THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
345 E. 47 St., New York, N.Y. 10017

85-GT-118

Inverse Design of Coolant Flow Passage Shapes with Partially Fixed Internal Geometries

STEPHEN R. KENNON
Graduate Research Assistant

GEORGE S. DULIKRAVICH
Assistant Professor

TICOM—Texas Institute for Computational Mechanics
Department of Aerospace Engineering and Engineering Mechanics
The University of Texas at Austin

ABSTRACT

A method is described for the inverse design of complex coolant flow passage shapes in internally cooled turbine blades. This method is a refinement and extension of a method developed by the authors for designing a single coolant hole in turbine blades. The new method allows the turbine designer to specify the number of holes the turbine blade is to have. In addition, the turbine designer may specify that certain portions of the interior coolant flow passage geometry are to remain fixed (eg. struts, surface coolant ejection channels, etc.). Like the original design method, the designer must specify the outer blade surface temperature and heat flux distribution and the desired interior coolant flow passage surface temperature distributions. This solution procedure involves satisfying the dual Dirichlet and Neumann specified boundary conditions of temperature and heat flux on the outer boundary of the airfoil while iteratively modifying the shapes of the coolant flow passages using a least squares optimization procedure that minimizes the error in satisfying the specified Dirichlet temperature boundary condition on the surface of each of the evolving interior holes. Portions of the inner geometry that are specified to be fixed are not modified. A first order panel method is used to solve Laplace's equation for the steady heat conduction within the solid portions of the hollow blade, making the inverse design procedure very efficient and applicable to realistic geometries. Results are presented for a realistic turbine blade design problem.

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 = ∇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 i'th hole ($1 \le i \le N_h$)
q = specified heat flux
R^N = the N-dimensional real vector space

Presented at the Gas Turbine Conference and Exhibit

s = arc length along a panel

T = specified temperature

x = x-coordinate of the z-plane

X = vector containing both x and y coordinates of points in the z-plane

y = y-coordinate of the z-plane

z = coordinate in the complex plane $z=x+iy$

$\bar{z}$ = panel control point coordinate

z^* = panel end point coordinate

i,j,m,p = indices

Greek Symbols

α = line search parameter in optimization procedure

$\beta = \begin{cases} 1 & \text{if end point is allowed to move} \\ 0 & \text{if end point is fixed} \end{cases}$

ϵ = $a \epsilon b \rightarrow$ 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/airfoil interface surface (inner hole boundary)

o = particular point

Superscripts

n = iteration number

T = transpose of a vector or matrix

$\bar{}$ = panel control point

$*$ = panel end point

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 airfoils has traditionally been accomplished using various approximate and empirical techniques. At the present time, the shapes and locations of internal (coolant) flow passages are determined from a complex system of interconnected computer codes capable of analyzing coolant flow, hot gas flow, heat transfer, aeroelasticity, weight etc. Nevertheless, this represents a design method that is based on repetitive analysis and modifications. An inverse design on the other hand offers substantial savings and exact fulfilment of specific design objectives.

This paper describes a method for the inverse design and analysis of complex, multiholed coolant flow passages taking into account only the steady heat conduction in the solid walls of internally cooled turbine airfoils. In particular, this work is an extension of a method[1,2,3] developed by the authors for the inverse design of a single coolant flow passage shape in an internally cooled turbine cascade airfoil. The present method can be used for more realistic problems where portions of the coolant flow passage contours can be fixed to represent struts and surface coolant ejection channels. This inverse design technique allows the airfoil designer to specify the desired temperature or heat flux at each point on the turbine airfoil 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 solid portions of the hollow airfoil. Potential savings from using this design method include the possibility of achieving 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 airfoil cascades can also be analyzed 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 only one thermal boundary condition on the airfoil outer suface. This condition can be either the temperature or the heat flux distribution. The hot gas flow field exterior to the blade can be calculated numerically using, say, an inviscid flow model[4] coupled with an appropriate boundary layer model.[5, 6] Instead of using these viscous/inviscid coupling concepts the Navier-Stokes equations[7] 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 airfoil 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 airfoil outer surface can be obtained and are assumed to be given. Consequently, our inverse design procedure focuses solely on the steady state heat conduction problem in the solid portions of the airfoil.

