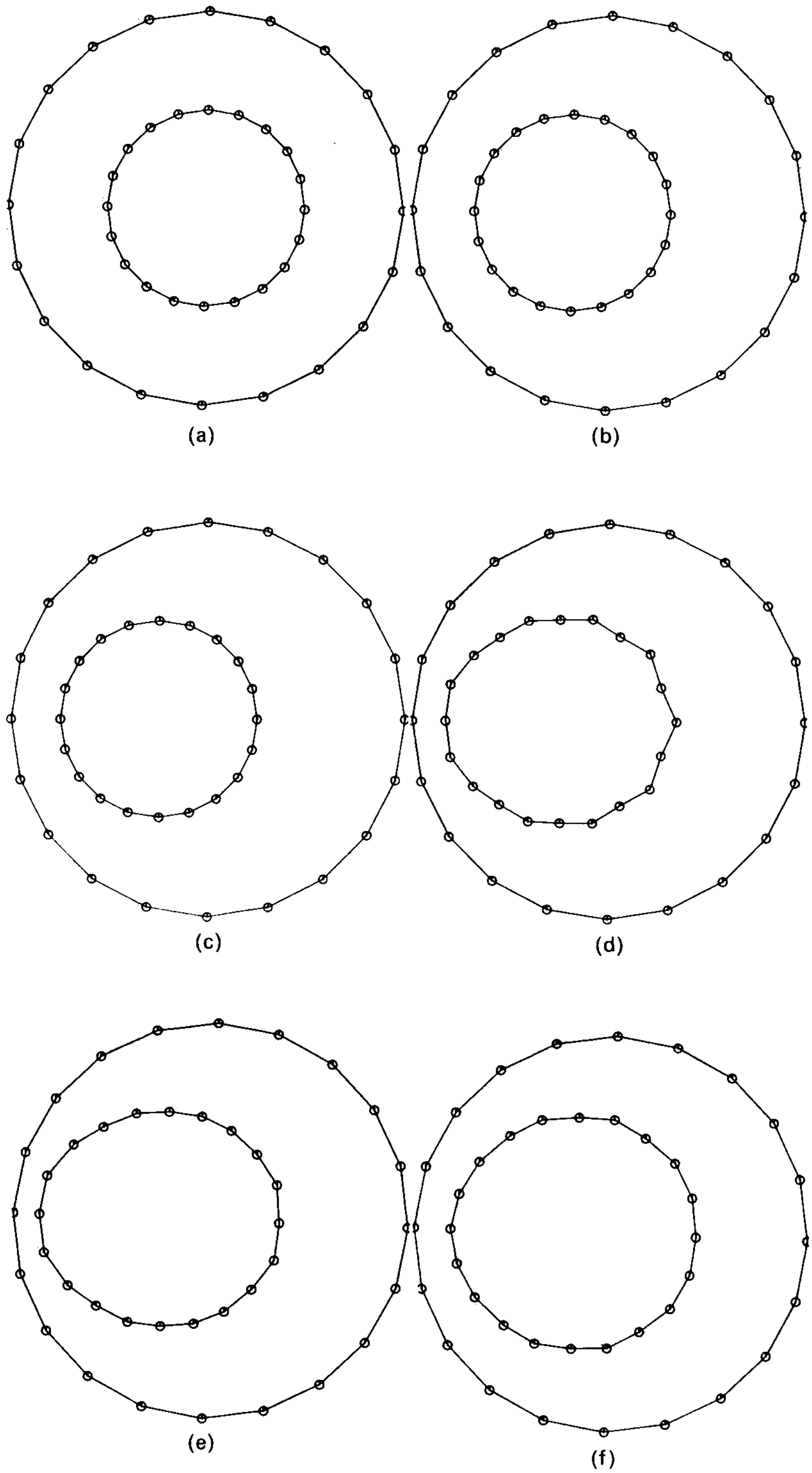

Figure 3. (a) Test case 1—initial guess; (b) solution after 3 iterations; (c) solution after 5 iterations; (d) solution after 6 iterations; (e) solution after 7 iterations; (f) solution after 9 iterations (converged solution)

