Transient Conjugate Heat Transfer of Turbine Rotor-Stator System

Yoji Okita
Ishikawajima-Harima Heavy Industries
229, Tonogaya, Mizuho-Machi, Tokyo, Japan
youji_ookita@ihi.co.jp

Keywords: Jet-Engine, Gas Turbines, Heat Transfer, Secondary Air System

Abstract

A fluid-solid conjugate solver has been newly developed and applied to an actual engine disk system. Most of the currently available conjugate solvers lack the special thermal modeling for turbomachinery disk system applications. In the present new code, these special models are implemented to expand the applicability of the conjugate method and to reduce the required computational resources.

Most of the conjugate analysis work so far are limited to the axisymmetric framework. However, the actual disk system includes several non-axisymmetric components which inevitably affect the local heat transfer phenomena. Also the previous work devoted to this area usually concentrate their efforts on the steady-state thermal field, although the one in the transient condition is more critical to the engine components.

This paper presents full 3D conjugate analysis of a single stage high pressure turbine rotor-stator disk system to assess the three-dimensional effects (Fig. 1). The analysis is carried out not only in the steady-state but also in the engine accelerating transient condition. The predicted temperatures shows good agreement with measured data.

Introduction

Interest in use of CFD in thermal modeling for aero-engine secondary air system has increased considerably in recent years. This trend is based on the recognition that the current practice of thermal modeling in industry highly depends upon empiricism and generally lacks the universal validity required for arbitrary configurations and conditions.

While stand-alone CFD calculations may be beneficial enough to define the convective heat transfer boundary conditions for the model, combining the CFD with heat conduction in the solid region offers greater accuracy and reduces the turn-around-time of the analysis in total. This is an area of current interest to industry, with a number of publications over the last few years [1-3] and the present work also tries to make some contribution with unstructured grid and with several new features.

There are basically two approaches to fluid/solid heat transfer; the 'conjugate' method in which a base CFD code is extended to include thermal energy equation in the solid and the 'coupled' method which couples separate software for each region by passing information between the two. The conjugate method is adopted here mainly because it allows fluid and solid to link in a fully conservative manner, whereas the coupled procedure, which inevitably needs an inter- or extrapolation, tends to have difficulties in highly distorted regions, typically around ridges or corners [3]. While the 'effective conductance' approach, first proposed as 'harmonic mean' for Cartesian grids, and later extended to non-orthogonal coordinates [4], is well known for dealing with fluid solid interfaces, it is not straightforward to apply it to an unstructured grid. An approach assuming a 'dummy' cell at the interface was devised and implemented in the present conjugate code.

A disadvantage of currently available conjugate solvers is that they generally lack the special thermal modeling for turbomachinery disc systems. This may be a reason why many industry researchers preferred the coupled method in which they can adopt CFD while keeping existing thermal practices [2,3]. In the present code, another approach was tried; special thermal models were implemented in the conjugate solver to expand its applicability and to reduce the required computational resources.

The 'conjugate' or 'coupled' studies in disc system so far have been limited to axisymmetric computations. While the axisymmetric assumption is valid for some regions, the disc system in an actual engine is fully three-dimensional, especially in the region affected by circumferentially discrete components like bolt protrusions or orifices for the cooling air. This paper presents the full three-dimensional conjugate analysis of a disc system to assess these effects on the flow and thermal field.

Another issue discussed in this paper is the detailed evaluation in transient condition. The previous studies generally focused on the accuracy of predicted temperature in the steady-state condition, but little attention has been paid to the accuracy in transient condition which is more important in the actual engines because it determines the maximum thermal stress and consequently the fatigue cycle of the components.

Numerical and experimental methods

Numerical methods

Flow solver

The governing equations in the flow field are the time-dependent, compressible, three-dimensional

Reynolds-averaged Navier-Stokes equations. In the solid region, only the heat conduction equation is solved. The equations are discretized with an unstructured finite volume method. It uses the pressure-correction procedure (SIMPLE) and employs a collocated grid arrangement with mass flux modification [6].

Turbulent flow computations are made using the high-Reynolds-number k-epsilon model [7] and the standard wall function with a modification to include viscous heating effect [9]. Based on the rotational Reynolds number at the outer radius in each cavity, the laminar region, if any, should be confined only in a small area. The assumption of fully turbulent flow in the calculation was thus considered reasonable. While the use of wall function in rotating cavity problems may be controversial, it was reported to give acceptable results, provided the rotational Reynolds number is sufficiently high and care is taken in near wall mesh spacing[10].

