Proceedings of ASME Turbo Expo 2004
Power for Land, Sea, and Air
June 14-17, 2004, Vienna, Austria

GT2004-53680

CONJUGATE SIMULATION OF FLOW AND HEAT CONDUCTION WITH A NEW METHOD FOR FASTER CALCULATION

Takashi Yamane, Toyoaki Yoshida, Shunji Enomoto, Ryoji Takaki, Kazuomi Yamamoto
Institute of Space Technology and Aeronautics,
Japan Aerospace Exploration Agency
7-44-1 Jindaiji-higashi, Chofu, Tokyo 182-8522, JAPAN
E-mail: yamane.takashi@jaxa.jp

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 improved by introducing 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 configuration changes the cooling efficiency.

NOMENCLATURE

A	area		Subscripts
k	thermal conductivity	g	gas
l	length	c	cooling air
M	Mach number	w	wall
Q	heat flux		
Re	Reynolds number		
T	temperature		
η_c	cooling effectiveness		

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 conflicting demand more precise aerodynamic and thermodynamic design methods are 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 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 [1]. The rotor-stator interaction effect against temperature distribution of solid turbine blade has been calculated [2]. Influences of flow unsteadiness has also been investigated using conjugate simulation of flow and heat conduction [3].

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 tends to be larger. 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 to accelerate the 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. The UPACS, *Unified Platform for Aerospace Computational Simulation*, is a project to develop a common CFD program since 1998 at National Aerospace Laboratory of Japan. The aim of the project was not only to overcome the increasing difficulties in recent CFD code development on various parallel computers, but also to accelerate the development of CFD technology by sharing a common base code among research scientists and engineers. A necessity of a common CFD program has been recognized among younger CFD researchers at National Aerospace Laboratory of Japan around 1996/1997 and discussions have started. The programming of a prototype code has begun in 1998 with the name of UPACS then the first version has been released on October 2nd, 2000 with the capability to solve compressible flows with the multi-block grid method [4]. The development of the UPACS is now succeeded by the "Institute of Space Technology and Aeronautics" of "Japan Aerospace Exploration Agency" which is an integration of the National Aerospace Laboratory of Japan (NAL), the Institute of Space and Astronautical Science (ISAS), and the National Space Development Agency of Japan (NASDA).

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
- Graphic visualization during calculation

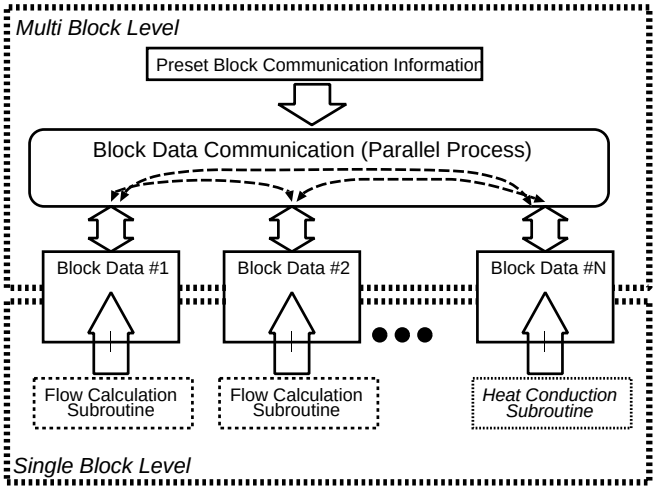


Figure 1. Concept of UPACS

In the UPACS the flow calculation inside each block is treated like a single block solver while data communications for parallel computing are controlled independently of solver subroutines (Fig.1). The flow solver subroutine of the UPACS is quite similar to a program for single block grid problems. The subroutine is called through an interface subroutine *"doAllBlocks"* which manages the distribution of calculation blocks to CPUs (Fig.2).

Flow Only

```
program main
 ......
 call MPL_init
 call doAllBlocks(initialize1)
 call doAllBlocks(transfer,"grid")
 call doAllBlocks(initialize2)
 ......
 do i=1,iteration_max
  call doAllBlocks(step)
  call mpi_reduce(residual,"sum")
  call doAllBlocks(transfer,"q")
 end do
 ......
 call doAllBlocks(finalize)
 call MPL_end
end program
```

Coupled with Heat Conduction

```
program main
 ......
 call MPL_init
 call doAllBlocks(initialize1)
 call doAllBlocks(transfer,"grid")
 call doAllGroups(initialize_flow, initialize_heat)
 ......
 do i=1,iteration_max
  call doAllGroups(step_flow, step_heat)
  call doOneGroup(flag, flow_only_procedure)
  call mpi_reduce(residual,"sum")
  call doAllBlocks(transfer,"q")
 end do
 ......
 call doAllBlocks(finalize)
 call MPL_end
end program
```

Figure 2. Interface Subroutines

For the conjugate simulation, a solver changing mechanism has been newly introduced. A flag has been given to each block in order to distinguish solid blocks from flow blocks. The modified interface subroutine switches solver subroutines by the value of the flag in each block. For this purpose, two interface subroutines, "*doAllGroups*" and "*doOneGroup*", have been added to the original solver program. The block-to-CPU allocation mechanism for parallel calculation of the original program is not affected by this modification, thus the users can freely distribute flow and solid blocks to CPUs.

