Proceedings of XIII International Symposium on Airbreathing Engines - XIII ISABE (editor: F. Billig)
Chattanooga, TN, September 8-12, 1997.

Aero-Thermal Analysis and Optimization of Internally Cooled Turbine Airfoils

T. J. Martin[1] and G. S. Dulikravich[2]
Department of Aerospace Engineering, 233 Hammond Building
The Pennsylvania State University, University Park, PA 16802

Abstract

This paper presents the results of a multi-disciplinary study on the computer automated design of internally cooled turbine airfoils. A constrained hybrid optimization algorithm was developed and used to iteratively modify the virtual geometric model of the entire turbine airfoil profile, thicknesses of one or more thermal barrier coatings, thickness distribution of one or more coolant flow passages, and internal strut shapes and locations. Two optimization objectives were formulated to account for the aerodynamic and thermal optimization goals of the design process. The behaviors of three distinct aerodynamic objective functions and three distinct thermal objective functions were studied. The computer simulation software included a heat conduction analysis using the Boundary Element Method (BEM) within the multi-component turbine airfoil, an aerodynamic analysis of the gas flow within the turbine cascade using a quasi-three-dimensional Navier-Stokes finite differencing solver, and an iterative coupling of the two codes for the conjugate heat transfer prediction of airfoil surface temperatures and heat fluxes.

Introduction

The greatest efficiency improvements in gas turbine engines are likely to be accomplished by increasing the temperature of the combustion.. Today's combustor exit temperatures are above the melting point of the nickel-based turbine blade alloys, while the turbine blades are subject to extremely large and ever-increasing aerodynamic and centrifugal loads. If high temperature combustion products are generated, the heat transfer into the turbine blade should be minimized in order to reduce the coolant mass flow rate, but this increases the temperature on the surface of the blade. At the same time, the maximum temperature within the turbine blade should be kept below a certain value to avoid thermal creep, melting, and oxidation problems. In the absence of new advances in high-temperature materials, the way to accommodate these extreme conditions is through the improvement of internal cooling techniques.

When needed in the initial turbine stages, cooling air can be made to impinge on the internal cooling passage surface to enhance convection at the leading edge. This type of impingement cooling demands large leading edge diameters which can subsequently increase aerodynamic losses. Surface film cooling has provided significant improvements to turbine cooling schemes, but they also result in greater aerodynamic losses primarily due to a reduction in the boundary layer momentum. In addition, film cooling creates additional unwanted nitrus oxide emissions at high combustion temperatures. Although larger cooling flow rates, film cooling schemes, and complex internal structures with miniature heat exchangers provide better cooling, the required compressor air bleed reduces the overall performance of the engine.

The walls of the turbine blade between the hot gas and coolant flow passages are made thin so as to accommodate the required coolant flow rate, but thick enough to sustain the high centrifugal and aerodynamic loads. High thermal strains can also harm the durability of the blades, so it is desirable to keep as uniform a temperature field as possible. Heat conduction in the turbine blade, as well as the thermo-elastic deformation of the blade, in turn, affects the aerodynamics, especially in the transonic case where the position of the shock is an important determinant of the aerodynamic performance. Ultimately, turbine designers are faced with a very complex aero-thermo-elastic problem with several composite goals.

Constrained Hybrid Optimization

The aerodynamic and thermal shape optimization strategy that is being presented in this paper will utilize a constrained numerical optimization algorithm. A scalar objective function will provide a mathematical model of the goal and this function is a set of design variables that includes all the parameters needed to generate the turbine blade geometry. The optimization algorithm iteratively modifies the aerodynamic shape and coolant flow passage configurations, calls for an aerodynamic and/or thermal analyses of the new configuration, quantifies the efficiency of the new design, and then provides a new set of design variables which will be more efficient (one than minimizes the

---

[1] NASA Graduate Student Fellow. Student member AIAA.
[2] Associate Professor. Associate Fellow AIAA.

optimization problem can be mathematically stated as follows.

$$\text{minimize} \quad F(\vec{V}) \tag{1}$$

$$\text{where} \quad \vec{V} = \{V_1 \quad V_2 \quad \cdots \quad V_{N\,var}\} \tag{1a}$$

$$\text{subject to:} \quad \vec{V}_{min} < \vec{V} < \vec{V}_{max} \tag{1b}$$

$$g_j(\vec{V}) \leq 0 \qquad j = 1, N_{con} \tag{1c}$$

$$\left|h_j(\vec{V})\right| \leq \varepsilon \qquad j = 1, N_{eqc} \tag{1d}$$

Here, $F(\vec{V})$ is the objective function, $\vec{V}$ is the vector of N_{var} design variables, $\vec{V}_{min}$ is the vector of lower limit constraints, $\vec{V}_{max}$ is the vector of upper limit constraints, g_j is the set of N_{con} inequality constraint functions, h_j is the set of N_{eqc} equality constraint functions, and ε is a very small number called the constraint thickness.

The solution of an optimization problem is the set of design variables for which the objective function takes on its global minimum value. In most inverse design and optimization problems, the design variables are restricted to one or more constraint functions. There are two different types of constraint functions that could be placed upon these variables; inequality and equality constraints. Inequality constraints are commonly required for geometric regularity. The engineer may also require them for structural integrity, durability, or to provide some minimum or maximum weight, volume, area, or length. Equality constraints confine the variables to a line in the design variable space. Examples of equality constraints might include the throat area of a gas path or a required load. A set of design variables that does not violate any constraints is said to be feasible, while one that violates one or more constraints is infeasible. If a constraint is on the verge of being violated (within the constraint thickness, ε), it is said to be an active constraint.

A constrained evolutionary hybrid optimization scheme has been developed in the FORTRAN programming language. This algorithm creates sequential populations of feasible designs that evolve with each new optimization cycle by minimizing the objective function associated with various members of the population. There are many optimization algorithms in the open literature and various techniques have been shown to provide faster convergence over others depending upon the size and shape of the mathematical design space, the nature of the constraints, and where it is during the optimization process.

