Gas Turbine Main Shaft Internal Flow and Heat Transfer

DAVID V. ROSCOE and RICHARD C. BUGGELN
Scientific Research Associates, Inc.
Glastonbury, CT 06033

and

PETER M. MUNSELL and F. C. HSING
Pratt & Whitney Aircraft
United Technologies Corporation

ABSTRACT

A CFD analysis of the cooling flow through a gas turbine engine low pressure turbine shaft is presented. Three cases are considered in which throughflow and rotation rate are varied. The primary objective of the analysis was to derive improved heat transfer coefficient information, over those obtainable via semi-empirical means. The coefficients so obtained were then used in a one-dimensional, time-dependent analysis for use in predicting shaft wall temperature throughout a snap acceleration phase of the engine. A second objective was to obtain insight into the flow structure within the shaft with a view to possible design input in future engine programs. Results presented include detailed velocity vector plots at select locations, heat transfer coefficient distributions for each case and finally, for Case 2 predicted wall temperature vs. time is shown in conjunction with engine test data.

INTRODUCTION

During recent years the use of sophisticated numerical simulations, such as Navier-Stokes techniques, has become increasingly common for gas turbine engine primary flow path analysis and design; however, application of these techniques to the secondary gas path components is less established. Secondary gas path components such as seals, disk cavities, bearings and the main shaft are critical components in engine operation. In order to maintain the structural integrity of critical engine components it is necessary to control peak temperature as well as both temporal and spatial temperature gradients within the material of the component. This is accomplished by efficient distribution of cooling air to the high temperature structures. Due to the obviously critical nature of many of these components, current design philosophy is necessarily conservative in the amount of coolant air utilized and in the component construction. Withdrawing excessive amounts of cool air increases parasitic losses and decreases engine efficiency. Increasing component size to withstand thermal stress increases weight, performance, and decreases the engine thrust-to-weight ratio.

Although flow through secondary gas path components has received considerably less attention than flow through their primary counterparts, numerical simulation through the Navier-Stokes equations has been applied both to flow through cavities and seals. Some examples of recent cavity applications include the work of Roscoe et al. (Ref. 1), Owen (Ref. 2), Chew (Ref. 3) and Morse (Ref. 4). An example of labyrinth seal analysis is given by Buggeln et al. (Ref. 5). The problem considered here, flow through the main shaft, represents a critical engine component which has received little previous attention in regard to Navier-Stokes modelling. Control of the coolant flow is very important in the main shaft secondary gas path. Inadequate coolant flow can cause thermal expansion of components leading to insufficient clearance between adjacent surfaces in relative motion or unacceptable thermal stress levels. Therefore, current design philosophy, which is based upon simpler analyses than that presented here and extensive testing, may indicate the need for more coolant air than is actually required, leading to excessive parasitic loss or to designs heavier than required. Clearly, optimum main shaft cooling design represents an area where significant gains in engine efficiency still can be realized.

As will be discussed, this problem has difficult numerical aspects due to the very high aspect ratio of the flow regime and the complex flow patterns which occur. The present paper discusses a state-of-the-art numerical analysis applied to this problem, presents some detailed results and assesses the analysis via comparison with engine test data. It is worth noting that while it is usual to show details of grid independence studies in papers dealing with numerical prediction; in this case where the purpose of the calculation was practical rather than academic, these studies were not performed. The authors have, however, established a reasonable level of confidence in the approach through application of the technique to other secondary gas path problems detailed in the references.

Analysis

The present simulation is based upon a solution of the full ensemble-averaged Navier-Stokes equations, in conjunction with a two-equation turbulence model. The continuity equation is given by

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0$$

(1)
The momentum equations are
\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \mathbf{\tau} + \mathbf{F} \tag{2}
\]
where \(\mathbf{\tau}\) is the stress tensor, including both the molecular and Reynolds stress, written in the appropriate coordinate system.

The energy conservation equation is written as
\[
\frac{\partial (\rho h)}{\partial t} = -\nabla \cdot (\mathbf{u} h) - \nabla \cdot (\mathbf{q} - \bar{q} \mathbf{u}) + \mathbf{F} \cdot \nabla \mathbf{u} \tag{3}
\]
where in this case the stagnation enthalpy form of the energy equation is used. The turbulent contribution is obtained from the solution of two partial differential equations for the turbulent kinetic energy and the dissipation of turbulent kinetic energy basically following Jones and Launder (Ref. 6); details of the model used are given in Ref. 1.