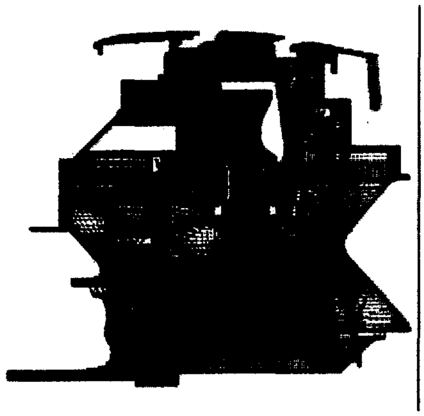


Fig. 1 Computational grid for conjugate analysis of the turbine rotor-stator system (meridional section)

Geometry modeling and computational grid

Figure 1 shows a meridional section of the computational grid for the conjugate analysis of the turbine rotor-stator disc system. Hexahedral cells were employed in the entire region. Grid expansion ratio was kept less than 1.3.

Although some regions in the disc system can be assumed axisymmetric, there are a number of parts which have fully three-dimensional geometry. Some of these, that is, discrete bolt protrusions, blade-dovetail, cooling air orifices, etc., especially around the rotor, were modeled in faithfully three-dimensional. The calculation region was confined to 6.55deg.-sector and periodicity was imposed at the side boundaries.

Special thermal models

As mentioned above, various special thermal models are available in the present conjugate solver. Several examples related to the present analysis are described in this section.

First, a one-dimensional network model connecting representative air and / or metal nodes called the "Compact Thermal Model (CTM)", which can interact with the main conjugate region via convective heat transfer and / or thermal contact resistance, is implemented. Figure 2 shows only the part, relevant to the main conjugate region, of whole CTM. A secondary air system spreads over the entire region of a engine and each sub-system interacts each other. Applying full conjugate method to the entire region is prohibitively expensive. Nevertheless, it is crucial to take into account the whole system in some way, otherwise the solution may not be consistent with flow and thermal balance of the whole system. The CTM is very useful to solve this difficulty. It allows a compact network to the region where the detailed flow and thermal phenomena does not need to be resolved, then most of the computational resources can be spent in the main region of interest with the full conjugate method.

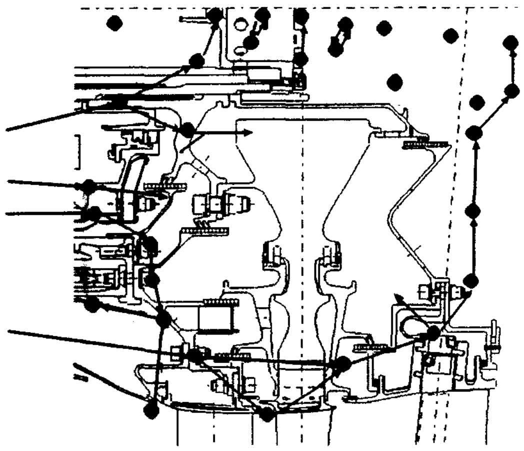


Fig. 2 The Compact Thermal Model linked with the main conjugate domain

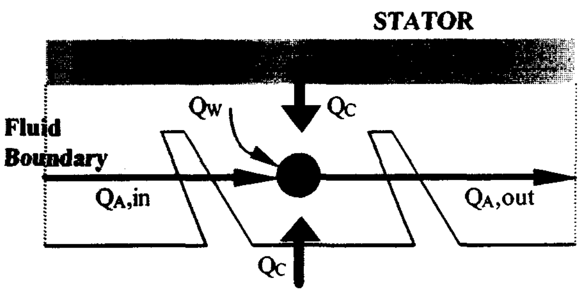


Fig. 3 Energy balance in the labyrinth seal model

Second, a labyrinth seal model is proposed. It is possible to calculate accurately the flow through a labyrinth seal and it is sometimes very informative to understand the flow phenomena in the region. Such an approach, however, inevitably demands sufficiently fine mesh inside the seal. While the present solver can directly resolve the seal region, it also provides another option to simplify the seal region (Fig. 3). In this model, an air node is first located in each cavity between neighboring teeth and then the links connecting the nodes and the flow boundaries upstream and downstream of the seal are formed. The temperature at each node is obtained by balancing energy advection through the links, heat transfer with solid boundaries surrounding the cavity, and windage heat generation due to frictional torque caused by the rotor / stator surface which is calculated by solving an equation of angular momentum conservation within the cavity [12].