The designer must specify the number of holes that the cooled airfoil is to have, an initial guess for the shapes and positions of these holes, the portions of the holes that are to remain fixed and the desired temperature on the surface of each of the holes.

The steady conduction of heat within the solid portions of the airfoil is found by solving Laplace's equation while imposing the dual boundary conditions of temperature and heat flux on the outer surface of the airfoil. Since both Dirichlet and Neumann conditions are imposed at each point of the airfoil 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 guessed shapes and locations of the coolant flow passage contours 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 using a first order surface panel integral approach. From this solution, the temperatures at the control points of each of the panels comprising the coolant fluid passage 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 on the surface of each coolant flow passage. Throughout the optimization procedure, the dual outer surface thermal boundary conditions are rigidly enforced and the airfoil outer surface shape is kept fixed. When the temperature difference is below a certain tolerance the optimization process is halted and the heat flux on the surface of each coolant flow passage is calculated.

This heat flux alone then serves as an input for the solution of the coolant fluid flow field which is not the objective of discussion of this paper. 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 airfoil 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 airfoil inner surface temperature distributions calculated from the coolant flow field solution match the temperatures calculated from Laplace's equation governing the steady state heat conduction inside the solid portions of the cooled airfoil.

This paper only addresses the two-dimensional heat conduction 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

Keeping in mind that we are addressing only the heat conduction problem inside the solid portions of the cooled airfoil, we will assume that the airfoil outer surface shape is to be kept fixed.

Given the temperature and heat flux distributions on the outer surface, Ω_s, of a given turbine blade cross-section (fig. 1), the problem is then to find the correct shapes and positions of the inner holes that satisfy all the specified thermal boundary conditions. The three boundary conditions to be satisfied are: 1) the airfoil outer surface temperature, T_s, 2) the airfoil outer surface heat flux, q_s, and 3) the temperature on the surface of each inner hole, Tc_m $(1 \le m \le N_h)$. The solid portions of the hollow airfoil 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

$$\lambda \nabla^2 \phi = 0 \qquad (1)$$

where ϕ is the local temperature in the solid 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 basic principle that panel or singularity distribution integral 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 field method, are that panel methods are inherently very efficient, and they are especially suited for discretization of geometrically irregular domains. In addition, the method is well understood, it 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, $1 \le m \le N_h$ (fig. 2). The temperature induced at a point z_o by a panel of strength k is given by

$$\phi(z_o) = k \int \ell n |z_o - z| ds \quad (2)$$

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. 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

$$\Sigma_{m=1}^{Nh} (N_m) = N \quad (3)$$

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

$$j(m,i) = \Sigma_{l=1}^{m} (l-1)N_l + i \qquad (1 \le j \le N) \quad (4)$$

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

$$\phi(z_o) = \Sigma_{j=1}^{N} \{ks_j \int \ell n|z_o - z| ds_j + kc_j \int \ell n|z_o - z| ds_j\} \quad (5)$$

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

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

$$-\lambda \frac{\partial}{\partial n}\phi(z_o) = \quad (6)$$
$$= -\lambda \, \Sigma_{j=1}^{N} \frac{\partial}{\partial n}\{ks_j \int \ell n|z_p - z| ds_j + kc_j \int \ell n|z_p - z| ds_j\}$$

The airfoil 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}. 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 eq. 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, $1 \le j \le N$. Satisfying the 2N boundary conditions produces four N X N influence coefficient matrices that multiply the source strengths as

$$[Is_{ij}]\{ks_j\} + [Ic_{ij}]\{kc_j\} = \{Ts_i\} \quad (7)$$
$$[Js_{ij}]\{ks_j\} + [Jc_{ij}]\{kc_j\} = \{-qs_i/\lambda\} \quad (8)$$

This can also be written as the 2N X 2N partitioned matrix:

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

Here, Is_{ij} and Ic_{ij} denote the influence of the i'th outer or inner panel on the temperature at the control point of the j'th outer panel, and Js_{ij} and Jc_{ij} denote the influence of the i'th outer or inner panel on the heat flux at the control point of the j'th 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 eq. 2.

