Proceedings of GT2006
ASME Turbo Expo 2006: Power for Land, Sea and Air
May 8-11, 2006, Barcelona, Spain

GT2006-90646

ADVANCED TURBINE AERODYNAMIC DESIGN UTILIZING A FULL STAGE CFD

Eisaku Ito*
Takasago Research &Development Center,
Technical Headquarters,
Mitsubishi Heavy Industries, Takasago, Japan
Email*: eisaku_ito@mhi.co.jp

Sunao Aoki
Technical Headquarters,
Mitsubishi Heavy Industries, Tokyo, Japan

Akimasa Muyama, Junichiro Masada
Takasago Machinery Works,
Mitsubishi Heavy Industries, Takasago, Japan

ABSTRACT

Gas turbines for power generation are required to operate more efficiently than ever before for both economic and environmental reasons. Because of this situation, an advanced multistage turbine design and optimization system is required to improve upon existing turbine designs where viscous CFD codes had already been applied on a single row or single stages basis. An advanced CFD code for multistage design applications has been developed at Mitsubishi Heavy Industries (MHI) and has been applied to the redesign of a four stage single shaft turbine. The front 3 stages of the turbine are highly cooled using about 20% cooling air. The outstanding performance of this redesigned turbine has been demonstrated at MHI's engine test facility.

This paper focuses on the customization of the Denton code [5] for industrial usage, the validation of the customized code employing experimental data, and finally the use of the code in executing a successful redesign. Code development and validation are discussed in terms of prediction accuracy for the basic aerodynamic design parameters such as exit flow angle and cascade losses. Through-flow design parameters such as pressure ratio and reaction of each stage are also addressed. Especially important in modern high temperature turbines is the location and distribution of cooling and leakage air being introduced into the main gas-path. The proper treatment of these flows is very important because of the mixing losses and the temperature migration downstream. These important considerations in any analysis approach are discussed and it is shown how they are treated in the customized CFD code. Consistency between the customized CFD code and other parts of the existing aerodynamic design procedure are carefully examined. This is important because aerodynamic parameters have different modeling fidelities in the different parts of the design system. Computer execution times are a very important consideration when utilizing advanced CFD codes. This issue is addressed from the perspective of an industrial design organization.

In validating the customized code, special attention was placed on tip clearance leakage flow behavior and seal air migration from the hub wall. Local changes of total pressure and temperature distributions affect the local velocity triangles and local static pressure distributions on the airfoil and end-wall surfaces. Airfoil section geometry and three-dimensional stacking to maximize the turbine efficiency are also considered and discussed.

The validated code was subsequently used to execute a redesign of a large frame industrial turbine. This is discussed in some detail. The redesigned turbine has completed full scale engine testing and has been shown to have met all design goals. The CFD predictions are compared with special measurements taken in the engine such as the inter-stage span-wise total pressure and temperature distributions as well as the efficiency trend versus engine load. These comparisons prove the capability of the advanced multistage CFD code.

NOMENCLATURE

Pt	total pressure
m	mass flow rate
γ	specific heat ratio
M	Mach number
T	total temperature
V	velocity
Φ	angle between main flow and seal air injection
η_{cool}	cooled turbine efficiency
P	turbine power

subscripts

actual	actual work
seal	seal air
gas	main gas flow
ideal	isentropic work based on expansion ratio of each flow

INTRODUCTION

Many numerical models introducing higher and higher fidelities have been proposed for the computational fluid dynamics (CFD) analysis of turbines to improve the simulation accuracy. In these reports, already designed turbine geometries were analyzed and very detailed flow phenomena were studied. Several of these papers (references) focused on the design point of view rather than on detailed flow phenomena have been found. The following are some examples for this point of view.

It is known that turbine flow capacity, especially of the first vane, is important to not only the turbine aerodynamic design but also to the cycle matching of the compressor. It is noted that in the design process, a full turbine performance analysis is necessary in order to predict accurate flow capacity of the turbine, because of both the flow channel geometry and 3D flow phenomena [1]. This is an important consideration in the design of a high temperature turbine, due to the high flux of cooling air addition. Here the mass flow rate at any axial location of a turbine must be closely accounted for.

Seal or leakage flows from a secondary air system changes the flow field of main stream as well and can have significant effects on the reaction and work split in a turbine. The effect of secondary air injection on the performance of a transonic turbine stage is investigated. Improvement of the performance by optimizing the direction of injection into gas path is shown to have an efficiency impact of from 0.3% to 1% for one percent of seal air [2]. This includes two aspects. One is a change of tangential component of injected velocity which appears in the Euler turbine equation and the second is the change in mixing loss due to injection.

Also, film cooling impact on turbine efficiency have been directly measured and assessed in the blow-down test rig [3]. Efficiency drops of 0.5% with one percent additional cooling flow have been reported.

In applying CFD to a multistage turbine, with or without cooling air addition, gas property change through turbine from inlet to exit is not negligible. Properly accounting for gas property changes is essential for the prediction of rear stage gas temperatures, although it is considered to have small effect on efficiency. This has been studied [4].

Each of the above well executed studies was focused on an important phenomenon in the turbine, however very few studies have focused on the whole turbine design application. The authors in this paper have taken a turbine design point of view. They have developed an advanced turbine aerodynamic design method utilizing a customized version of Denton's well known multistage CFD code.

After a careful study it was found that Denton's code is the most suitable code for this purpose because important cascade parameters and through-flow parameters had already been verified. Also the code is not a "black box" code, so customizing the code to the users' special requirements is possible.

