Numerical simulation of turbulence flow in a Kaplan turbine
-Evaluation on turbine performance prediction accuracy-

P Ko1 and S Kurosawa2
1,2 Toshiba Corporation
20-1 Kansei-cho Tsurumi-ku Yokohama, 230-0034, Japan
E-mail: pohan.ko@toshiba.co.jp, sadao.kurosawa@toshiba.co.jp

Abstract. The understanding and accurate prediction of the flow behaviour related to cavitation and pressure fluctuation in a Kaplan turbine are important to the design work enhancing the turbine performance including the elongation of the operation life span and the improvement of turbine efficiency. In this paper, high accuracy turbine and cavitation performance prediction method based on entire flow passage for a Kaplan turbine is presented and evaluated. Two-phase flow field is predicted by solving Reynolds-Averaged Navier-Stokes equations expressed by volume of fluid method tracking the free surface and combined with Reynolds Stress model. The growth and collapse of cavitation bubbles are modelled by the modified Rayleigh-Plesset equation. The prediction accuracy is evaluated by comparing with the model test results of Ns 400 Kaplan model turbine. As a result that the experimentally measured data including turbine efficiency, cavitation performance, and pressure fluctuation are accurately predicted. Furthermore, the cavitation occurrence on the runner blade surface and the influence to the hydraulic loss of the flow passage are discussed. Evaluated prediction method for the turbine flow and performance is introduced to facilitate the future design and research works on Kaplan type turbine.

1. Introduction
Efficiency performance usually plays a main role on evaluating a hydro turbine because it is directly reflected to the power output. But besides the efficiency performance of the whole turbine system, the detailed flow behaviour inside the flow passage especially the runner can be crucial. As a hydro turbine is in operation, unstable flow behaviour is the key generating noise and vibration. This instability is often induced by the occurrence of cavitation and the pressure fluctuation caused by vortex. It erodes and fatigues the material of the turbine including runner blades by the continuous impulsion released by the collapsing cavitation bubbles and the swirling vortex. This no doubt shortens the lifespan of the turbine and leads to a bad cost performance. Also, when the occurrence of cavitation is in a critical condition as the tail-water level drops, it significantly deteriorates the efficiency performance. On top of the above disadvantages, besides the efficiency-related hydraulic loss, the phenomena of cavitation and pressure fluctuation also need to be captured. Substantial way often leads to a conventional model test, but the test tends to (1) a significantly high manufacturing cost, (2) a significantly long manufacturing time, (3) possibility of the unavoidable human error during the model apparatus design and assembling, and most importantly (4) the lack of appropriate or easy means detecting the detailed insight and mechanism of the unstable flow behaviour such as cavitation and vortex, therefore a numerical simulation is preferable.
The most common type of hydro turbine in use today is the Francis type, but regarding the needs of the rapidly growing market in emerging countries such as China and Brazil, Kaplan type hydro turbine is more hydraulically efficient to the rivers of lower water head. Therefore, a study on Kaplan type turbine is certainly required. Different from Francis type turbine, Kaplan type turbine blade features with additional tip edge moving immediately adjacent to the stationary shroud wall. As water flows around the tip edge, secondary flow occurs and triggers cavitation inside the swirl [1]. Aside from the Kaplan blade of hydro turbine, ship propeller is also no exception. Due to the complexity of the flow behaviour near the ship blade tip, the study was also carried out numerically. The comparison between the experimental and numerical data was made. It was found that despite the complexity of the flow with smaller scale flow pattern, other than the grid refinement the turbulence model plays a more crucial role. The anisotropy of RSM was suggested the key helping accurately predict the radial velocity gradient near the tip and was applied to numerically reproduce the vortex tube experimentally measured and observed [1, 2].

