Numerical Analysis of the Flow Field through a Turbine Stage with Bucket Tip Clearance*

Yoshio SHIKANO**

A numerical technique for the computation of three-dimensional turbulent flows through a turbine stage with bucket tip clearance is presented. To calculate nozzle and bucket flow regions simultaneously, steady interaction is assumed to occur at the connecting boundaries. To analyze the exact flow-region geometry of the bucket tip clearance, an H-type computational grid system, which is also used in the nozzle and bucket flow regions, is used in the tip clearance region. A finite volume method is used to obtain the spatially discretized governing equations, while the damping surface technique is employed for the time integration. In the present analysis, a two-equation model of turbulence is introduced to estimate the turbulence effect. In order to verify the validity of the present method, computations are carried out for the flow through turbine stages of different tip clearance heights. The results show clear simulation of secondary flow phenomena such as the tip leakage vortex and passage vortex motions in the bucket flow region. The effects of the clearance height on the turbine stage flow fields are also well predicted qualitatively.

Key Words: Numerical Analysis, Cascade, Three-Dimensional Flow, Compressible Flow, Turbulent Flow, Turbomachinery

1. Introduction

Many turbomachines such as gas turbines, and axial and centrifugal compressors have bucket tip clearances in their structures and leakage flows through these clearances occur. These leakage flows affect the machine performance so that estimations of their effects are required during design.

Fortunately, recent progress in computer hardware technology has enabled computation using Navier-Stokes equations for three-dimensional cascade flows including the tip clearance flows. Numerical methods for predicting the tip clearance effects in cascade flows have been studied by Dawes⁽¹⁾ using the Baldwin-Lomax turbulence model and Jameson-type artificial viscosity, by Yamamoto et al.⁽²⁾ using the total variation diminishing method, by Hah⁽³⁾ using the algebraic Reynolds stress model, and by Watanabe et al.⁽⁴⁾ and Nozaki et al.⁽⁵⁾ using the multi-domain method. The numerical results obtained show the

effects of tip leakage flows on cascade flows such as a leakage vortex.

On the other hand, the flow interaction between nozzle and bucket flows caused by the short axial distance between them should be considered if the prediction accuracy for real machines is to be increased. This means that stage flow computation is required to estimate nozzle flow effects on the bucket flow and bucket flow effects on the nozzle flow. The method for this is called stage flow analysis and it reduces the difficulty of setting boundary conditions in the radial direction on the nozzle outlet and bucket inlet, including the effect of the boundary layers on the nozzle and bucket casings and side walls.

In this paper, a numerical technique for the analysis of three-dimensional turbulent flows in a turbine stage with bucket tip clearance is presented using a steady flow interaction technique⁽⁶⁾ between nozzle and bucket flow regions. The effects of turbulence are estimated by introducing a two-equation model of turbulence proposed by Launder and Spalding⁽⁷⁾. The numerical technique used is a finite volume method, which is implemented using an H-type computational grid system, for the spatial discretization and a damping surface technique⁽⁸⁾ for time integration. A multi-domain technique is also introduced to compute

* Received 13th January, 1995. Japanese original: Trans. Jpn. Soc. Mech. Eng., Vol. 59, No. 563, B (1993), pp. 2202-2208. (Received 9th December, 1992)

** Mechanical Engineering Research Laboratory, Hitachi, Ltd., 1-1 Saiwai-cho 3-chome, Hitachi-shi, Ibaraki-ken 317, Japan

bucket tip clearance flows of a thick blade using the H-type computational grid system in the clearance region, the same as is used in main flow regions.

