JSME CENTENNIAL GRAND CONGRESS
International Conference on Fluid Engineering
Tokyo, Japan : July 13-16, 1997

JSME ICFE-97-613

Unsteady Flow Analysis for Cross-Flow Turbine by Use of MAC Method

Takaya KITAHORA, Dept. of Mechanical Engineering, Yokohama National University
79-5 Tokiwadai, Hodogaya-ku, Yokohama 240, Japan
Junichi KUROKAWA, Yokohama National University,
and Jun MATSUI, Yokohama National University,

ABSTRACT

Flow in a cross-flow water turbine is more difficult to calculate than that of other shaft symmetry inflow type turbines, because the flow in the runner, that enters non-symmetrically from a nozzle, is unsteady even it is observed on the relative coordinate system. And the free surface in the runner also makes the analysis difficult. In this study, the flow in the runner and nozzle is solved simultaneously on the basis of absolute coordinate system. This flow is assumed to be unsteady Euler's one, and is analysed by MAC method. The time marching is made using implicit method of Crank-Nicholson and a new method of boundary condition on walls is developed.

INTRODUCTION

Flow across the runner of cross-flow turbine is approximated to a two-dimensional flow because of its shape. And it is difficult to obtain the flow using the relative coordinate system because the flow has free surfaces and is not axis symmetry. It differs from other water turbines in which water in the runner flows axial-symmetrically.

Till now, the inner flow of the runner has been treated as one-dimensional flow assuming one representative stream line. One of prime cause of hydraulic loss at the maximum efficiency point is discharge loss at the runner exit which occur by large tangential velocity around the outer nozzle side. It does not appear in the case of other symmetrical flow type turbines. Therefore, the efficiency cannot be predicted precisely by one-dimensional calculation. On the other hand, calculation[1] has been performed on each area of the runner, separated to calculate, under the assumption of infinite number of vanes. However, these methods are not suitable for practical use. Because results with sufficient accuracy are not obtained by the former method and the latter is too complex.

Moreover, it can be thought that the influence of pressure distribution around the entrance to the runner upon the upstream flow in the nozzle is large because the flow is not axis-symmetrical. Therefore, the method considering the mutual influence of the runner and the nozzle had been desired. However, an analysis method considering the influence of nozzle shape had not been performed till now.

In this paper, the flow across the nozzle and the runner is analyzed using an absolute coordinate system. Unsteady Euler's equations are solved by using the MAC method. To obtain pressure easily, the method of imitated compression is utilized. Moreover, for quick calculation, implicit method of Crank-Nicolson is utilized and an original method is tried to achieve slip wall condition. The theoretical discharge loss can be calculated from the change in angular momentum on the runners periphery obtained by the above method. Thus the performance can be predicted.

The authors have already designed a suitable shape of a model turbine experimentally [2]. In this report, the accuracy of this analysis is verified sufficient by comparison between the calculated results and the experimental results.

NOMENCLATURE

a : Sonic speed of water

n	:	Vector component into normal direction on a wall
p	:	Pressure
U	:	Flow velocity on a wall
U'	:	Velocity component into tangential direction along wall
u	:	Velocity component to x (horizontal) direction
V	:	Velocity component to normal direction of a wall
v	:	Velocity component to y (vertical) direction

Subscript

f	:	Value on a free surface:
X	:	Component of horizontal direction
Y	:	Component of vertical direction

ANALYSIS METHOD

FDM and calculation grids

Unsteady Euler's equations are used for this calculation. The MAC method is used with staggered square grids, which is adopted in order to prevent divergence of the results. The Crank-Nicolson's implicit method is used in calculating the unsteady phenomenon in order to improve the convergence of the results and to increase calculational speed. These equations are transformed for FDM, then the velocity u in the x direction at k+1 time step is :

$$u_{k+1}=\frac{B}{A}$$
$$A=\frac{1}{\Delta t}+\frac{1}{2}\frac{\partial u}{\partial x} \tag{1}$$
$$B=u_k\left(\frac{1}{\Delta t}-\frac{1}{2}\frac{\partial u}{\partial x}\right)-v\frac{\partial u}{\partial y}-\frac{1}{\rho}\frac{\partial p}{\partial x}$$

Similarly, the velocity v in the y direction is :

$$v_{k+1}=\frac{B'}{A'}$$
$$A'=\frac{1}{\Delta t}+\frac{1}{2}\frac{\partial v}{\partial y} \tag{2}$$
$$B'=v_k\left(\frac{1}{\Delta t}-\frac{1}{2}\frac{\partial v}{\partial y}\right)-u\frac{\partial v}{\partial x}-\frac{1}{\rho}\frac{\partial p}{\partial y}-g$$

where, g, k and t denote acceleration due to gravity, time step number and time interval of one time step respectively. All variables without subscript indicate values at k time step. Upwind difference scheme of first order approximation is used for the differential calculus concerning flow velocity. Central difference method of first approximation is used for differential calculus about pressure.