Inner Contour Correction

These calculated temperatures on the coolant flow passage walls will in general be different from the specified temperatures Tc_m, $1 \le m \le N_h$. 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 j'th inner panel control point by

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

and the global error function E by

$$E(x^*_n, y^*_n) \equiv \Sigma_{j=1}^{N} (e_j)^2 = e^T e \qquad (11)$$

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 coordinates of the end-points of the hole panels.

Thus, we use the formulation

find $(x^*, y^*) \in R^N$ such that

$$E(x^*, y^*) \text{ is minimized} \qquad (12)$$

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

$$X^{*n+1}_i = X^{*n}_i - \alpha^n d^n_i \beta(i) \qquad (13)$$

where X^*_i is the vector of length 2N formed from the two vectors of length N, x^*_i and y^*_i (the coordinates of the panel end-points), d_i is the so-called 'search direction vector' (of length 2N), α is the scalar 'line search' parameter, and $\beta(i)$ is the index function that specifies which panel end-points are to remain fixed. The vector d is determined using either Cauchy's method of steepest descent (d = g) or by the Davidon-Fletcher-Powell 'variable metric' method[8] (d = Hg; H is an updated approximation to the inverse of the second partial derivative, or Hessian, matrix $\partial^2 E/\partial X^*_i \partial X^*_j$, $1 \le i \le 2N$, $1 \le j \le 2N$). The line search parameter α is determined using Powell's quadratic search algorithm[8].

The gradient g is calculated analytically. This calculation involves the solution of a 2N X 2N linear system with the same coefficient 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 specified tolerance.

RESULTS

A computer program was developed to implement the inverse design procedure. Input to the program includes the outer contour coordinates, 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 and the index function $\beta(i)$, $1 \le i \le N$, must be specified in the input data.

Results are presented for the case of an actual turbine airfoil design problem. The design problem we considered is a turbine airfoil with three coolant holes (fig. 1) which was used 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. The inner hole geometry was modified to include two struts and a trailing edge coolant ejection channel as shown in fig. 2.

The holes were discetized with 10 panels on each hole resulting in a total of 30 panels discretizing the blade surface (fig. 2). Our program was placed in analysis mode and the resulting temperatures on the surface of each coolant hole were calculated. Figure 3 shows a plot of constant temperature contours for the initial configuration. Next, we specified that the desired design temperatures were to be 20°C less than the calculated ones on each inner hole panel. The program was then run in design mode using the new specified coolant hole surface temperatures. After 29 iterations, the solution converged to the shape shown in fig. 4. We also ran the program with temperatures that were specified to be 40°C less than the calculated ones. The resulting converged solution (after 34 iterations) for the second design case is shown in fig. 5, while fig. 6 shows the optimization procedure convergence history. From these figures one can conclude that physically realistic blade designs can be efficiently obtained that produce lower hole surface temperatures while maintaining the structural integrity of the fixed struts and coolant ejection channel.

6. FUTURE RESEARCH

There are many possibilities for extending the present method to more complex problems with increased accuracy, namely using a higher order panel technique.[10] 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 analyzed 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.[4] 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 possibly be used to delay the onset of boundary-layer transition[13]. Most importantly, future research should involve experimental verification of the outlined inverse design procedure.

7. SUMMARY

A procedure has been developed for the efficient design and analysis of complex coolant flow passage shapes in internally cooled configurations. The method is particularly applicable to cooled turbine airfoil cascade 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 airfoil outer surface and to specify the desired temperature at the coolant/blade interface surfaces. It is also possible to fix portions of the geometry of the coolant flow passages to represent struts and surface coolant ejection channels. The result of the design procedure is the precise shape and location 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 and stress analysis code, 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 airfoil cascade design, but can be used for the design of coolant flow passages in fully three-dimensional blades.