2. Governing Equations and Boundary Conditions

The computational domain used for the stage flows with bucket tip clearance is shown in Fig. 1. The nozzle flow region is set upstream and the bucket flow region with tip clearance is set downstream. The governing equations used to describe the three-dimensional compressible turbulent flow are written in a fully conservative form with respect to a relative coordinate system (x, y, z) which rotates with angular velocity Ω ($\Omega=0$ in the nozzle domain). The resulting equations based on the finite volume method are

$$\frac{\partial}{\partial t} \iiint_V \vec{U} \cdot dV = - \iint_S (n_x \cdot \vec{F} + n_y \cdot \vec{G} + n_z \cdot \vec{H}) dS + \iiint_V \vec{I} \cdot dV \quad (1)$$

where n_x , n_y and n_z are the directional cosines of the normal unit vector on the boundary surface S of control volume V , and t is time. The vectors $\vec{U}$, $\vec{F}$, $\vec{G}$, $\vec{H}$ and $\vec{I}$ are expressed by

$$\left. \begin{aligned} \vec{U} &= \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \\ \rho q \\ \rho \varepsilon \end{bmatrix}, \quad \vec{F} = \begin{bmatrix} \rho u \\ \rho u u + \sigma_{xx} \\ \rho v u - \tau_{yx} \\ \rho w u - \tau_{zx} \\ u(\rho E + \sigma_{xx}) - v\tau_{yx} - w\tau_{zx} + Q_x \\ \rho q u + A_x \\ \rho \varepsilon u + B_x \end{bmatrix} \\ \vec{G} &= \begin{bmatrix} \rho v \\ \rho u v - \tau_{xy} \\ \rho v v + \sigma_{yy} \\ \rho w v - \tau_{zy} \\ v(\rho E + \sigma_{yy}) - u\tau_{xy} - w\tau_{zy} + Q_y \\ \rho q v + A_y \\ \rho \varepsilon v + B_y \end{bmatrix} \\ \vec{H} &= \begin{bmatrix} \rho w \\ \rho u w - \tau_{xz} \\ \rho v w - \tau_{yz} \\ \rho w w + \sigma_{zz} \\ w(\rho E + \sigma_{zz}) - u\tau_{xz} - v\tau_{yz} + Q_z \\ \rho q w + A_z \\ \rho \varepsilon w + B_z \end{bmatrix} \\ \vec{I} &= \begin{bmatrix} 0 \\ \rho f_x \\ 0 \\ \rho f_z \\ \rho(u f_x + w f_z) \\ I_q \\ I_\varepsilon \end{bmatrix} \end{aligned} \right\} \quad (2)$$

where ρ is density, u , v and w are velocity components in the x , y and z directions, E is total energy, q

Fig. 1 Computational domain and coordinate system

is turbulent kinetic energy, and ε is turbulent kinetic energy dissipation rate. In the column vectors $\vec{F}$, $\vec{G}$ and $\vec{H}$, σ is normal stress, τ is shear stress, Q is heat flux and A and B are fluxes of turbulent variables. In the column vector $\vec{I}$, f represents the sum of Coriolis and centrifugal forces, and I_q and I_ε are the source functions of the turbulence model equations⁽⁸⁾. In these notations, the subscripts x , y and z mean that the quantities are related to the x , y and z directions, respectively.

In this study, the perfect gas equation of state is used to relate pressure and temperature, and Sutherland's equation⁽⁹⁾ is introduced to calculate molecular viscosity.

The boundary conditions for the stage flow analysis with tip clearance are given as follows.

Stage inlet boundary: Total pressure, total temperature of the fluid, and two flow directions in a spanwise direction are specified and are assumed to be constant in the circumferential direction. Other unknown variables at this boundary such as kinetic energy of the mean flow, turbulent kinetic energy and turbulent kinetic energy dissipation rate are set equal to those of adjacent elements in the downstream direction.

Stage outlet boundary: Static pressure of the fluid at the root location is described at this boundary and the static pressure distribution in a radial direction is calculated by solving the pitch-wise averaged radial momentum equation. Other unknown variables at this boundary such as density, velocity components, turbulent kinetic energy and turbulent kinetic energy dissipation rate are set equal to those of adjacent elements in the upstream direction.

Periodic boundary: All unknown variables and their derivatives in the normal direction of this boundary are assumed to be the same at corresponding locations. As shown in Fig. 1, the pressure side of the main flow region and the pressure side of the tip flow region also form a periodic boundary, and the suction side of the main flow region and the suction side of the

tip flow region are also periodic in this analysis.