The numerical procedure used to solve the governing equations is a consistently split linearized block implicit (LBI) scheme of Briley and McDonald (Refs. 7 and 8). The method uses a formal linearization technique adapted for the integration of initial value problems. The governing equations are replaced by an implicit time difference approximation. Terms involving nonlinearity at the implicit time level are linearized by Taylor series expansion in time about the solution at the known level, and spatial difference approximations are introduced. The result is a system of coupled multidimensional linear difference equations for the dependent variables at the implicit time level. To solve these difference equations, the Douglas-Gunn procedure for generating alternating direction implicit (ADI) schemes as perturbations of the fundamental implicit difference schemes is introduced. This technique leads to systems of coupled linear difference equations having block-banded matrix structure which can be solved efficiently by standard block elimination methods.

For the subsonic inflow boundary of the current problem, total temperature, total pressure, and flow angle are set as boundary conditions. Static pressure is set at the outflow boundary and no-slip conditions on solid walls. In the vicinity of solid walls, high near-wall resolution is used to allow resolution of the wall gradients. In this problem the computational grid, consisting of 345 streamwise and 75 radial points was constructed to place at least one point within \(y^+\) of 10. The use of high near-wall resolution is considered particularly important in the flow application considered here. As will be shown, the resulting flow patterns are very complex with the appearance of multiple recirculation regions. The major item of interest emerging from these simulations is the near-wall heat transfer. The ability to accurately compute this heat transfer rate requires a viable near-wall flow modelling and for these complex flows viable near-wall flow modelling requires a highly resolved flow in which velocity and temperature distributions in the near wall region emerge from the solution.

**Problem Description**

The problem considered here is the prediction of flow and heat transfer in the gas engine turbine main shaft secondary flow path. As shown in Fig. 1, the flowpath has a large length-to-radius ratio. The flow enters at the left side of the shaft through discrete holes and exits at the right side again through discrete holes. Although a complete simulation would require a three-dimensional analysis, including the discrete hole inflow and outflow, the present approach assumes an axisymmetric simulation in which flow both enters and exits the domain through circumferential slots. Thus, the present simulation is that of axisymmetric throughflow with a swirl component derived from the shaft rotation. Boundary conditions were set to match the engine test total pressure and total temperature at inflow with no preswirl. Static pressure, obtained from the engine test, was set at the downstream boundary. It should be noted that this specification does not set the mass flux, but rather allows the mass flux to adjust to the upstream total pressure, the downstream static pressure and the flow losses implied by the predicted flow field.

Three cases were considered. In the first case upstream and downstream conditions, as well as a wall temperature profile and a shaft rotation speed, were specified. The second case varied the wall temperature profile, increased the inflow and outflow areas and specified a slightly different shaft rotation speed. The third case considered a third wall temperature schedule and the original inflow and outflow areas. Details of these three cases follow in Table 1 and the three wall temperature schedules are shown in Fig. 2.

<table>
<thead>
<tr>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inflow Area (in²)</td>
<td>2.872</td>
<td>5.11</td>
</tr>
<tr>
<td>Outflow Area (in²)</td>
<td>2.405</td>
<td>4.811</td>
</tr>
<tr>
<td>(T_{\text{in}}) (*F)</td>
<td>270*</td>
<td>304*</td>
</tr>
<tr>
<td>(P_{\text{in}}) (psi)</td>
<td>29.58</td>
<td>30.11</td>
</tr>
<tr>
<td>(P) (psi) out</td>
<td>26.16</td>
<td>23.8</td>
</tr>
<tr>
<td>Rotation Rate (rpm)</td>
<td>3632</td>
<td>3856</td>
</tr>
<tr>
<td>Mass Flow Rate (lbm/sec)</td>
<td>0.1489</td>
<td>0.75</td>
</tr>
</tbody>
</table>