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 for flow blocks.

The heat conduction subroutine for solid blocks has been newly developed. It has been derived from the heat conduction term in the energy equation of the flow solver. All original vari-

ables of the flow solver has been kept by setting 1.0 to density and 0.0 to velocities in order to minimize the modification of the program. It is a memory consuming approach but quite beneficial because the file format of the original UPACS can commonly used and many flow-visualization programs can handle flow blocks and solid blocks simultaneously. The time integration methods for the heat conduction are the Euler explicit and the Runge-Kutta. In this study the Euler explicit method has been mainly used.

Coupling of Flow and Heat Conduction

Each calculation block has auxiliary cells which are copies of cells in neighboring block (left figures of Fig.3) and the values of the auxiliary cells are transferred from the neighboring blocks so that the block-to-block data communication can be achieved.

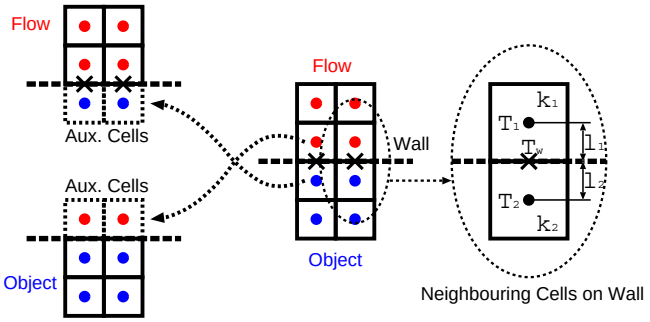


Figure 3. Connecting Boundary of Flow and Object

The flow block and the solid object block should be connected like the central figure of Fig.3 on solid surfaces. Physical values are defined at the center of cells and the values on the object surfaces ($\times$ in the figure) do not appear explicitly because the UPACS is based on the Finite Volume Method, thus appropriate values are being extrapolated and set in auxiliary cells so that the values on the wall become correct.

The thermal conditions which must be satisfied between flow blocks and solid blocks in the conjugate simulation 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, thermal conductivity 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 all cell-to-cell faces, thus no special treatment is necessary along the connecting boundary for heat conduction term calculation if values in auxiliary cells are received and converted appropriately between flow blocks and solid blocks. This approach was suitable and straightforward for the original UPACS program.

The developed program at present has a limitation that the cell size of the solid block along the connecting boundary should be equivalent to the size of the neighboring cell of the flow block.

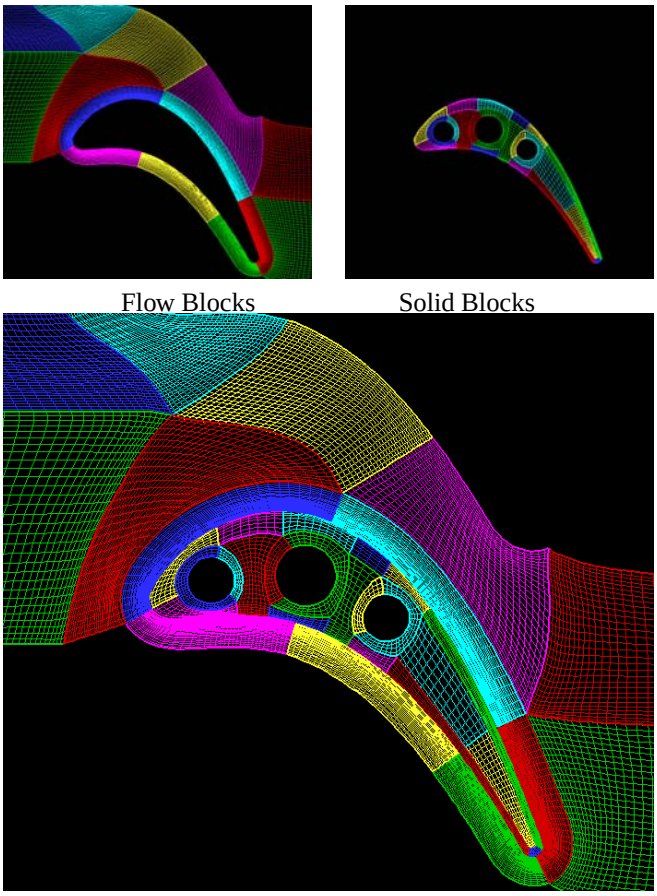


Figure 4. Numerical Grid of 2D Turbine

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 imaginary circular cooling passages.

Figure 4 shows the numerical grid that consists of 14 flow blocks and 21 solid blocks for heat conduction.

The inlet total temperature has been set to 300 deg.C and the pressure ratio is 2.0. Reynolds number by the axial length of the blade is 1.8×10^6. The temperature along the cooling passages has been set to be a constant value of 30 deg.C in this calculation.

