Large eddy simulation of pressure fluctuations at off-design condition in a Francis turbine based on cavitation model

W T Su\textsuperscript{1,2}, X B Li\textsuperscript{1}, F C Li\textsuperscript{1}, W F Han\textsuperscript{1}, X Z Wei\textsuperscript{1,2} and J Guo\textsuperscript{3}

\textsuperscript{1}Harbin Institute of Technology, School of Energy Science and Engineering, Harbin 150001, China;
\textsuperscript{2}State key laboratory of hydro-power equipment, Harbin 150040, China;
\textsuperscript{3}Zhejiang University, Institute of Chemical Process Machinery, Hangzhou 310027, China

E-mail: suwentao@hit.edu.cn

Abstract. To study the instability of a Francis turbine at off-design operating condition, a hydraulic model was established and the flow characteristics at the off-design point were studied based on large eddy simulation (LES). The simulation was conducted for both single phase model and cavitation model. The results were compared with the experimental data. Results show that the simulation based on cavitation model can capture more channel vortex structures than single phase calculation. The result of vortex rope by cavitation model is similar to the experimental result. The dominant frequency can be obtained by these two methods, while the result based on cavitation model can capture the high frequency component at the inlet of draft tube. Great difference can be seen from the internal flow of the two simulation results. These conclusions can provide a basis for the study of instability of Francis turbine.

1. Introduction

The phenomena of unsteady cavitating flow is complex, experimental study of the cavitation is hard to perform in many fields of cavitating flow. The mechanism of the occurrence of cavitation, the parameters related to the development of cavitation and the basic rules of cavitation erosion are investigated mainly by qualitative analysis. There has no breakthrough evolution for quantitative analysis of the cavitating flow. Thus, numerical simulation of the unsteady, three-dimensional flow in the turbomachineries has become a key technology in the study the cavitating phenomenon.

At present, lots of researches have been done to investigate the cavitation in turbines. Due to the limit of the computational technology and the computational fluid dynamic (CFD), studies of cavitation in turbines were focusing on the instable flow in draft tubes\textsuperscript{1-4}. With the development of numerical technology, the unsteady flow in the runner, between the stay vanes and guide vanes were studied by computational methods. Grein\textsuperscript{5} simulated the channel vortex and compared the cavitating flow with the results of model test. Peter\textsuperscript{6} predicted the pressure fluctuations in an draft tube by the methods of characteristics, and analyzed the reason for causing the pressure fluctuation. Grindoz\textsuperscript{7} studied the cavitating flow in a turbine by cavitation model. Kurosawa\textsuperscript{8} predicted the shape of channel vortexes alone the runner blade and in the middle of the passage.

In order to capture the shape of channel vortex in the runner and the vortex rope in the draft tube, and study the relationship with instability, unsteady flow in the turbine was calculated by CFD software. The shapes of the channel vortex and vortex rope were analyzed. The reason for causing
such unsteady cavitating flow was investigated. The study of cavitating flow can provide a basis for
the detailed experimental research, such as particle image velocimeter test. Reynolds averaged Navier-
Stocks (RANS) can’t capture all scales of vortex, it will not predict the internal flow and the external
characteristics accurately[9]. Thus, Large eddy simulation (LES) were used for the study of cavitating
flow in the turbine.

2. Turbine structure and mesh

2.1. Parameters of the Francis turbine

Parameters of the Francis turbine is show in the Tab.1. Figure 1 shows the structure of the turbine. The
model Fracis turbine contains casing, stady vanes, guide vanes, runner and draft tube.

| Parameter | Value |
|-----------|-------|
| \(D_1/\text{mm}\) | 420 |
| \(Z\) | 15 |
| \(Z_0\) | 24 |
| \(B_0/\text{mm}\) | 76.68 |
| \(Z_s\) | 23 |

**Table 1. Geometric parameters of Francis turbine**

2.2. Mesh

In special working condition, such as partial load, the flow in a turbine is so complex that the mesh of
the hydraulic region for predicting the unsteady cavitating flow must be refined. The turbine is divided
into five parts, and the structured meshes in the runner and casing are shown in Figure 2. \(D_1\) is the
diameter of runner; \(Z\) is the number of blade; \(Z_0\) is the number of guide canes; \(B_0\) is the height of guide
vanes; \(Z_s\) is the number of stay vanes.

3. Numerical method

3.1. Turbulence model and cavitation model

The governing equation for LES:

\[
\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \frac{\partial \tau_{ij}}{\partial x_j}
\]  

(1)

(2)
Where, $\tau_{ij} = 2(C_s\Delta)^2 S_{ij} (2 S_{ij} S_{ij})^{1/2} - \frac{1}{3} \tau_{kk} \delta_{ij}$ is the sub-grid-scale (SGS) stress, and the Smagorinsky model is used.