Pressure is obtained by repeating above calculations and the following at the same time step, until the pressure difference between left hand and right hand becomes zero[3].

$$p_{n+1}{}^{m+1}=C\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)+p_{n+1}{}^{m} \tag{3}$$

Here, m is the number of repeated calculations needed in the same time step k.

In the compressible unsteady continuity equation, the value of C can be predicted from sonic velocity a. However, to increase the stability of the calculation, the sonic velocity is set to be around 5 times of maximum velocity of flow in the solved area, which is a much smaller value than that of real fluid. The influence of this value ceases when the result converges.

Calculation process

In this analysis, the markers represent particles moving with the velocity of liquid phase flow and are used here to distinguish a liquid phase from an air phase. Velocity of the marker is obtained by first approximate interpolation between the velocities at four sides of the grid having the marker.

To avoid the existence of liquid phase cells without the markers, more than two generating points of the markers per cell are installed on the entrance of the flow area. Moreover, the markers are generated at time intervals of about 1/5 of the period taken by the marker to go through one cell. If 8 cells surrounding the center cell having no marker include markers, a marker is generated at the center of a cell, because a cause of the occurrence is that the markers become thin due to the decrease in density.

Markers which go out of calculated area are eliminated to save the computer memory. When the above-mentioned FDM equations are applied to liquid phase, velocity and pressure are obtained. However, there are the cells whose phases change from liquid to air. In this air phase case, the pressure is reset to a constant value (atmospheric pressure) and moreover the flow velocity is assumed to be zero, at every time step.

A inconvenient situation may occur that the pressure in the cell on the boundary between the liquid phase and air phase becomes high, when the velocity on the boundary is zero. This suggests the necessity to accelerate the fluid in the cell contacting with a free surface. Therefore, the flow velocity on the free surface side of the cell is exchanged with the value on the opposite side, in order that the flow in the cell moves with

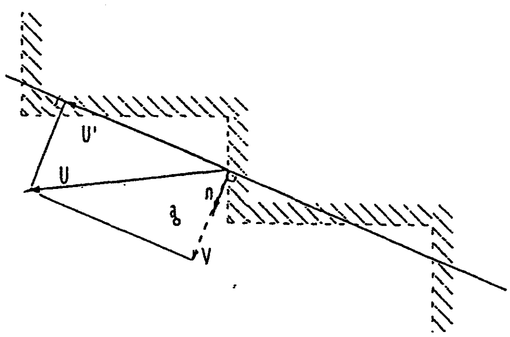


Fig. 1 Slip condition on a wall

equal speed.

The calculation of the three equations of FEM for variables u, v and p is repeated to satisfy continuity equation at the same time step. Then, each value at the next time step is calculated step by step. Time interval of one step is chosen around 1/20~1/40 of the time which the flow takes to go through one cell, considering the calculation stability and the speed of the solution.

<u>Boundary condition</u>

Slip condition is set on solid walls such as the nozzle wall. The following method is applied to active this condition.

It is necessary to compose the walls by setting the value of u and v on each side of the cell, because the square grids which do not fit correctly with the real wall shape are used. The real wall shown by a slanted line is expressed by dotted line on cell side for calculation such as Fig.1. If the flow velocities U_X and U_Y near the wall are obtained by calculation at this point then normal direction velocity components V_X and V_Y to the wall are obtained from normal direction vector n. The data of vector n are provided beforehand from real wall shape. In the calculation of the next step, the velocity U_X on the point where x-direction velocity is set, is replaced with $U_X' = U_X - V_X$.

In the case of y direction velocity U_Y on the free surface, the velocity is replaced with $U_Y' = U_Y - V_Y$. Therefore, the normal direction component of the velocity can be corrected without changing the parallel direction component to the wall in order that the water does not leak out from the wall. As a result; the smooth wall condition is obtained although the square cell is used. However, the velocity on point (a), which is passage side of the cell is used, because the velocity U_Y is not set at this point.