RSM was also applied by the second author for studying Francis type turbine. Turbine efficiency and occurrence of cavitation were accurately predicted [3]. Given by this achievement, the suggested numerical method is also expected for the study on Kaplan turbine. As one of the results of this work, RSM combined with Rayleigh-Plesset equation is found accurately predicting Kaplan turbine efficiency and cavitation but not pressure fluctuation at the runner outlet or at the upper draft tube. Kaplan turbine has blade tip like ship propeller, but instead of having the tip vortex flowing in the free stream, the tip vortex in a Kaplan turbine occurs and evolves with a stationary shroud wall immediately nearby. Furthermore, Francis turbine does not have blade tip. As a result, the pressure fluctuation becomes relatively more complex and unpredictable in Kaplan type turbine. Regarding Large Eddy Simulation (LES) was reported accurate for predicting the flow in draft tube [4], where pressure fluctuation of interest occurs, therefore a related study is also carried out and briefly discussed later in this paper.

2. Prediction method of model turbine performance

2.1. Fundamental equations and physical model

The fundamental equations for the performance characteristics of a Kaplan turbine are given as follows.

\[ H = H_{in} - H_{out} \]  \hspace{1cm} (1)

\[ H_{th} = \frac{2\pi \times N \times T}{60 \times Q} \]  \hspace{1cm} (2)

\[ \eta = \frac{H_{th}}{H} \]  \hspace{1cm} (3)

\[ n = \frac{N}{\sqrt{H}} \hspace{1cm} q = \frac{Q}{\sqrt{H}} \]  \hspace{1cm} (4)

Q, N, H and T are the discharge, rotational speed, effective head and runner torque, respectively. \( H \) is calculated by using the total pressure difference between turbine inlet (\( H_{in} \)) and turbine outlet (\( H_{out} \)). \( H_{th} \) is the theoretical head acting on runner blades and is represented by eq.(2).

Turbine efficiency (\( \eta \)) is predicted by dividing theoretical head (\( H_{th} \)) with the effective head (\( H \)). The prediction of the turbine performance in the operating range is done by the use of unit speed (n) and unit discharge (q) of eq.(4), which are derived from given actual rotational speed (N), discharge (Q), and predicted effective head (H).

Three-dimensional turbulence flow simulation

As for URANS simulation, it is important to employ the anisotropy of the Reynolds stresses as the turbulence model since the secondary flow and rotational force effect is strong in runner and draft tube.
flow. RSM model closes URANS equations by solving the transport equations of Reynolds stresses, together with an equation of the dissipated rate. [6]

As for the wall modeling, the non-equilibrium wall function is used. This wall function is more suitable in regions, where the mean flow and turbulence are subjected to severe pressure gradients and change rapidly, and it is effective for the off-design flow simulation, which is involving the separation and reattachment near blades. The non-equilibrium wall function is based on two layers concept in computing the budget of turbulent kinetic energy at the wall adjacent cells, which is needed to solve the k equation at wall-neighboring cells. [7]

As for an additional simulation on pressure fluctuation at the runner outlet later in this paper, LES solver is applied. The governing equations of LES for incompressible flow are spatially filtered continuous equation and Navier-Stokes equations, which refers to the second author’s prior work. [4]

**Cavitation modelling**

As for the evaluation of critical cavitation point that turbine performance begins to drop when cavitation coefficient is decreasing, it is necessary to take into consideration the influence of the unstable behaviour of cavitation bubbles in the turbine flow passage. The basic equation of cavitation flow analysis is URANS equation that is expressed the cavitation appearance region by Volume of Fluid (VOF) method. About phenomenon, such as growth and collapse of the cavitation nuclei and the bubbles, it is modelled by the modified Rayleigh-Plesset equation shown below. In this equation, the effect of bubble interaction is taken into account by adding the first term at the left-hand side of the Rayleigh-Plesset equation.[8]

\[
\frac{d}{dt} \left( \sum_i \frac{1}{r_i} \frac{dR}{dt} R^2 \right) + R \frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 = \frac{P_{\text{crit}} - P}{\rho_L} \tag{8}
\]

Where \( r_i \) is the distance to bubble \( i \) center, \( \rho_L \) is the density of the surrounding liquid, \( P \) is the pressure in the surrounding liquid away from the bubble and calculated by the URANS equation, and \( P_{\text{crit}} \) is the pressure in the liquid at the bubble boundary calculated by the equation as follow. \( P_v \) is the pressure in the bubble, and \( T \) and \( \mu \) are the surface-tension constant and the coefficient of the liquid viscosity.

\[
P_{\text{crit}} = P_v - \frac{2T}{R} - 4\mu \frac{\partial R}{\partial t} \tag{9}
\]

The volume fraction of vapour \( f_g \) in the URANS equation is determined by the following formula based on the radius \( R \) of the cavitation bubbles calculated by (8) and (9) formula.

\[
f_g = n \frac{4}{3} \pi R^3 \tag{10}
\]

In addition, the initial bubble density \( n \) used as the generation source of cavitation is assumed to be constant all over the computational domain, though real cavity flows have a distributed bubble density.