Wall boundary: Velocity components and the normal derivative of temperature are set equal to zero. The normal derivative of pressure is set equal to zero on the blade surface, while the normal derivative of pressure on the casing and side wall is set by considering the effects of centrifugal and Coriolis forces. In the case of a stationary bucket tip casing, which has a motion relative to the bucket, the circumferential velocity component is set equal to $-r\Omega$, where r is the casing radius. For turbulence model equations, a wall function method⁽⁷⁾ is employed to reduce the number of computational grid points near the wall.

Connecting boundary: The last boundary condition to be described is for the connecting boundary where nozzle outlet and bucket inlet computational elements overlap in the axial direction at the same radial location. Boundary conditions are set at each radial section to calculate steady interaction flow field using variables of these overlapping elements. The pitchwise-averaged static pressure of the bucket inlet elements is used as the nozzle outlet boundary condition, while the pitchwise-averaged relative total pressure, total temperature and two flow directions of the nozzle outlet boundary are used for bucket inlet boundary conditions; all five of these values are assumed to be uniform in the circumferential direction. Other unknown variables in these elements are calculated using the same methods as employed at the stage outlet and inlet boundaries. For boundary conditions of turbulent kinetic energy and turbulent kinetic energy dissipation rate, the same method is used to connect each flow region. This method is used at each computational time step until a steady flow solution is obtained.

3. Numerical Scheme

The governing equations are solved using a finite volume method. As a result of the formulation based on the finite volume method, in which arbitrarily shaped hexahedral elements are used, spatially discretized equations involve derivatives with respect to unknown variables. To avoid the difficulty of computing derivatives, a body-fitted curvilinear coordinate system is introduced and the finite difference method is used to calculate the derivatives in a transformed domain. After this computation, the mathematical relation between the transformed domain and physical domain⁽⁸⁾ is employed to calculate the derivatives in the physical domain. To illustrate the spatially discretized governing equations, the resultant x -momentum equation is described below:

$$\begin{aligned} \text{RHS} = & - \sum_{i=1}^6 (n_x \tilde{u} + n_y \tilde{v} + n_z \tilde{w})_i (\rho \tilde{u})_i \Delta S_i \\ & - \sum_{i=1}^6 (n_x \tilde{\sigma}_{xx} - n_y \tilde{\tau}_{xy} - n_z \tilde{\tau}_{xz})_i \Delta S_i + (\rho f_x)_K \Delta V_K \end{aligned} \quad (3)$$

where RHS denotes the right-hand side of Eq. (1), ΔS_i is the area of surface S_i and ΔV_K is the volume of element K . For spatial discretization of the turbulence model equation, the donor cell method⁽¹⁰⁾, which is a kind of upwind method, is introduced to stabilize the computation. The stabilization technique for the mean flow equations is included in the time integration.

Next, the time integration method is considered. In this study, the damping surface technique is used for the mean flow equations. This time integration method stabilizes the computation by the explicit use of artificial viscosity. It can be written as follows:

$$\vec{U}_K^{m+1} = \frac{\delta t}{\Delta V_K} (\text{RHS}) + \frac{\sum_{i=1}^6 \vec{U}_{Ki}^m}{6} - \alpha \frac{\sum_{i=1}^6 \vec{U}_{Ki}^m - 6 \vec{U}_K^m}{6} \quad (4)$$

where δt is the time step interval, m represents the number of time steps, and RHS denotes the right-hand side of Eq. (1). The last term of the right-hand side in Eq. (4) is recalculated every J time steps (usually $J=1-20$) and fixed during the J time steps. The total artificial viscosity needed to stabilize the computation depends on the parameter α , which is usually set between 0.95 and 1.0. In this study, the parameter α is adjusted locally to increase the accuracy of the computation while maintaining the stability. If the base value of the parameter α is α_{base} , which is used in the entire flow region, and if ρ_0 is total density at the flow inlet, then the local parameter α_c is calculated by considering the density gradient and it can be written as follows:

$$\alpha_c = \alpha_{\text{base}} \left(1 - \left| \sum_{i=1}^6 \rho_{Ki} - 6\rho_K \right| / 6\rho_0 \right). \quad (5)$$

The use of α_c calculated using Eq. (5) may destroy the convergence of the computation due to the change of the value every J time steps. Therefore threshold values of α_{max} and α_{min} are introduced as described in Eq. (6):

$$\left. \begin{aligned} \alpha_c \geq \alpha_{\text{max}} : \alpha_c &= \alpha_{\text{base}} \\ \alpha_c \leq \alpha_{\text{min}} : \alpha_c &= \alpha_{\text{min}} \end{aligned} \right\} \quad (6)$$

where α_{max} is smaller than α_{base} . The parameter α_c calculated using Eq. (6) is used for the computation.