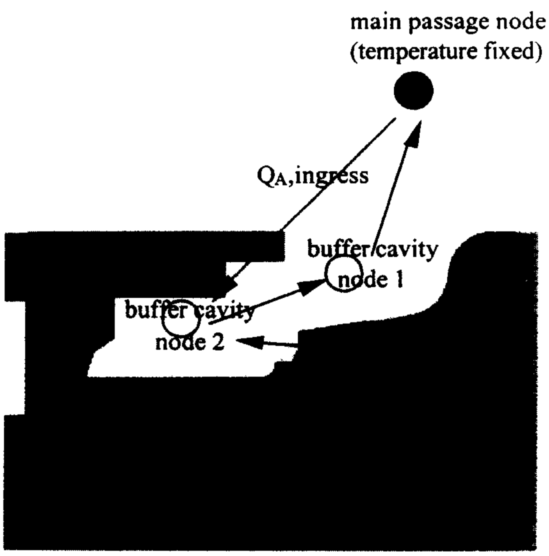


Fig. 4 Mainstream-ingress model for a buffer cavity

Finally, a mainstream-ingress model is described. Mainstream-ingress is known as a highly three-dimensional phenomena and moreover, recent works [e.g. 13] suggests that a full unsteady calculation is needed to predict seal effectiveness accurately. To avoid such a huge computation, a model similar to the labyrinth seal treatment is adopted in the conjugate code (Fig. 4). Again, the buffer cavity and main-passage are simplified with air nodes connected to the upstream fow boundary with links representing air flow, which results in a energy conservation equation for each node (the buffer cavity may be divided into several nodes, if needed). The temperature at the mainstream node is fixed, in other words, it is assumed the secondary air does not affect the mainflow thermal condition. The validity of this model depends on the determination of the amount of ingested gas. Ishida[14] studied seal performance in a generic turbine rotor-stator cavity with mainstream for various conditions. Their correlation of seal effectiveness was used in the current analysis. For most of the results presented in this paper, the ingress was assumed to be limited only in the buffer cavity. However, a case allowing the ingress to reach the rim cavity is also presented for comparison.

Boundary conditions

The secondary air system around the disc was divided into five cavities (Fig. 5) where inlet and exit boundary conditions were applied respectively.

At the inlet to each cavity, the mass flux, swirl, and temperature were automatically set to be consistent with the aforementioned CTM and / or the labyrinth seal model. Radial velocity was assumed zero. The turbulent kinetic energy was set with a turbulence intensity of 4% and the dissipation rate then obtained assuming turbulence Reynolds number as R_T=100.

At the exit of each cavity, pressure was imposed. For velocities, temperature, and turbulent quantities, derivatives across the exit plane were assumed to be zero.

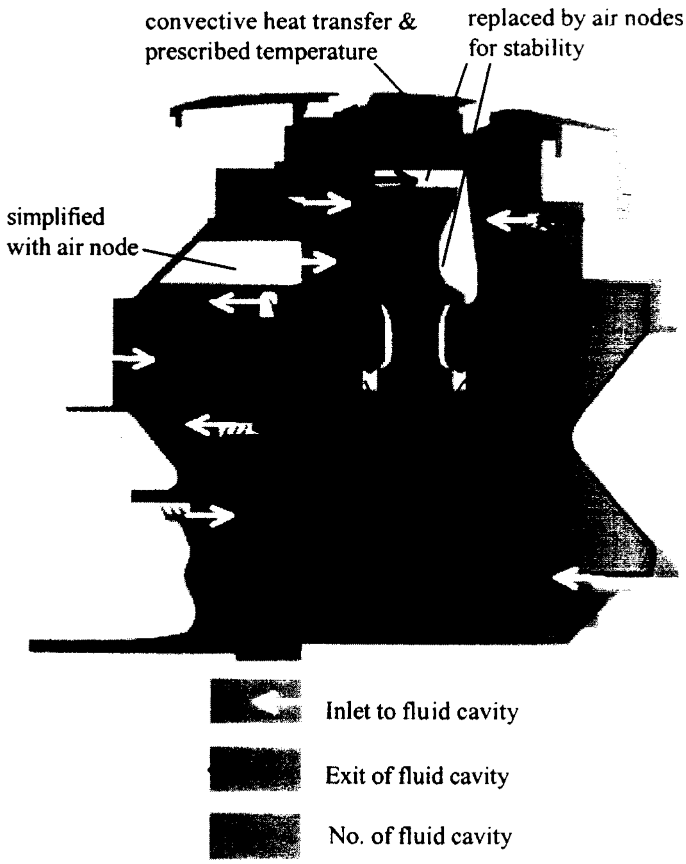


Fig. 5 Boundary conditions for the conjugate model

For the rotor platform, the region corresponding to the blade root was set at a prescribed temperature which was obtained from the blade cooling design, whereas convective heat transfer with mainstream air node was applied to the rest of the platform using a correlation of flow over a flat plate.

At the solid surface where different components contact each other, a thermal contact resistance, assuming empirical contact coefficient (approximately, 104 W/m2 K), was applied. All other solid boundaries facing the exterior of the conjugate model were linked with the aforementioned compact thermal model. Several interior solid boundaries were also linked with air nodes which simplified some of the fluid region, namely, closed cavities around the front- and rear-seal retainers and upstream of the pre-swirler (Fig. 5). It also should be noted that preliminary studies revealed that including a closed cavity inside the rear-seal and a dovetail slot passage into the conjugate model deteriorated the convergence of the #4-cavity, though it was confirmed that the instability was not attributed to the fluid-solid coupling Thus, those regions were omitted from the conjugate model and replaced by air nodes.