8. REFERENCES

1. Kennon, S. R., "Novel Approaches to Grid Generation, Inverse Design, and Acceleration of Iterative Schemes," Master's Thesis, Department of Aerospace Engineering and Engineering Mechanics, University of Texas at Austin, May, 1984.

2. Kennon, S. R. and G. S. Dulikravich, "The Inverse Design of Internally Cooled Turbine Blades," ASME Paper 84-GT-7, presented at 29th International Gas Turbine Conference, Amsterdam, The Netherlands, June 4-7, 1984.

3. Kennon, S. R. and G. S. Dulikravich, "Inverse Design of Multiholed Internally Cooled Turbine Blades," proceedings of the International Conference on Inverse Design in Engineering Sciences (ICIDES), pp. 217-240, G. S. Dulikravich ed., University of Texas at Austin, October 17-18, 1984.

4. Dulikravich, G. S. and H. Sobieczky, "Shockless Design and Analysis of Transonic Cascade Shapes," AIAA Journal, Vol. 20, No. 11, November 1982, pp. 1572-1578.

5. Gaughler, R. E., "Some Modifications to, and Operational Experience with, the Two-Dimensional, Finite-Difference, Boundary-Layer Code, STAN5," ASME Paper 81-GT-89, 1981.

6. Olling, C. R. and G. S. Dulikravich, "Transonic Cascade Flow Analysis Using Viscous/Inviscid Coupling Concepts," AIAA paper 84-2159 presented at AIAA 2nd Applied Aerodynamics Conference, August 21-23, 1984, Seattle, WA.

7. MacCormack, R. W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," AIAA 81-0110R, 1981.

8. Dahlquist, G. and A. Bjorck. Numerical Methods. Prentice-Hall inc., Englewood Cliffs, N. J., 1974.

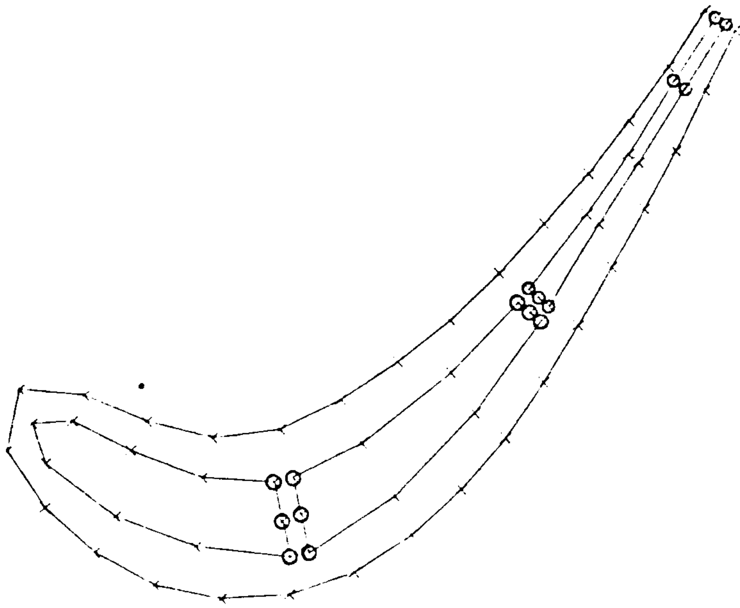
9. Nakata, Y. and T. Araki, "Application of Boundary Element Method to Heat Transfer Coefficient Measurements Around a Gas Turbine Blade," to be presented at ASME Winter Annual Meeting, New Orleans, December 8-13, 1984.

10. McFarland, E., "Solution of Plane Cascade Flow Using Improved Surface Singularity Method," Journal of Engineering for Power, Vol. 104, July, 1982.

11. Hess, J. L. and A. M. O. Smith, "Calculation of Potential Flow About Arbitrary Bodies," Progress in Aeronautical Sciences, Vol. 8, Pergamon Press, New York, 1966.