The time integration method for turbulence model equations is based on the Euler method and no stabilizing technique is employed.

4. Numerical Examples

In order to verify the validity of the present numerical method, computations are carried out for the stage flow with various tip clearance heights. The

computational parameters are listed in Table 1. The passage height represents the blade height without tip clearance and tip clearance is generated by shortening the blade height. In Table 1, the numbers of elements $51 \times 20 \times 25$ and $18 \times 12 \times 6$ denote the numbers of computational elements in the flow direction, circumferential direction and radial direction, respectively. When the tip clearance height is set equal to 1.0 mm, 4 elements in the radial direction are used for the tip clearance computation and 21 elements in the radial direction are used for the no tip clearance computation. The computational results are shown in Figs. 2 - 10. These results are expressed in terms of values in the stationary coordinate system and the relative coordinate system for the nozzle and the bucket flow regions, respectively.

The velocity vectors in the radial section near the tip casing, including those in the tip clearance region obtained from the computation of 1 mm tip clearance height ($G/H_B = 0.0128$), are shown in Fig. 2. The velocity vectors in the main flow region are plotted alternately in the flow direction. The velocity vectors in the nozzle flow region indicate that the fluid flows along the blade passage, while those in the bucket flow region indicate the occurrence of more complicated flow phenomena caused by the effects of a boundary

layer on the nozzle tip casing and tip clearance flow. The boundary layer on the nozzle tip casing changes the velocity triangle near the tip section drastically and the inlet flow angle is completely different from the bucket mechanical inlet angle. In the section shown in Fig. 2, the incidence angle is -50° and flow enters from the suction side of the bucket. As a result of this large incidence angle, fluid flows from the suction side to the pressure side of the bucket in the region upstream of the tip clearance region. The usual tip clearance flow caused by the pressure difference between the suction side and pressure side appears downstream of the clearance region where fluid flows from the pressure side to the suction side. Then, interaction between the main and clearance flows occurs in the downstream region of this part.

Figures 3(a)-(e) show the velocity vectors in a cross section after the bucket trailing edge. The figures are viewed from the mass-averaged flow direction. When the value of G/H_B is greater than or equal to 0.0256, the tip leakage flow from the suction side reaches the pressure side through the main flow region and a strong tip leakage vortex is built up in the main flow. When G/H_B is less than or equal to 0.0128, there is no through flow from the suction side to the pressure side and interaction between the tip leakage vortex and passage vortex occurs near the tip casing.

Table 1 Computational parameters

	Nozzle	Bucket
Inlet total pressure (kPa)	130.3	
Inlet total temperature (K)	330.1	
Outlet root pressure (kPa)	104.0	
Passage height H_B (m)	0.078	0.078
Root radius (m)	0.27	0.27
Rotational speed (rpm)		3700.0
Number of elements	$51 \times 20 \times 25$	$51 \times 20 \times 25$
Elements in clearance		$18 \times 12 \times 6$
Clearance height G (mm)	8.0, 4.0, 2.0, 1.0, 0.0	
G/H_B	0.1026, 0.0513, 0.0256, 0.0128, 0.0

Fig. 2 Velocity vectors near blade tip

Fig. 3 Velocity vectors after bucket trailing edge

The passage vortex is also observed near the root side wall. A pair of passage vortices near the tip casing and the root side wall is obtained from the computation in the case of no tip clearance ($G/H_B=0.0$), and no interaction between the main flow and the clearance flow is obtained.