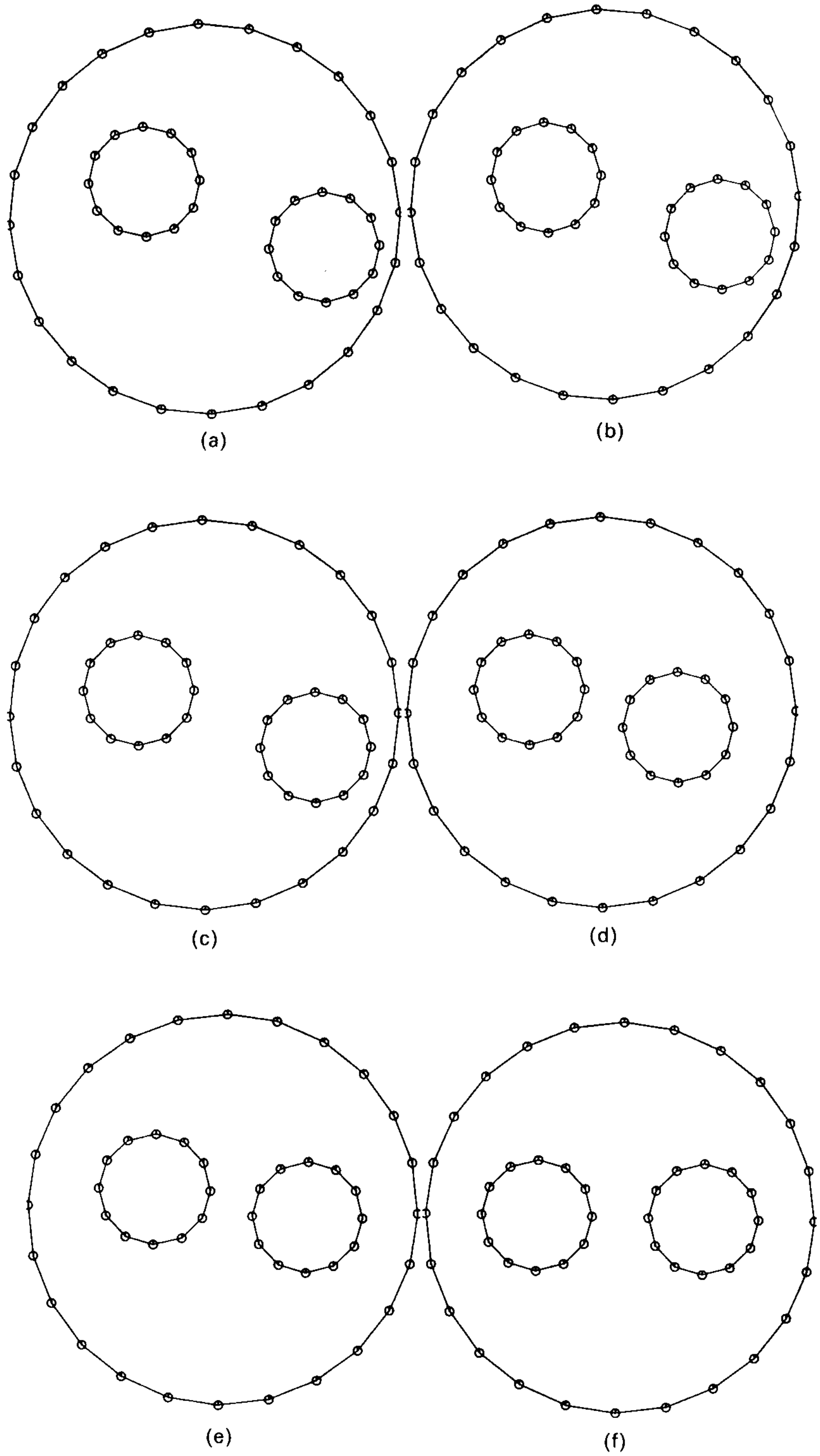

Figure 4. (a) Test case 2—initial guess; (b) solution after 1 iteration; (c) solution after 3 iterations; (d) solution after 4 iterations; (e) solution after 5 iterations; (f) solution after 11 iterations (converged solution)