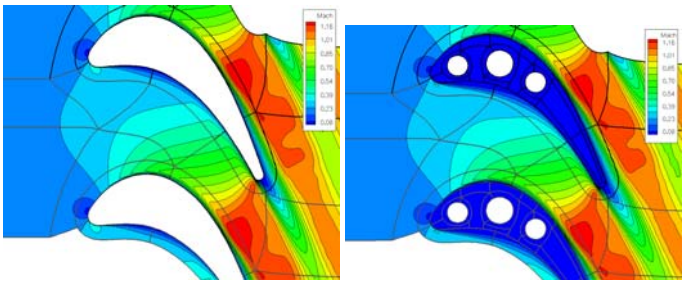
The results are given in Fig.5. The Mach number distributions are almost identical to normal flow-only calculation around turbine blade while the temperature distribution in the blade has been calculated by coupling between flow simulation and heat conduction of blade materials along the blade external surface.

Calculation Acceleration Technique

In the previous simulation the temperature along the cooling passages have been fixed, however, the actual temperature should also be the result of coupling with coolant flow. In this case the temperature at every part of the blade should be obtained as a result of the conjugate simulation.

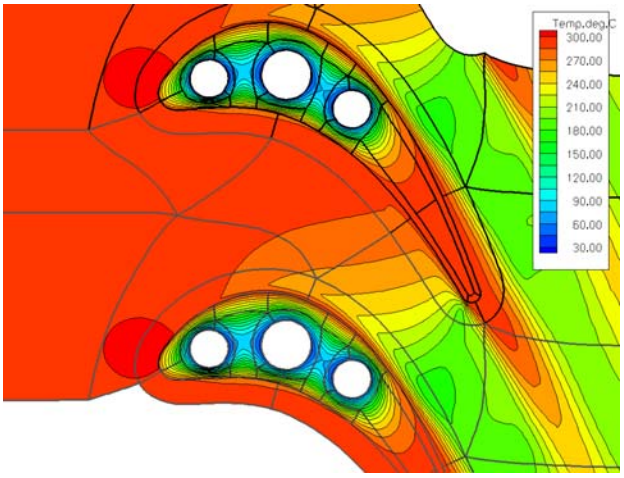
In order to simulate such a condition in 2-D calculation, a constant heat transfer coefficient and coolant temperature has been assumed along the cooling passages.

Figure 6 shows the temperature result where non-uniform temperature distribution has been obtained around cooling passage surfaces as a result of conjugate simulation.



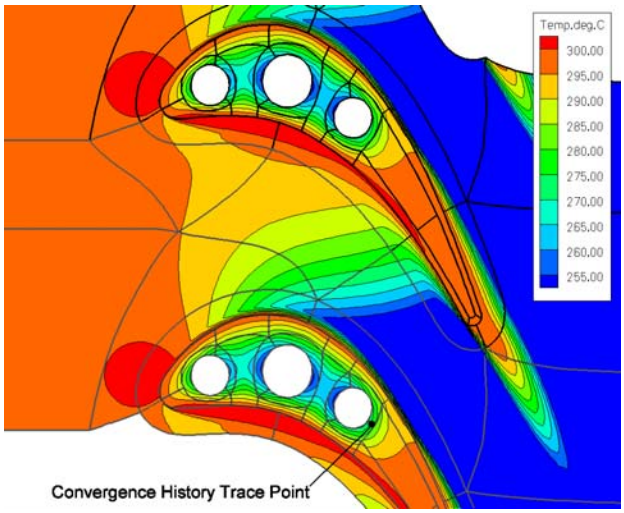
Flow Only Simulation Conjugate Simulation

(a) Mach Number Distributions



(b) Temperature Distributions

Figure 5. Conjugate Simulation of 2-D Turbine Blade

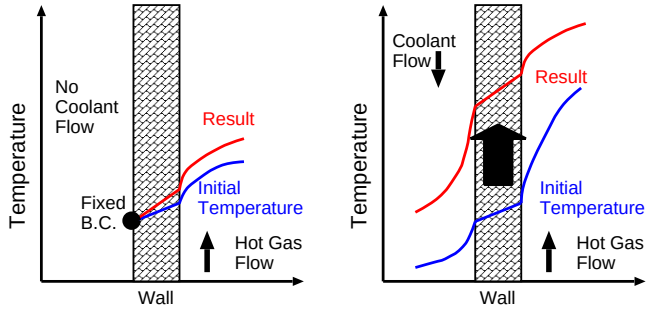


(Temperature scale is different from Fig.5)

Figure 6. Result with Emulation of Coolant

One of the serious problems in these simulations is a larger calculation time when the temperature boundary value 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 scale difference also appears in the conjugate simulation and requires many iterations until steady temperature result is obtained. 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. If some fixed temperature values are given in some parts of the solid object the resulting temperature will not be far from the given values, otherwise the result of all region are obtained as a result of the conjugate calculation and the temperature change during calculation tends to be larger and the necessary calculation time becomes longer (Fig.7).

One of the solutions to the problem is to use a larger time step for the heat conduction calculation in solid blocks than the flow calculation. In the simulation of Fig.6, the largest time step



(a) Fixed Temperature B.C. (a) No given B.C.