DESIGN SYSTEM UTILIZING MULTISTAGE CFD

Figure 1 is a work flow chart that shows the comparison between the conventional design procedure and the advanced design procedure utilizing the multistage CFD. To start the engine development, the cycle is selected employing a one dimensional turbine through-flow code (meanline) with an initial cooling flow schedule estimated from existing engine experiences. In the conventional design, turbine optimization was done with mainly one dimensional loss model which is incorporated into the one dimensional design tool. Then designers would go to an axis-symmetric through-flow code to establish the vortexing of the design. This would be followed by the designing of the airfoil sections. Although initial design trial goes through this procedure, it goes directly to the multistage CFD analyses to confirm feasibility of the design concept from a three dimensional flow standpoint.

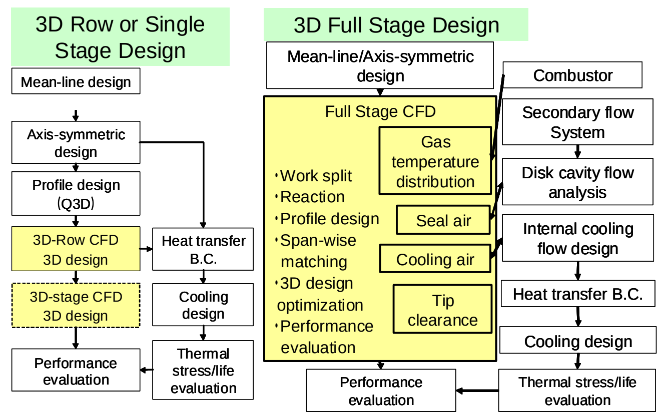

Fig.1 Conventional design procedure (left) and advanced procedure (right) utilizing three dimensional full-stage CFD

In this aerodynamic design procedure, rapid turnaround times is the key to first establishing the feasible efficiency level and then refining the aero design before detailed design begins. Just after meanline design and axis-symmetric design, using the preliminary vane and blade with typical geometries, cooling flow schedule should be easily transferred to the input of the full-stage cooling flow schedule without detailed heat transfer design, because heat transfer design follows aerodynamic design execution. Therefore a prescribed standard modeling which transforms a previously defined secondary flow schedule design into a typical distribution of cooling flow on the profile surfaces and end-wall surfaces. Then the detailed design study of cooling scheme can be started by using the initial 3D heat transfer boundary condition which is automatically outputted by the system from the multistage CFD computation (see Figure 1).

In terms of aerodynamic design, work-split and reaction, which are established by the spanwise loss and flow angle are compared between the meanline and axis-symmetric design simulations, and design iterations are executed to assure consistency.

MULTISTAGE CFD CODE

Features of CFD as a turbine design tool which is incorporated in the above mentioned design procedures are described here in detail.

First of all, flow phenomena to be solved are at least three dimensional steady viscous flows with the multi-stage environment. And they are sometimes critical or beyond steady design guideline, time-accurate phenomena are solved. Flow capacity is the key to solve stage work-split and reaction. Especially secondary air system like cooling/seal/leakage flows should be modeled and flow mixing with main flow should be calculated for loss prediction. And in full-stage condition, gas property should be properly considered. And predicted efficiency should be about within one percent of measured efficiency and most importantly, efficiency change should be predicted against design modifications and operational conditions.

Secondly, turn around time should be well less than one day to perform design refinement studies.. Mesh size may be preferable if it is less than computer memory to avoid data transfer between machines although it is not the limitation to a parallel code. In case that these conditions cause the computational mesh to be too coarse for important flow phenomena in the design, additional confirmation with finer grid may be necessary.

The above two features here are the requirements that were self imposed for the new CFD code for the design procedures described in Figure.1

As the CFD code which satisfies the above requirements, Denton's Multip code is selected. Multip is a finite volume solver of Reynolds Averaged Navier Stokes which bases on the explicit SCREE scheme [5]. The grid is structured H type mesh and mixing plane approach is adopted. These features are suitable for customizing the code as the design system, because pre/post processing of designs and analyses can be topologically similar for each row. In terms of the cooling/seal flow addition, a simple model is adopted which described below.

The computational mesh generator for the new code was developed in-house and connected to the existing design system. This mesh generator is based on one previously used in a single row viscous flow analysis code.

Cooling air and Seal air modeling are important aspects of the customization. In the high temperature gas turbines, about 20% of cooling flow is necessary to assure turbine durability. This amount of additional air affects the turbine through-flow parameters, such as pressure ratio of each stage (work split), reaction and total flow capacity determined by flow capacity characteristic of each row. Therefore, with the high flux ratio of cooling air, the mass flow rate at any axial position (or locally, if necessary) from inlet to exit of turbine should be simulated accurately in the design. Moreover, total efficiency is affected by losses generated in the mixing phenomena with main flows.

In the code, these cooling air and seal air are treated as the source term from the surfaces of the airfoils and end-walls. Total pressure, total temperature, injected flow angles of the cooling and seal air are specified as the input. Then the cooling and seal flow rates are calculated with the surface static pressure of main flow field. In the customization, mass flow rate can be chosen as the input parameter in the iteration between aerodynamic design and cooling design. In order to reduce turn around time, discrete rows of cooling holes are modeled as slots in this model and areas of cooling holes are specified shown in Figure 2. Secondary flow cavities and gaps are modeled as smooth surfaces and slope with the exact radii of hot operating condition at leading edge and trailing edge to keep the connectivity with conventional design concept.

