Proceedings of the International Gas Turbine Congress 2003 Tokyo
November 2-7, 2003

IGTC2003Tokyo TS-085

Conjugate Simulation of Flow and Heat Conduction for Turbine Cooling

Takashi YAMANE[1], Fujio MIMURA[1], Toyoaki YOSHIDA[1]
Shigemichi YAMAWAKI[2], Chiyuki NAKAMATA[2], Ryoji IMAI[2]

[1]Aeronautical Environment Technology Center,
Institute of Space Technology and Aeronautics, Japan Aerospace Exploration Agency
7-44-1 Jindaiji-higashi, Chofu, Tokyo 182-8522, JAPAN
Phone: +81-422-40-3421, FAX: +81-422-40-3440, E-mail: yamane.takashi@jaxa.jp
[2]Ishikawajima-Harima Heavy Industries Co., Ltd.

ABSTRACT

A new conjugate simulation program for flow and heat conduction has been developed based upon a common CFD platform UPACS. It connects flow calculation blocks and solid blocks without using surface temperature values explicitly. The time-lag between flow simulation and heat conduction calculation which is a severe problem in conjugate heat transfer has been overcome by incorporating a heat conduction sub-step method. The developed program has been applied to simulations of new turbine cooling structures which are the integration of impingement and pin cooling device and revealed that the pin configulation changes the cooling efficiency.

INTRODUCTION

Overall efficiency of gas turbine engines can be enhanced by higher turbine inlet temperature but to increase cooling air for turbine blades will lower the efficiency. In order to meet the confilicting demand more precise aerodynamic and thermodynamic design method is necessary.

The conjugate numerical simulation which combines CFD and heat conduction calculation is a solution to the problem. The first step of this technique was to couple different computer programs for flow and heat conduction by using heat transfer coefficient derived from flow simulation result as interface. This method, however, requires some thermal assumption in flow analysis which causes inaccuracies, thus the direct coupling methods of flow simulation and heat conduction have been developed.

One of the studies is the three-dimensional conjugate simulation of internal cooling turbine vane with thermal barrier coatings(Bohn et al., 2000). The rotor-stator interaction effect against temperature distribution of solid turbine blade has been calculated(Sondak and Dorney, 2000). Influences of flow unsteadiness has also been investigated using conjugate simulation of flow and heat conduction(Yamane et al., 1999).

One of the difficulties in conjugate heat transfer analysis is an increase in calculation time. Ultimately no fixed temperature is necessary because surface temperature on solid object is the result of the conjugate simulation, however, the speed of heat transfer is quite slow compared to the characteristic speed of flow, thus the required calculation time becomes enormous. A solution technique is indispensable for the conjugate heat transfer simulation to become an efficient thermal design tool.

In this study a new numerical technique for faster calculation of the conjugate heat transfer is introduced. The developed program is used to solve temperature distribution of new turbine cooling structures.

NUMERICAL METHOD

Base Flow Solver -UPACS-

The numerical simulation program in this study is based on the common CFD platform UPACS which has been developed at National Aerospace Laboratory of Japan since 1998(Yamane et al., 2000). UPACS stands for *Unifiled Platform for Aerospace Computational Simulation* and its major purpose is to cooperate and share knowledges and informations in numerical simulations through development and utilization of a common CFD program.

The UPACS has the following characteristics;

- Finite Volume Method
- Multiblock Structured Grid Method
- Coding by FORTRAN90
- Parallel Computation by MPI
- Various Computer Environment including Supercomputers, EWS, and PC linux cluster
- Automatic block connection
- Simple graphic visualization during calculation

In the UPACS the flow calculation inside each block are treated like a single block solver while data commu-

Manuscript Received on June 30,2003

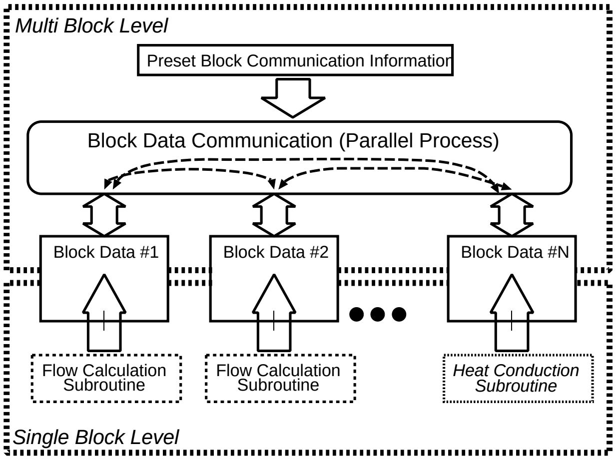


Fig.1: Concept of UPACS

nications for parallel computing are controlled independently of solver subroutines(Fig. 1). In this study, the flow solver subroutine is called in flow calculation blocks while the heat conduction program which has been derived from the heat conduction term of the original flow solver is called in solid blocks.

The original flow solver part of the UPACS can solve compressible flows of perfect gas with the selection of the following numerical schemes.