12. Siegel, R. and A. Snyder, "Heat Transfer in Cooled Porous Region with Curved Boundary," Journal of Heat Transfer, Vol. 103, November 1981, pp. 765-771.

13. Kahawita, R., and R. Meroney, "The Influence of Heating on the Stability of Laminar Boundary Layers Along Concave Curved Walls," ASME Paper 77-APM-4, 1977.



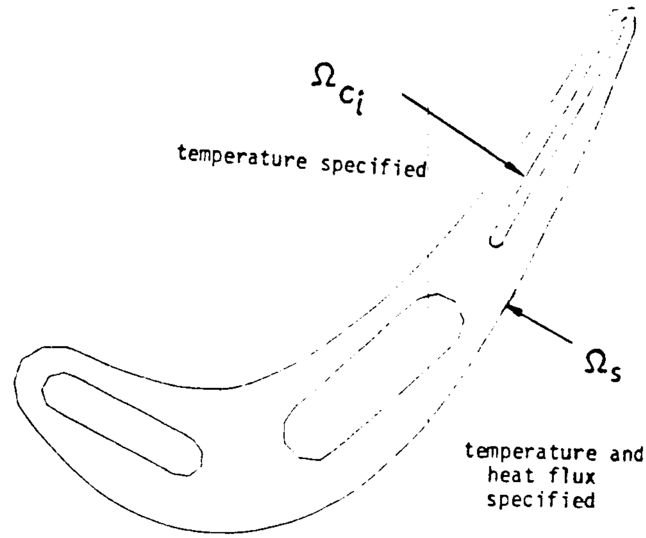


Fig. 1 Geometry and Boundary Conditions[9]

Fig. 2 Inner and Outer Contours Discretized with Panels
(○ denotes fixed end points)