To show differences in the flow fields due to differences in the tip clearance height, velocity contours for 10 m/s intervals after the bucket trailing edge (same location as shown in Fig. 3) of each solution are shown in Fig. 4. Figures 4(a) and (b), in which G/H_B is greater than or equal to 0.0513, show no low-velocity core in the leakage vortex region due to the strong leakage flow, while Fig. 4(c), in which G/H_B is 0.0256, shows a low-velocity core in that region due to a weaker leakage flow. For the smaller G/H_B value of 0.0128 (Fig. 4(d)), the low-velocity core moves to the suction side and the core area becomes smaller compared with those in the case of larger tip clearances. The velocity contours obtained for no tip clearance show no low-velocity core in the region close to the tip casing. The passage vortex of the root region affects the boundary layer flow on the root side wall and the thickness of the low-velocity area on the suction side becomes greater compared with that on the pressure side.

As shown in Figs. 3 and 4, differences in tip clearance height result in differences in the interaction flow field. Although the velocity vectors and contours are

Fig. 4 Velocity contours after bucket trailing edge

useful for understanding complicated flow phenomena, turbine designers require flow patterns at the outlets of each blade, such as the velocity distribution in the radial direction. In the following, computational results are discussed using radial distributions of various flow parameters. In the graphs, h/H_B denotes the blade height location h from the root section normalized by the passage height H_B and the chain lines represent the blade (bucket) tip location when the tip clearance is considered.

Absolute velocity V_N distributions at the nozzle outlet are shown in Fig. 5. The velocity near the tip casing and root side wall becomes lower due to the effect of boundary layer development on the wall. The degree of reaction in the radial direction affects the velocity distribution in that direction and the velocity near the root section becomes larger than that near the tip casing, following the theory of turbomachinery. When a larger value of G/H_B , such as greater than 0.0513, is set for the computation, the bucket flow area is increased and the degree of reaction is rather small compared with those in other cases. This flow pattern change results in a higher velocity at the nozzle outlet. The merit of the stage flow analysis is that this flow condition is obtained automatically and the inlet boundary condition can be defined without carrying out any experiments.

The axial velocity $(V_N)_{axial}$ distributions, which are related to the important mass flow distributions, are shown in Fig. 6. These results show that an increase in the value of G/H_B results in a slight increase in the axial velocity and that the tip clearance heights do not affect the axial velocity distributions significantly.

Next, the bucket outlet flow pattern is shown and

Fig. 5 Absolute velocity distributions at nozzle outlet

discussed, referring to Figs. 7 to 10.

The relative velocity V_B distributions in the radial direction at the bucket outlet are shown in Fig. 7. In the bucket flow region, a change in the tip clearance height results in a change in the flow pattern, especially near the tip casing. When the value of G/H_B is greater than 0.0256, the relative velocity is decreased near $h/H_B=0.9$ to 0.95 of the radial location due to the effect of the tip clearance and it is increased again in the location adjacent to the tip casing due to the relative motion of the tip casing. For the lowest G/H_B value of 0.0128, no such large decrease in relative velocity occurs in that location. The results for no tip clearance ($G/H_B=0.0$) show no decrease in relative velocity, but there is an increase in the relative velocity due to the relative motion of the tip casing. The tip clearance height affects the value of the relative

Fig. 6 Axial velocity distributions at nozzle outlet

Fig. 7 Relative velocity distributions at bucket outlet

Fig. 8 Axial velocity distributions at bucket outlet

Fig. 9 Relative flow angle distributions at bucket outlet

Fig. 10 Swirl angle distributions at bucket outlet

velocity itself. When a larger tip clearance is considered in the computation, a lower relative velocity is obtained. This means that the degree of reaction becomes lower when the larger tip clearance height is introduced, as described previously.

Figure 8 shows the axial velocity (V_B)_{axial} distributions in the radial direction, which are related to the mass flow distributions, at the bucket outlet. For different tip clearance heights, the axial velocities at the nozzle outlet are almost the same (Fig. 6), while those of the bucket outlet show different distributions (Fig. 8). In the region of h/H_B of less than 0.8, where the influence of tip clearance is small, a lower axial velocity is obtained for larger G/H_B values. On the other hand, larger G/H_B values produce higher axial velocities when $h/H_B \geq 0.8$. These results indicate that an increase in the tip clearance height causes the mass flow near the tip casing to increase. This important flow pattern can be obtained from the computation which considers the tip clearance.