Convective term	Roe scheme, AUSMDV
Time integration	Runge-Kutta, Matrix Free Gauss Seidel(MFGS)
Turbulence	Baldwin-Lomax, Spalart-Allmaras

In this study, Roe's approximate Riemann solver with the MFGS time integration method and Spalart-Allmaras turbulence model has been used.

Coupling of Flow and Heat Conduction

Each calculation block has imaginary cells which are copies of cells in neighbouring block (left figures of Fig. 2) and the values of the imaginary cells are transferred from the neighbouring blocks then block-to-block data communication is achieved.

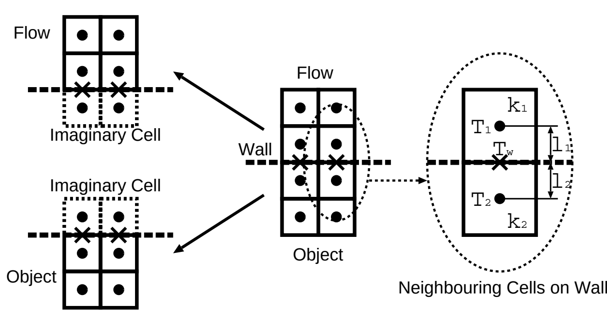


Fig.2: Connecting Boundary of Flow and Object

On solid surfaces the flow block and the solid object block should be connected like the central figure of Fig. 2. Physical values are defined at the center of cells and the values on object surfaces (× in the figure) do not appear explicitly because the UPACS is based on the Finite Volume Method, thus appropriate values should be set in imaginary cells so that the values on the wall become correct.

The thermal conditions which must be satisfied between flow block and solid block are;

1. Temperature is continuous.
2. Heat flux between surface and flow cell coincides with heat flux between solid cell and surface.

These conditions can be written using cell temperature T_1, T_2, wall temperature T_w, heat conduction ratio k_1, k_2 and length between cell center and wall l_1, l_2,

$$\frac{Q}{A} = k_1 \frac{T_1 - T_w}{l_1} = k_2 \frac{T_w - T_2}{l_2} \tag{1}$$

By eliminating T_w the following equation is derived.

$$\frac{Q}{A} = \frac{k_1 k_2 (T_1 - T_2))}{k_1 l_2 + k_2 l_1} \tag{2}$$

Thus the wall surface temperature is not necessary even at the connecting boundary of flow and solid heat conduction.

The conditions above applies not only to connecting boundary but also to every cell-to-cell face, thus no special treatment is necessary along the connecting boundary if values in imaginary cells are received and

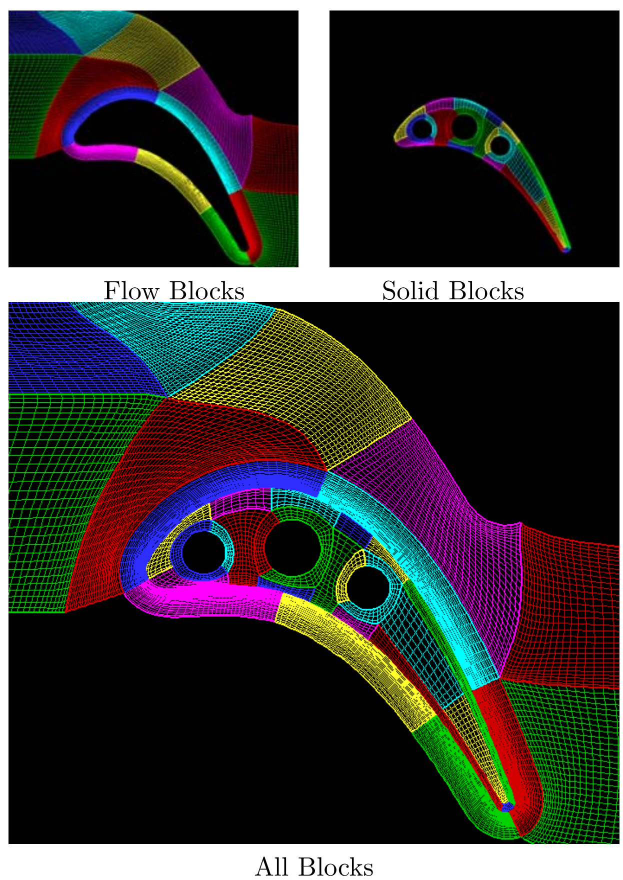


Fig.3: Numerical Grid of 2D Turbine

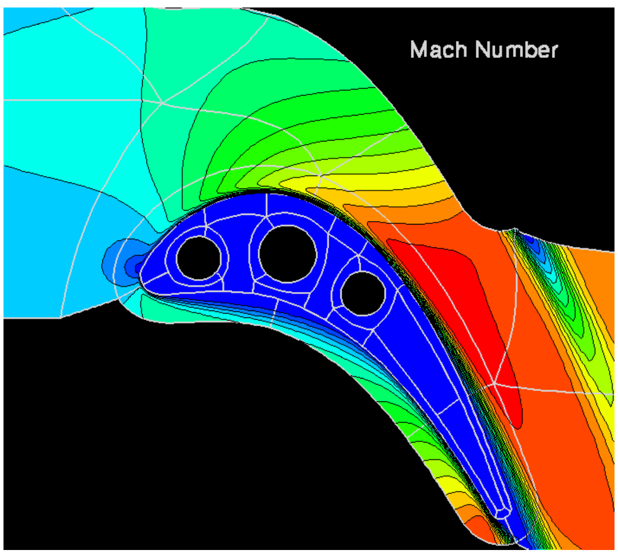
converted appropriately between flow blocks and solid blocks.