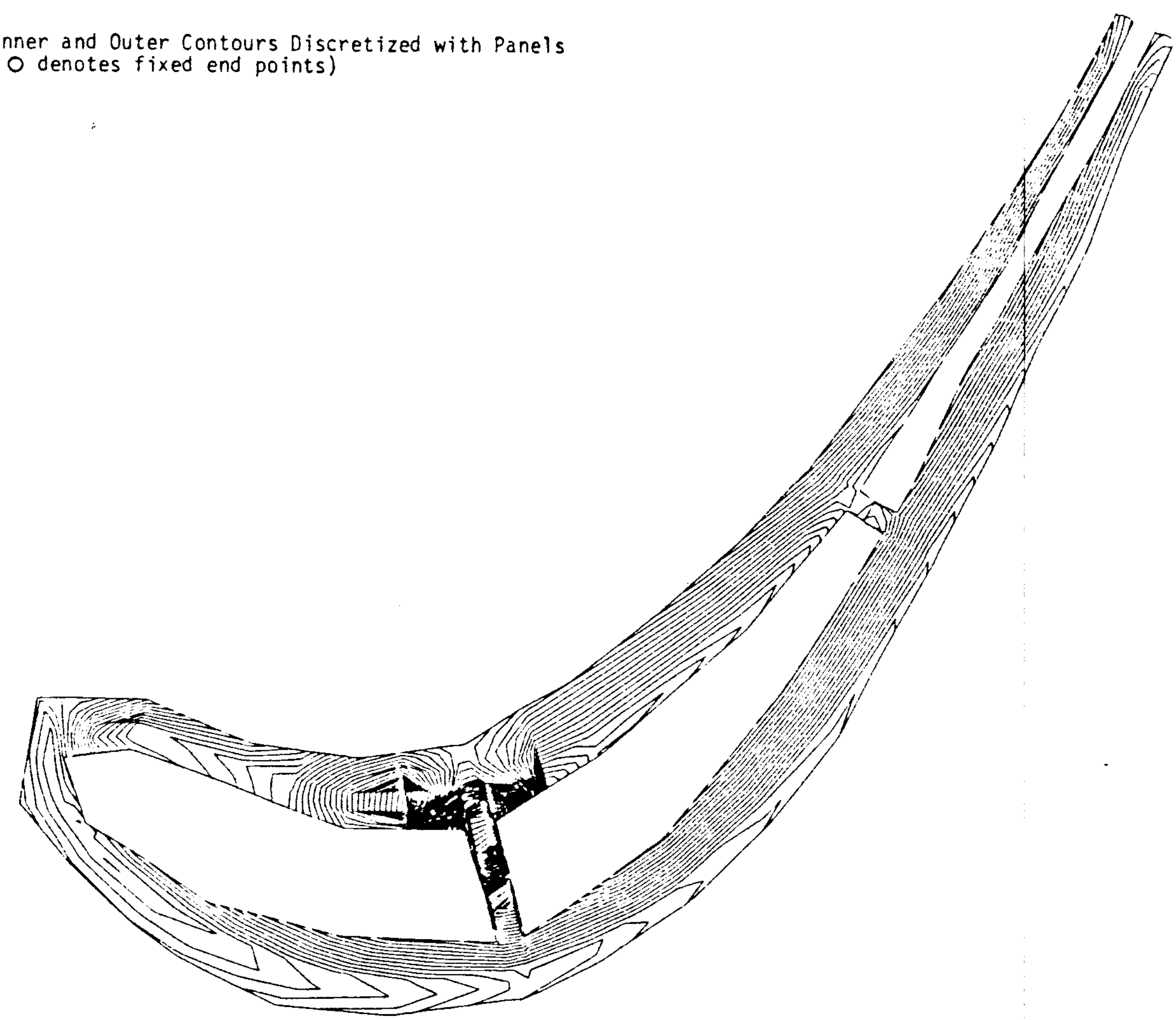


Fig. 3 Constant Temperature Contours for Initial Configuration

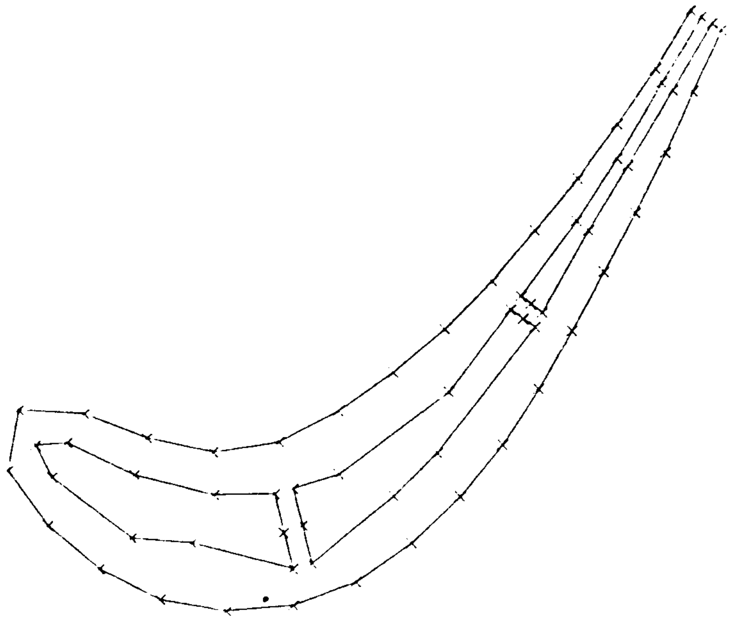


Fig. 4 Turbine Design for Case # 1