At this point, it should be noted that conservation of energy should be kept for the cooled turbine system in the scope of design. It is dependent on the inlet boundary location to be analyzed. Usually, detailed heat transfer design comes after aerodynamic design of the vane and blade geometries. So, before the heat transfer design started (concurrent design being studied), boundary conditions from secondary flow system design which usually proceeds side by side with cycle study are convenient to be used. Most simple modeling is that total temperature of cooling air is specified as the value at the inlet temperature into internal cooling structure like serpentine passages, and cooled surface of the vanes and blades are modeled as adiabatic surface. In this simple modeling, whole energy balance is conserved and whole cooled turbine efficiency is easily assessed without detailed cooling structure. However, local gas temperature near cooling holes may be too low because input temperature of cooling air does not include the heat exchange with cooled metal surfaces. Another modeling is that cooled surface is modeled using enthalpy source term and cooling air temperature is input as the cooling holes exit temperature. In the latter case, input is more complicated and total energy conservation depends on the exactness of the cooling modeling but in the author's experience does not create any significant problems.

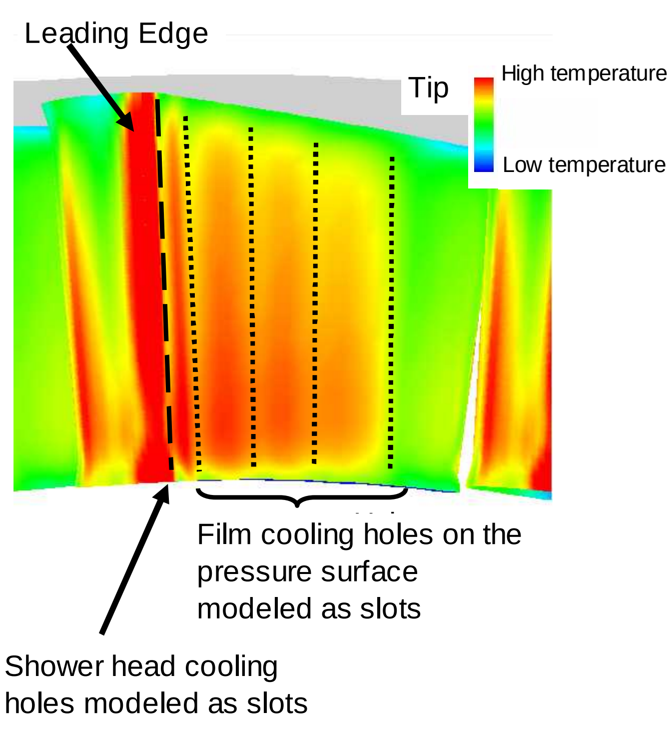

Fig.2 Film cooling model of CFD in the design iterations on first vane pressure surface (Color: Total temperature)

Tip clearance modeling is also very important because clearance flow changes the downstream flow field. The most basic modeling is to distribute several span-wise meshes in the tip clearance to allow the leakage flow from pressure side to suction side of the tip. These are currently investigated by many researchers. For example, most recent work was reported by Rosic et al(2005).

VALIDATION OF CFD

In a similar approach to that used to validate single row CFD codes, two dimensional cascade data was used to verify basic cascade prediction accuracy such as exit flow angle, profile loss level and loss due to mixing between cooling flow and main flow are confirmed. Secondly, mass flow capacity. using annular cascade test data. In the range from subsonic to transonic region which covers single shaft heavy duty gas turbines, mass flow rates are predicted within about 1% accuracy in Figure 3 which is good considering the uncertainties in the engine assembly.

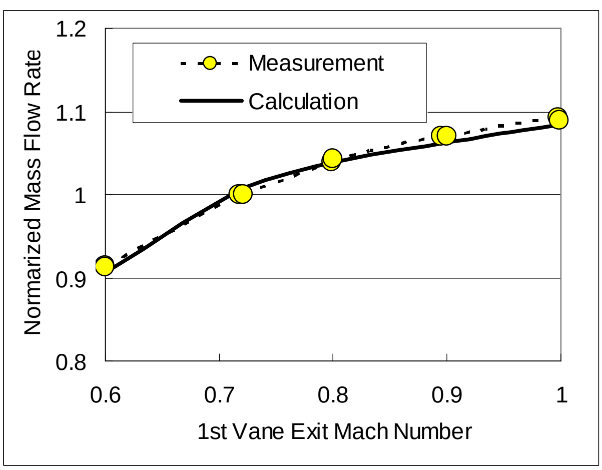

Fig.3 Measured and predicted mass flow rate for an annular cascade of first vane

The CFD modeling for multistage turbines is verified using two stage turbine rig data. (see Figure 4 and Table 1) This cold test rig is the scaled model for the typical front stages of the high temperature gas turbine which have the high hub-to-tip ratio 0.85. Spanwise pressure and temperature rakes are used at the inlet and exit of the turbine to measure the efficiency characteristics against various operating conditions. Extracted power is absorbed and rotating speed is controlled by a dynamometer. Pressure ratio is controlled by the inlet pressure from the air source compressor with the atmospheric turbine exhaust.