Our hybrid algorithm incorporates four of the most popular optimization approaches; the Davidon-Fletcher-Powell (DFP) gradient search method (Vanderplaats, 1984), a genetic algorithm (GA) (Goldberg, 1989), the Nelder-Mead (NM) simplex method (Nelder & Mead, 1965), and simulated annealing (SA) (Press et al., 1986). Each technique separately provides a unique approach to optimization with varying degrees of convergence, reliability, and robustness at different cycles of an iterative optimization procedure.

Our new evolutionary hybrid scheme handles the existence of constraints in three ways; Rosen's projection method, a feasible search, and random design generation. Rosen's projection method (Haftka and Gurdal, 1992) provide corrects search directions and guides the descent direction tangent to active constraint boundaries. If newly generated designs violate one or more constraints, a feasible search automatically restores any designs to feasibility via the minimization of the active global constraint functions (Foster and Dulikravich, 1997). If at any time this constraint minimization fails, a random design is generated about a desirable and feasible one until a new feasible design is reached. The random design generator uses a Gaussian-probability density distribution that is allowed to contract or expand depending upon the variance in the population.

The feasible set of design variables from the current optimization cycle are saved into an array called the population matrix. This population is updated every iteration with new designs and ranked according to the value of the objective function. The optimization problem is completed when the best design in the population is equivalent to a target design, when the variation in the population's design characteristics becomes very small, or whenever every optimizer fails to improve the upon the objective.

Development of the Virtual Turbine Blade Geometry from the Design Variables

The development of the realistic multi-holed coolant passage geometry and coated turbine airfoil section will now be discussed. The set of optimization design variables completely defines the shape of the turbine airfoil section, thermal barrier coating, coolant flow passages, and struts.

First, the turbine airfoil section shape is developed from four distances and six angles that fix two circles and four tangency points near the leading and trailing edges on the airfoil contour. The airfoil profile shape is completed by connecting two conical curves between these leading and trailing edge tangency points. One curve is drawn on each pressure and suction side in order to connect the two leading and trailing edge circles. These ten parameters, which are illustrated in Figure 1, are also listed below.

from point to point on the airfoil surface, allowing for greater thermal barrier protection in areas which are hotter, but thin in other areas that are more sensitive to delamination problems.

The first step in the development of the multiple coolant flow passages is the description of the cooling wall thickness function. The wall thickness function's ordinate, W(s), determines the thickness of the wall between the hot gas and coolant fluid. The abscissa follows counter-clockwise along the metal/coating interface, s, all the way around the airfoil, from trailing edge to trailing edge and forming a close loop. It is shown as the dotted line in Figure 2. W(s) is also described by a piecewise-continuous β-spline curve that varies in the direction normal to the metal/coating interface to a distance controlled by one design variable per β-spline control vertex.

Next came the specification of the locations of the strut centerlines. The x-coordinate of the intersections of the strut centerlines with the outer turbine airfoil curve were defined as x_{Ssi} and x_{Spi}, for the suction and pressure sides, respectively. The index i varies from one to the number of struts, N_{strut}. The range over which each strut could vary was $\pm \Delta x_{Si}$. The number of coolant flow passages in the turbine blade was specified, $N_{hole} = N_{strut} + 1$. In addition to the coordinate of the struts, the strut thickness, t_{Si}, and a filleting exponent on either the trailing or leading edge sides, e_{Sti} and e_{Sli}, respectively, are used to complete the geometric modeling of each strut.

Since the number of holes may also be a design variable, a problem would arise when computing the gradient of the objective function. Additional holes cannot easily be added since the searching capabilities of the optimizer could never include all the possible combinations of hole location and geometry. Therefore, a simple and straightforward approach was utilized by starting the optimization with a large number of holes (limited by computer memory and computational efficiency) and then reducing the number of holes during the optimization procedure (Dulikravich and Kosovic, 1992). Any particular coolant passage was eliminated whenever the hole was reduced to such a small size that it had a negligible effect on the outer boundary heat flux.

Constraints

Several constraints must be incorporated into the aero-thermal optimization for realistic engineering of the turbine blade and coolant passage geometries. By category, these are listed as follows; structural integrity, durability, cooling scheme, and aerodynamic. The primary constraint on the designer is the integrity of the turbine blade, as the turbine blade coring must be able to carry the required loads without plastic deformation. Structurally speaking, the maximum stress within the blade should not exceed the yield stress. Thus, an analysis of the three-dimensional stress field subject to the centrifugal and aerodynamic loads on the turbine blade would be an ideal inclusion into the constrained optimization algorithm. This will be left for another publication on aero-thermo-elastic optimization. Instead, the following constraint functions have been incorporated into our turbine blade optimization algorithm to account for the structural integrity.

Chord Angle. The twisting of the three-dimensional turbine blade requires that the angle between the chord line and the principal axis not be too large. This is to ensure that the torquing stresses at the attachment to the platform do not exceed the yield.

Wall Thickness A minimum wall thickness will be enforced so that the turbine blade can accommodate the centrifugal load.

Moment of Inertia. The moment of inertia about each principal axis must be kept large enough to avoid the blade's vibrational modes.

High temperature loads can cause creep and corrosion. Therefore, turbine blade durability can be improved significantly by ensuring that the maximum temperature within the turbine blade does not exceed its melting or oxidation temperature. In our thermal optimization code, thermal durability has been reflected in the objective function and not as a separate constraint function.

Turbine blade coring has become very expensive because it relies upon exotic manufacturing techniques such as laser drilling and electro-chemical machining (ECM). In order to limit manufacturing costs, as well as to guarantee feasibility, geometric constraints were applied to the sizes, shapes and locations of the turbine coolant flow passages. The following constraint functions are common to turbine design groups.

Coolant Passage Cross-Sectional Area. The smallest cross-sectional areas of the coolant flow passages are limited by the compressor air bleed pressure. Thus, the cooling holes must be large enough to allow the required coolant mass flow rate. This constraint also reflects the stiffness of the airfoil and its tendency to untwist.

Leading Edge Radius. The leading edge circle or ellipse size is limited by its cooling requirements. The leading edge radius must be large enough to achieve the required amount of shower-head or impingement cooling air. In addition, larger elliptical leading edges are less sensitive to off-design incidence.