In the case of moving walls such as runner vanes, the modified relative velocities U_X' and U_Y' are estimated from relative velocity vector, which is obtained by subtracting peripheral velocity from the calculated absolute velocity vector. The modified absolute velocities used for variables of FDM calculation are obtained by adding the peripheral velocity to these velocities. It is very difficult to provide the positional coordinate data of moving walls for a computer program because the positions change at every time step. The moving walls at an

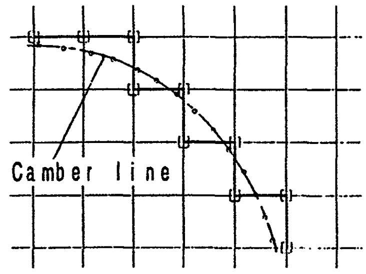
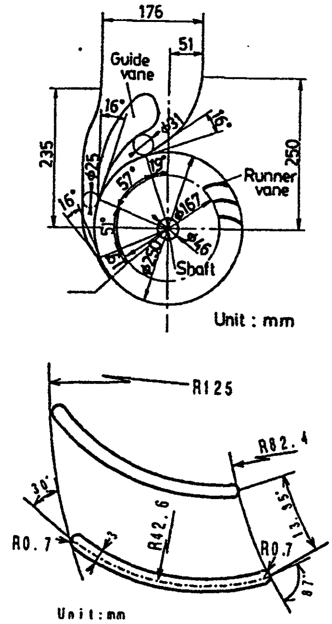


Fig.2 Method obtaining position of moving walls



(a) Nozzle and runner

(b) Runner vanes

Fig.3 Tested model turbine

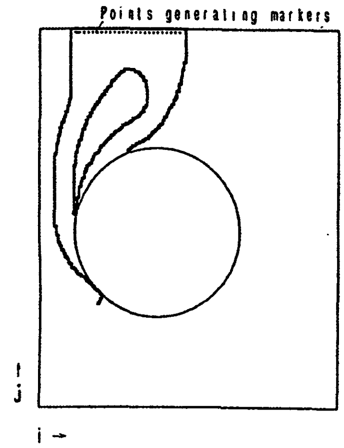


Fig.4 Calculated area and shape of solid wall

arbitrary moment are expressed by rows of points which line up in a shorter interval than cell size such as in Fig.2. The moving positions of these points at each time step are obtained easily because they can be determined by using peripheral velocity of each radius. The cell sides forming moving wall at each time step are decided by connecting the corners of the cells which are closest to these points.

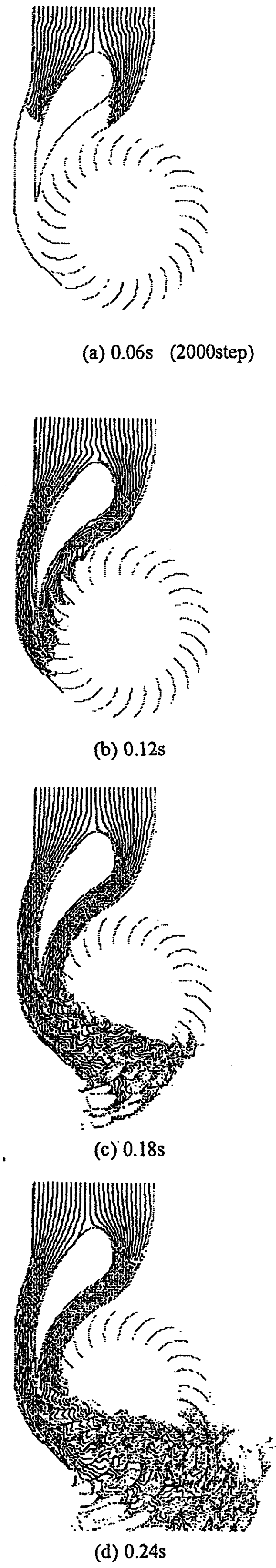
APPLICATION OF THE METHOD TO CROSS-FLOW TURBINE

Fig.3 shows the shape of a model cross flow turbine which is used in this analysis. Twenty six circular arc vanes are installed in the runner whose diameter is 250mm. Equations in this analysis are solved in a rectangular area consisting the cells whose size is 2.66mm×2.66mm square such as Fig.4. The markers are generated over the whole range of the nozzle entrance. The runner vanes are assumed to be thin, and are represented by 72 points on the camber line. Then the moving vanes form the cell boundaries of walls as explained above paragraph. The pressure at the runner exit is assumed to be atmospheric pressure. It corresponds to conditions of the calculation that the guide vane is full open and the efficiency is maximum with a net head of H=2.97m.