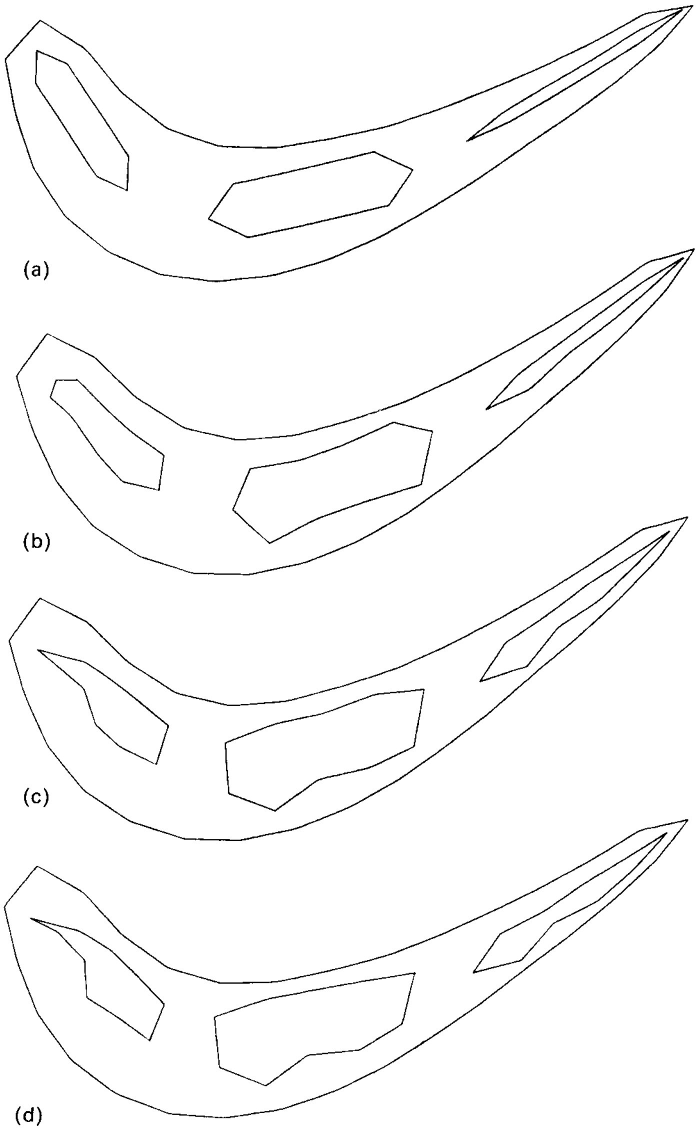

Figure 5. Iteration sequence for turbine design case 1: (a) initial configuration; (b) solution after 6 iterations; (c) solution after 14 iterations; (d) solution after 18 iterations

by Nakata and Araki[9] as a test case for their boundary element analysis method. The geometry, temperature and heat flux data as given in their paper formed the input and initial guess for the design problem. Our program was placed in analysis mode and the resulting temperatures on the surface of the coolant holes were calculated as shown in Table I.

Next, we specified that the desired design temperatures were to be as shown in Table I. The program was then run in design mode with gradient averaging and using the new specified coolant hole surface temperatures. The iteration sequence for the turbine blade design problem is shown in Figures 5(a–d). Clearly, we asked too much of the procedure since the hole nearest

Table I. Turbine blade design: case 1

Hole number	Calculated		Specified
	Minimum surface temperature (°C)	Maximum surface temperature (°C)	design surface temperature (°C)
1	561·2	729·2	625·0
2	581·7	727·2	625·0
3	430·4	814·3	625·0