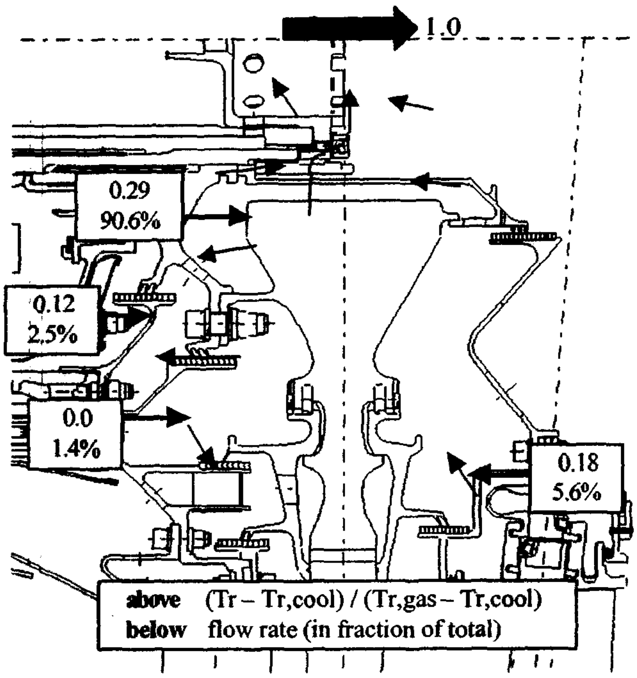


Fig. 6 Secondary air system of the HP-turbine disc

Conventional FE analysis

Conventional practice in industry for calculation of component temperature is typically a finite element (FE) code which solves the steady / transient heat conduction equation with some thermal modeling. In this work, this type of analysis was also carried out on the same disc system to compare the results with the conjugate solution. The thermal modeling adopted was very similar to the method provided in detail by Monico and Chew [15].

Experimental aspects

Since the detail of the experiment has already been provided in [16], several points relevant to the current objective will be only briefly described.

A single stage high pressure turbine (Fig. 6), installed in a high temperature core engine, was instrumented at multiple locations for metal and secondary air temperatures. A nominal condition, where the turbine inlet temperature was 1870K (which was carefully calculated based on the measured engine exit temperatures and a detailed performance analysis) and compressor discharge temperature was about 830K, was used in the present steady-state calculation and time-dependent data where the engine accelerates from start to the nominal condition were used in the transient calculation.

The surface temperature on the discs were measured by embedded thermocouples. The spot welded junctions of the sheathed thermocouple edges were covered with thin plate of Inconel. The uncertainty in the temperature measurements was estimated using a simplified thermal model where assuming the thermocouple as a fin, the temperature at the junction was calculated by balancing heat transfer with the surroundings and heat conduction with base metal. The resulting uncertainty was about 3K. An error due to reconnection of the thermocouples to other wires inside the engine cannot be estimated and thus is not included in this uncertainty value. The effect, however, should be negligible because the material with same thermal and electrical property as the thermocouple is used for the extension wire.

No.	Re? /10^6	Cw / 10^4	$?_T$	G (min / max)
1 (front)	0.62	0.83	0.192	0.092 / 0.202
1 (mid)	0.32	1.24	0.491	0.191 / 0.262
1 (aft)	1.12	1.17	0.169	0.137 / 0.247
2	1.13	0.95	0.136	0.038 / 0.093
3	1.30	0.40	0.052	0.037 / 0.080
4 (front)	2.10	7.95	0.697	0.053 / 0.191
4 (aft)	2.06	7.10	0.630	0.013 / 0.050
5	1.28	0.28	0.037	0.119 / 0.348

Table 1 Flow conditions and geometry parameters of fluid cavities

The air temperature were measured by protruding sheathed thermocouples sufficiently into the cavity to minimize the heat conduction error. Bare wires were exposed at the tip. A recovery factor of 0.68 was assumed for the wires normal to flow, 0.86 for parallel [17] The uncertainty due to velocity and conduction error could not be quantified.

The secondary air system around the turbine rotor stator system is summarized in Fig. 6 in which non-dimensional supplied air temperature and fractional flow rate are presented. Main cooling air was supplied by the pre-swirler located inside the nozzle guide vane. Flow and geometry parameters in each cavity at the steady-state condition is shown in Table 1.

Results

Steady-state condition

The metal temperature contours at the steady-state condition predicted by the present conjugate method is shown in Fig. 7. The figure also shows the locations where the thermocouple data are available. Fig. 8 shows differences between the engine thermocouple data and the predictions by the conjugate code and also by the conventional FE method. The accuracy of the conjugate solution is within 17K on the rotor disc, the region of main interest. This is a fairly good result in comparison with other works [1, 2]. The errors increase on the stator (-30K / +24K). This is partly because there is some incompleteness in the compact thermal model the stators directly connect with their boundaries.