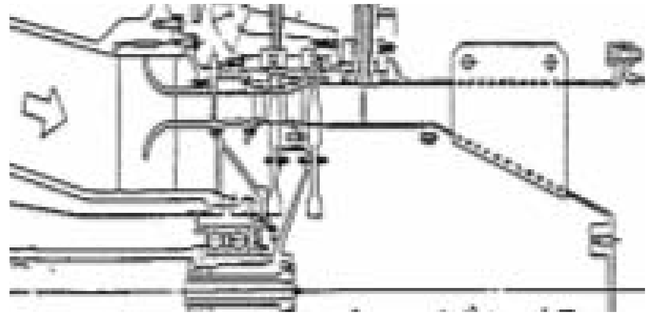

Fig.4 Schematic of Model Turbine

Table 1 Geometrical and Operational Parameters of Model Turbine

1st Blade Height(mm)	47
2nd BladeHeight(mm)	65
1st Blade Hub to Tip Ratio	0.85
Design speed(rpm)	7500
Number of Stages	2
Inlet Pressure(Mpa)	0.4
Inlet Temperature(K)	480
Mass Flow(Kg/s)	13

Important measurement items for the rotating rig test are first of all span-wise distributions of pressure, temperature and swirl angles, and secondly efficiency change against the operational conditions. Span-wise distributions predicted by the CFD with computational mesh in Figure 5 are compared with the rig data in Figure 6-8. These comparisons verified the capability of the tool. Most important item is the capability of efficiency change with design change. Instead of this, Efficiency trend curves are plotted against measured data in Figure 9. This figure shows that the code predictions are quite accurate even for off design conditions and it is confirmed that the modeling and mesh size are acceptable for design purposes.

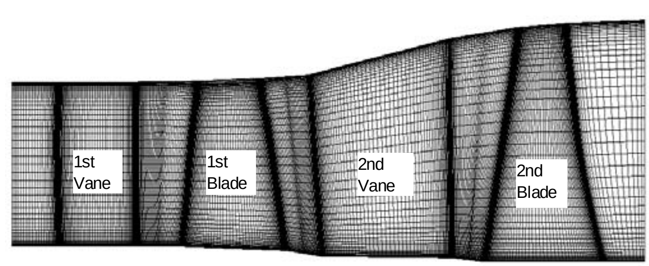

Fig.5 Computational grid of CFD

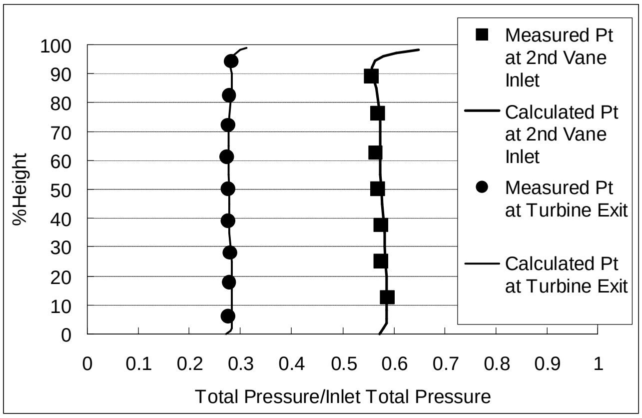

Fig.6 Measured and predicted total pressure

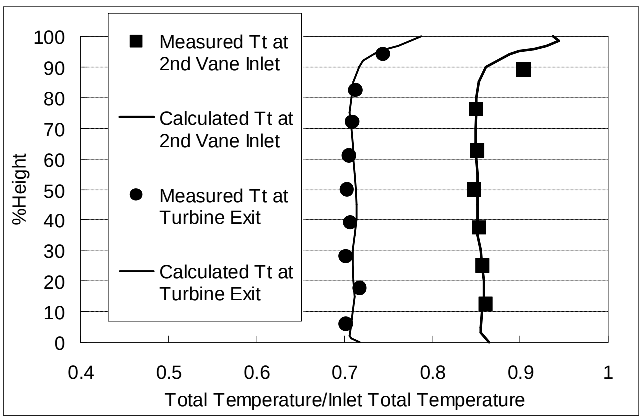

Fig.7 Measured and predicted total temperature

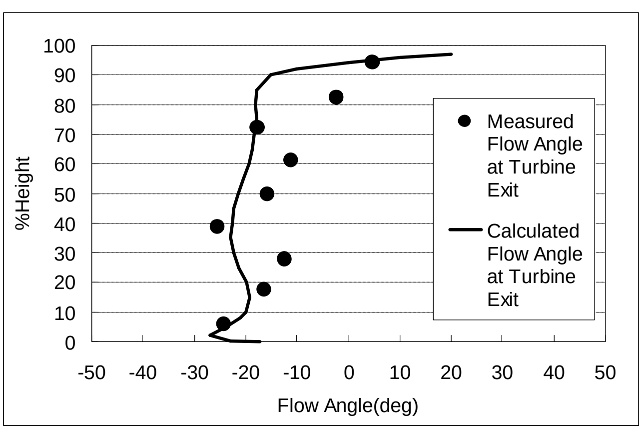

Fig.8 Measured and predicted turbine exit swirl angle

In addition to the above verifications, several important flow phenomena which a multi-stage CFD can predict to improve aerodynamic design are discussed here. One is the effect of blade hub inlet seal air on the cascade flow. Another is the effect of the tip clearance flow on the downstream vane flow.