2.2. **Numerical method**

The discretization of URANS equations is done by finite volume method. Convective terms of URANS equations are approximated by the third order MUSCL scheme and differential term with respect to time is approximated by second order time difference. As for numerical algorithm to solve the algebraic finite volume equations with continuity, the PISO method is used. Due to applying the sliding mesh interface, the relative motion between the rotational part and the stationary part is simulated.

In addition, as for the prediction of cavitation performance, the Courant number is kept less than 0.3 to stabilize the calculation since the time scale in a growth and collapse of cavitation bubbles is small 3 - 5 order as compared with a mainstream phenomenon.
2.3. Computational grid and boundary condition

Ns 400 Kaplan type model hydro turbine is studied. Kaplan type turbine is widely applicable to the market in countries or regions such as China, Brazil, and south-east Asia but due to its relatively large discharge, cavitation is severe. Although the numerical prediction of turbine efficiency is made at both design and off-design points, only off-design point which has 1.4 times of design discharge is focused for cavitation and pressure fluctuation studies. Configuration and summery of this turbine is shown in Figure 1. The calculation conditions are summarised in Table 1.

![Figure 1. Configuration of Ns 400 Kaplan model turbine](image1)

| Specific speed | 400 (min⁻¹, m-kW) |
|----------------|-------------------|
| Runner outlet diameter (De) | 350 mm |
| Stay vane number | 24 |
| Guide vane number | 24 |
| Runner blade number | 5 |

![Table 1. Calculation conditions](image2)

The computational model is shown in Figure 2. The number of grid points is about 30 million. In the numerical simulation, it adopts the high order numerical scheme and the high accuracy turbulence model. Therefore higher grid quality needs to be kept as well to avoid the instability in the calculation. To achieve the high quality of the computational model, an automatic multi-block grid generation method minimizing the skewness of the block and cell is used. [9]

The computational boundary conditions are applied at the inlet surface and at the outlet surface of the computational domain. About the inlet boundary condition, a uniform velocity distribution is assumed. As for the outlet boundary condition, the average pressure is set to fix. Furthermore, about the surface of the passage wall, the non-slip boundary condition is prescribed, i.e. the velocity components are set to zero.

![Figure 2. Overview of computational model](image3)
Toshiba has developed an in-house software package integrating meshing, flow simulation and performance prediction described here.

3. Experiment

Validation data for numerical results are measured on a test rig at Hydraulic research laboratory in Toshiba Corporation. The test rig for model turbine is shown in Figure 3 as a typical example. The model test was conducted on the basis of IEC standard. The following performance data were measured and the inaccuracy of measurement was estimated less than 0.3%.

- Efficiency
- Pressure fluctuation
- Critical cavitation (Efficiency break down)

![Photograph of model turbine](image1)

(a) Photograph of model turbine

![Outline of model turbine test stand](image2)

(b) Outline of model turbine test stand

Figure 3. Model test equipment for Model hydro turbine

4. Simulation results and discussion

4.1. Model turbine efficiency

A study of an Ns 400 Kaplan model turbine is carried out experimentally and numerically. Figure 4 shows the comparison of the effective head (H) and efficiency (η) between the results of numerical simulation and experiment. Those at both the design and off-design points are made. The torque (T) acting on the runner, discharge (Q), and rotational speed (N) are numerically predicted and experimentally measured, and the efficiency values are then calculated by eq.(2) and eq.(3). In these figures, the values are normalized by the experimental results. From these figures, it shows that the proposed numerical simulation method gives accurate results comparing with those measured experimentally.

![Figure 4](image3)

Figure 4. The comparison of the effective head and efficiency between the numerical and experimental results in both operating conditions at design (A) and off-design (B) points.