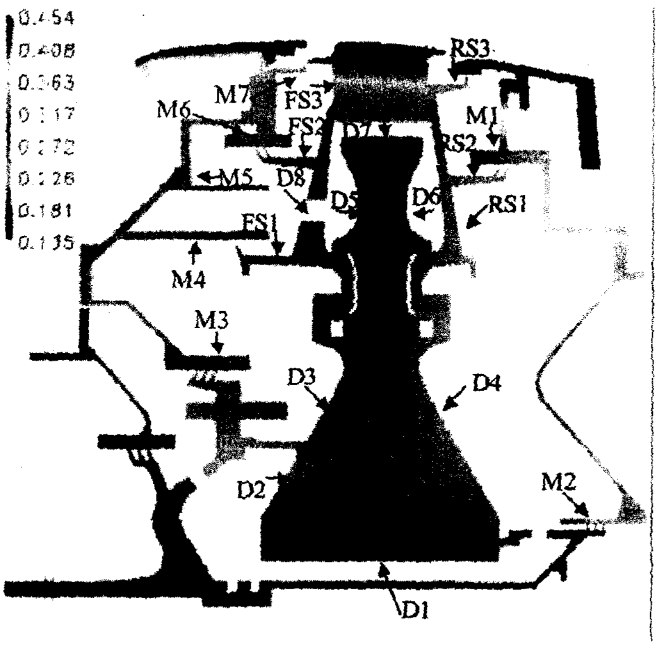


Fig. 7 Predicted metal temperature (T - Tr,cool) / (Tr,gas - Tr,cool)

Figure 9 shows the relative total temperature contours predicted by the conjugate method. It also shows errors with measured data. The largest errors occur in the front rim cavity. Except for those, the accuracy is within 13K which is comparable to the metal temperature accuracy. One of the possible explanations for the under-prediction in the rim cavity is that the mainstream ingress may not be modeled properly in the current analysis in which the ingress is assumed to be limited in the buffer cavities.

It is widely recognized the circumferential pressure asymmetry in the mainstream causes the ingress. The ingress model used in the above analysis assumes that the hot gas is ingested into the buffer cavity, which locates just inside the mainstream. But it also assumes the gas cannot reach the rim cavity because the flow mixing in the buffer cavity disperse the pressure asymmetry. However, the ingested gas may reach the rim cavity if the mixing is not enough to uniform the asymmetry completely. More works will be needed to establish a better ingress modeling.

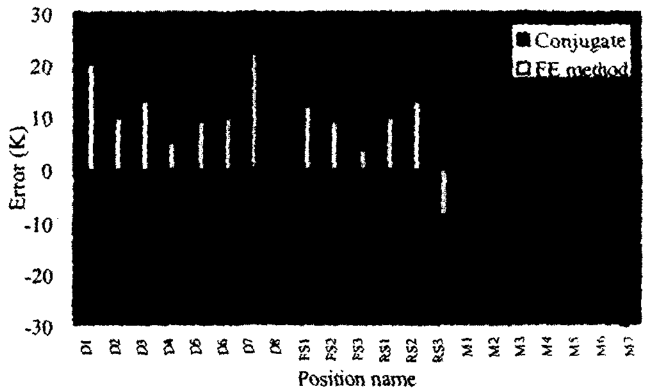


Fig. 8 Error (= predicted - measured) of metal temperature Note:Results of FE method are not available at stator (M1 to M7)

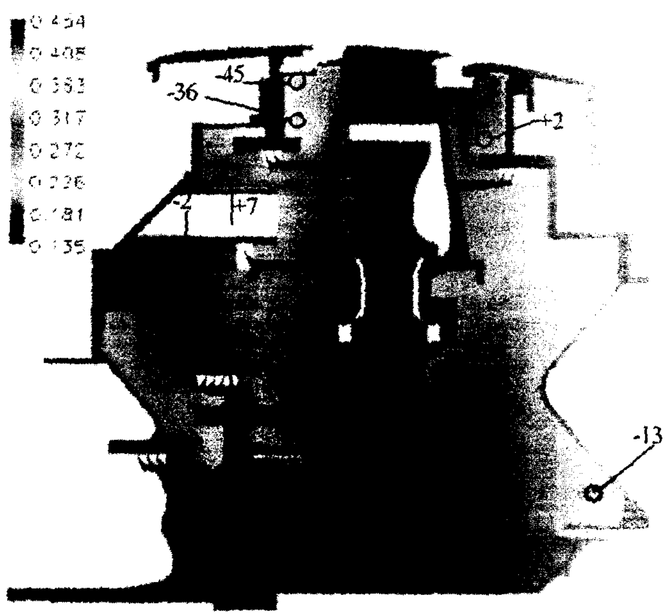


Fig. 9 Predicted relative total air temperature distribution and errors at thermocouple points contour values: (Tr - Tr,cool) / (Tr,gas - Tr,cool)