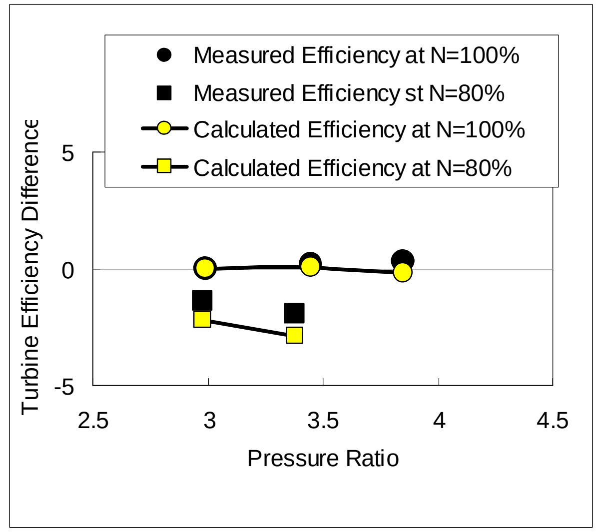

Fig.9 Measured and predicted turbine efficiency at different operational conditions

Effects of blade hub inlet seal air migration on cascade flow

In the high temperature gas turbine, it is impossible to reduce seal air to zero to prevent hot combustion gas into the rotor cavity. Interaction of secondary flow system and main flow, and flow field in cavity itself are studied by many researchers. Recently, analyses which can compute main stream and cavity flows at the same time are introduced for research work. In the Multip code discussed in this paper, secondary flows are simply modeled with source term from simple slits on the imaginary surface between stator shroud and blade platform. Similar to the way of input of film cooling air, input direction or tangential velocity can be specified. Importance of method is explained. [6] Potential interaction between vane exit and blade LE stagnation can be analyzed by unsteady CFD computations.

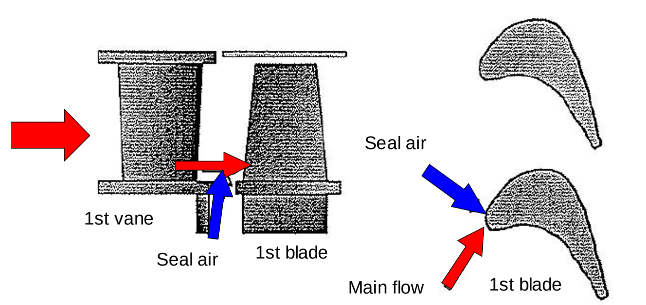

Fig.10 Image of blade hub seal air

Important phenomenon from the aerodynamic view is the effect of the seal air amount on the stage performance.

Usually, seal air mixing loss is assessed by the simple Shapiro's model [7] Equation (1) is very good modeling and verified empirically. This is widely used in industry in the mean-line codes.

$$\frac{dPt}{Pt} = \frac{dm}{m}\left[-\frac{\gamma M^2}{2}\left(\frac{Tseal}{Tgas}-1\right) + \gamma M^2\left(1-\frac{Vseal}{Vgas}\cos\Phi\right)\right] \tag{1}$$

However, this loss is only for mainstream loss and loss of seal air and effect of the disturbance of the flow field should be taken into account [8, 9]. This is extremely difficult to express by the simple equation. In order to assess these effects, cooled efficiency is defined.

$$\eta_{cool} = \frac{Pactual\_total}{Pmain\_ideal + \Sigma Pseal\_ideal + \Sigma Pcool\_ideal} \tag{2}$$

In this cooled efficiency, film cooling air and seal air are treated as the mass flow and energy injection which expands from injected pressure to turbine exit pressure. All of injections are summed up in the definition.

In the rotating rig test, for example, in case of blade inlet hub seal air, measured efficiency drop is sometimes larger because suction surface flow field is disturbed by the locally negative incidence. It is thought due to the low temperature and low tangential velocity of the incoming seal flow. In this case, Shapiro's model under predicts the efficiency drop. With applying Multip code, this effect can be counted with better accuracy showed in Figure 11. This is plot for a single stage cooled efficiency. It should be noted that difference between measured data and a simple model becomes larger with amount of seal air injection. This effect of course depends on the cavity structure and seal location in the secondary flow system. For each location of incoming flow, specified input models should be examined carefully.

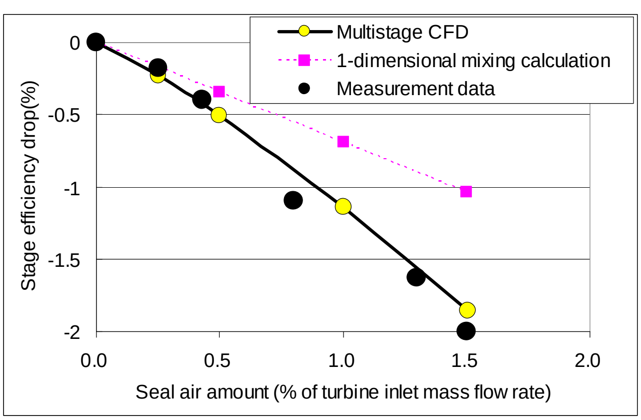

Fig.11 Cooled efficiency drop due to the seal air injection at hub inlet of rotating blade. Comparison between one dimensional mixing model and CFD

Effects of tip clearance flow on the downstream vane flow

Tip leakage flow of row-1 blade is a negative incidence jet on the suction surface of row-2 vane tip region like Figure 12. With this flow impingement, suction surface static pressure distribution is high locally and this pressure distribution drives the surface boundary layer from tip to hub side due to the span-wise pressure gradient at trailing edge. This is confirmed by the picture of the row-2 vane after operation from an existing power plant. It is seen that the flow trace on the suction surface which was driven from the leading edge tip region to downward (see Figure 13). In comparison with the observed flow trace on the used vane and the surface streamline computed by the CFD, the tip leakage effect on the surface flow direction on the downstream vane seems to be simulated well.