In all cases converged solutions were obtained. Two criteria were used for convergence. Previous experience with similar problems has indicated that when (i) the maximum flow field residual has decreased by three or more orders of magnitude and (ii) the maximum change in any dependent variable from time step to time step is less than .005 of its reference value for a significant time step, the calculation is converged for engineering purposes. The results in this effort dropped the maximum residual by four orders of magnitude and showed normalized change in dependent variables from time step to time step to be less than .0001 within approximately 1500 time steps, thus indicating convergence, based upon the author's experience in other applications.
Since the prime areas of interest for this particular investigation are well removed from the inflow and outflow locations, the axisymmetric assumption should be viable for determination of the wall heat flux in regions removed from the injection locations. Turbulence modelling always raises questions; however, the model used here is a well-accepted one. Good results have been obtained by a variety of investigators using the k-ε model for a range of problems. Finally, the code used has given good comparisons with data for a wide variety of secondary gas path flows, including seals and disk pump cavities. Therefore, the results are expected to represent the actual flow physics fairly accurately. In evaluating the heat transfer characteristics of the problem the following definition of heat transfer coefficient, h, used in the present study was:

\[ h = \frac{q}{\Delta T} \]

where \( \Delta T = T_{wall} - T_{ref} \)

and

\[ q = -k \frac{\partial T}{\partial n} \]

and the normal heat flux \( T_{ref} \) is a local radially integrated mass averaged temperature.

**Results**

Results of the three simulations are given in Figs. 3-5. For each set of Figures, e.g. Figs. 3a-3e, each figure represents a segment of the flow with the approximate location of the segment indicated on Fig. 1. Prior to discussing these figures, it should be noted that Cases 1 and 3 are similar with differences in the inflow and stagnation conditions and the outflow static conditions leading to a twenty per cent change in mass flow rate. Case 3 also has a lower shaft rotation speed; however, none of these differences are extreme and the overall flow patterns are expected to show similar features. Case 2 has a significantly different mass flow rate differing from Case 1 by a factor of five. This is expected to lead to a significantly different flow pattern. Each case is presented as a series of figures showing velocity vector plots for axial segments of the computational domain starting from the inflow boundary and ending with the outflow boundary. The velocity lengths represent \( (v^2 + v^2 + v^2)^{1/2} \) and the vector direction indicates the flow direction in the r-z plane.

Figures 3a-3e represent the Case 1 flow field, Figures 4a-4e represent the Case 2 flow field and Figs. 5a-5e represent the Case 3 flow field. In all cases the flow proceeds from left to right with injection coming from the top port. Figs. 3a, 4a and 5a show the inflow regions. In Case 1, flow entering the upstream plenum forms three distinct vortices confined to the outer radius region. By comparison with Case 3, where the inlet area is the same and 81% of the flow rate of Case 1, it is clear that the rotation rate has a strong influence on the flow structure in the plenum. In Case 3 the jet maintains its integrity almost to the centerline and forms a much larger recirculation on the downstream side of the jet. In some respects Case 2 resembles Case 3, insofar as the jet maintains its integrity further into the plenum. In this case the mass flow is five times that of Case 1 at a 6% higher rotation rate. Clearly, as expected, the balance between radial injection and the centrifugal effects of rotation affects the flow structure in the plenum.

On leaving the plenum and entering the main part of the shaft (Figs. 3b, 4b and 5b), the flow in each case behaves somewhat differently. In Case 1 the flow tends to slow down at the centerline, while in Case 3 the flow recirculates at the centerline and consequently accelerates toward the wall. The flow in Case 2 is less affected by rotation and accelerates as a result of entering the shaft and shows no sign of recirculation near the centerline. All three cases show recirculation at the rearward facing step located at the right side of section b.

The flow in all cases reattaches rapidly and begins to redevelop boundary layer type profiles. However, at the shaft contraction, shown in section c, Cases 1 and 3 again show flow retardation at the centerline and acceleration near the wall, while Case 2 shows only the general retardation expected through a diverging section (Figs. 3c, 4c and 5c). Interestingly, the flow does not separate at the wall, as would almost certainly occur in a non-rotating conical diffuser of this angle.

Following this contraction and divergence, the flow enters the first of two long constant area sections in the shaft. Here in all three cases, though the peak velocities are quite different, each flow field straightens and takes on the appearance of a classical turbulent boundary layer profile. The approach flow to the contraction in section d once again shows the similarity between Case 1 and 3, in that in both of these cases the flow at the centerline accelerates and gives rise to a velocity profile appearing to be on the point of separation, while the flow in Case 2 behaves more like the flow in a non-rotating pipe (Figs. 3d, 4d and 5d).