Figure 5 shows the surface flow on the runner blade, of which the comparison between the numerical and experimental results is made. In these figures, the numerical results are presented by using the velocity data of the cells adjacent to the blade surfaces. From these figures, it shows that the numerical method successfully simulates the distorted surface flow experimentally visualized near the
inlet region of the pressure surface and the outlet region of the suction surface. Therefore it is thought that the internal flow field of the runner is well predicted.

![Pressure surface](image1)

(a)

![Suction surface](image2)

(b)

Figure 5. The blade surface flow of numerical simulation (a) and experimental surface oil flow visualization (b). Both pressure and suction surfaces are presented.

Figure 6 shows the flow velocity distributions numerically simulated and experimentally measured. They are the velocity distribution at the runner outlet. In these figures, the velocity value is normalized by the averaged axial velocity value corresponding to each operating condition, and the measure point locates 400mm downstream of the turbine center. As for the direction of tangential velocity, the negative value designates the rotational direction of runner. It is found a satisfactory agreement between the results of numerical simulation and experiment.

![Design point](image3)

![Off-design point](image4)

Figure 6. Flow velocity distributions numerically simulated and experimentally measured at runner outlet

4.2. Cavitation Performance

For studying influences of cavitation, it is common by looking at the relation between model turbine efficiency $\eta$ and cavitation coefficient $\sigma$. At an operating point with stable net head and discharge, the turbine efficiency is stable and independent of cavitation coefficient. When cavitation coefficient drops to a certain point, the turbine efficiency begins to act unstably. Cavitation coefficient drop may sometimes lead to an efficiency rise temporarily, but at the end, leads to a rapid efficiency breakdown when cavitation coefficient goes below the so-called critical cavitation point $\sigma_c$. $\sigma_c$ is determined by the intersection of line A and line B shown at the left hand side of Figure 7. Line A indicates the averaged efficiency of those points with the $\sigma$ values ample and away from the potential $\sigma_c$. And line B indicates the best fit line of those points with the efficiency values below line A and with the $\sigma$ values roughly equal and smaller than the potential $\sigma_c$. The cavitation coefficient $\sigma$ is the dimensionless number describing the elevation of tail-water level.

The relation between the model turbine efficiency and the cavitation coefficient as well as the component hydraulic loss of the Ns 400 Kaplan model turbine are numerically predicted and plotted in Figure 7. Off-design point, of which the discharge is 1.4 times of that of design point, is selected for the study due to the operating condition of larger discharge leads to a severer cavitation influence. At the left hand side of Figure 7, it compares the results of both experiment and numerical simulation. It shows the experimentally measured relation between turbine efficiency and cavitation coefficient.
Meanwhile, the numerical results show a good agreement with those of experiment. The numerical simulation also captures the critical cavitation point $\sigma_c$ measured experimentally with satisfactory accuracy. These agreements validate the proposed numerical simulation method. For further study on the cavitation influence, the numerically predicted hydraulic losses of flow passage components are shown at the right hand side of Figure 7. The components include the flow passage upstream of the runner, the runner, and the flow passage downstream of the runner, i.e. draft tube. Studying from the result, it is known that the cavitation leads to an increasing loss at the runner but plays a relatively minor role to the stationary parts including both up and downstream of the runner.

Figure 7. Turbine efficiency and cavitation coefficient comparison between experimental and numerical results (left), and the component hydraulic loss (right)