Numerical Result of 2D Turbine

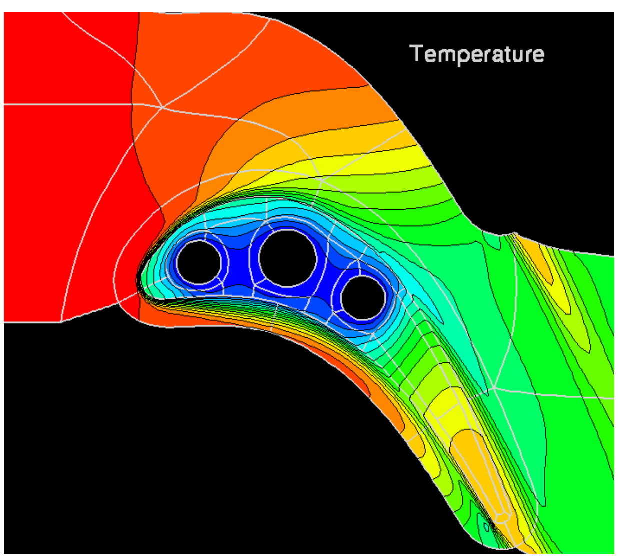
The developed program for conjugate simulation of flow and heat conduction has been tested with a two-dimensional turbine blade which has three circular cooling passages.

Figure 3 shows numerical grid that consists of 14 flow blocks and 21 solid blocks for heat conduction. The temperature along the cooling surfaces has been set to be constant in this calculation.

The results are given in Fig. 4. The Mach number distributions are almost identical to normal flow-only calculation around turbine blade while the temperature distribution in the blade has been calculated by the fixed temperature on the cooling surfaces and the coupling between flow simulation and heat conduction of blade materials along the blade external surface.



(a) Mach Number Distributions



(b) Temperature Number Distributions

Fig.4: Conjugate Simulation of 2-D Turbine Blade

Calculation Acceleration Technique

In the previous simulation the temperature along the cooling surfaces have been fixed, however, the actual temperature should also be the result of coupling with coolant flow. In this case no temperature values in blade materials are known in advance.

In order to simulate such a condition, a constant heat transfer coefficient and coolant temperature has been assumed on the cooling surfaces so that temperature value will be obtained from numerical simulation result.

Figure 5 shows the temperature result where non-uniform temperature distribution has been obtained around cooling passage surfaces as a result of conjugate simulation.

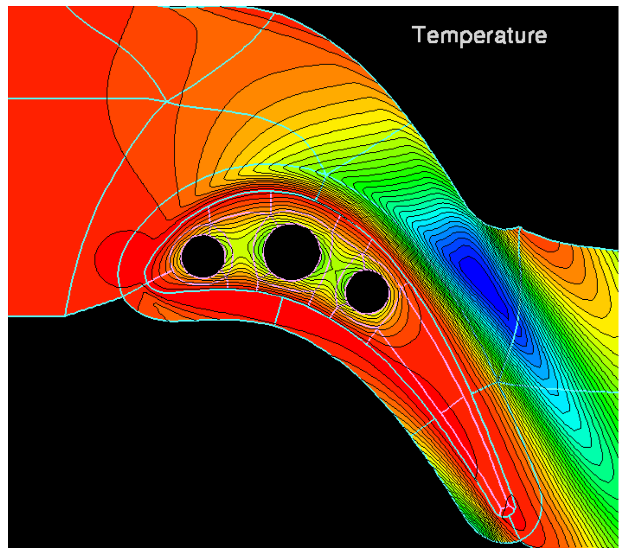


Fig.5: Result with Emulation of Coolant

One of the serious problems in these simulations is a huge calculation time that is due to the boundary condition that temperature is not known in advance. In actual physical phenomena high subsonic flows usually come to nearly steady state in some milli-seconds while temperature of object in the flow continues to change for more than several seconds until it becomes steady state. This time difference also appears in the conjugate simulation and requires enormous iterations until steady temperature result is obtained but the flow calculation should be continued only for the consistency with the heat conduction calculation of solid region even after the flow field has been mostly converged.