Trailing Edge Radius. To reduce aerodynamic losses, the thinnest possible trailing edge is desirable. However, castability, trailing edge life, and pedestal or cutback cooling schemes require some finite thickness.

x_{te} difference in the sectional x-coordinate between the centers of the leading and trailing edge circles

y_{le} difference in the sectional y-coordinate between the centers of the leading and trailing edges circles

r_{le} leading edge radius

α_{le} inlet air angle between the inlet gas path and the leading edge point

βp_{le} metal angle between the inlet gas path and the leading edge circle for the tangency point on the pressure side

βs_{le} metal angle between the inlet gas path and the leading edge circle for the tangency point on the suction side

r_{te} trailing edge radius

α_{te} exit air angle between the exit gas path and the trailing edge point

βp_{te} metal angle between the exit gas path and the trailing edge circle for the tangency point on the pressure side

βs_{te} metal angle between the exit gas path and the trailing edge circle for the tangency point on the suction side.

This set of 10 design variables will be referred to as the set of conic section parameters.

Although nearly any combination of these design variables will produce an airfoil shape, the design space is only partially captured, and the control which an engineer desires is limited. Therefore, an additional capability has been added to improve the aerodynamic shape optimization. In cases where greater control and flexibility of the design space is desirable, the turbine airfoil generated by conic section parameters provides only a first reference airfoil shape. This shape, being a series of (x,y) coordinates forming a closed loop curve which is clustered towards the leading and trailing edges, can then be slightly modified using a piecewise second-order continuous β-spline curve (Barsky 1988).

The β-splined curve has as its ordinate the distance normal to the reference airfoil, B(s), versus the abscissa of the airfoil contour following coordinate, s. This spline generates a new distance function that is allowed to vary in the direction normal to the boundary of the reference airfoil shape to a distance controlled by one design variable per β-spline control vertex. Beta-spline vertices are an ordered sequence of points, generally not lying on the curve, but instead form a control polygon. The β-spline curve is also connected to itself at the leading or trailing edge to form a closed loop. The piecewise curve can be represented by the following series of cubic polynomials.

$$B(s) = \sum_{i=1}^{N_{vert}} G_i(s) \tag{2a}$$

where

$$G_i(s) = \sum_{k=-1}^{3} b_k(\beta_1, \beta_2, s) V_{i+k} \tag{2b}$$

Here, V_{i+k} is the (i+k)th control vertex coordinate and $b_k(\beta_1,\beta_2,s)$ are called the basis functions. Each basis function is a cubic polynomial. The shape parameter β_1 is referred to as the bias parameter which can produce clustering towards the end vertices. The second parameter β_2 is called the tension parameter and it should always be positive. For high values of β_2, the curve will be strongly pulled towards the control vertices and, in the limit as $\beta_2 \to \infty$, the β-spline will be identical to the control polygon.

Beta-splines are closely related to the more common v-splines, B-splines and Bernstein polynomials, but, unlike its relatives, the β-splines exhibit local control over the geometry. That is, its defining parameters affect only a portion of the overall curve without altering the rest of it. Local control of the shape is a desirable feature because any perturbation of the geometry should have a distinct effect on the objective function in an optimization.

The number of β-spline vertices was user-specified (N_{vert}), as well as the number of points of the piecewise continuous airfoil curve (N_{af}). For simplicity, N_{af} was kept the same as the number of points generated by the conic section format. Generally, the number of vertices can be much less than the number of (x,y) coordinates defining the airfoil curve.

Figure 2 illustrates the complete geometric modeling of the coolant flow passages and thermal barrier coating. One or more coatings can be easily added to the multi-component turbine airfoil by generating additional airfoil shapes with a specified thickness inward to the outer airfoil surface. The thickness of the coating can also be fixed, or it can be allowed to change during the optimization process. Coating thicknesses were described as a wall thickness function, C(s), versus the airfoil contour following coordinate, s. The interface between the coating and the metal blade is drawn as a thin solid line in Figure 2. The coating thickness distribution could be constant around the perimeter, or it can be described by a curve using the β-splines, which was discussed earlier. In the latter case, the thickness of the coating can vary

The trailing edge is normally enforced as a half circle with a fixed radius.

Finally, aerodynamic constraints may also be required. Usually, these can be reflected in the aerodynamic objective function, but two that do not require a CFD analysis are listed below.

Axial Chord. A specified axial chord must be maintained. It arises from the preliminary design of the turbine by the meanline and streamline analyses.

Throat Area. A proper hot gas flow rate through the turbine cascade is necessary for high Mach number flow regimes. If the required throat area, sometimes called the gage, is not met, the mass flow rate will not be synchronized with the compressor and a surge could occur.

Inverse Thermal Shape Design

During the past several years, Dulikravich and his research team of graduate students have developed a fully-automatic computational inverse shape design algorithm that helps a cooling systems designer to determine the proper numbers, sizes, shapes and locations of arbitrary coolant flow passages within internally cooled configurations. This is accomplished by over-specifying a desirable variation of heat flux over the external surface of the turbine blade in addition to the boundary conditions of a well-posed heat conduction problem. An optimization algorithm was then used to iteratively modify the shapes of the coolant passages in an attempt to achieve the least square sum of the difference between the user-specified and computed heat fluxes on the outer boundary. The physics was provided for by a steady-state, nonlinear heat conduction solver using the BEM.

The methodology has been successfully demonstrated on multi-holed two-dimensional turbine airfoils with thermal barrier coatings (Kennon & Dulikravich, 1985; Chiang & Dulikravich, 1986; Dulikravich & Kosovic, 1992, Dulikravich & Martin, 1994), single-holed three-dimensional turbine blades (Dulikravich & Martin, 1995), scramjet combustor struts (Dulikravich, 1988), and periodic rocket nozzle wall sections (Dulikravich & Martin, 1993). The BEM has powerful advantages over other numerical techniques in that it only requires discretization over the surface of the solid, it is faster, and it is more accurate than the other numerical methods having the same surface mesh.