The continuity equation and momentum equations for calculating the cavitation flow are shown as follows,

$$\frac{\partial (\rho_i \alpha_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho_i \bar{u}_j \alpha_i \right) = \dot{m}^+ - \dot{m}^- \quad (3)$$

$$\frac{\partial (\rho_i \bar{u}_j \alpha_i)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho_i \bar{u}_j \alpha_i \right) = 0 \quad (4)$$

The exchange of mass flow between the vapor and liquid for a unit volume is,

$$\dot{m} = N_b \frac{d \left( \rho_v 4\pi R^3 / 3 \right)}{dt} = 4\pi N_b \rho_v R^2 \frac{dR}{dt} \quad (5)$$

The source terms for vaporation and condensation are,

$$\dot{m}^+ = C_\epsilon \frac{3 \rho_v \left( 1 - \alpha_v - \alpha_g \right) \text{Max}(\alpha_g, \alpha_{nuc})}{R_b} \sqrt{\frac{2}{3} \text{Max}(p_v - p, 0)} \quad (6)$$

$$\dot{m}^- = C_\epsilon \frac{3 \rho_v \alpha_v}{R_b} \sqrt{\frac{2}{3} \text{Max}(p - p_v, 0)} \quad (7)$$

where $R_b = 10^{-6}$ m is the radius of cavity bubble; $C_\epsilon = 50$, $C_c = 0.01$.

3.2. Boundary condition

Inlet boundary: total pressure was used at the inlet of the casing, which was defined by model test result.

Outlet boundary: static pressure was set at the outlet of draft tube, which was set according to the cavitation coefficient.

$$\sigma = \frac{H_a - H_{va} - H_s - H_v}{H} \quad (8)$$

where, $H_{va}$ is the vacuum value of the draft tube; $H_a$ is the head related to atmospheric pressure; $H_s$ suction height of the turbine; $H_v$ is the head related to saturated vapor pressure at the test temperature.

Wall: No slip boundary conditions were used.

The time step for unsteady calculation was 0.0002 s, which the runner rotated for 1 degree. The rotational speed of the runner was show in Tab.3.

3.3. Operating condition

The operating condition of the turbine was shown in Table 2, which the channel vortex can be found in the runner. The opening of guide vanes is $a = 10$ mm (relative opening is 55.6%). Single phase flow and multiphase flow were used to investigate the cavitation phenomenon. $N_{11}$ is the unit speed, $Q_{11}$ is the unit flow rate, $\sigma$ is the cavitation coefficient, $T$ is the temperature of the water.

| $N_{11}$       | $Q_{11}$ | $\sigma$ | $T$ |
|---------------|---------|---------|-----|
| 63.58 r/min   | 255 L/s | 0.1009  | 20°C|

3.4. Verification of grid independence

Five kinds of scales are used for the verification of grid independence. The amplification factor ($\alpha$) for mesh scale is 1.2. The efficiencies of the five cases are shown in Fig.3. The efficiency is close to a certain value when the number of mesh is more than 8 million, and the calculation result agrees well.
with experimental data ($\eta=94.8\%$). The mesh used for the calculation of unsteady cavitating flow is shown in Table 3.

![Figure 3. Verification of grid independence](image)

**Table 3.** Grid details for computational domain

| Domain          | Casing and stay vanes | Guide vanes | Runner | Draft tube |
|-----------------|-----------------------|-------------|--------|------------|
| Nodes           | 1104122               | 1919160     | 3551122| 970830     |
| Cells           | 1228125               | 2054640     | 3768512| 993876     |

### 4. Results and discussions

#### 4.1. Pressure fluctuations in draft tube

To analyze the influence of the channel vortex and the vortex rope on the pressure fluctuation in draft tube, HD1 and HD2 were monitored and the positions of the two points were set according to IEC standard. HD1 and HD2 were 0.3D2 far from the runner outlet, and the positions were shown in Fig.4. Pressure fluctuations were shown be peak to peak value, and compared with experimental results. Numerical simulations based on single flow and multiple flows were performed, and the results were compared with experimental data.

![Figure 4. Locations of monitor points](image)

The pressure fluctuation at time domain of the two point were shown in Fig.5 and Fig.6. The dominant frequency of pressure fluctuation is $0.18 \, f_n$ ($f_n$ is the rotational frequency of the runner), which is in good agreement with the experimental result ($0.2 \, f_n$). The result based on single phase couldn’t capture the high frequency except the dominant frequency. The result of multiphase flow has the component of high frequency which is more than rotational frequency. The result of multiphase flow is larger than the experimental result, and it may be caused by the vibration of the unit. In Fig.7,
it can be seen that the pressure fluctuation in HD1 differs half a cycle from the HD2 point, which is caused by the geometric position.