In order to overcome the problem, heat conduction sub-step method has been introduced. Figure 6 shows the flowchart of the program. In the normal iterations, the boundary condition is set in each block first, physical values for imaginary cells are exchanged among blocks, eddy viscosity and heat conduction rate are calculated in flow blocks then transferred similarly, and finally flow and heat conduction term is solved in flow blocks and solid blocks respectively. In the heat conduction sub-iterations, only heat conduction is calculated even in flow blocks after setting boundary conditions and exchanging values for imaginary cells.

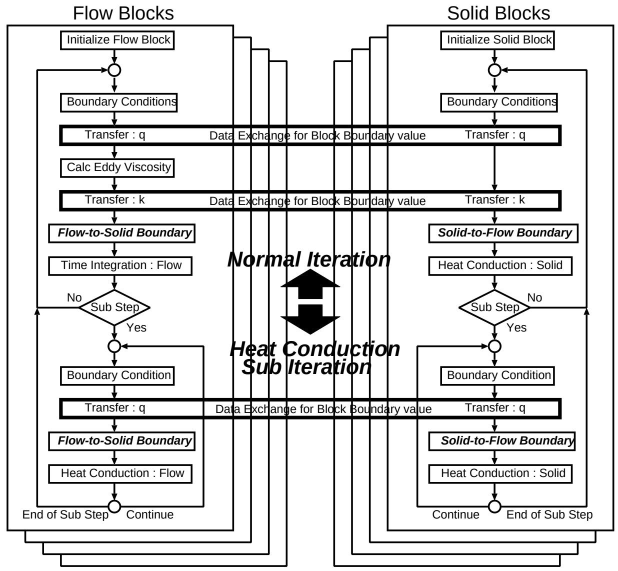


Fig.6: Program Flow with Sub-iteration

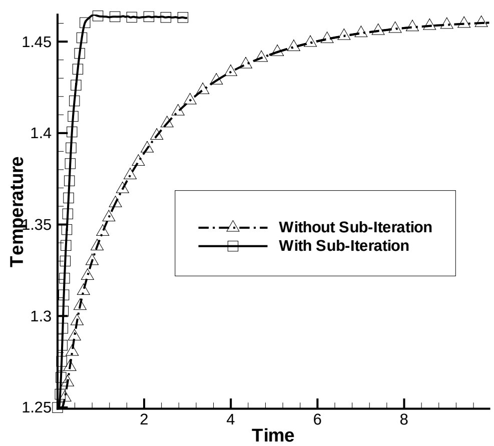


Fig.7: Convergence History with/without Sub-iteration

The effect of the sub-iteration is compared in Fig. 7 where temperature history with and without 5 sub-iterations in each normal iteration is displayed. The heat conduction sub-step method showed 20 times faster convergence speed than the normal-iteration-only calculation.

It should be noted that the calculation with heat conduction sub-iteration method is not physically correct, thus the difference between the results with and without sub-iteration has been checked. Figure 8 displays temperature value differences between the result with 10 sub-iterations and without sub-iteration. The errors are minimum and negligible in all over the calculation region and a bigger error area in the later half of the blade has been proved to be due to the flow fluctuations. It is assumed that the errors generated by the sub-iteration are not so big that they are canceled in the normal iteration steps.

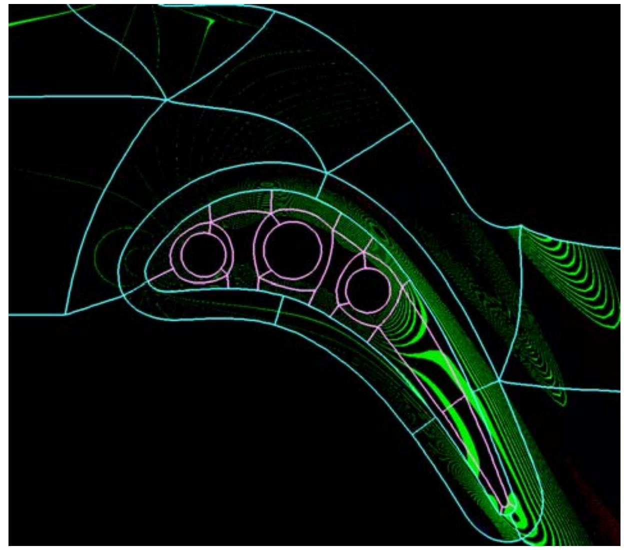


Fig.8: Difference of Temperature with/without Sub-iteration

CONJUGATE SIMULATION FOR TURBINE COOLING STRUCTURE

Integrated Cooling Configuration

A cooling configuration which integrates impingement cooling and pin cooling device into one body (Fig. 9)has been introduced(Funazaki et al., 2001) and experiments in order to investigate the effect of the pin density of the new cooling configuration against the cooling performance have been conducted (Yamawaki et al., 2003).

