Large-Eddy Simulation of Non-Cavitating and Cavitating Flows in the Draft Tube of a Francis Turbine

Yang GUO*, Chisachi KATO** and Kazuyoshi MIYAGAWA***

1. INTRODUCTION

Under the partial load conditions in the turbine mode of a reversible pump-turbine, the whirl of a vortex rope is produced in the draft tube and under certain operation conditions it may cause flow instability that leads to vibration or noise. On the other hand, the operation range of a pump-turbine is strongly related to the cavitation phenomena, which may occur either in the vanes of the runner or in the stationary parts. The success of extending the operation range depends on the knowledge about the cavitation physics in the machine.

Due to the above reasons, many studies have been conducted on the characteristics of the vortex rope and the cavitation phenomena in pump-turbines. Nishi et al. studied the behavior of the vortex rope by experiments and proposed simple numerical models for predicting the behavior of the vortex rope. Avellan, Miyagawa et al. investigated the draft tube flow within a Francis turbine by experiments as well as CFD analysis and clarified the flow patterns in the draft tube. Sick et al. investigated the flow instability in the draft tube of a pump-turbine by means of unsteady Reynolds-averaged Navier-Stokes simulation (URANS) and the time-averaged quantities agree fairly well with experiments. Wang and Nishi proposed a quasi-3D analytical model to simulate swirling flow with spiral vortex rope in a pipe to predict the rotating frequency of the vortex rope. Sirok et al. did a simultaneous structural dynamics analysis and static analysis of static pressure pulsation of a cavitating vortex rope in the draft tube of a Francis turbine. Nishi et al. investigated the cavitating vortex rope by experiment. They found that when the natural frequency of the draft tube vibrating system with cavitation region approaches the rotating frequency of the cavitating rope in the elbow, the intensive pressure pulsation called draft-tube-surging occurs due to the resonance.

Spatially-filtered incompressible Navier-Stokes equations were solved by our finite element code “FrontFlow/Blue”. Dynamic Smagorinsky Model (DSM) with modification by Lilly was used to account for the Subgrid-Scale (SGS) stresses. An explicit time-accurate streamline-upwind scheme with second-order accuracy both in time and space was applied for the time integration. Fractional Step (FS) method was employed to solve the pressure equation. Residual Cutting Method (RCM) combined with Bi-CGSTAB method was used as the matrix solver of the global linear system of equations that results from the FS method.

2. COMPUTATIONAL METHODS

2.1 Numerical Schemes

Spatially-filtered incompressible Navier-Stokes equations were solved by our finite element code “FrontFlow/Blue”. Dynamic Smagorinsky Model (DSM) with modification by Lilly was used to account for the Subgrid-Scale (SGS) stresses. An explicit time-accurate streamline-upwind scheme with second-order accuracy both in time and space was applied for the time integration. Fractional Step (FS) method was employed to solve the pressure equation. Residual Cutting Method (RCM) combined with Bi-CGSTAB method was used as the matrix solver of the global linear system of equations that results from the FS method.

2.2 Cavitation Model

For the calculation of cavitating flows, the cavitation model of Okita et al. was adopted. The evolution of cavitation is represented by the source/sink of the vapor phase in incompressible liquid flows, and weak compressibility is taken into account through the low-Mach-number assumption. The formulation of the cavitation model is represented as:

$$\frac{DF_c}{dt} = \left[ c_k (1-f_L) + c_m f_L \right] \left( \rho - \rho_v \right)$$
In Eq. (1), $f_L$ is the liquid volume fraction, $p$ is the static pressure, $p_v$ is the vapor pressure and $C_g$, $C_l$ are model constants. Here we adopted the values of these model constants as recommended in Ref. 14): if the static pressure is lower than the vapor pressure, $C_g=1000$ and $C_l=1$, otherwise $C_g=100$ and $C_l=1$.