Transient condition

Prediction of temperature is much more difficult for transient conditions than for the steady-state. This is primarily because the accuracy of the heat transfer coefficient (HTC) affects the thermal field more in the transient situation. As shown in Table 2, the conventional FE method did fairly a good job, comparable to the more expensive conjugate method in the steady-state, where conduction primarily determines the thermal field. The standard correlation of the HTC used in the FE method, however, often gives rather unrealistic value in some regions, especially the field affected by three-dimensional flow features, which results in a more significant temperature error in the transient situation. Below transient results for several thermocouple points are shown to clarify the effect of HTC.

Transient results at the rear side of disc bore (D4) are first shown for an example in good agreement with the FE method. Fig.10 is a plot of temperature against time for a test bed cycle, i.e. start, accelerate to low power (idle), followed by a further acceleration to stabilized high power. Throughout the cycle, there obtained good agreement (within 20K) using the FE method. The flow field at this region is the one of typical rotor-stator cavity with radial outflow and thus is supposed nearly axisymmetric (the CFD results also supported this observation).

In contrast, as shown in fig. 15, the results at the front side of disc bore (D3) show more discrepancies, with the largest errors being 50K during the acceleration. The standard correlation of HTC for the bore is typically formulated as ;

Nu ? ?EF?? Re; Pr'

where, Re is usually based on the cavity circumferential velocity magnitude relative to the surface and a representative length scale of the cavity meridional section and , and EF are empirical factors. Even with the factors carefully adjusted to match the experimental data, considering its basis, the validity of the correlation is clearly limited, at most, in the axisymmetric situation.

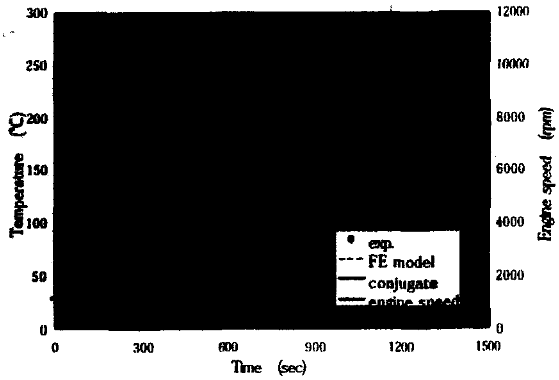


Fig. 10 Comparisons of measured and predicted transient temperature for the point D4

Fig.15 shows that the conjugate method improves the accuracy at the bore. It gives a closer agreement during the acceleration, while it still under-predicts the level of temperature. Swirl ratio around the corresponding bore location predicted by the conjugate method is 0.85 which is consistent with the FE method. The value suggests hat the empirical HTC correlation, which only considers the swirl velocity, cannot give any higher heat transfer at this position. Figure 12(a) shows the velocity vectors in a meridional plane through the bolt axis predicted by the conjugate method. It clearly reveals the strong secondary motions shift the fluid from the cavity core to the disc surface and then direct it towards the inner and outer side along the surface. As a result of this process the heat transfer at the measured point, which agrees with the location the secondary flow impinges, is enhanced. Such an vortical flow features cannot be seen in the calculation without a bolt as shown in Fig. 17(b), which clearly suggests this secondary motion is driven by the circumferentially-discrete bolt protrusions. The present conjugate method captures the important three-dimensional flow pattern, which results in the aforementioned more correct heat transfer level.

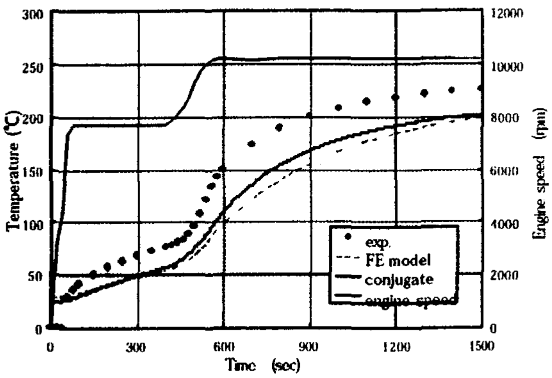


Fig. 11 Comparisons of measured and predicted transient temperature for the point D3