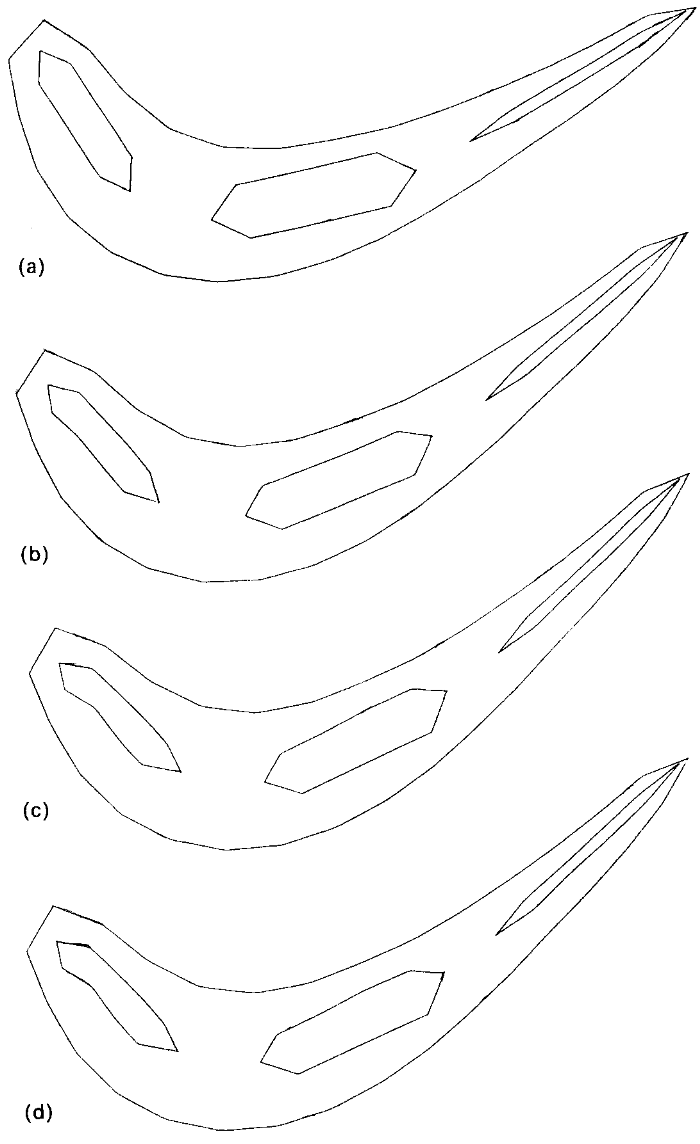

Figure 6. Iteration sequence for turbine design case 2: (a) initial configuration; (b) solution after 6 iterations; (c) solution after 14 iterations; (d) converged solution (15 iterations)

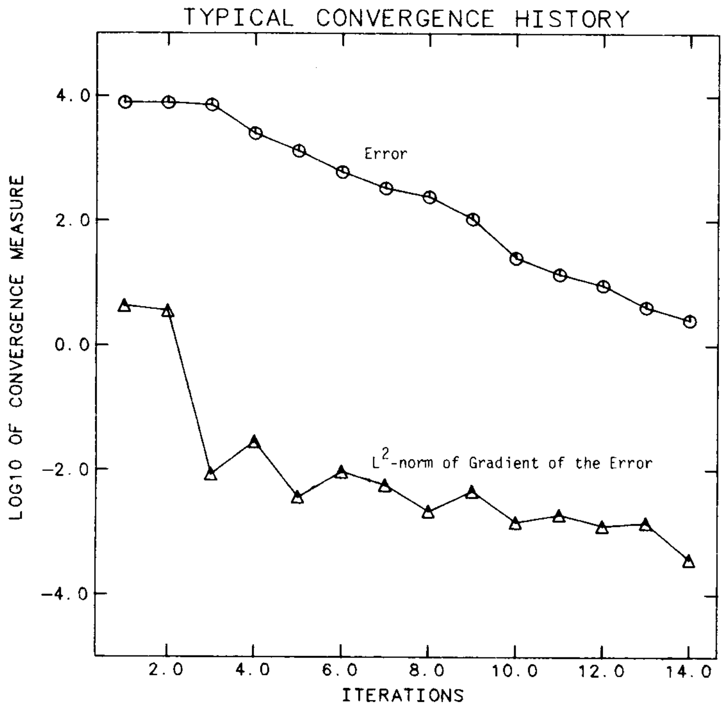

Figure 7. Convergence history for turbine design case 2

the leading edge began to overlap itself. Thus, we relaxed the design requirements and specified that the hole temperatures were to be 20°C less than those calculated for the original configuration and the blade was redesigned using our program (Figures 6a–d). In this case, we were able to achieve a physically realizable coolant flow passage design that had blade hole surface temperatures lower than the initial configuration.

A typical convergence history for the turbine blade design problem is shown in Figure 7.

FUTURE RESEARCH

There are many possibilities for extending the present method to more complex problems with increased accuracy. A major extension of the method would be to make it fully three-dimensional by using quadrilateral panels in place of the outer blade surface (from hub to tip) and the unknown inner holes.[11] Transpiration cooled configurations could be designed and analysed also since the coolant and heat flow through a porous medium has been shown by Siegel[12] to be derivable from a potential. One of the most intriguing applications for this design method would be to attempt an optimum design of an almost entirely shock-free cooled turbine cascade outer flow field.[13] This quasi-shock-free design procedure would not entail any modification of the overall shape of the turbine blade. The shock-free character of the flow field could possibly be maintained over a range of operating inlet temperatures, Mach numbers and stagger angles by varying the coolant flow rate and temperature and keeping the blade geometry fixed. In addition, the method could be used to delay the onset of boundary-layer transition.[14] Most importantly, future research should involve experimental verification of the outlined inverse design procedure.

SUMMARY

A procedure has been developed for the efficient design and analysis of coolant flow passage shapes in internally cooled configurations. The method is particularly applicable to turbine blade inverse design, but can also be used for the design of other configurations with non-adiabatic boundaries such as missile cone tips and internal combustion engine cylinders. The designer is able to specify and fix the temperature or the heat flux at the turbine blade outer surface and to specify the desired temperature at the coolant/blade interface surfaces. The result of the design procedure is the shape and position of each of the interior coolant flow passage contours (holes) that satisfy all three of the above thermal boundary conditions. When coupled with an appropriate flow solver, the method provides the gas turbine engine designer with an efficient and accurate tool for the design of coolant flow passages. The method is not limited to cascade design, but can be used for the design of coolant flow passages in fully three-dimensional blades.