As for the Ns 400 Kaplan model turbine in this study, along with the change of cavitation coefficient $\sigma$, an efficiency breakdown is predicted when $\sigma$ drops roughly below 0.9. The $\sigma$ drop reflects the tail-water level drop of a turbine. The lower tail-water level decreases the fluid pressure around the runner and facilitates the occurrence of the cavitation bubbles on the blade surface and the surrounding. The evolution of the cavitation bubbles on the blade surface along with the dropping $\sigma$ is shown in Figure 8. It is the numerically simulated results showing the contours of both liquid-vapour volume fraction and pressure distribution in colour. Each main configuration shows the volume fraction contour with a sub-configuration showing the corresponding pressure distribution contour. For volume fraction contour, the blue colour designates the water vapour while the orange colour designates the water liquid. The colour in between designates the transient between water vapour and liquid. For pressure distribution contour, the blue designates the low pressure area while the red colour designates the high pressure area. Cavitation usually occurs downstream of the runner, so only the contours on runner blade and hub are snapped for the discussion. There are six snapped configurations which are of the same operating point but different cavitation conditions of $\sigma = 1.6 \sim 0.7$ also shown in Figure 7, respectively. As in the condition of $\sigma=1.6$, which is of the highest tail-water level and highest sounding pressure at runner outlet among the all, it shows that nearly no cavitation on the blade surface. As $\sigma$ drops, it is clear that cavitation bubble starts to occur and the amount of vapour is inversely proportional to $\sigma$. Starting from $\sigma = 1.6$, despite pressure drop happens near the leading edge of the blades, the pressure has not yet gone below the saturated vapour pressure and therefore no cavitation occurs. When $\sigma$ reaches 1.3 and 1.1, somewhere at the leading edge near the tip starts to have the surface pressure drops and cavitation occurs. When the cavitation bubbles get dragged to the downstream, it increases the low pressure area on the suction surface of the blades. On the contrary, this induces bigger load acting on the blades. This contributes a bigger runner output and a better turbine performance via an efficiency rise seen in Figure 7. When $\sigma$ goes around 0.9 and below, somewhere at the suction surface near the hub also starts to have the pressure dropping below saturated vapour pressure. Consequently, the pressure drop also permeates on hub surface. The hub produces nearly no output, so this pressure drop mainly leads to a hydraulic loss at the runner outlet. This explains the turbine efficiency breakdown when $\sigma \leq 0.9$ in Figure 7. Critical cavitation coefficient $\sigma_c$ is about 0.9 here.
4.3. Pressure fluctuation characteristic

Cavitation on the blade surface is often induced by the pressure drop in an operating condition of lower $\sigma$. Aside from the discussed near-blade cavitation, cavitation is also induced by the vortex in the flow passage. The vortex swirls up and generates a low pressure area at its rotating center, and then cavitation occurs when the center pressure drops below saturated vapour pressure. The vortex generates cavitation and often forms in a tube shape fluctuating the flow in the flow passage. The fluctuation is recognizable by measuring the near-wall pressure inside the flow passage over a period of time. The numerical and experimental results for the pressure fluctuation characteristics are plotted at the left hand side of Figure 9. The pressure fluctuation was experimentally measured by using the pressure transducers set on the upper draft tube wall. It is clearly that there is a huge discrepancy between the numerical and experimental data. As for the turbulence model, RSM, which is URANS-based, is used. Different from the energy-based turbulence models such as two-equation, and SST model, its anisotropy helps accurately resolve the complex vortex and cavitation phenomena from the blade tips [2]. But different from ship propeller blades, the tip vortex in a Kaplan turbine additionally encounters a stationary wall immediately nearby. The flow behaviour becomes even more complex and increases the numerical difficulties. RSM consequently damps out the tip-induced vortex before it reaches the runner outlet, where the pressure fluctuation is measured. By studying the eddy viscosity contour at the right hand side of figure 9, it is known that the eddy viscosity rises at the runner outlet and therefore damps out the vortex and fluctuation phenomena.

Figure 9. Runner outlet pressure fluctuation comparison between experimental and RSM-based numerical results (left) and eddy viscosity contour (right)
Large Eddy Simulation of pressure fluctuation

The flow behaviour at the runner outlet and further down to the draft tube is complex and harder to predict. Aside from the tip-induced vortex in Kaplan turbine, the unsteady secondary flow essentially exists [4]. Referring to the prior work by the second author, Large Eddy Simulation is applied for Kaplan runner outlet pressure fluctuation simulation here. The comparison between the experimental and numerical results is shown at the left hand side of Figure 10. The result shows that the pressure fluctuation phenomenon is not damped out. And the right hand side clearly shows LES has resolved a relatively low eddy viscosity throughout the flow passage. By comparing with Figure 9, LES is relatively successful on capturing the amplitude of the fluctuation in a Kaplan turbine.