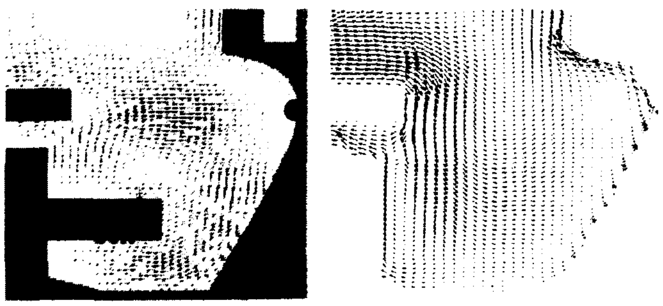


Fig. 12 Predicted velocity vector around disc bore (meridional plane through the bolt axis) (right) with bolt (left) without bolt

In Fig. 13, metal temperature in the solid region and relative total air temperature as well as velocity vectors in the fluid region predicted by the conjugate method are shown in several different time points. The change in the flow field and the resultant thermal field during the acceleration is more clearly understood with the conjugate results and it will be very useful tool for secondary air and thermo-structural design of engine hot section components.

Conclusion

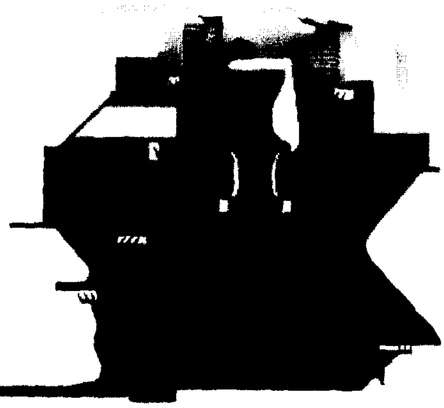
A computational code has been developed to solve the fluid-solid conjugate problem. Various special thermal models particularly designed for a turbomachinery secondary air system were devised and implemented in the code, which allowed the time-dependent and full three-dimensional conjugate analysis of an actual engine disc system in industry-affordable computational resources by partly simplifying computationally expensive region without loosing physical consistency in the areas of main interest.

Application of the code to a single stage high pressure turbine rotor stator disc system demonstrated that the accuracy of the surface temperature in the steady-state condition was good enough. It was believed to be the best level of accuracy with the present CFD methods. It appeared that the ingress model will need to be improved to obtain better agreement in the thermal field around disk rim region.

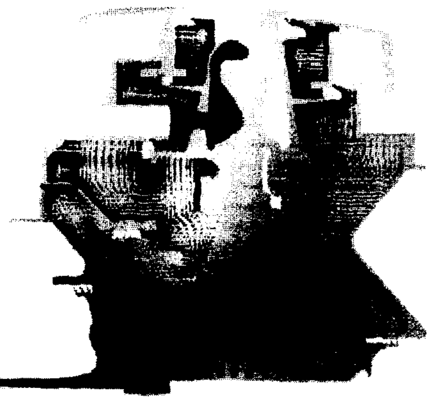
While a clear superiority of the conjugate method to the conventional FE method could not been seen in the steady-state, the validation with transient condition suggested the advantage of the present method. Particularly in the three-dimensional flow field, the FE method using the standard correlation showed poor accuracy and the physically more reasonable surface heat transfer provided by the conjugate method significantly improved the accuracy of transient temperature profile. The transient thermal field is generally much more important than the steady-state field because it is not the thermal stress in steady-state but the one in transient which determines the fatigue cycle of the disc components. Thus it is of great benefit to use the present conjugate method in a three-dimensional framework.

Acknowledgements

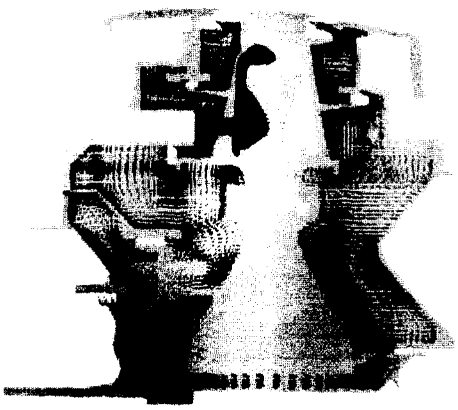
The authors would like to express their thanks to the New Energy and Industrial Technology Development Organization (NEDO) and the Ministry of Economy, Trade and Industry (METI), who gave them the opportunity to conduct "Research and Development of Environmentally Compatible Propulsion System for Next-Generation Supersonic Transport (ESPR) project".