Figure 7. Temperature Disctibution along Wall

has been set in the solid blocks but its value was not satisfactory due to an instability along the connecting boundary of flow and solid. The Runge-Kutta method has also been tested but the available time step value was less than twice with the Euler explicit method. Spreading the widths of cells along the connecting boundary in the solid blocks may be effective to take a large time step, but the current program has the limitation about the cell size which has been stated above and the equivalent cell size between flow and solid will be preferable when the conjugate simulation is used for a study of the unsteady flow effect against the heat transfer in the future. Thus, a new method for faster calculation has been considered.

The problem is that the time consuming calculations of flow which include the convective terms and the implicit time integration should be continued until temperature distributions catch up with velocity distributions. In order to overcome the problem, a heat conduction sub-step method that skips the time consuming term calculations has been introduced.

Figure 8 shows the overall flowchart of the program. In the normal iterations, the boundary condition is set in each block first, physical values for auxiliary cells are exchanged among blocks, eddy viscosity and heat conduction rate are calculated in flow blocks then transferred similarly, and finally flow in flow blocks and heat conduction in solid blocks are calculated respectively. In the heat conduction sub-steps, the program flows similarly with the normal iteration but the subroutine used for the flow blocks is different. Instead of the flow solving subroutine, the heat conduction subroutine which has been originally developed for the solid blocks is called even in the flow blocks during the sub-step. This method reduces the calculation load in the flow blocks and the heat conduction calculations in both flow and solid blocks are being advanced (Fig.9).

The effect of the sub-step is compared in Fig.10 where temperature histories at the point shown in Fig.6, with and without 5 sub-steps, are compared (solid lines). The time step for the flow blocks has been set that the CFL number becomes 10.0 and the maximum available constant time step has been used in the solid blocks with the Euler explicit method. The calculation

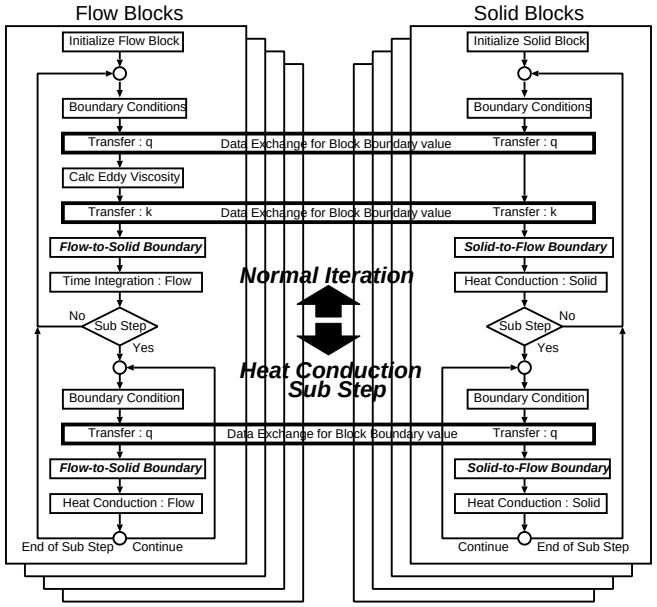


Figure 8. Program Flow with Sub-step

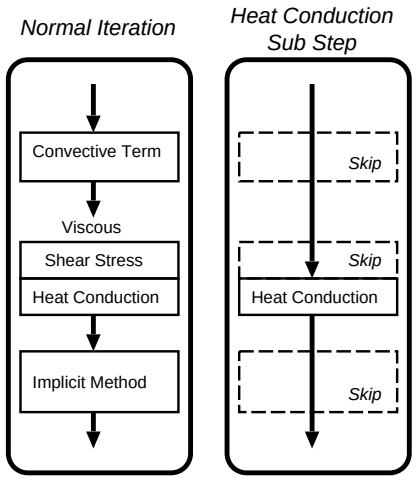


Figure 9. Comparison of Normal-step and Sub-step for Flow Blocks

started with the flow only simulation results in the flow blocks and a constant temperature of 100 deg.C in the solid blocks. The calculation time in Fig.10 is the actual time using a Pentium4 3.06GHz processor. The convergence of calculations has been determined by monitoring the temperature value change at the point. The heat conduction sub-step method showed three times faster speed than the normal-iteration-only calculation until the difference between the temperature trace value and the initial value reached 99% of the convergence. By the calculation with one tenth Reynolds number of the original, the convergence speed ratio was similar but the absolute convergence time has be-