Although the inverse thermal design problem provided a very fast and robust method of automatic coolant passage design, it lacked generality. Primarily, the engineer does not know in advance the temperature and heat flux distributions on the external turbine blade surface. Specifying these variations is similar to the idea of designating a target pressure distribution on an airfoil surface for inverse aerodynamic shape design. Many years of expensive test rigging, experimentation, and engineering experience have been spent trying to determine what should be the optimal pressure distribution on an airfoil to minimize just profile pressure losses. It would not be surprising that an equal endeavor for the heat flux distributions would be exigent. Hence, this paper is presenting a new and superlative method, a conjugate or aero-thermal turbine cooling shape optimization.

The aero-thermal optimization will be presented in three parts: (1) thermal shape optimization involving a heat conduction analysis in the multi-component turbine blade and using heat transfer coefficients at the solid/gas boundary, (2) aerodynamic shape optimization involving a steady, compressible, viscous solution to the Navier-Stokes equations in a turbine cascade and a constant wall temperature at the turbine blade wall, and (3) a conjugate thermal shape optimization involving an iterative coupling of the thermal and aerodynamic analyses at the solid/gas interface.

Thermal Shape Optimization

The temperature field within the multi-component turbine airfoil was computed using a two-dimensional steady-state BEM heat conduction algorithm. The following partial differential equation governs the conduction of heat within an inhomogeneous solid made up of one or more materials having constant material properties.

$$k_m \nabla^2 T = 0 \tag{3}$$

Here, T is the temperature and k_m is the coefficient of thermal conductivity in the mth domain.

In order to handle the multi-component turbine blade, the computational domain Ω must be subdivided into a number of subdomains, Ω_1, Ω_2, ..., Ω_M, each with a different thermal conductivity, $k_1, k_2, \ldots, k_M$. The solutions to the partial differential equation for each subdomain are combined using the compatibility relations on the interface boundaries. For instance, between the subdomains Ω_1 and Ω_2, the compatibility relations equate the temperatures and heat fluxes,

$$T_{I1} = T_{I2} = T_I \tag{4a}$$

$$-k_1\left(\frac{\partial T}{\partial n}\right)_{I1} = k_2\left(\frac{\partial T}{\partial n}\right)_{I2} = -k_1\left(\frac{\partial T}{\partial n}\right)_{I} \tag{4b}$$

where the subscript I refers to the interface boundary between the subdomains. This system of partial differential equations can be solved efficiently using the Boundary Element Method (Brebbia & Dominguez, 1989). Laplace's equation is cast into a weighted residual statement and integrated by parts twice to form a Boundary Integral Equation (BIE).

$$c(x)T(x)+\int_{\Gamma} q^{*}(x,\xi)T(\xi)d\Gamma(\xi)=\int_{\Gamma} u^{*}(x,\xi)q(\xi)d\Gamma(\xi) \quad (5)$$

The integration over the boundary Γ is with respect to the source coordinate system $\vec{\xi}$ which is independent of the real-space coordinate $\vec{x}$. Since one BIE is developed for each coordinates $\vec{x}$ on the boundary, the coefficient of the free term is physically the interior angle at that coordinate normalized by 2π. The flux was defined by $q = \partial T / \partial n$, and the two-dimensional fundamental Green's function solution is well known.

$$u^{*}=\frac{1}{2\pi}\ln\left(\frac{1}{r}\right) \quad (6a)$$

$$q^{*}=\frac{\partial u^{*}}{\partial n}=-\frac{\partial r/\partial n}{2\pi r} \quad (6b)$$

$$r=\left|\vec{x}-\vec{\xi}\right| \quad (6c)$$

The boundary Γ was discretized with linear boundary elements (panels), and the variations of T and q were also assumed to be linear over each element. The BEM does not require an internal mesh. The computation of temperatures in the interior are done in the post-processing phase.

Discretization of the boundary into N_{BE} isoparametric linear boundary elements results in a system of N_{BE} linear algebraic equations that can be solved after the boundary conditions are applied. In the thermal shape optimization problem, the outer surface of the turbine blade, which contacts the hot gas flow field, was given a Robin-type boundary condition.

$$q'' = -k_B\frac{\partial T}{\partial n} = h_{conv}\left(T - T_{inlet}\right) \quad (7)$$

Here, k_B is the coefficient of thermal conductivity in the solid blade region bounding the gas. It is either the thermal conductivity of the metal blade or the outer-most thermal-barrier coating, if one or more exists. In the above equation, h_{conv} is the convective heat transfer coefficient variation and T_{inlet} is the turbine inlet temperature. The variation of h_{conv} over the outer airfoil boundary can be taken from a computational flow field prediction, or it can be taken from experimental data.

For the lack of a better model, the internal boundaries of the sectional turbine shape, which are in contact with the coolant flow field, will also be fixed with a convective heat transfer boundary condition.

$$q'' = -k_M\frac{\partial T}{\partial n} = h_{cool,n}\left(T - T_{cool,n}\right) \quad (8)$$

Here, k_M is the coefficient of thermal conductivity of the metal blade, $h_{cool,n}$ is the convective heat transfer coefficient on the surface of the nth coolant passage, and $T_{cool,n}$ is the ambient fluid temperature in the nth passage.

Because a realistic thermal barrier coating is very, very thin, a special concern was addressed when integrating over boundary elements in close proximity to the real-space node at $\vec{x}$. On these boundary elements, the integrand can be nearly singular due to the nature of the fundamental solution. Therefore, a cubic variable transformation (Telles, 1987) was employed on all nearly singular panels. This approach was verified against a simple analytic solution made up of two concentric circular regions of different thermal conductivity. The temperatures and heat fluxes at the interface boundary were compared against those of the analytic solution. As the radii of the two circles approached each other, the error versus the analytic solution was substantially reduced by the transformation, and solutions with an error of less than 1% were produced even when the thickness of the outer shell was less than 0.00001 times the radius.

In order to come up with the most effective thermal shape optimization strategy, three different thermal objective functions have been utilized. The usefulness of each objective function have been tested and documented here.

(1) Domain Integrated Temperature Difference. The user specifies the desired or target temperature within the metal turbine blade, $\overline{T}$. The optimizer modifies the coolant passage scheme in order to minimize the squared difference between the local computed temperature and the desired temperature .

$$F(V_i) = \int_{\Omega} (T - \overline{T})^2 d\Omega \quad (9)$$

As this function is minimized, the temperature field within the turbine blade will approach the desired value, hopefully reducing temperature gradients and producing a more uniform temperature distribution. It has the added advantage of simultaneously minimizing weight.