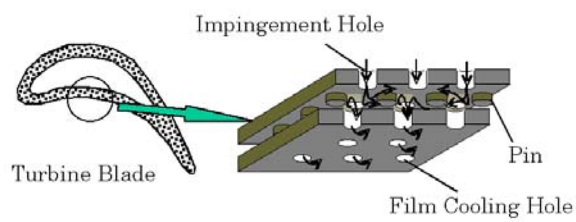
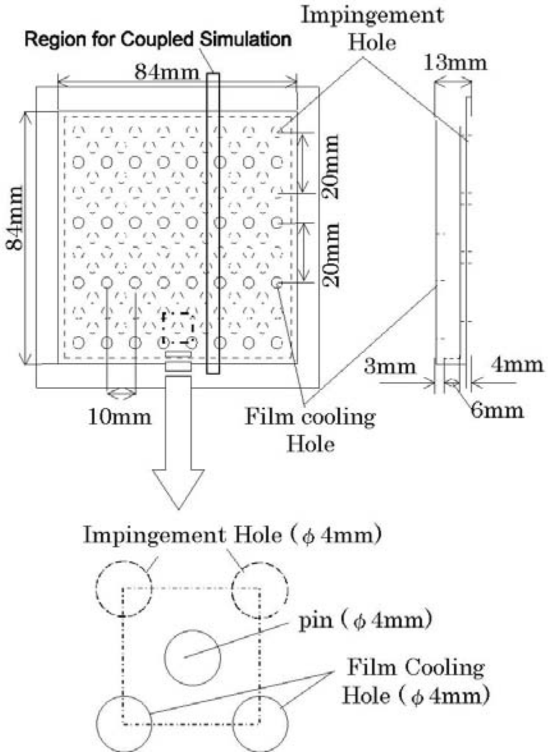


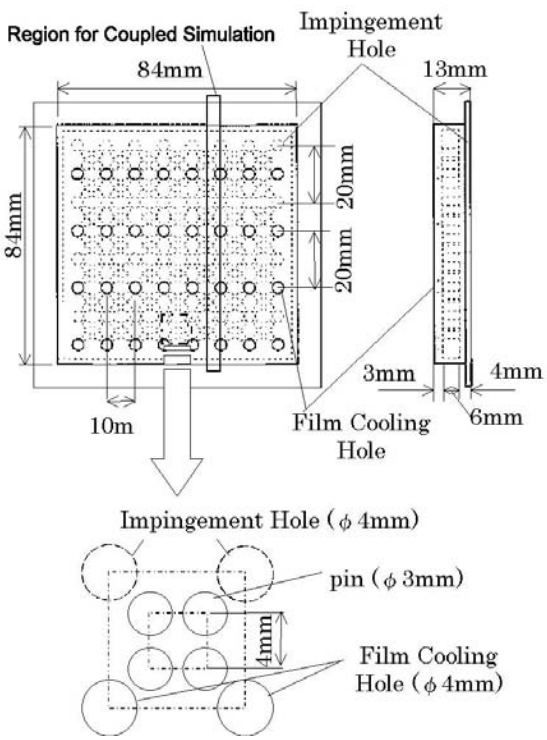
Fig.9: Concept of Integrated Cooling Configuration

The conjugate simulation program for flow and heat conductions of this study has been applied to two configurations with coarse and fine pin density. A coarse pin specimen called the basic specimen, shown in Fig. 10(a), has one pin of 4.0mm diameter spaced between an impingemnet hole and a film cooling hole while another specimen called the fine specimen, shown in Fig. 10(b), has four pins of 3.0mm diameter.

Two specimens have been examined in the test configuration in Fig.11 that hot gas and cooling air are provided and temperature measurements have been made by an IR camera and a thermocouple on the surface of specimen. Figure 12 shows an example of IR temperature image. IR temperature calibrations with the thermocouple were carried out for each specimen in order to get accurate area averaged temperature to obtain



(a) Basic Specimen



(b) Fine Specimen

Fig.10: Configuration of Basic andFine Specimen

cooling effectiveness(Yamawaki et al., 2003).

Numerical Grid

To express hole and pin shapes of the specimens precisely, two dimensional mesh with circles which represent hole and pin positions has been created first then the 2-d mesh has been extended vertically(Fig. 13(a)).