As a calculated result Fig.4 shows the liquid phase region in which the markers exist at each time step. Flow conditions are shown in Figs.5(a) to (d) at each time step of 0.06 seconds interval after the moment when water begins to enter the nozzle. In real flows, such undisturbed inlet flow is not realized. However, these results are shown in order to evaluate the calculation of unsteady flow.

The line patterns seen in this figure are formed by the rows of markers from several generating points, that is streak lines. The pictures of the flow across the nozzle, entering the runner and discharging from the runner are well expressed.

Water region and velocity vector diagram after 0.36 seconds are shown in Fig.6(a) and (b). The dots which can be seen thinly in the vicinity of the runner are water drops caused by the vanes. These fall gradually because of gravity. The results are confirmed to correspond well with than that of after 0.51 seconds as shown in Fig.6(c), if small disorder due to rotating vanes are ignored. Therefore it can be inferred that these flow phenomena have reached steady operating condition. Turbulences on the free surface in the downstream of the nozzle are caused by the vanes passing across the water. Fig.6(d) shows



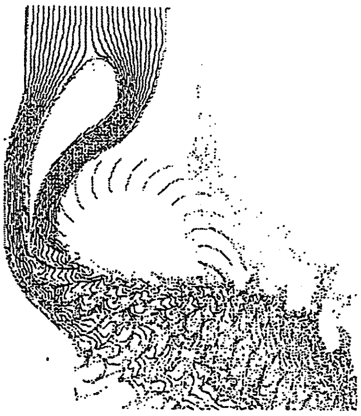
(a) 0.06s (2000step)

(b) 0.12s

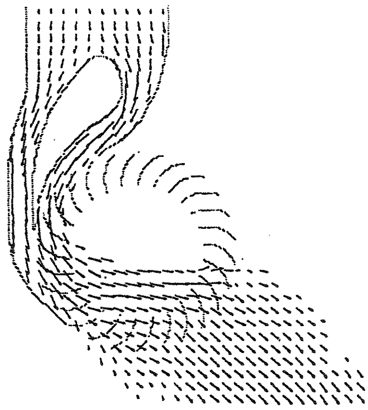
(c) 0.18s

(d) 0.24s

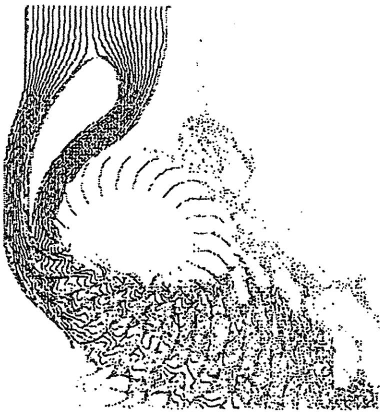
Fig.5 Region of water phase varying with time



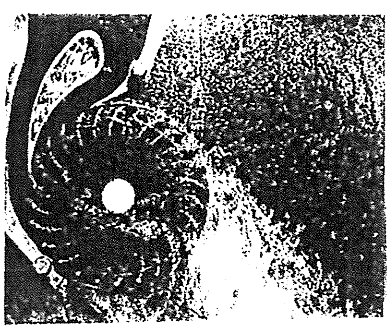
(a) Water region (0.36s)



(b) Velocity vector (0.36s)

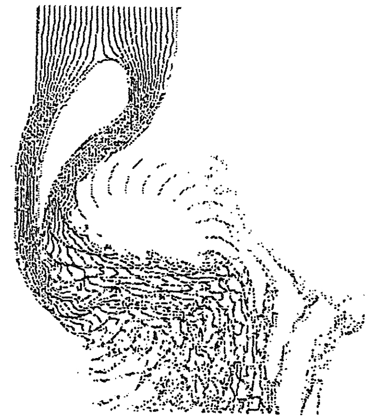


(c) Water region (0.51s)

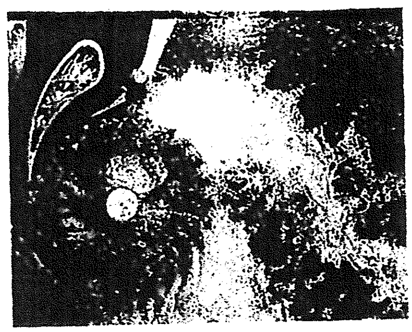


(d) Flow pattern in experiment

Fig.6 Flow patterns in steady operating condition (30rad/s)