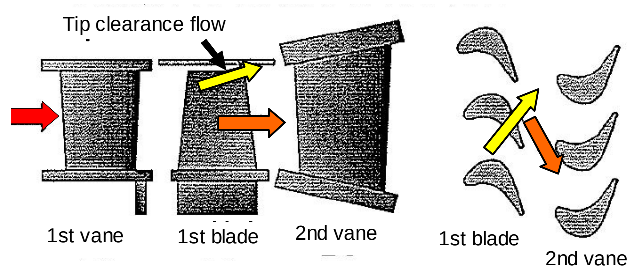

Fig.12 Image of tip clearance flow and effect on the downstream vane

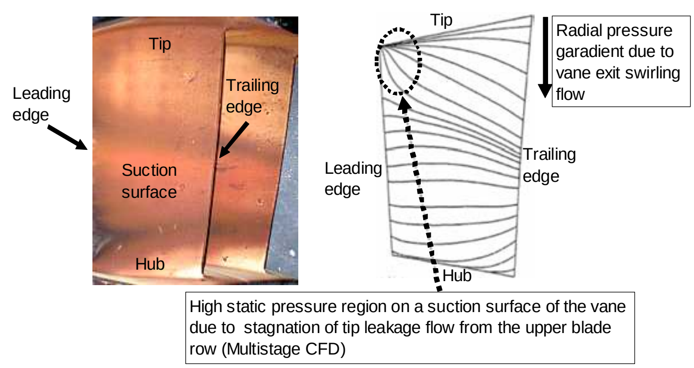

Fig.13 Flow trace of suction surface of second vane after operation and CFD prediction of streamline

This tip flow phenomenon makes it difficult to control the static pressure distribution and causes the strong secondary flow. As well as aerodynamic difficulties, "hot jet" through tip clearance without being extracted of turbine work i.e. with higher total temperature than that of flow through row-1 blade throat, special care should be taken to heat transfer design. Operational evidence of the surface trace is seen on the TBC coating shown in Figure 14.

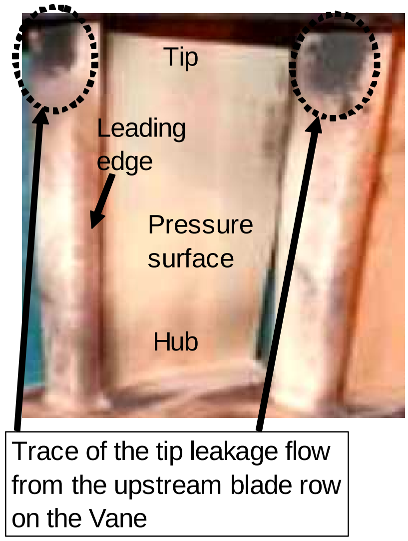

Fig.14 Trace of the impingement of tip clearance flow on the suction surface of second vane

M701G2 TURBINE DESIGN GOALS

Full-stage CFD design is applied to M701G2 design which is redesign from original M701G. The design target is to increase efficiency by 0.5%. Design specification is shown in Table 2. Pressure ratio is increased by 24% and mass flow rate is increased by +13 % from M701G1 with the same turbine inlet temperature. In addition to these, design restrictions are to use the exactly same gas path geometry and to try to use the same vanes and blades as much as possible to meet the very tight design schedule and to maintain the reliability which has been proven by M701G1.

These specifications and restrictions cause an increased aerodynamic load factor. In this condition, usual approach is to increase the axial chord of critical rows. However, because of the restriction of using the same gas path geometry, very tight axial gaps are limiting the possibility of this approach. Moreover, to use the same vanes and blades as much as possible, mismatching of local velocity triangles cannot be avoided. Consequently, it becomes a very difficult aerodynamic design.

To meet the requirement, it is necessary to simulate and understand the full stage three dimensional turbine aerodynamic matching. It is because of these demanding requirements that a validated multistage CFD design tool is strongly required.

Table 2. Design Conditions of M701G2

Performance	M701G	M701G2
G/Tpower(MW)	271	334
Pressure Ratio	17	21
Air Flow	Base	Base+13%
G/Tthermal Efficiency (%-LHV)	38.7	39.5

APPLICATION OF FULLSTAGE CFD TO THE M701G2 TURBINE DESIGN

After these preparations, M701G2 turbine design was done with the boundary condition of the secondary systems in Figure 15. Full-stage CFD grids is shown in Figure 16. All cooling flows are modeled as slits on the surface of profiles, and all seal and leakage flows are modeled as slots on the surface of end-walls.

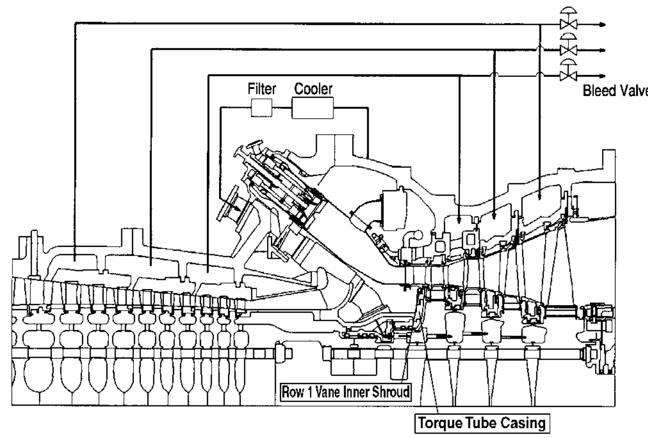

Fig.15 M701G2 turbine gas path and secondary flow system [10]

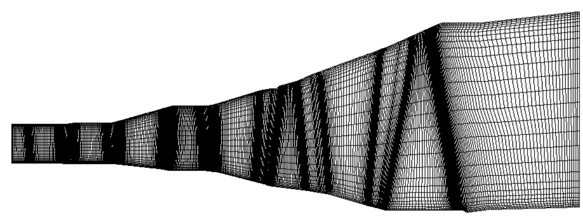

Fig.16 Computational grid for design iteration

With multistage CFD code, design iterations were done on a three dimensional basis by changing the each profile geometry considering the multistage interactions and matching. Some design results of interest are shown below.