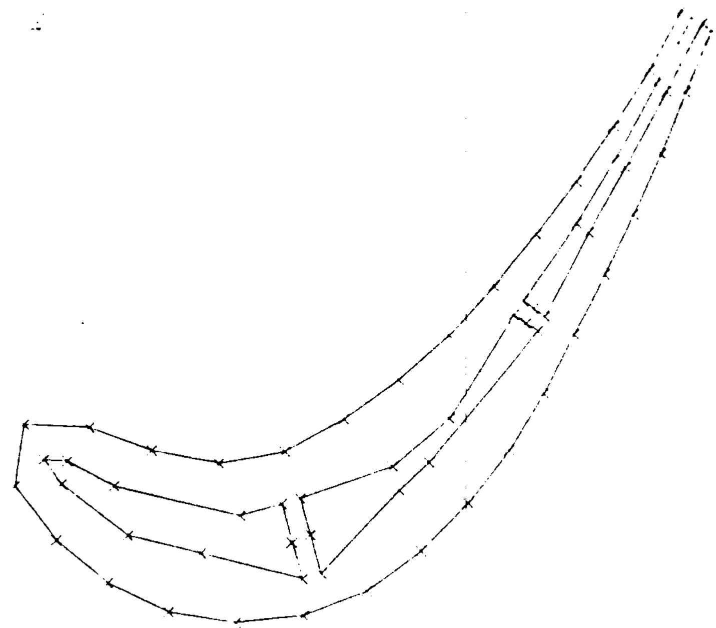


Fig. 5 Turbine Design for Case # 2

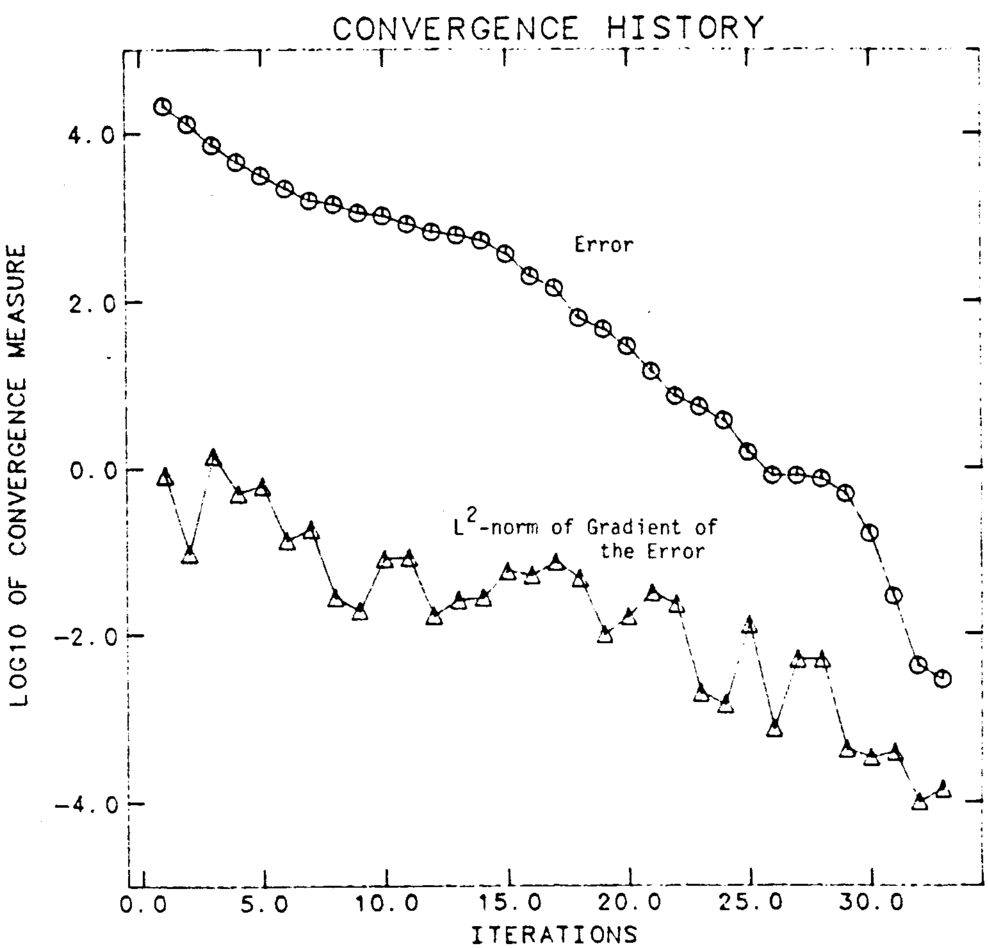


Fig. 6 Convergence History for Second Turbine Design Case