Figure 10. Runner outlet pressure fluctuation comparison between experimental and LES-based numerical results (left) and eddy viscosity contour (right)

Figure 11 shows the numerically predicted vortex by LES turbulence model. It shows that LES does not damp out the tip-induced vortex at the runner outlet. Tip vortex in free stream, like that downstream of ship propellers, usually evolves and forms a tube in shape [1, 2]. The vortex tube can be simple and relatively easy to predict. On the other hand, this vortex tube is also found evolving from the blade tip in a Kaplan turbine, but instead of in free stream, the tip vortex is induced in an extremely narrow clearance between rotating runner and stationary shroud wall. The tip vortex occurs due to the radial velocity gradient along the rotating direction of runner. But right after that, instead of being flown to the downstream freely, the vortex runs into adjacent shroud stationary wall, which is relatively rotating in an opposite direction. And a component of peripheral velocity gradient is therefore added to the vortex. Consequently, the vortex pattern in a Kaplan turbine becomes relatively complicated and difficult to be predicted.

Figure 11. LES-resolved vortex tubes downstream of the runner of a Kaplan model turbine
5. Conclusions
The high accuracy prediction method based on the whole flow passage model is applied for the study of a Kaplan turbine and the prediction accuracy is evaluated with RSM and then LES. The results obtained are as follows:

(1) As for the efficiency characteristic of a Kaplan turbine, the relation with cavitation coefficient is accurately captured by the proposed numerical method with RSM turbulence model. The critical cavitation coefficient is predicted and the turbine efficiency breakdown is found with the pressure drop on the hub surface.

(2) As for the pressure fluctuation characteristic, RSM turbulence model is found not accurate for the prediction on a Kaplan turbine. The blade tip induced vortex is numerically damped out before reaching the runner outlet. The pressure fluctuation at the runner outlet is therefore underestimated.

(3) LES resolves the blade tip induced vortex tube at the runner outlet in a Kaplan turbine. LES does not damp out this phenomenon and resolves the induced pressure fluctuation at the runner outlet.

Numerical simulation on a whole flow passage of a Kaplan turbine is carried out. Proposed numerical method with RSM is found sufficient to have a satisfactory accuracy on efficiency and cavitation prediction. Its accuracy replaces the burdensome preparation and cost for model test, and also facilitates future research and design works. For pressure fluctuation prediction, despite LES is still needed, a rapid improvement on computer system today is expected to resolve the difficulties on calculation time and data storage in near future.

Acknowledgments
This work was completed by the help of the project members in hydraulic research laboratory of Toshiba Corporation and the support of members in Toshiba Information System Corporation. The authors would like to acknowledge Kiyoshi Matsumoto, Suzuki Toshiaki, Nakamura Takanori, Akira Shinohara for helpful advice, and Kunie Ochiai and Oo Thanda for the technical support of the large scale computer system.

References
[1] Arndt R E A et al 1991 Some observation of tip-vortex cavitation Journal of Fluid Mechanics 229 269-289
[2] Frank T et al 2008 Investigation of pressure fluctuations caused by turbulent and cavitating flow around a P1356 ship propeller NAFEMS Seminar (Wiesbaden, Germany)
[3] Kurosawa S et al 2010 Virtual model test for a Francis turbine Proceedings 25th IAHR Symposium (Timisoara Romania)
[4] Kurosawa S et al 2006 Turbulent flow simulation for the draft tube of a Kaplan turbine Proceedings 23rd IAHR Symposium (Yokohama Japan)
[5] JSME-S008 1999 Performance conversion method for hydraulic turbines and pump-turbines JSME, ed (Tokyo Japan)
[6] Fu S et al 1987 Modeling strongly swirling recirculating jet flow with Reynolds-stress transport closures Sixth Symposium on Turbulent Shear Flows (Toulouse, France)
[7] Kim S et al 1995 A near-wall treatment using wall functions sensitized to pressure gradient ASME FED Separated and Complex Flows 217
[8] Kurosawa S et al 2003 Numerical prediction of critical cavitation performance in hydraulic turbines Proceedings, 4th ASME-JSME Joint Fluids Engineering Conference (Hawaii, USA)
[9] Peter R et al 1994 Application of multiblock grid generations with automatic zoning Numerical Grid Generation in Computational Fluid Dynamics and Related Fields