3. COMPUTATIONAL RESULTS

3.1 Non-Cavitating Flow

An operation point with an effective head of 50 m, rotational speed of 1,150 rpm, flow rate of 0.15 m$^3$/s and guide vane opening of 50 % was adopted for the computation. Figure 1 shows the computational models. Two computational models were adopted in the computation of the non-cavitating flow: draft tube only and draft tube with the runner. In the draft tube only case, there are about 1.8 million hexahedral elements in the computational grid of the draft tube. Steady axial and circumferential velocities were assumed at the inlet of the draft tube in the current computation. The velocity profiles are the RANS results of Miyagawa et al.$^{3}$. Non-slip condition was prescribed on the wall and convective boundary condition was used at the outlet. For the computation of this draft tube only case, the time increment was set such that 1,400 time steps corresponded to one revolution of the runner.

For draft tube with the runner case, the computational domain consists of the runner, the draft tube and an outlet buffer region (not shown in Fig. 1) that is added to reduce the influence of the outlet boundary condition on the flow field. The computational grids are composed of about 1.9 million hexahedral elements in the runner and about 1.2 million in the draft tube. The inlet condition of the runner was specified by a uniform velocity profile along the circumferential direction. The moving boundary interface between the runner and the draft tube was treated with overset grids from multiple dynamic frames of reference. The details of the algorithm can be found in Kato et al.$^{15}$. In the computation of flow with low-Mach-number assumption, non-reflecting boundary condition$^{16}$ was used at the outlet of the computational domain. For the system with the draft tube and the runner, 4,000 time steps corresponded to one revolution of the runner. The computation cost of runner/draft tube system is much higher than that for the draft tube only case.

Figure 2 compares the time-averaged velocity components at the inlet of the draft tube in the case with draft tube only (also denoted as Case A) and in the case with the draft tube and the runner (also denoted as Case B). At the inlet of the draft tube the radial velocity is small in both of the two cases. The differences can be seen in the tangential velocity and the axial velocity. Near the crown side of the runner ($r/R_{inlet}<0.5$), the tangential velocity and axial velocity in Case A is larger than that in Case B. On the other hand, near the shroud side ($r/R_{inlet}>0.5$) the tangential velocity in Case B is larger than that in Case A.

Figure 3 plots the velocity profiles along a sampling line at the sampling section located at the downstream of the inlet of the draft tube (see Fig. 1 (bottom) for the position of the sampling line). The sampling line is perpendicular to the meridional plane and the arrow of the line indicates the positive direction of the line. It can be seen from Fig. 3 (b) that the axial velocity profile of Case B is closer to the experimental data of the co-authors. Near the crown side of the runner ($r/R<0.5$), the tangential velocity in Case B is also closer to the experimental data (see Fig. 3 (a)). Overall, we think the results of Case B are better than that of Case A at the

---

Fig. 2 Comparison of time-averaged velocity profiles at the inlet of the draft tube ($U_r^*$: radial velocity, $U_t^*$: tangential velocity, $U_m^*$: axial velocity, all the velocity components are normalized by the tip speed of the runner $U_t^*$)
However, near the shroud side \( (r/R > 0.5) \), the tangential velocity in Case B is larger than the experimental one. It can be inferred that at the inlet of the draft tube, the tangential velocity near the shroud side \( (r/R_{inlet} > 0.5) \) in Case B is also overpredicted. This may be attributed to insufficient grid resolution.

Figure 4 compares the instantaneous pressure at the meridional plane. In the draft tube only case, it is found that the minimum pressure in the draft tube is higher than the vapor pressure at the plant cavitation number, for which the occurrence of cavitation has been confirmed in the experiment. That means it is impossible to predict the cavitating flow in this case. While it can be found in Fig. 4 (b) that in the case with the runner, the minimum pressure point is located at the cone of the runner. The minimum pressure at the cone of the runner is lower than the vapor pressure in the case of the plant cavitation number. It is thus possible that the cavitation will be initiated near this place. It should be noted that it is also possible to calculate the cavitating flow with only the draft tube in theoretical sense if we have the correct profiles of velocity and liquid fraction at the inlet of the draft tube. However, in practical sense, it is difficult to obtain the unsteady distribution of velocity and liquid fraction at the inlet of the draft tube in advance. Thus we concluded that it is better to include the runner in the computation to predict the cavitating flow.