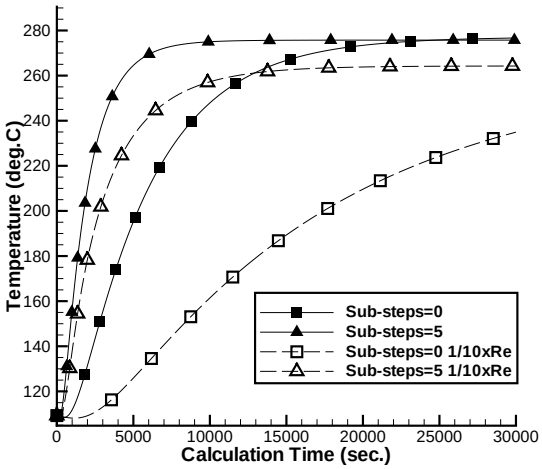


Figure 10. Convergence History with/without Sub-step

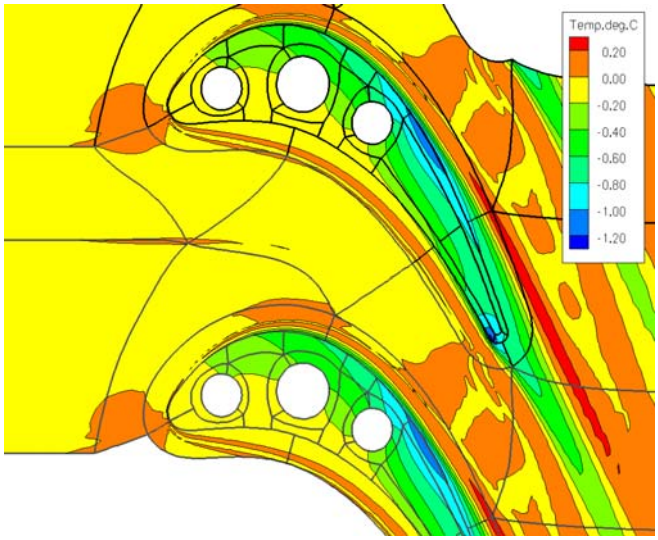


Figure 11. Difference of Temperature with/without Sub-step

come much longer (the dashed line). It is said that the sub-step method is more helpfull when the mesh size along the connecting surface is smaller.

It should be noted that the calculation with heat conduction sub-step method is not physically correct, thus the difference between the results with and without sub-step has been checked. Figure 11 displays temperature value differences between the results with 5 sub-steps and without sub-step. The largest difference was 1.2 deg.C which is only 2.7% of the maximum temperature difference in the solid blocks. The errors are generally larger near the blade suction surface than the pressure surface. Some large errors appeared at the trailing edge where a quick turn of the flow has been observed and at the suction surface where the flow is choked. It can be estimated that a lack of the mesh density in flow boundary layer causes larger temperature errors but the criterion of the mesh density has not figured out yet. In the flow blocks the errors are much smaller than in the solid blocks. It is assumed that the errors generated by the sub-step have been canceled in the normal iteration step.

CONJUGATE SIMULATION FOR TURBINE COOLING STRUCTURE

The developed simulation program for the conjugate heat transfer problems with a new calculation acceleration method has enables larger scale simulations and has been applied to study turbine cooling structures.

Integrated Cooling Configuration

A cooling configuration which integrates impingement cooling and pin cooling device into one body (Fig.12) has been introduced [5] and experiments in order to investigate the effect of the pin density of new cooling configurations against the cooling performance have been conducted [6].

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 specimen, which is called the basic specimen, has one pin of 4.0mm diameter spaced between an impingement hole and a film cooling hole as in Fig.13(a), while another specimen, which is called the fine specimen, has four pins of 3.0mm diameter as shown in Fig.13(b).

Two specimens have been examined in the test configuration in Fig.14 that hot gas and cooling air are provided and temperature measurements have been made by an infra-red (IR) camera and a thermocouple on the surface of specimen. IR temperature calibrations with the thermocouple were carried out for each specimen in order to get accurate area averaged temperature to obtain cooling effectiveness [6].

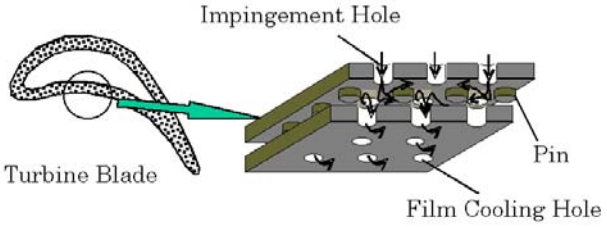
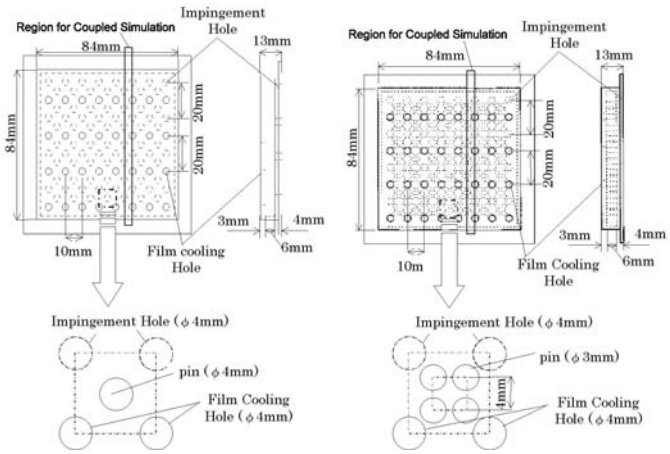
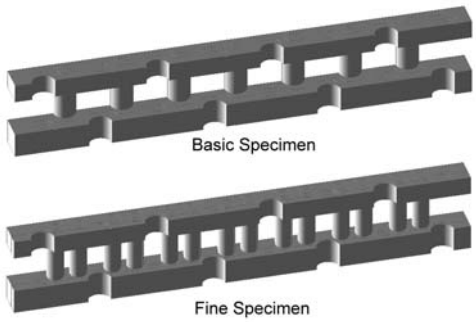