The flow through the second constant area section though at a high peak axial velocity, has a form similar to the previous constant area section. Further downstream the shaft wall again diverges, giving rise once again to a wall acceleration region in Cases 1 and 3, with the now familiar recirculation at the centerline (Figs. 3e, 4e and 5e). In Cases 1 and 3 the flow does not recover and reaccelerates at the centerline, following this last change in section, and begins to stagnate as the effect of the endcap of the shaft is felt. Figs. 3f, 4f and 5f show the outflow patterns.

**Heat Transfer Behavior.** In the gas turbine engine shaft, the bleed air extracted from the low pressure compressor is utilized in two ways. This cool air is used primarily to provide a cold buffer for the rear bearing compartment, and secondly, to cool the low speed shaft. As the demand for high thrust advanced engines further increase, the projected shaft torque will be much higher than in today's engines and the shaft is also anticipated to operate in an elevated temperature environment.

Without resorting to an engine test with extensive instrumentation, CFD analysis is the only analytical tool that can provide engineers an insight into the internal flowfield in the shaft cavity and how it affects the heat transfer mechanism at a range of engine operating conditions. This information will allow designers to deliver a proper amount of cold buffer air to the bearing compartment, and to control the thermal environment of the shaft such that it is not susceptible to unacceptable thermal stress.
The primary purpose of performing the CFD analysis described in the text was to obtain some understanding of the effect of flow structure on heat transfer behavior in the particular shaft in question. The CFD analysis was used in conjunction with other empirical and semi-empirical tools commonly used in secondary gas path design. Comparable analysis using estimated heat transfer coefficients while within the normally accepted bounds of uncertainty nonetheless prompted the CFD analysis. The purpose of performing the analysis was to attempt to raise the level of confidence in the design, particularly with regard to its transient wall temperature response.

The distribution of heat transfer coefficients predicted and shown in Figure 7 was subsequently used to apply boundary conditions in a one-dimensional finite element heat transfer code. The transient prediction and measurement of wall temperature of the shaft at three locations for case 1 are shown in Figure 8. The plots show excellent agreement between the measured and predicted wall temperatures.

CONCLUSIONS

A state-of-the-art Navier-Stokes solver has been applied to the problem of flow and heat transfer within the main shaft secondary gas path. Three cases were considered, all showing relatively complex flow patterns. The nature of these flow patterns, as well as the heat transfer, were dependent upon both the throughflow and the shaft rotational speed. The results indicate the ability of this Navier-Stokes solver to provide insight into the expected flow structure in this important engine component.

The analysis has important implications in engine development. The CFD analysis provided information very difficult to obtain experimentally, and beyond the capability of other simpler analytical means. Furthermore, the results clearly show that experimental determination of the air temperature near the shaft wall would be very difficult, because of the thin thermal boundary layer present there. The results can be very useful in providing guidance to instrumentation engineers on the selection and installation of temperature probes for engine verification tests, and to heat transfer engineers on interpretation of the results.

With sufficient experience, we anticipate that CFD simulation can help reduce, though not replace, expensive and time-consuming testing for the entire engine and for complex configurations where instrumentation is very costly or not possible. The study described formed a relatively small fraction of the effort required in the analysis of this important engine component. The scope of the study was relatively limited and from an academic point of view leaves many interesting questions unanswered. However, in the context of the engine development program, of which if formed part, the analysis served its purpose well.

Much more work would be required to have broader impact on the general level of understanding of components of this type.

REFERENCES

Fig. 2 - Wall Temperature Schedules.

Fig. 3a - Case 1 Velocity Vectors for Segment a.

Fig. 3b - Case 1 Velocity Vectors for Segment b.

Fig. 3c - Case 1 Velocity Vectors for Segment c.

Fig. 3d - Case 1 Velocity Vectors for Segment d.

Fig. 3e - Case 1 Velocity Vectors for Segment e.

Fig. 4a - Case 2 Velocity Vectors for Segment a.

Fig. 4b - Case 2 Velocity Vectors for Segment b.
HEAT TRANSFER COEFFICIENT BASED ON MASS AVERAGE FLUID TEMPERATURE

CASE #1: 1898, 2100 RPM
CASE #2: 1900, 3856 RPM
CASE #3: 1200, 2700 RPM

Fig. 6 - Distribution of Heat Transfer Coefficients.
Fig. 7 - Radial Temperature Profile.

Fig. 8 - Transient Wall Temperature Prediction Using 1-D Finite Element Analysis and CFD Heat Transfer Coefficients.