(2) Net Heat Flux. The integrated heat flux into the turbine blade from the hot gas is minimized. This objective can concern itself directly with the reduction of the coolant flow requirements by reducing the heat transfer absorbed by the coolant fluid. The integration is carried out over the outer airfoil boundary.

$$F(V_i) = \int_{\Gamma_O} q'' d\Gamma \quad (10)$$

(3) Thermal Penalty. The thermal penalty function will include a boundary integrated temperature difference in order to achieve the most uniform temperature as possible on the coating/metal interface. This will be multiplied by a penalty so that the objective function will be adversely affected by temperatures above the limiting temperature, T_{melt}.

$$F(V_i) = \left\{ \int_{\Gamma} (T - \overline{T})^2 d\Gamma \right\} \left\{ \exp\left(\frac{T_{max}{}^2}{T_{melt}{}^2} \right) \right\} \quad (11)$$

In this equation, T_{max} is the maximum temperature computed by the BEM. For the most effective cooling scheme, $\overline{T}$ and T_{melt} could take on the same value.

A generic turbine blade was developed in order to demonstrate the capabilities of the thermal shape optimization algorithm. Table 1 gives some important parameters and boundary conditions of the present test case.

T_{inlet}	2000.0 K
k_I	1.0 W/m K
k_M	30.0 W/m K
C	0.0001 m
T_{cool} (all passages)	1000.0 K
h_{cool} (all passages)	300.0 W/m^2 K
$\overline{T}$	1200.0 K

Table 1. List of boundary conditions and constant geometric dependencies for the thermal optimization of a turbine blade having four coolant flow passages and a thermal barrier coating.

Figure 3 shows the initial guess to the turbine cooling scheme. The turbine blade was assumed to be protected by a thermal barrier coating of constant thickness around the perimeter although it is too thin to be seen in this figure. Also shown are the temperature contours for the initial guess in a flooded gray scale. These contours were computed using the BEM. The convective heat transfer coefficient, h_{conv}, on the outer airfoil surface was approximately taken from experimental measurements of a similar turbine blade. The variation of this heat transfer coefficient is shown in Figure 4. Convective heat transfer boundary conditions were also applied to the inner coolant flow passage surfaces.

The constrained hybrid optimization algorithm was used to minimize the domain integrated temperature objective function, with a target temperature of 1200 K. The optimization problem had 16 design variables; 2 x-coordinates per each of the 3 struts, 3 strut thicknesses, 2 filleting exponents per each of the 3 struts, and 1 constant coolant wall thickness. The outer turbine airfoil and the coating/metal interface boundaries were discretized with 60 linear isoparametric panels each. The four coolant flow passages required an additional 260 more panels. In this example, the thickness of the thermal barrier coating was kept fixed. The population size was kept fixed at 16 members, and the percentage chance of mutation in the genetic algorithm was specified to be 3% chance per breeding pair. The crossover process of the GA used a random linear interpolation between two population members selected by a roulette wheel.

The entire optimization procedure took almost 400 objective function evaluations (BEM heat conduction solutions). The convergence history of the objective function versus the number of objective function evaluations is shown in Figure 5. Of the 32 optimization cycles, the hybrid optimization algorithm required 19 cycles with the DFP gradient search technique, 1 cycle with the NM, and the remaining cycles used the GA. Notice that a large improvement in the objective function occurred at the beginning of the optimization. This is characteristic of most steepest descent optimizations. After reaching a local minima, the optimizer proceeded with the GA, returning several times to the DFP. Although it appears that there was only a slight improvement in the objective function during these later cycles, in fact, these cycles were characterized by noticeable changes in the geometry of the coolant flow passages. Each BEM solution required about 1.5 sec of CPU time on a single processor CRAY C-90. Therefore, the entire optimization used about 10 minutes of CPU time.

The final configuration shown in Figure 6 was the best design created by the thermal shape optimization after every optimizer failed to improve the population. The turbine blade is obviously cooler and lighter. Although no constraints were enforced, the thicknesses of the walls and struts converged to a finite value. We have found that, as the target temperature is decreased, the tends to make the thicknesses of the coolant walls and struts smaller.

The behavior of the net heat flux minimization objective was examined next. The optimization process was started from the same initial guess. The minimum net heat flux objective function forced the sizes of the coolant flow passages to become as small as possible until the geometric feasibility and the minimum cross-sectional area of the coolant flow passages became a limiting constraint. The final configuration obtained by the optimizer, shown in Figure 7, stalled in a on a constraint boundary. It was, therefore, concluded that thermal objectives (1) and (2) can have opposite effects, although both appear to be desirable outcomes. The thermal penalty objective (3) will be discussed in the section covering the conjugate thermal optimization.

Aerodynamic Shape Optimization

Optimal aerodynamic shape design of compressor and turbine blades provides decreased fuel consumption, better performance, safety, and ultimately lower operating costs. In order to achieve these desirable performance characteristics, the aerospace engineer must be equipped with a thorough understanding of the aerodynamics. The classical way of obtaining this knowledge is through the accumulation of a wealth of personal experience, through trial and error, and after extensive testing and verification. But the development of test rigs and experiments are both time consuming and expensive.

The introduction of high speed computers has led companies to invest large amounts of capital into the development of Computational Fluid Dynamics (CFD) analysis tools which provide their engineers with a preliminary analysis of new and redesigned products. Most of this software is run manually, relying on the engineer's intuition in an attempt to result with a better design. With the advent of high performance optimization algorithms, this repetitive task can be performed automatically using a computer. Therefore, many more designs can be tried in a shorter period of time and sometimes conceptually new designs are discovered by the optimization algorithm. In addition, after the engineer submits this computer job, while it is running, the engineer can perform other, less redundant, tasks. These algorithms still require an engineer's expertise, but they are employed during the planning and decision making stages when the engineer formulates the problem.