Figure 12. Concept of Integrated Cooling Configuration



(a) Basic Specimen (b) Fine Specimen



Simulation Models

Figure 13. Configuration of Basic and Fine Specimen

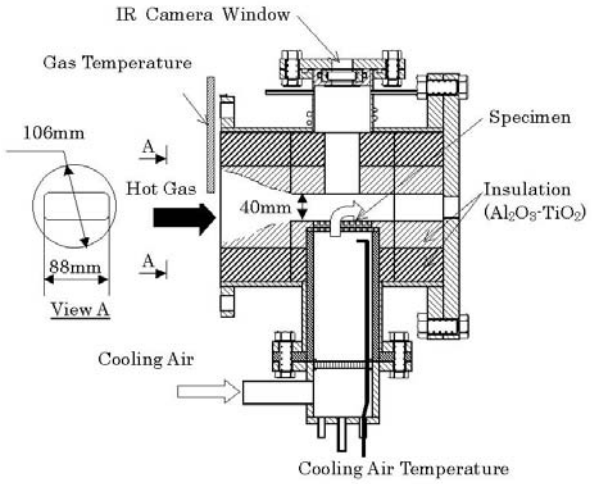


Figure 14. Test Section Detail

Numerical Grid

In order to express hole and pin shapes of the specimens precisely, two dimensional mesh with circles which represents hole and pin positions has been created first. The initial 2-D mesh has been extended vertically to create the inlet flow region of the cooling air, the lower plate with impingement holes, flow and pin region between the lower and the upper plate, the upper plate with film cooling holes, and the hot gas flow region over the upper plate. All mesh blocks has been re-grouped to flow blocks and solid blocks, then the in-flow and exit region of the hot gas has been attached finally (Fig.15).

The number of flow and solid blocks for the basic specimen is 310 and 207 respectively and the total number of grid points is approximately 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. The mesh width along all conjugate boundaries is 0.01mm that is 1/400 of the hole diameter and the Y^+ values of the first cells from the walls were within 2.

The hot gas temperature at the inlet stagnation condition is 417 deg.C while the cooling gas is 30 deg.C. The Reynolds number of the hot gas is 7.0×10^5 by using the specimen length as the reference.

Numerical Results

Two conditions for each specimen has been calculated by changing the cooling air inlet pressure. Figure 16 compares cooling effectiveness η_c which is defined as

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

where T_g is gas temperature, T_w is averaged wall temperature, and T_c is cooling air temperature.

The reason of the η_c differences between fine and basic specimens in experiments in larger cooling air flow ratio conditions have been estimated that the effective observation area by the IR camera changed by the experimental conditions and the differences between fine and basic should be closer to each other [6]. The conjugate simulation results are based on T_w that has been averaged from the hot gas side wall temperature between the 2nd film hole and the end of the specimen as shown in Fig.17. The quantitative correlations between experiments and simulations are not very good but the simulation results clearly showed almost identical cooling effectiveness between two specimens at a higher cooling air flow ratio condition (Case A) and a better cooling effectiveness by the fine specimen than the basic specimen at a smaller cooling air (Case B) similarly to the experiments.

In order to clarify the advantage of the conjugate simulations, temperature distributions of both surfaces of the upper