Next, the relative flow angle β_o distributions at the bucket outlet are shown in Fig. 9. The relative flow angle is measured from the axial direction and it is defined as negative when the fluid flows along the computational region shown in Fig. 2. The relative flow angles are almost the same except near the tip casing. The difference in flow angles appears when $h/H_B \geq 0.8$. Near the tip casing, a smaller absolute value of the flow angle is obtained from the results of the larger tip clearance height due to the absence of flow reflection caused by the bucket profile. It is reasonable that the absolute flow angle adjacent to the tip casing gives a larger value again due to the influence of the relative motion of the casing.

Figure 10 shows changes in the bucket outlet swirl angle γ , which directly relates to the amount of leaving loss of the turbine stage, with changes in h/H_B . A small swirl angle produces a small leaving loss. These swirl angle distributions obtained from the calculations performed using the different tip clearance heights are quite different. Larger tip clearances produce larger swirl angles. From these results, it is recommended that the tip clearance height be set as small as possible to obtain a small swirl angle.

As exemplified by the results of the test problems, the present numerical method gives encouraging results for analyzing the flow field in a turbine stage with tip clearance and can be a powerful tool in the design of turbomachines.

5. Conclusions

A numerical technique for computation of the turbulent flow through a turbine stage with tip clear-

ance was presented. To consider the tip flow region together with the main flow region, a multi-domain method was introduced in the technique. A steady interaction between nozzle and bucket flow regions was assumed in order to obtain a steady flow solution.

As seen from the results of the test problems using different tip clearance heights, boundary conditions at the nozzle outlet and bucket inlet were computed automatically. Complicated three-dimensional flow fields such as the leakage vortex and passage vortex in a bucket flow region were also simulated by the present method. The effects of differences in tip clearance height on the turbine stage flow were clarified from the radial distributions of variables such as velocities and flow angles.

References

- (1) Dawes, W. N., Development of a 3D Navier-Stokes Solver for Application to all Types of Turbomachinery, ASME Paper, 88-GT-70 (1988).
- (2) Yamamoto S., Daiguji H. and Ikeda S., A Navier-Stokes Analysis of 3-D Transonic Cascade Flow and Its Visualization, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 56, No. 523, B (1990), p. 765.
- (3) Hah, C., A Numerical Modeling of Endwall and Tip-Clearance Flow of an Isolated Compressor Rotor, Trans. ASME, J. Eng. Gas Turbines and Power, Vol. 108, (1986), p. 15.
- (4) Watanabe, T., Nozaki, O., Kanazawa, I. and Tamura, A., A Numerical Analysis of the Flow Passing through a Cascade with Tip Clearance, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 56, No. 527, B (1990), p. 1985.
- (5) Nozaki, O., Tamura, A., Watanabe, T. and Kanazawa, I., Numerical Simulation of the Flowfield through a Turbine Blade Row with Tip Clearance, Special Publication of National Aerospace Laboratory, (in Japanese), SP-10, (1989), p. 147.
- (6) Shikano, Y., Ikegawa, M., Nakano, S. and Satoh, T., Numerical Analysis of Three-Dimensional Compressible Turbulent Flows in a Turbine Stage, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 53, No. 496, B (1987), p. 3622.
- (7) Launder, B. E. and Spalding, D. B., The Numerical Computation of Turbulent Flows, Comput. Method Appl. Mech. Eng., Vol. 3, (1974), p. 269.
- (8) Shikano, Y., Ikegawa, M. and Nakano, S., Numerical Analysis of Two-Dimensional Compressible Turbulent Flows in a Turbine Stage, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 51, No. 471, B (1985), p. 3675.
- (9) Ikui, T. and Inoue, M., Viscous Fluid Mechanics, (in Japanese), (1978), p. 5, Rikougakushya, Tokyo.
- (10) Roache, P. J., Computational Fluid Dynamics, (in Japanese Trans. by Takahashi, R., et al.), (1978), p. 107, Kouzou-Keikaku-Kenkyujo, Tokyo.