In aerodynamic shape optimization (ASO), this decision making process can begin with development of an effective ASO methodology involving three general tasks; generation of an aerodynamic configuration from a set of design variables, enforcing any number of geometric, manufacturing, durability and aerodynamic constraints, and quantifying the aerodynamic goal into a single scalar objective function.

Turbine and compressor airfoils are typically developed from a set of about 10-15 parameters that are unique and familiar to aerodynamic designers. Greater flexibility and control over the design space can be accomplished with splines or floating coordinates, but these alone can lead to airfoil contours that are not smooth or cross over themselves. In this section, we will detail a method that can guarantee geometric regularity while effectively capturing a large portion of the design space.

Constraints will need to be quantified into a finite number of inequality or equality constraint functions. Various constraint functions have been discussed in a previous section.

The quantification of the high-temperature flow field about a complex three-dimensional turbine or compressor blade into an all-encompassing aerodynamic loss function is rather difficult. Most aerodynamic engineers use the pressure distribution on the surface of the airfoil section as their first measure of the performance of the design. Airfoil design processes use a handful of rules to classify various characteristics of the pressure curve, such as acceleration, over-speeds, and diffusion, and then the undesirable characteristics are minimized or removed. If more than one of these characteristics are of concern, the engineer must also specify weighting factors. Unfortunately, this approach can lead to a discontinuous objective function, and often two performance characteristics will oppose each other so that the weighting functions must be chosen very carefully.

In inverse aerodynamic shape design (IASD), the user specifies a desirable target pressure distribution and then an IASD algorithm attempts to find an airfoil that produces a similar curve. When an optimizer is used, the integrated difference between the two curves is minimized. This approach produces a smooth scalar objective function, but the main problem with this technique is caused by the fact that the engineer usually does not know exactly what the best pressure curve is exactly before hand. For example, the pressure curve could be shifted by the angle of attack of the airfoil and a good design would be reflected by a poor objective function.

In an attempt to resolve these problems, we have studied several of the more common ASO methodologies used in contemporary aerodynamic shape design. Dulikravich (1992) has surveyed several of the more prominent methods of ASO and IASD. Although the approach that we are currently presenting (ASO using an optimization algorithm) is not as efficient as, for example, an adjoint operator/control theory or surface transpiration technique, it does have its advantages: (1) the flow field analysis routine does not need to be modified, (2) the modular nature of the ASO accepts any CFD solver, (3) optimization algorithms, such as the genetic algorithm, are less sensitive to local minima, and (4) an arbitrary ASO methodology can be easily developed.

In this paper, we will compare the behavior of several different objective functions listed below.

(1) Maximization of Lift/Drag. This objective function is developed by integrating the pressure and viscous forces over the turbine airfoil boundary.

$$F(\vec{V}) = -\frac{\int_{\Gamma}\left(-pn_y + (\mu+\mu_t)\frac{\partial v_y}{\partial n}\right)d\Gamma}{\int_{\Gamma}\left(-pn_x + (\mu+\mu_t)\frac{\partial v_x}{\partial n}\right)d\Gamma} \qquad (12)$$

Here, p is the static pressure, $\hat{n}$ is the outward unit vector normal to the airfoil surface, μ is the laminar viscosity, μ_t is the turbulent eddy viscosity, and v_x and v_y are the velocity vector components.

(2) Minimization of Vorticity. The square of the local vorticity is integrated over the fluid flow domain of the turbine cascade, averaged, and non-dimensionalized.

$$F(\vec{V}) = \frac{|\vec{v}_{inlet}|}{x_{len}}\sqrt{\frac{\int_{\Omega}(\nabla\times\vec{v})\bullet(\nabla\times\vec{v})d\Omega}{\int_{\Omega}d\Omega}} \qquad (13)$$

Here, x_{len} is the characteristic length of the turbine cascade and v_{inlet} is the inlet flow velocity.

(3) Minimization of the Maximum Mach number This objective function can be used whenever the CFD prediction solves only the Euler equations. The objective function is simply the maximum Mach number computed on the surface of the airfoil.

$$F(\vec{V}) = M_{max} \qquad (14)$$

Since ASO programs usually require several hundred objective function evaluations, each requiring a CFD analysis, the algorithm should use the fastest and most robust CFD solver available. We have chosen to work with an existing quasi-3-D CFD code for viscous flow through rows of turbine rotor cascades. This rotor viscous code (R.V. Chima, 1987) accounts for the quasi-3-D effects of rotation, radius change, and stream surface thickness variation on a two-dimensional C-grid. It has been used extensively at NASA Lewis Research Center and in industries and universities throughout the U.S.. The CFD algorithm uses a multi-grid technique, Runge-Kutta time integration, and implicit residual smoothing. A Baldwin-Lomax turbulence model is also incorporated in the code. Due to the highly specialized needs of the coupled aero-thermal shape optimization for internally cooled turbine blades, we have developed our own robust C-grid generator that uses the BEM.

Table 2 lists some relevant aerodynamic parameters and boundary conditions that were required for the aerodynamic analyses. All of these values were kept fixed during the entire aerodynamic shape optimization. N_{blade} is the number of turbine blades in the rotor circumferentially around the engine axis. R_{blade} is the mean radius of the turbine blade section (shown here as a reference since the stream surface thickness and radial coordinate are functions of the stream surface coordinate along the engine axis). p_{inlet} is the inlet static pressure. p_{exit} is the exit static pressure. T_{inlet} is the turbine inlet temperature. T_{wall} is the constant temperature on the turbine blade wall. v_{inlet} is the speed of the gas flow at the inlet. ω is the rotational speed of the blade row. α_{inlet} is the absolute inlet gas flow angle measured from the negative x-axis. μ_{ref} is the coefficient of absolute viscosity at a reference temperature. k_{ref} is the coefficient of thermal conductivity of the gas at a reference temperature. γ is the ratio of specific heats. Sutherland's model for the temperature dependence of viscosity was used in the viscous CFD code.

An initial guess to the turbine airfoil geometry was provided by the conic section design variables listed in Table 3. For each objective function evaluation, the complete system of quasi-3D Navier-Stokes equations used the boundary conditions shown in Table 2 and was converged fully to the steady-state. The velocity vector field predicted by the rotor viscous code on the initial guess configuration is shown in Figures 8a and 8b. Notice that there is a large separation region on the suction side revealing that this turbine airfoil shape is a very poor design.