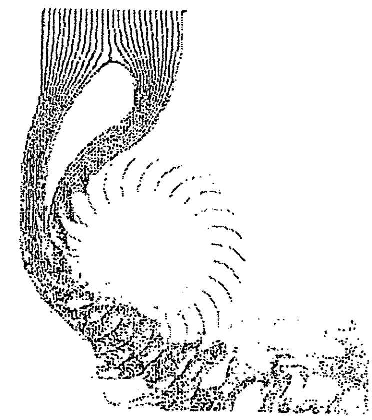


(a) Water region calculated

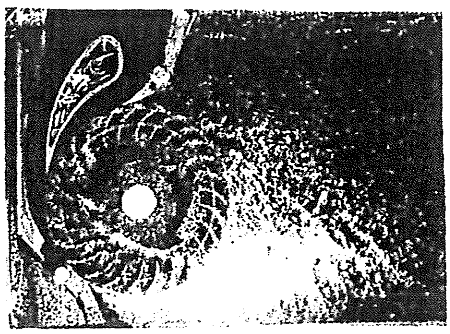


(b) Flow pattern in experiment

Fig.7 In case of low rotational speed (15rad/s, 0.36s)



(a) Water region calculated



(b) Flow pattern in experiment

Fig.8 In case of high rotational speed (45rad/s, 0.36s) a picture of flow pattern in a experimental model turbine, which is the same in both the shape and the size with the calculation, under the same operating condition. The general appearance also mostly agrees with the calculated results.

Now, theoritical shaft power W given to the runner can be calculated from the angular momentum change through following equation.

$$W=\rho u_2 R \oint v_r v_\theta d\theta \tag{4}$$

Here, the cyclic integration is performed along the runner periphery. V_r and V_θ are velocity components in radial and tangential direction on the runner periphery respectively. Data from 6 time steps between 0.36 seconds and 0.51 seconds are averaged. To do so, velocities in the area where the water phase period is less than half of whole period are assumed to be zero. Direction of the radial velocity V_r is defined as plus sign when the flow is entering the runner. Direction of the tangential velocity V_θ is defined as plus sign when it refers to the direction of the runner rotation.

Theoretical hydraulic efficiency considering only discharge loss is 79.6%, and is obtained from dividing the valuet of shaft power by ρgQH. This result is reasonable because it is about 5% higher than the experimental value. This difference can be attributed to leakage loss, friction loss and collision loss.

Moreover, the change of the flow by the difference of the rotating speed is also examined. Fig.7(a) shows the area of liquid phase and velocity vectors after 1.2 seconds in the case of rotational angular velocity 15rad/s of the runner. It can be confirmed, as well as shown in the experimental results, that the flow angles in the entrance to the vane-less area of the center and in the exit of the runner are close to the vane outlet angle, because the rotational speed is slow. Fig.7(b) shows the experimental result at the same rotational speed. The flow is separated in the runner, that is different from calculation, because a shaft is installed in the center of a real runner. But an overall tendency of the flow angle is almost the same with the calculation.

In addition, the result for the case of 45rad/s is shown in Fig.8(a). The opposite tendency to the case of 15rad/s can be observed in this result owing to large influence of the rotating speed. Fig.8(b) of an experimental flow pattern shows that the jet width becomes thin as such as calculational result. The turbulence of free surface is larger than that of the calculation.

CONCLUSION

The flow in the nozzle and the runner of the cross flow turbine is analyzed by the MAC method, and the perfomance is predicted from the angular momentum change.

(1) The flow across the runner and the nozzle could be analysed irrespective of the difficulties caused by free surface, the non-axial symmetry and the flow unsteadiness owing to vane rotation. It has been confirmed that the results obtained here can simulate important aspects of actual flow.

(2) The power received the runner is calculated from the angular momentum difference between the entrance and the exit of the runner. Then analyzed unsteady results are averaged over several time steps. Theoretical performance is obtained by considering discharge loss. The value thus obtained is reasonable. Consequently, the present method is useful for the prediction of hydraulic efficiency at the design stage of similar machineries.

REFERENCES

1. Fukutomi J,et al.,JSME Int. J Ser.2, Vol.34, No.1 pp.44-51 (1991)
2. Kitahora T,et.al.,Trans. Jpn. Soc. Mech. Eng.,Vol.61, No.585, pp.1744-1749 (1995)
3. L.W.B.Browne, Numerical Simulation of Fluid Motion, pp.223 (1978)