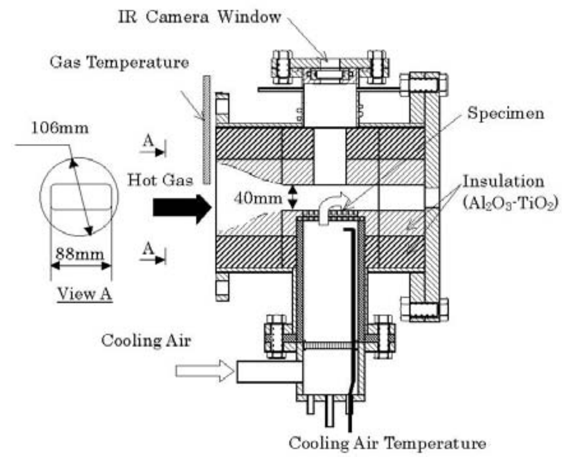


Fig.11: Test Section Detail

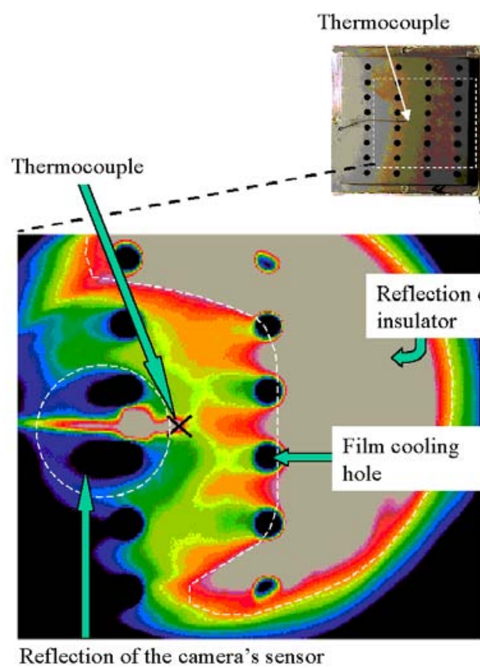


Fig.12: IR Image

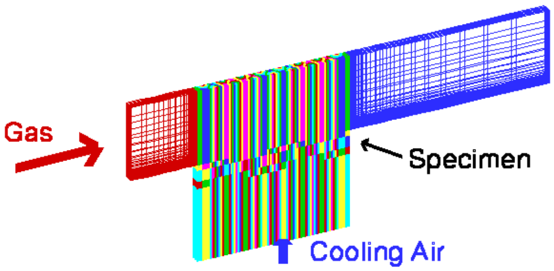
Solid calculation blocks of the basic specimen for heat conduction is shown in Figure 13(b). The flow region and solid region have 310 and 207 blocks respectively and the total number of grid points is apporximately three million. For the fine specimen 558 flow blocks and 434 solid blocks has been used and the number of grid points has exceeded five million.

Numerical Resutls

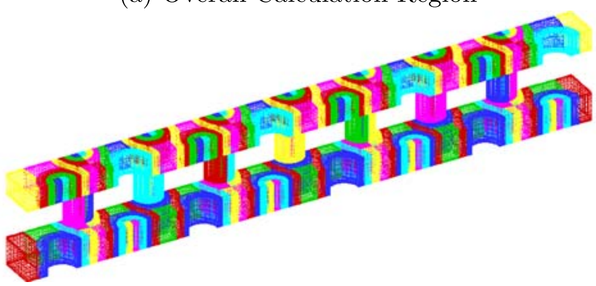
One calculation condition for each specimen has been chosen where the gas flow speed is rather faster and the cooling air flow is larger because it is easier for compressible flow solvers. Figure 14 compares cooling effectiveness η which is defined as

$$\eta = \frac{T_g - T_w}{T_g - T_c} \tag{3}$$

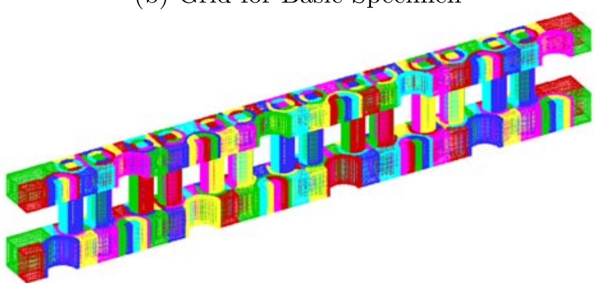
where T_g is gas temperature, T_w is averaged wall tem-



(a) Overall Calculation Region



(b) Grid for Basic Specimen



(c) Grid for Fine Specimen

Fig.13: Numerical Grid for Basic and Fine Specimen

perature, and T_c is cooling air temperature.

Numerical results are very close to the experimental results. The differences between fine and basic specimens in experiments in larger cooling air ratio conditions have been estimated to be due to the effective observation area by the IR camera(Yamawaki et al., 2003) while the conjugate simulation results showed almost identical cooling effectiveness between two specimens. Current numerical results are few for discussion thus further simulation cases with various cooling air flow rates should be necessary.

In order to examine the difference between two specimens, surface distributions of cooling effectiveness of the numerical results are compared in Fig. 15. It is clearly observed that the surface temperature distributions are similarly obtained in both results.

Figure 16 compares both results by temperature distributions on solid surfaces and streamtraces of cooling air. It is clearly and similarly observed in both results that the cooling air lowers plate temperature by impinging beneath the plate, then it flows around pins, and finally blows out to form the cooling film. The difference has been found in the area where the impingement air hits the upper plate. In the basic specimen result the lower temperature region spreads widely while the impingement flow is prevented by larger number of pins in the fine specimen result thus higher temperature region can be observed beneath the upper plate of the fine specimen. On the other hand the surface area of pins is three times larger in fine specimen than basic specimen, thus it can be estimated that the disadvantage in the impingement cooling of the fine specimen has been compensated by pin surface heat transfer then the cooling effectiveness of the fine specimen resulted in the same as the basic specimen.