The hybrid optimization algorithm was used to minimize the integrated vorticity objective function. Figure 9 shows the geometric history of the optimization process. The initial guess is the thick solid line. The best design from each of the 36 optimization cycles are shown as dotted lines. The converged turbine airfoil shape is the thin solid line. All ten conic section parameters were allowed vary, although the range of these design variables was specified to the program.

N_{blade}	50
R_{blade}	50.0 cm
p_{inlet}	124,132 N/m^2
T_{inlet}	1500.0 °C
T_{wall}	1200.0 °C
p_{exit}	120,000 N/m^2
v_{inlet}	75.0 m/s
ω	100 rad/s
α_{inlet}	60.0°
μ_{ref}	5.4x10^{-5} kg/m s
k_{ref}	0.0946 W/m K
γ	1.4

Table 2. Boundary conditions and fixed quantities used for the aerodynamic shape optimization of a generic turbine blade.

The design variables of the final converged configuration are also listed in Table 3. Notice that the flow turning angle increased by about 12 degrees. The velocity vector field for the converged geometry is illustrated in Figures 10a and 10b. The minimization of the integrated vorticity resulted in the removal of the large separation region on the suction side of the turbine airfoil, thereby substantially reducing the pressure drag on the turbine blade.

Design Var.	Initial Guess	Converged
x_{te}	5.0 cm	5.47 cm
y_{le}	4.0 cm	4.97 cm
r_{le}	0.5 cm	0.547 cm
r_{te}	0.1 cm	0.150 cm
α_{le}	20.0°	22.60°
α_{te}	65.0°	72.08°
βs_{le}	70.0°	70.53°
βp_{le}	80.0°	81.68°
βs_{te}	80.0°	79.19°
βp_{te}	90.0°	93.20°

Table 3. Initial guess to conic section design variables for the aerodynamic shape optimization of a generic turbine blade.

The convergence history of the objective function is shown in Figure 11. In this figure, the value of the objective function is plotted versus the CPU time on a single processor Cray C-90. The entire optimization utilized 400 calls to the viscous CFD subroutine. Each CFD analysis required about 25 seconds of CPU time plus about 1 second for the C-grid generation. The hybrid optimization algorithm shared time amongst three optimization techniques, DFP (5 cycles), GA (17 cycles), and NM (14 cycles).

Some of the most challenging types of turbine airfoil optimizations are the transonic airfoils. Their performance is primarily dominated by shock losses. In order to study how our ASO methodology behaves in these situations, an airfoil shape was initially generated and optimized using the conic section parameters. The flow field characteristics of this optimization are given in Table 4.

N_{blade}	62
R_{blade}	11.92 in
$p_{loss}=(p_{inlet}-p_{exit})/p_{inlet}$	0.9662
T_{inlet}	2195.62 R
p_{inlet}	14.7 psi
α_{inlet}	55.674°
M_{exit}	1.2197
α_{exit}	18.065°
γ	1.338
R_{gas}	1716.3 Btu/lbm R

Table 4. Flow field description and boundary conditions for the high Mach number turbine airfoil optimization.

The minimization of the maximum Mach number was used as the aerodynamic objective. The problem was fully constrained and incorporated many of the constraint functions described previously. These are listed in Table 5. Because of the cooling requirements, the diameter of the trailing edge was kept fixed at 0.04 in and a minimum thickness of 0.0623 in was held 0.135 in back from the trailing edge along the camber line. Impingement and shower head cooling kept the leading edge diameter above 0.1 in.

When optimizing with the conics only, the optimization process improved the design only marginally, with the maximum Mach number remaining at about 1.5. Figure 12a shows the geometry of the high Mach number airfoil (solid line) that was optimized by the conic section design variables.

Then, the conic section design variables were kept fixed to the previously converged aerodynamic configuration and 32 β-spline vertices were used to alter the airfoil with respect to that reference shape. The CFD analyses were provided for by a solution to the quasi-three-dimensional Euler system.

Perimeter <	4.0 in
Cross-Section >	0.22 in^2
Chord <	1.5
Moment Inertia >	0.0014 in^4
r_{le} >	0.05 in
Thickness >	0.0623 in
λ_{gage} =	0.3344 in
Axial Chord =	0.835 in
r_{te} =	0.02 in

Table 5. Constraints on the high Mach number turbine airfoil optimization.

After nearly 2400 function evaluations, a significant improvement on the original design was realized by modifying the β-spline vertices. Figure 12a shows the converged β-spline design plotted as a dotted line over the converged reference airfoil. Notice the bulge near the trailing edge on the pressure side. The optimizer wanted to make the trailing edge radius as small as possible, but the trailing edge equality constraint prevented this from happening. The optimizer succeeded in reducing the maximum Mach number to 1.35. The resulting static to total pressure ratio (p/p_0) versus the axial coordinate, x, for the initial guess and converged turbine airfoil are shown in Figure 12b. The Navier-Stokes equations were also solved to verify that the results from a turbulent compressible viscous flow analyses also showed improvement. The initial and final p/p_0 curves predicted by the viscous flow solver are very similar to those predicted using the Euler solver.

To understand why this new design performed better, we can look at the initial and converged pressure contours. In the initial guess geometry, an expansion wave fans outward from the trailing edge on the pressure side and impacts the suction side of the neighboring airfoil in the cascade. This over-expansion creates a rapid acceleration causing sharp pressure losses as the flow field diffuses in the uncovered turning. The optimized geometry slows down then speeds up the flow by alternately thinning and bulging the airfoil on the pressure side. The result is a weaker shock wave and the static pressure around the trailing edge is reduced so that the recompression is pushed farther downstream. This rather successful optimization resulted with a conceptually new design.

Conjugate Aero-Thermal Shape Optimization

In the standard practice of computational modeling of turbomachinery blades, aerodynamic engineers solve the system of fluid flow conservation equations in the hot gas flow region separately and fully decoupled from the heat conduction within internally cooled turbine blades. The thermal boundary condition applied to this CFD analysis is usually a constant wall temperature or an adiabatic heat flux condition. If any coupling is attempted, it is usually through the use of heat transfer coefficients when the heat conduction in the internally cooled turbine blade is analyzed. Any further thermal-stress analysis of the blade is carried out afterwards.