APPENDIX I: NOMENCLATURE

d = Optimization search direction vector.
e = Vector of errors in satisfying the hole boundary conditions.
E = Global error function = L^2-norm of e.
g = Gradient of $E = \nabla E$.
H = Hessian of $E = \partial^2 E/\partial X_i \partial X_j$.
i = Complex number $i = \sqrt{-1}$ (also used as integer index).
I = Temperature influence coefficient.
J = Heat flux influence coefficient.
k = Strength of a source or sink.
N = Number of panels on the inner and outer contours.
N_h = Number of holes.
N_i = Number of panels on the ith hole ($1 \leqslant i \leqslant N_h$).
q = Specified heat flux.
R^N = the N-dimensional real vector space.
s = Arc length along a panel.
T = Specified temperature.
x = x-Co-ordinate of the z-plane.
X = Vector containing both x and y co-ordinates of points in z-plane.
y = y-Co-ordinate of the z-plane.
z = Co-ordinate in the complex plane $z = x + iy$.
$\bar{z}$ = Panel control point co-ordinate.
z^* = Panel end-point co-ordinate.
i, j, m, p = Indices.

Greek symbols

α = Line search parameter in optimization procedure.
ε = $a \in b \rightarrow a$ is an element of b.
λ = Coefficient of heat conduction.
Ω = Closed contour in the z-plane.
ϕ = Temperature.

Subscripts

s = Airfoil surface.

c = Coolant/aerofoil interface surface (inner hole boundary).
0 = Particular point.

Superscripts
n = Iteration number.
T = Transpose of a vector or matrix.
$^{-}$ = Panel control point.
$*$ = Panel end-point.

REFERENCES

1. S. R. Kennon and G. S. Dulikravich, 'The inverse design of internally cooled turbine blades', *ASME Paper 84–GT–7*, presented at 29th Int. Gas Turbine Conf., Amsterdam, The Netherlands, 4–7 June 1984.
2. S. R. Kennon, 'Novel approaches to grid generation, inverse design, and acceleration of iterative schemes', *Master's thesis*, Dept. of Aerospace Engineering and Engineering Mechanics, Univ. of Texas at Austin (May, 1984).
3. D. S. Dulikravich, 'CAS2D-Fortran program for nonrotating, blade-to-blade, steady, potential transonic cascade flows', *NASA TP* 1705 (1980).
4. Ron Ho Ni, 'A multiple-grid scheme for solving the Euler equations', *A.I.A.A., J.*, **20** (11), 1565–1570 (1982).
5. R. E. Gaughler, 'Some modifications to, and operational experience with, the two dimensional, finite-difference, boundary-layer code, STAN5', *ASME Paper* 81–*GT*–89 (1981).
6. R. W. MacCormack, 'A numerical method for solving the equations of compressible viscous flow', *A.I.A.A.* 81–0110*R* (1981).
7. G. Dahlquist and A. Bjorck, *Numerical Methods*, Prentice-Hall, Englewood Cliffs, New Jersey, 1974.
8. E. R. G. Eckert and R. M. Drake Jr., *Analysis of Heat and Mass Transfer*, McGraw-Hill, N.Y. 1972.
9. Y. Nakata and T. Araki, 'Application of boundary element method to heat transfer coefficient measurements around a gas turbine blade', to be presented at ASME Winter Ann. Mtg., New Orleans, 8–13 Dec. 1984.
10. E. McFarland, 'Solution of plane cascade flow using improved surface singularity method', *J. Eng. Power*, **104,** (July 1982).
11. J. L. Hess and A. M. O. Smith, 'Calculation of potential flow about arbitrary bodies', in *Progress in Aeronautical Sciences*, **8,** Pergamon Press, New York, 1966.
12. R. Siegel and A. Snyder, 'Heat transfer in cooled porous region with curved boundary', *J. Heat Transfer*, **103,** 765–771 (Nov. 1981).
13. G. S. Dulikravich and H. Sobieczky, 'Shockless design and analysis of transonic cascade shapes', *A.I.A.A. J.* **20** (11), 1572–1578 (Nov. 1982).
14. R. Kahawita and R. Meroney, 'The influence of heating on the stability of laminar boundary layers along concave curved walls', *ASME Paper 77–APM–4* (1977).