Reduction of tip leakage loss

Clearance leakage flow and vortices are driven mainly by the pressure difference between pressure surface and suction surface. So it is considered that reducing the aerodynamic loading is preferable. So vortex design which reduce the tip loading and wide chord design are adopted.

Fig.17 Designed first blade geometry [12]

Reduction of tip clearance flow effects on row-2 vane

Considering the tip leakage effects shown in Figure 13 and Figure 14, geometry of tip section of the row-2 vane can be optimized. Firstly, inlet metal angle of the tip section is designed to match the tip leakage flow direction from row-1 blade. Secondly, the leading edge wedge angle of the tip section is designed larger than usual to achieve robust characteristic against unsteady incidence change among leakage. Thirdly, leading edge radius is larger than usual to reduce heat load of the hot jet impingement. Designed second vane is shown in Figure 18. Similar approach is taken in the first blade hub geometry against the seal air migration.

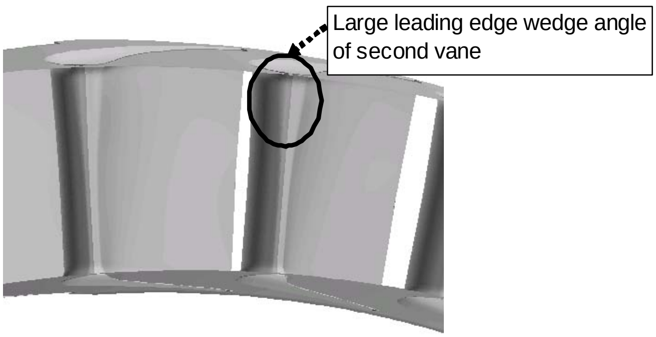

Fig.18 Optimized geometry of second vane

ENGINE TEST VERIFICATION OF DESIGN

Figure 19 is the designed turbine rotor of M701G2 on the lower casing. Although it is important to consider the cooling flow in the full-stage CFD simulation, verification in laboratory test is very difficult due to the complexity of cooling scheme for rotating rig test. In this situation, engine test is an only opportunity to acquire the data like Figure 20. Span-wise total temperature and total pressure overall performances of the turbine, like a flow capacity and pressure ratio, reaction and efficiency are confirmed.

Engine data in Figures 21 and Figure 22 show that predicted accuracy is in quite good agreement with measurements. At the inlet of the third vane, measured total temperature data are scattered in the circumferential locations. This effect can be analyzed by an unsteady CFD.

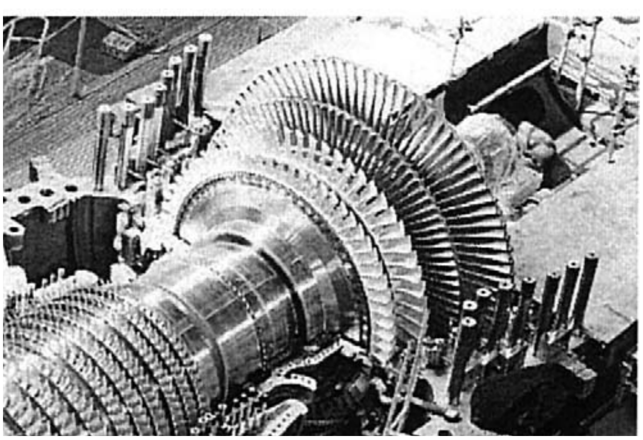

Fig.19 Turbine rotor of M701G2 [11]

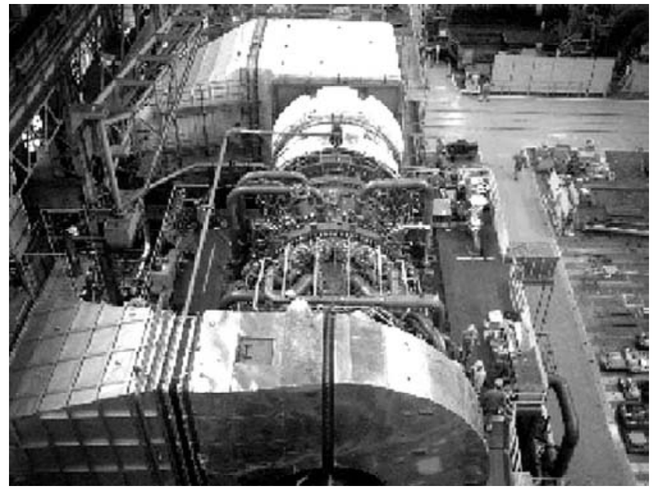

Fig.20 Shop test of M701G2 [11]