Besides some basic theory and mainly from results of experimental observations, the wall temperatures, surface heat fluxes, and the heat transfer coefficients on the turbine blade's external surface are not known a priori. In real compressible and high temperature flow fields, however, the temperature field will affect the velocity field, the shock locations, and the shape of the boundary layer. The modified gas dynamics, in turn, affects the heat transfer as well as the resulting thermal loads on the internally cooled turbine blade. Therefore, the mutual thermal interaction between the external hot gas flow field and the structure must be taken into account.

The conjugate heat transfer problem is defined as the solution of the energy conservation equation within a fluid flow simultaneously coupled to the heat conduction equation within the solid region in contact with the fluid. Conjugate heat transfer problems have been studied for many years (Yeung and Liburdy, 1995). With the improvement of computer performance and numerical schemes, coupled computer simulations are becoming an attractive feature of future aero-thermal prediction software.

In most publications, conjugate heat transfer predictions involve an explicit coupling between the two computational regions with separate analysis programs. Conjugate heat transfer applications focus mainly on the coupling of Finite Element (FEM) and Finite Difference (FDM) algorithms for both fluid and solid domains. However, FEM and FDM require the generation of grids within the interior of the turbine blade. This can be very difficult, especially when the blades have complicated internal coolant flow passages and thin trailing edges.

A coupled analysis using the BEM and the Finite Volume Method (FVM) was developed for the solution of the conjugate problem of heat transfer over and within turbine blades with cooling holes (Li & Kassab, 1994). The fluid flow and thermal convection were resolved by solving the time-dependent Navier-Stokes equations on a non-skewed, shifted periodic grid. The temperature distribution in the solid portion of the blade was determined at each time step using the steady-state BEM.

The advantages of using the BEM are that it requires minimal computational effort and it eliminates the need for a computational grid within the solid. The blade's surface temperature obtained from the solution of the Navier-Stokes system was used as the boundary condition of the BEM. The heat flux computed by the BEM was then iteratively enforced as a boundary condition to the Navier-Stokes energy equation. Hildebrand et al. (1995) have presented a similar coupled FDM/BEM strategy for high-temperature, hypersonic reentry vehicles.

We have chosen to employ a variation of the FDM/BEM method (He, et al., 1995). This conjugate coupling procedure consists of the following steps.

(1) Make an initial guess to the temperature on the turbine blade wall as a boundary condition to the FDM.
(2) Solve the FDM analysis using the rotor viscous CFD code.
(3) Use the heat fluxes predicted by the FDM and apply them as boundary conditions to the BEM algorithm. For stability, the original authors suggested the following relaxation method.

$$\left(q''_s\right)^{n+1} = \frac{w_s\left(q''_s\right)^n + w_f\left(q''_f\right)^n}{w_s + w_f} \tag{15}$$

(4) The BEM will compute temperatures on the outer wall of the turbine blade.
(5) Apply these new wall temperatures as a variable boundary condition on the turbine blade wall for the FDM.
(6) Iterate until the temperatures and heat fluxes predicted on the turbine blade wall converge.

In the relaxation equation (15), the subscripts s and f refer to the wall heat fluxes in the solid and fluid regions, respectively. The parameters w_s and w_f are weights. To ensure stability, the original authors found that the following relation must be satisfied.

$$\frac{w_f}{w_s} < \frac{3}{5} \tag{16}$$

The thermal boundary condition on the inner coolant flow passage surfaces was accomplished by specifying convective heat transfer coefficients and ambient fluid temperatures of the coolant fluid. It would be highly desirable, and theoretically feasible, to construct a conjugate heat transfer analysis that couples the hot gas flow field, the heat conduction within the turbine blade, and the coolant flow field simultaneously. Unfortunately, the analysis and subsequent coupling of the internal coolant flow has not yet been attempted by the current authors due to the limitations of computer resources and the need for a CFD prediction on an unstructured mesh.

The gas flow within turbine blade coolant passages is three-dimensional, with complex recirculating flows caused by the existence of miniature heat exchangers, skewed trip strips, bifurcations of the flow stream, U-shaped turning of the coolant channels, impingement and film cooling schemes, etc. The complexity of the CFD computation becomes even greater because of the contribution of the thermal buoyancy within the boundary layer caused by the centrifugal acceleration. In addition, Coriolis forces affect each coolant passage differently, depending upon whether the coolant stream is traveling within the radially-outward or radially-inward portions of the serpentine-shaped coolant flow channels. Therefore, a fully three-dimensional conjugate heat transfer analysis, which includes the fluid flow predictions within the coolant flow passages, is beyond the scope of this paper.

The conjugate heat transfer prediction was demonstrated on the generic internally cooled turbine blade described previously. The boundary conditions and geometric parameters used to develop this conjugate FDM/BEM example are the same as those given by the initial guess to the generic turbine blade (see Tables 1-3). During the iterative FDM/BEM coupling procedure, the fluid/solid interface temperature converged very rapidly (to less than 0.1 degree temperature difference in 8 or less iterations). The convergence in the heat flux was somewhat slower. In order to improve the performance of the conjugate coupling, subsequent calls to the rotor viscous code were restarted from the previously converged analysis, thus requiring less iterations overall. The entire viscous FDM/BEM conjugate solution required about 1 minute of CPU time on a single processor CRAY C-90.

Figure 13 shows an example of how a constant initial guess to the wall temperature (1200°C) converged to the more realistic temperature variation. Figure 13a shows the initial guess as a thick solid line, intermediate wall temperatures as dotted lines, and the final converged wall temperature as a thin solid line. The s-coordinate follows along the turbine airfoil contour clockwise from the trailing to leading edge and back to the trailing edge. As a comparison, the wall temperature distribution for the converged generic turbine blade configuration of the minimum vorticity objective is shown in Figure 13b. We even attempted to start the iterative conjugate process with a bad initial guess to the wall temperature. The iterative