4.2. Flow in the draft tube
Volume fraction of 10% \cite{10} is used to obtain the shape of vortex rope in the draft tube for multiphase calculation results, and saturated vapor pressure is used for single phase result. The shapes of the vortex ropes in draft tube are shown in Fig.8. The shape of vortex rope in draft tube based on multiphase flow agrees well with experimental results shown in Fig.9. The vortex rope has a heliciform shape. The pressure fluctuation in the draft tube has a direct relationship with the rotation of the draft tube.
4.3. Vortex channel in runner

Vorticity criterion is used to show the shape of channel vortices in the runner \[11\].

Vorticity denotes the rotation of velocity, and it shows the vector of the flow field, which shows the vector of maximum rotational intensity.

\[
\Omega = \nabla \times \mathbf{u}
\]

(9)

Figure 10 shows the computational results of single phase and multiphase. More channel vortexes can be captured by cavitation model at the same criterion. The channel vortexes in the runner based on cavitation model distributes along the flow direction, while the results of single phase are along the axial direction. During the experiment, it can be found that the channel vortexes in the runner agree well with the results of multiphase calculation. Thus, results based on cavitation model can capture the channel vortexes accurately.

(a) Cavitation model
(b) Single phase

**Figure 10.** Channel vortex display by vorticity criterion

4.4. Streamlines in the runner

Streamlines in the runner on the blade to blade surface are shown in Fig.11. The flow in the runner by cavitation model is great different from the result based on single phase. There have lots of large scale vortexes at the runner inlet of pressure side. The incidence angle between the flow direction and runner inlet is negative value, which leads to the flow separation at the pressure side of the blade, even causing the channel vortex in the runner. The results based on cavitation model are in line with the actual situations.
5. Conclusions

Numerical simulation of a Francis turbine was performed by LES combined with cavitation model. The computational results of channel vortexes, the vortex rope and pressure fluctuations in the draft tube are investigated and compared with the experimental result, which are in good agreement with each other. The results can provide a basis for engineering design.

Results of this paper are summarize as follows:

(1) LES combined with cavitation model are used to predict the cavitating flow in a Francis turbine, and the results are compared with results based on single phase. Results of cavitation model are in good agreement with experimental data.

(2) Pressure fluctuations based on cavitation model can predict the high frequency component at the inlet of draft tube.

Model test of a Francis turbine can see the actual shape of the vortexes in the turbine, and it can be also used to check the performance of the turbine. Due to the reason for causing the vortex rope is complex and the position of channel vortexes are difficult to observe, PIV or laser doppler velocimetry (LDV) test is one of the most useful method to measure the flow fields in a turbine. It can show the channel vortex and the vortex rope in the draft tube clearly. The results of LES can provide an advice for the the measurement surface in the turbine.

References

[1] Vu T C and Shyy W 1990 Fluids Engineering 2 199-204.
[2] Ruprecht A, Helmrich T and Scherer T 2002 Simulation of Vortex Rope in a Turbine Draft Tube Proc. of the 21st IAHR Symp. (Lausanne, Switzerland, 9-12 September 2002).
[3] Xiao R F 2004 Study on Flow Field Simulation and Performance Improvement of the Medium-High Specific Speed Francis Turbine (Wuhan: Huazhong University of Science and Technology) (in Chinese)
[4] Zhang S Q, Xue F W, Liu Z H, et al 2005 Hydroelectric Energy 23 50-52 (in Chinese).
[5] Grein H and Goede E 1994 Site Experience with Francis Turbines Operation under Very Large Head Variations Proc. of the 17th IAHR Symp. (Beijing, China, 15-19 September 1994)
[6] Peter K and Doepler J E 2009 Evaluating of 1-D models for vortex-induced pulsation in Francis turbine Proc. of the 3th IAHR int. meeting of the workgroup on cavitation and dynamic problems in hydraulic machinery and system (Brno, Czech Republic, 14-16 October 2009).
[7] Grindoaz B 1991 Lois de Similitudes dans les Essays de Cavitation des Turbines Francis (Switzerland: Lausanne Federal Polytechnic University).
[8] Kurosawa S 2003 Numerical Prediction of Critical Cavitation Performance in Hydraulic Turbines Proc. of the 4th ASME-JSME Joint Fluids Engineering (Honolulu, Hawaii, 6-10 July 2003)
[9] Su W T, Li F C, Li X B, et al 2012 *Engineering Applications of Computational Fluid Mechanics* 6 356-65.

[10] Liu J T, Liu S H, Wu Y L, et al 2012 *Computers and Fluids* 68 105-111.

[11] Liu D M 2011 *Numerical analysis of the channel vortices and vortices in draft tube of a Francis turbine based on the cavitation calculation* (Beijing: Tsinghua University) (in Chinese)