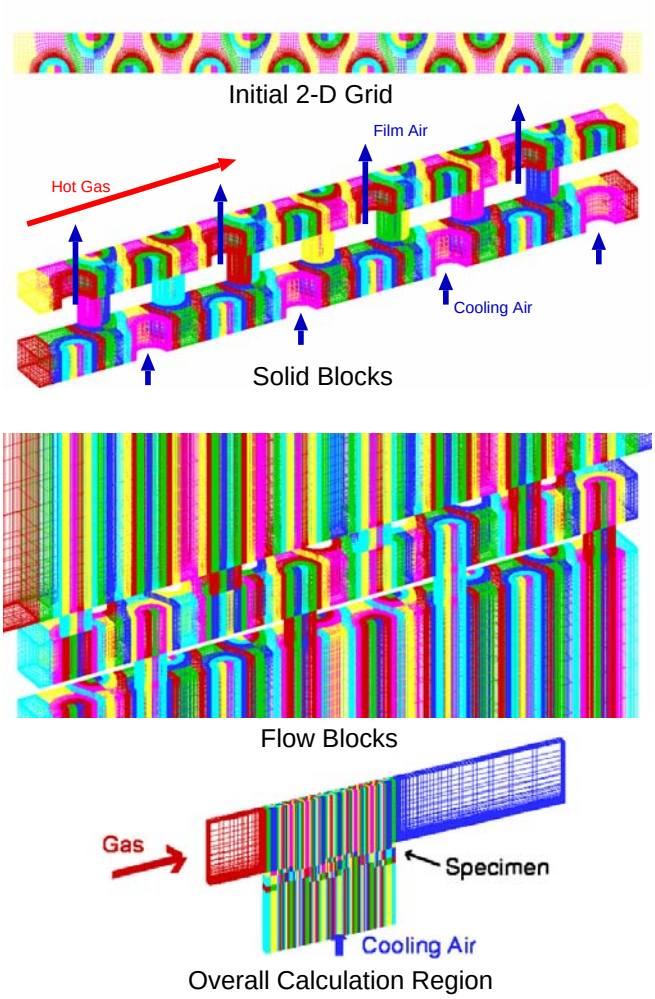


Figure 15. Numerical Grid for Basic Specimen

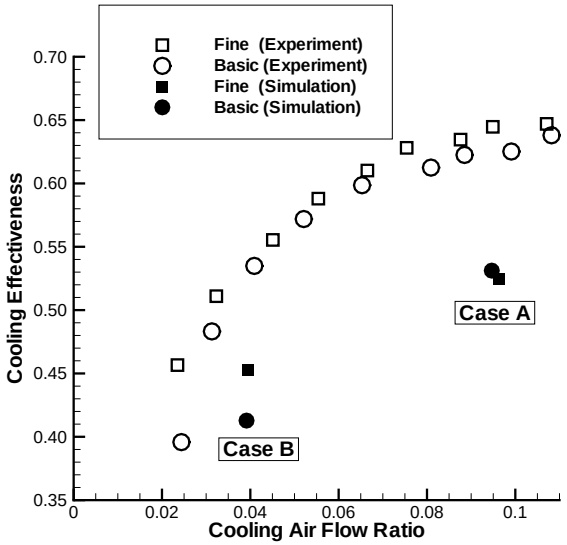


Figure 16. Comparison of Cooling Effectiveness at Gas Speed of $Re = 7.0 \times 10^5$

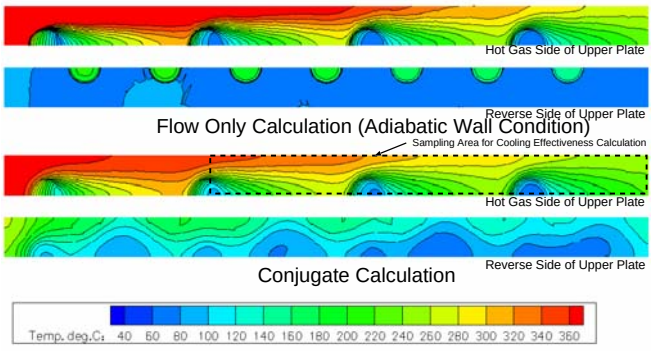


Figure 17. Surface Temperature Distribution of Upper Plate (Case B)

plate of the basic specimen are compared with the flow-only calculation of adiabatic wall condition (Fig.17). Temperature patterns on the gas flow side looks similar in both calculations but the values are lower all over the hot-gas side surface by the conjugate result. The distributions on the reverse side where impingement jets hit are completely different. The temperature by the flow-only calculation is dominated by the coolant flow temperature. By the conjugate simulation, the temperature on the reverse side is generally higher by the heat conduction from the hot gas surface and the effect of the impingement cooling is clearly observed.

Figures 18 and 19 compare the differences of the pin configurations by the temperature distributions on solid surfaces and streamtraces of cooling air at the smaller cooling condition (Case B). It is clearly and similarly observed in both results that the cooling air lowers plate temperature by impinging beneath the upper plate, then it flows around pins, and finally blows out forming the film. The difference has been found in the area where the impingement air hits the upper plate. The lower temperature region by the impingement is wider with the basic pin configuration than with the fine pins because the impingement flow is blocked by the pin roots in the fine specimen, thus some higher temperature regions can be observed beneath the upper plate of the fine specimen. With this fact the hot gas side temperature of the upper plate can be lower by the basic specimen than the fine specimen but the results in Fig.16 are the opposite. The surface area of pins is three times larger in the fine specimen than the basic specimen, thus it can be estimated that the disadvantage in the impingement cooling of the fine specimen has been compen-

sated and overcome by pin surface heat transfer then the cooling effectiveness of the fine specimen resulted in a better value than the basic specimen.

CONCLUSIONS

A conjugate simulation program of flow and heat conduction has been developed based upon the flow solver "UPACS" by introducing a mechanism to switch solver subroutines in each calculation block.

A new calculation acceleration technique has also been developed. The sub-step, which calculates the heat conduction terms only both in flow blocks and solid blocks, has been inserted among normal calculation steps and improved the convergence speed even when a large time step in the solid block heat conduction calculation cannot be taken. The sub-step method has a side effect that causes small errors in temperature field inside solid blocks especially when the mesh density is not enough in the boundary layer of the neighboring flow blocks.

The developed program has been applied to two turbine cooling structure configurations with no fixed temperature boundary conditions on solid surfaces. The numerical results have clearly revealed the difference of the overall cooling performance as well as the difference of the impingement effect between two configurations which have different pin cooling devices.