(a) T=500s



(b) T=900s



(c) T=1500s

Fig. 13 Predicted transient flow and thermal field

References

[1] Ho, Y. H. et al., 1996, "Numerical Simulation of Secondary Flow in Gas Turbine Disc Cavities, Including Conjugate Heat Transfer", ASME Paper 96-GT-67

[2] Verdicchio, J. A., Chew, J. W., and Hills, N. J., 2001, "Coupled Fluid/Solid Heat Transfer

Computation for Turbine Discs", ASME Paper 2001-GT-0205
[3] Montenay, A., Pate, L., and Duboue, J., 2000, "Conjugate Heat Transfer Analysis of an Engine Internal Cavity", ASME Paper 2000-GT-282
[4] Kelkar, K. M., Choudhury, D., and Ambrosi, M., 1991, "Numerical Method for the Computation of Conjugate Heat Transfer in Nonorthogonal Boundary-Fitted Coordinates", Numerical Heat Transfer, B, Vol. 20, pp.25-40
[5] Karki, K. C., and Patankar, S. V., 1989, "Pressure Based Calculation Procedure for Viscous Flows at All Speeds in Arbitrary Configurations", AIAA J., Vol. 27, pp.1167-1174
[6] Rhie, C. M., and Chow, W. L., 1983, "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", AIAA J., Vol. 21, pp.1525-1532
[7] Launder, B. E., and Spalding, D. B., 1974, "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp.269-289
[8] Chieng, C. C., and Launder, B. E., 1980, "On the Calculation of Turbulent Heat Transport Downstream from an Abrupt Pipe Expansion", Numerical Heat Transfer, Vol. 3, pp.189-208
[9] Viegas, J. R., and Rubesin, M. W., 1985, "On the Use of Wall Functions as Boundary Conditions for Two-Dimensional Separated Compressible Flows", AIAA-85-0180
[10] Virr, G. P., Chew, J. W., and Coupland, J., 1993, "Application of Computational Fluid Dynamics to Turbine Disc Cavities", ASME Paper 93-GT-89
[11] Okita, Y., and Iacovides, H., 2001, "Comparisons of High-Reynolds-Number EVM and DSM Models in the Prediction of Flow and Heat Transfer in Internal Cooling Passages of Turbine Blades", Proceedings, ECCOMAS CFD Conference, Swansea, U.K., Sep. 2001
[12] Mcgreehan, W. F., and Ko, S. H., 1989, "Power Dissipation in Smooth and Honeycomb Labyrinth Seals", ASME Paper 89-GT-220
[13] Hills, N. J., Chew, J. W., and Turner, A. B., 2001, "Computational and Mathematical Modeling of Turbine Rim Seal Ingestion", ASME Paper 2001-GT-0204
[14] Ishida, K., 1999, "Effect of Fin Overlap on Seal Characteristics of a Gas Turbine Rotor Stator System", Proceedings, International Gas Turbine Congress, Kobe, Japan, Nov., 1999
[15] Monico, R. D., and Chew, J. W., 1993, "Modeling Thermal Behaviour of Turbomachinery Discs and Casings", Proceedings, Heat Transfer and Cooling in Gas Turbines, AGARD-CP-527, pp.24.1-24.9
[16] Fujimura, T. et al., 1998, "Research and Development of High Temperature Core Engine for HST", AIAA Paper 98-3279
[17] Saravanamuttoo, H., 1990, "Recommended Practices for Measurement of Gas Path Pressures and Temperatures for Performance Assessment of Aircraft Turbine Engines and Components", AGARD-AR-245
[18] Schlichting, H., 1979, "Boundary Layer Theory", 7th edition, McGraw Hill, NY
[19] Kao, K-H., and Liou, M-S., 1997, "Application of Chimera/Unstructured Hybrid Grids for Conjugate Heat Transfer", AIAA J., Vol. 35, pp.1472-1478
[20] Pozzi, A., and Lupo, M., 1990, "The Coupling of Conduction with Forced Convection in Graetz Problems", J. Heat Transfer, Vol. 12, pp.323-328
[21] Rahaim, C. P., Kassab, A. J., and Cavalleri, R. J., 2000, "Coupled Dual Reciprocity Boundary Element/Finite Volume Method for Transient Conjugate Heat Transfer", J. Thermophysics and Heat Transfer, Vol. 14, pp.27-38
[22] Roy, R. P., Xu, G., and Feng, J., 2000, "Study of Main-stream Gas Ingestion in a Rotor-Stator Disk Cavity", AIAA-2000-3372

Nomenclature

b	outer radius of cavity
Cp	specific heat at constant pressure
Cw	non-dimensional mass flow rate
G	gap ratio (=s / b)
h	heat transfer coefficient or channel half-height
k	turbulent kinetic energy
l	chord length
	mass flow rate
Nu	Nusselt number
Pe	Peclet number
Pr	Prandtl number
q	heat flux
r	radius
	rotational Reynolds number
RT	turbulence Reynolds number
s	axial width of rotating cavity
T	temperature
Tr	relative total temperature
Tr,cool	minimum cooling air temperature
Tr,gas	mainflow gas temperature
u	axial velocity
y+	non-dimensional wall distance

Subscripts

A	advection
ad.	adiabatic
c	convetive heat transfer
conj.	conjugate
f	fluid region
i	inner
in	inlet
o	outer
s	solid region
w	windage