Figure 5 compares the pressure pulsation at a pressure sampling point on draft conical wall (see Fig. 1 for the position of the sampling point). In Case B, the swirling frequency of the draft tube vortex rope is approximately 0.52 times the rotational frequency of the runner, while in Case A, the swirling frequency of the vortex rope is approximately 0.28 times the rotational frequency of the runner. The frequency in Case B agrees reasonably well with the experimental data measured by Miyagawa et al., which is 0.48 times the rotational frequency of the runner. The amplitude of the pressure pulsation (peak to peak) at this frequency in Case B is
about 1.9% of the effective head. This amplitude also agrees reasonably well with the experimental data \(^{16}\), which is 2.1% of the effective head.

### 3.2 Cavitating Flow

According to the results of non-cavitating flow, the runner/draft tube system was used to calculate cavitating flows. Computations of cavitating flows at two cavitation numbers were conducted: the plant cavitation number \(\hat{\sigma}=0.146\) and a lower cavitation number \(\hat{\sigma}=0.09\). In the current computations, the cavitation number is defined as: 

\[
\hat{\sigma} = \frac{p_e - p_v}{\frac{1}{2} \rho L U_0^2},
\]

where \(p_e\) is the hydrostatic pressure at the shroud casing of the runner (the axial position of the shroud casing is shown in Fig. 1 (bottom), but the shroud casing itself is not shown in the figure) and the reference velocity is com-
puted by \( \overline{U} = \sqrt{2gH_e} \), where \( g \) is the gravity acceleration and \( H_e \) is the effective head of the Francis turbine.

Figure 6 shows the instantaneous iso-surfaces of 5% void fraction in LES computation as well as the experimental photographs obtained by the co-authors. In both cases, a whirl cavity develops from the cone of the runner to the draft tube. The size of the cavity is dependent on the cavitation number \( \sigma \), but the vertical motion is similar. This can be confirmed by the pressure pulsation at the sampling point on draft conical wall shown in Fig. 7. In both of the cases, no cavities were found within the runner vanes. The current LES can predict the occurrence of the cavitation inception and the change of cavity size with the change of cavitation number. However, the size of the cavity in the computation seems smaller than that in the experiment, especially in the case of the plant cavitation number (see Fig. 6 (a) and Fig. 6 (c)). We assume that this is due to the insufficient grid resolution in the runner and the draft tube, especially in the region of the vortex rope. As a result, the computed instantaneous minimum pressure at the center of the vortex rope may not be very accurate.

The pressure pulsation shown in Fig. 7 indicates that the amplitude of the pressure pulsation at the sampling point decreases when the cavitation number becomes smaller. The effects of cavitation will be investigated in detail after a more accurate cavitation field is obtained.

4. CONCLUSIONS

Large-eddy simulations of non-cavitating and cavitating flows in the draft tube of a Francis turbine under a partial load condition were performed. Computational results in the non-cavitating flow with two computational models were compared: the draft tube only case and the case with the draft tube and the runner. The velocity profiles at a sampling line in the case with the runner are closer to those in the experiment of the co-authors, although the tangential velocity is still overpredicted in this case. Moreover, the minimum pressure point is near the runner cone so that it is better to include the runner in the computation to predict cavitating flows. In the case with the runner, the computed swirling frequency and the dominant pressure fluctuation of the draft tube vortex rope agree reasonably well with the experimental data. Results of cavitating flows indicate that the cavitation phenomena can be predicted qualitatively. However, the computed unsteady pressure in the vortex rope may not be accurate. We assume this is due to insufficient grid resolution especially in the vortex rope. The computation with finer grid is in progress.

ACKNOWLEDGEMENTS

A part of this research was done in “Revolutionary Simulation Software for the 21st century (RSS21)” project supported by Research and Development for Next-generation Information Technology of Ministry of Education, Culture, Sports, Science and Technology (MEXT), Japan. A part of the computation was done at The Earth Simulator Center under a collaborative project “Research and Development of Revolutionary Simulation Software”. The computation of the flow in the draft tube was initiated by Mr. J. Kanazawa while he was staying at The University of Tokyo. The authors would like to thank Dr. K. Okita of RIKEN for the beneficial discussion of cavitation models and non-reflecting boundary conditions.

(Manuscript received, December 8, 2006)

REFERENCES