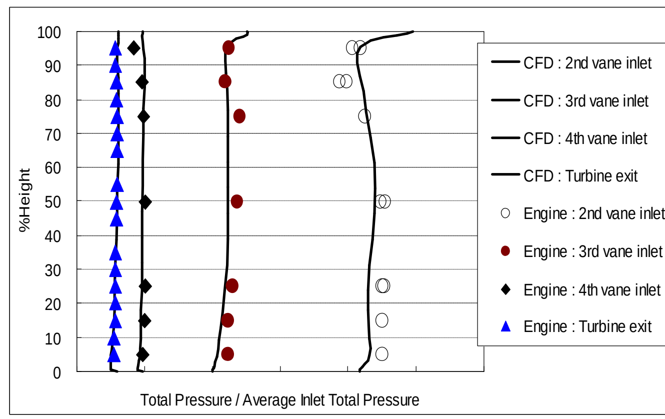

Fig.21 Measured and predicted total pressure

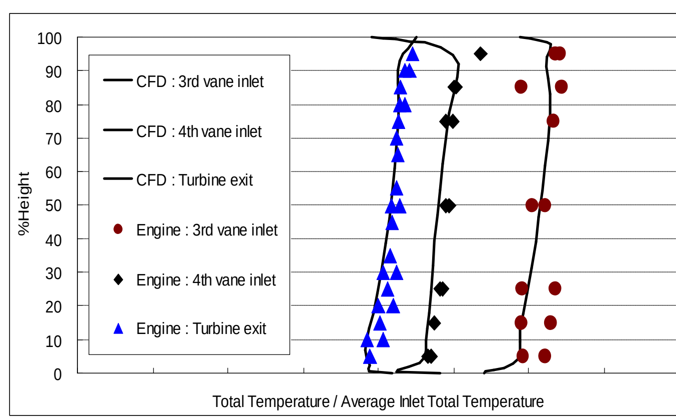

Fig.22 Measured and predicted total temperature

Figure 23 is a picture of the first vane which operated in M501G1in commercial operation. M701G2(50Hz engine) first vane is also used in the M501G1(60Hz engine), and flow field around the vane is very similar. Flow traces from the discrete film cooling holes are observed. Comparison with the surface gas temperature distribution in Figure 2 shows that even with the slots model of film cooling holes in an iterative design period before the detailed cooling design is very good. Fullstage CFD aerodynamic design results predicted more than 0.5% improvement in efficiency. Engine testing confirmed these predictions.

Fig. 23 Trace of film cooling air on the pressure surface of the first vane [12]

CONCLUSION

Denton's multistage CFD has been customized in terms of cooling flow and seal flow injection and gas property to apply to the design of high temperature gas turbine as the CFD design tool. The code has subsequently been verified employing cold flow rig test data. In order to apply CFD to an iterative aerodynamic design optimization before the detailed heat transfer design, it is necessary to assume a proper cooling scheme by reviewing the existing engine from the simple secondary flow schedule in the cycle deck.

In a multistage environment, important flow phenomena which affect on a turbine aerodynamic performance are discussed and compared with the conventional design.

To compensate these phenomena, new design concepts are withdrawn from full-stage computation. These concepts are applied to the actual engine aerodynamic design and adopted to the large scale high temperature gas turbine.

Engine testing has demonstrated that all performance goals for the turbine redesign were met. These results also show that the multistage CFD prediction is in reasonably good agreement with measurements.

References

[1] Afanasiev, I.V., Granovski, A.V., Karelin, A.M., Kostege, M.K., 2004, Effect of 3D Vane Shape on the Flow Capacity, ASME GT2004-53095

[2] Girgis, S., Vlasic, E., Lavoie, J.P. and Moustapha, S.H., 2002, The Effect of Secondary Air Injection on the Performance of a Transonic Turbine Stage, ASME GT-2002-30340

[3] Keogh, R.C., Guenette, G.R., Spadaccini, C.M., Sommer, T.P., Florjancic, S., 2002, Aerodynamic Performance Measurement of a Film-Cooled Turbine Stage-Experimental Results, ASME GT-2002-30344

[4] Yao, Y., Amos, I. G., 2004, Numerical Study of Axial Turbine Flow with Variable Gas Property, ASME GT2004-53748

[5] Denton, J.D., 1992, The Calculation of Three-Dimensional Viscous Flow Through Multistage Turbomachinery, ASME Journal of Turbomachinery, Vol. 114

[6] Rosic, B., Denton, J D., Pullan, G., 2005, The Importance of Shroud Leakage Modeling in Multistage Turbine Flow Calculations, GT2005-68459

[7] Shapiro, 1953, The Dynamics and Thermodynamics of Compressible Fluid Flow: Volume Ⅰ, Ronald Press. New York, pp219-231

[8] Denton, J.D., 1993, Loss Mechanisms in Turbomachines, ASME J. Turbomachinery, 115,621-656

[9] Greitzer, E.M., Tan, C.S., Graf, M.B., 2004, Internal Flow, Cambridge University Press, pp234-241

[10] Maekawa, A., Magoshi, R., Iwasaki, Y., Development and In-house Shop Load Test Results of M701G2 Gas Turbine, Proceedings of the International Gas Turbine Congress 2003 Tokyo November 2-7,2003 , IGTC2003Tokyo-TS-100

[11] Fukuizumi, Y., Masada, J., Kallianpur, V., Iwasaki, Y., 2003, Application of "H Gas Turbine" Design Technology to Increase Thermal Efficiency and Output Capability of the Mitsubishi M701G2 Gas Turbine, GT-2003-38956

[12] Arimura, H., Iwasaki, Y., Fukuizumi, Y., Shiozaki, S., Kallianpur, V., Upgraded M501G Operating Experience, ASME GT2005-69135

[13] Ito, E., 2004, Trend of the latest technology development of turbine aerodynamics, GTSJ 33rd Gas Turbine Seminar