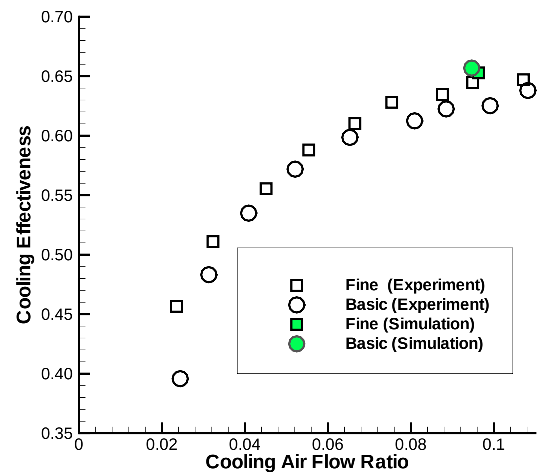


Fig.14: Comparison of Cooling Effectiveness at Gas Speed of $Re = 7.0 \times 10^5$

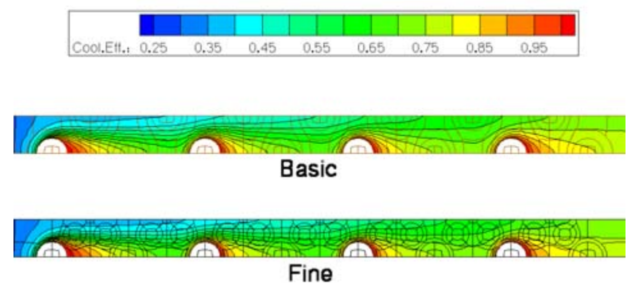
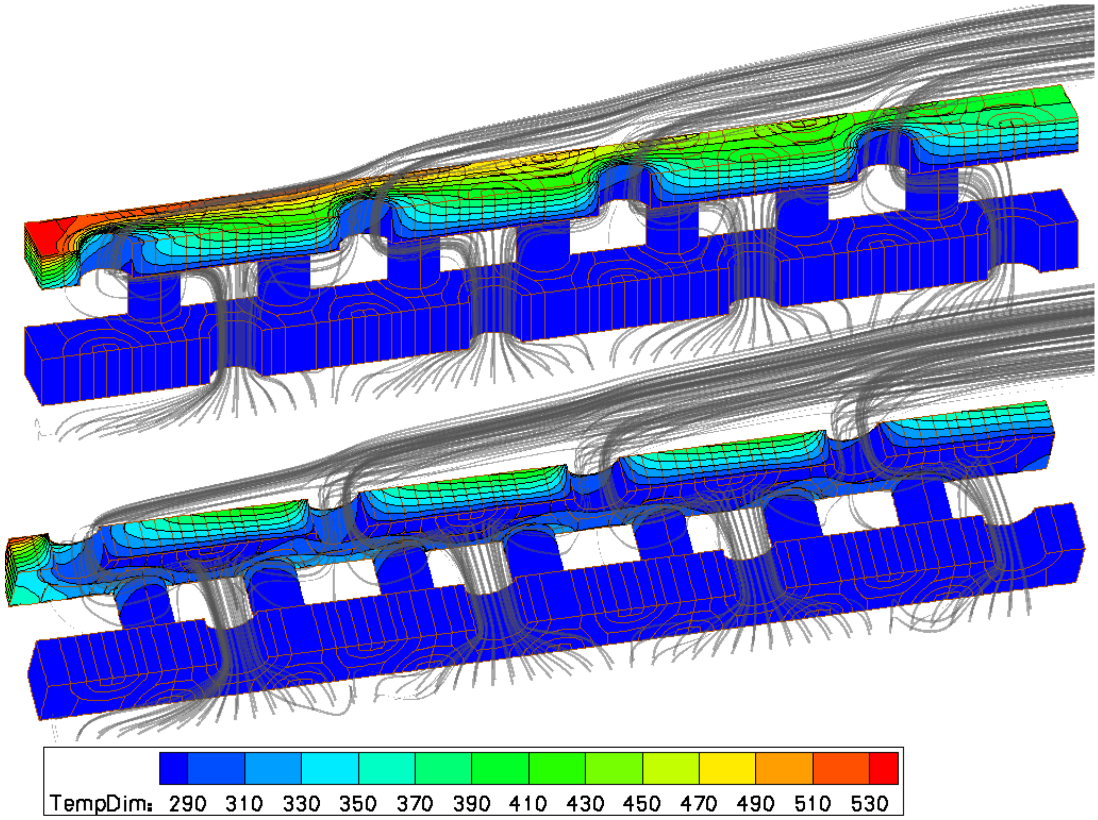