ACKNOWLEDGMENTS

This work is partly supported by the "Research and Development of Environmentally Compatible Propulsion System for Next-Generation Supersonic Transport (ESPR) project" sponsored by the New Energy and Industrial Technology Development Organization (NEDO) and the Ministry of Economy, Trade and Industry (METI). The experimental data of the integrated cooling structures are the results of the joint research with Ishikawajima-Harima Heavy Industries Co., Ltd. [6]

REFERENCES

[1] 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

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

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

[4] 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

[5] 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

[6] 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

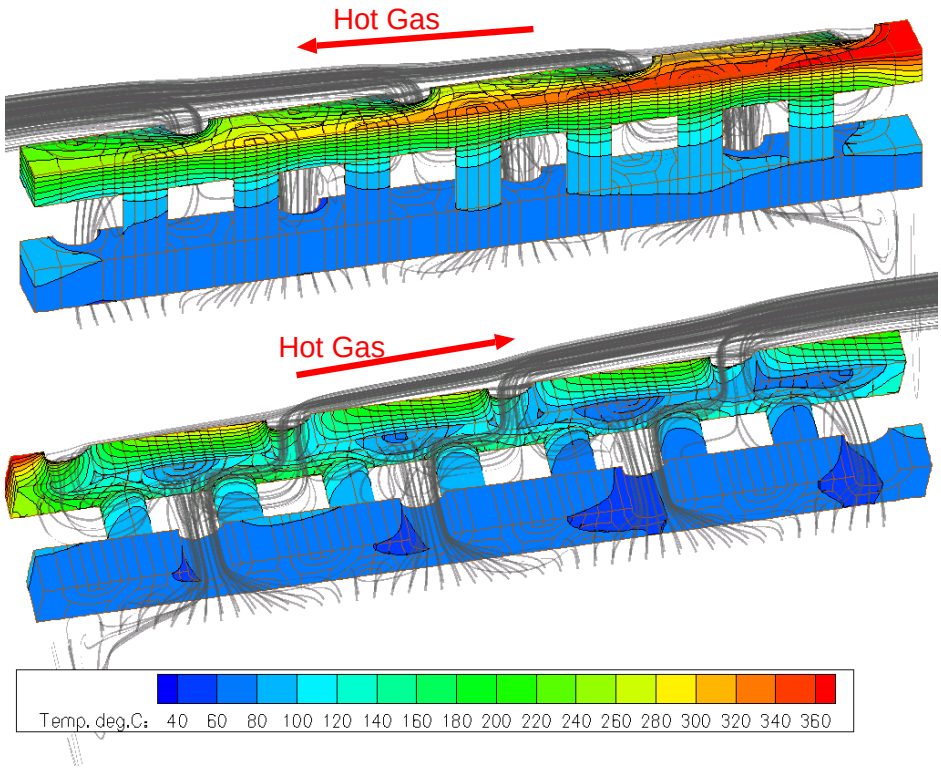


Figure 18. Surface Temperature and Streamtrace for Basic Specimen (Case B)

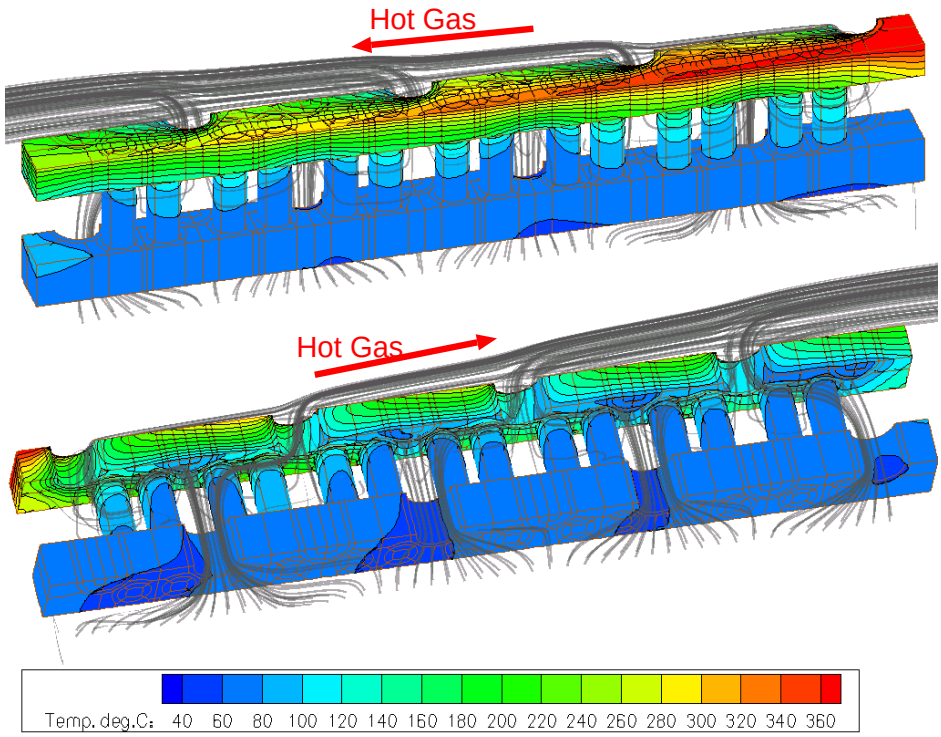


Figure 19. Surface Temperature and Streamtrace for Fine Specimen (Case B)