Fig.15: Cooling Effectiveness Distribution on Specemen Surface

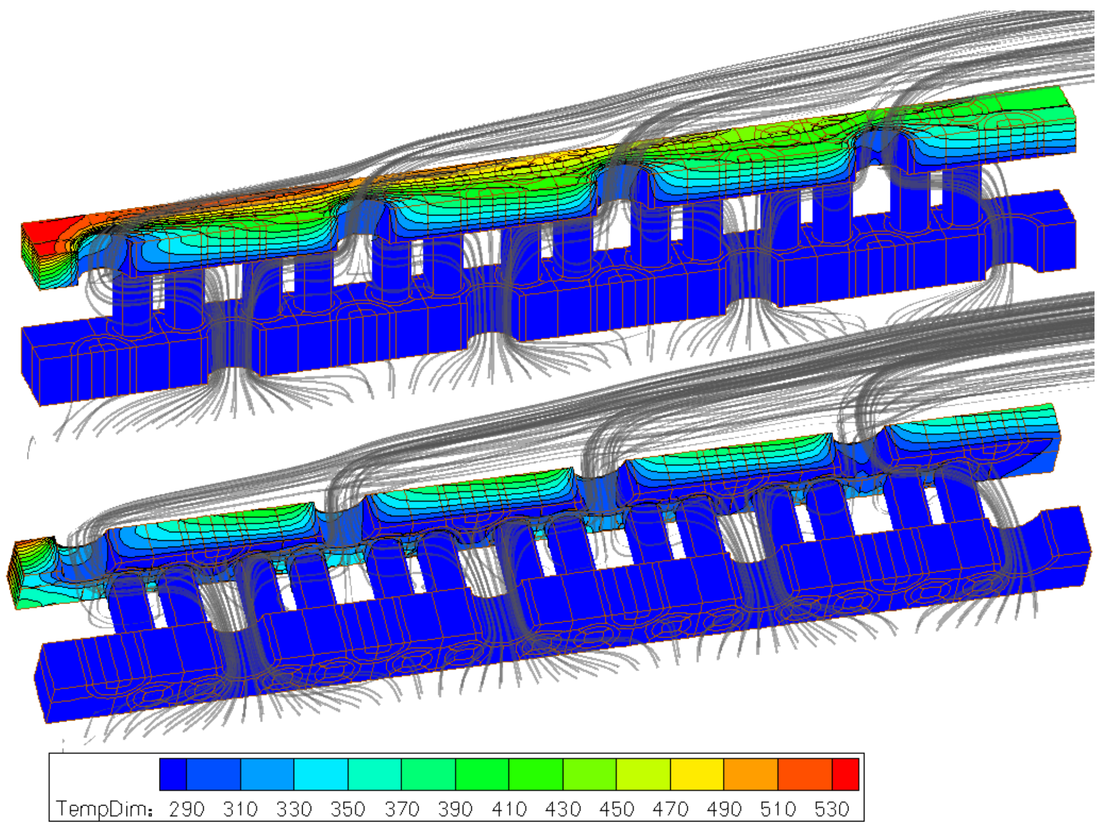
CONCLUSIONS

A conjugate simulation program has been developed with a new calculation acceleration technique. The technique, which is not physically accurate, has proved its great improvement in convergence speed and the numerical error by the technique has turned out to be minimum.

The program has been applied to two turbine cooling strucutre configurations with no fixed temperature boundary conditions on solid surfaces. The numerical results have cleary revealed the difference in the impingement effect between two configurations which have different pin cooling devices.



(a) Basic Specimen



(b) Fine Specimen

Fig.16: Surface Temperature and Streamtrace

References

Yamane, T., Yoshida, T., Mimura, F., Taki, M., "Influences of Flow Unsteadiness on Conjugate Simulation of Thermofluid-dynamics and Thermal Conductivity", Proceedings of the International Gas Turbine Congress 1999 Kobe, IGTC'99Kobe TS-45

Bohn, D., Heuer, T., Kortmann, J., "Numerical Conjugate Flow and Heat Transfer Investigation of a Transonic Convection-Cooled Turbine Guide Vane with Stress-Adapted Thickness of Different Thermal Barrier Coatings", AIAA 2000-1034

Sondak, D. and Dorney, D., "Conjugate Unsteady Heat Transfer Simulation in a Turbine Stage", AIAA 2000-3360

Yamane, T., Yamamoto, K., Enomoto, S., Yamazaki, H., Takaki, R., Iwamiya, N., 2000, "Development of a Common CFD Platform -UPACS-", Proceedings of the Parallel CFD Conference 2000, Trondheim, Norway

Funazaki, K., Tarukawa, Y., Kudo, T., Matsuo, S., Imai, R. and Yamawaki, S., 2001, "Heat Transfer Characteristics of an Integrated Cooling Configuration for Ultra-High Temperature Turbine Blades: Experimental and Numerical Investigations", ASME Paper 2001-GT-148

Yamawaki, S., Nakamata, C., Imai, R., Matsuno, S., Yoshida, T., Mimura, F., Kumada, M., "Cooling Performance of an Integrated Impingement and Pin Fin Cooling Configuration", Proceedings of ASME TURBO EXPO 2003, GT2003-38215