The optimization of j-groove shape in the draft tube of a francis turbine to suppress the draft surge

Q S Wei and Y D Choi
(61 Dorim-ri) 1666 Youngsan-ro, Cheonggye-myeon, Muan-gun, Jeonnam, South Korea
E-mail: sidda306@126.com

Abstract. Suppression of draft surge caused by vortex and cavitation surge in the draft tube is very important to improve the turbine performance when the turbine is operated in the range of partial load condition. In present work, a series of CFD analysis have been conducted in the range of partial load, design condition and over load of a Francis turbine model with a kind of J-Grooves. The pressure contours, circumferential velocity vectors and vortex core regions in the draft tube are compared by the conditions with or without J-Grooves. Study results show that the J-Grooves can suppress the abnormal phenomena to some extents on the condition of maintaining the efficiency. In the second stage, the shape of J-Groove is optimized step by step considering the groove length, depth and width normalized by the diameter of outlet of turbine runner.

1. Introduction
Quite often turbines tend to be operated at a regime of far away from the design point associated with demand on the energy market. Particularly, Francis turbine, experience an abrupt decrease in efficiency and severe pressure fluctuations at off-design operating regimes [1]. Moreover, it is also underwent draft tube surge at partial load [2, 3] and even at full load [4]. With the development of computational methods, the analysis of the swirling flow were investigated both focus on radial and axial velocity component by computational and experimental methods [5, 6], and some correspondence suppressing methods were proposed. However, it is still a big challenge and a lot of job remained unfinished. Among the more important is cavitation. Cavitation is a three-dimensional, unsteady and discontinuous phenomenon of formation, growth and rapid collapse of bubbles. It makes the hydraulic machinery present unwanted consequences such as flow instabilities, excessive vibrations and damage to material surfaces and degradation of machine performance [7]. Here we only talk about draft tube swirl cavitation though leading cavitation, travelling bubble cavitation, von Karman cavitation are also common in Francis turbine.

CFD analysis takes the advantage of less time consuming and low cost, and it gradually becomes the most popular method to simulate cavitation flows. Latest study dates back to the CFD validation of performance improvement of a 500kW Francis turbine which is conducted by Choi in 2013. It’s pointed out that numerical prediction, including the evaluation of the local flow and the global variables, of the turbine model at optimum and off design conditions is critical to ensure the satisfaction of the desired performance specifications [8]. However, since the cavitation is such a complicated physical phenomenon, it is impossible to treat all factors which may affect the cavitation. Assumption of treating the flow as two phases with a mixture density is popularly used. Based on that,
Zhang conducted a numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation and showed that cavitating flow was well predicted [9]. However, most studies are related to the noise and the investigation of the two phase flow of the full operating range is barely conducted. In this study, we target at the suppression of the two phase flow surge in the draft tube by jet groove, J-Groove in short. Large amount of previous studies on the draft surge of the full range and cavitation analyses were deeply checked and the CFD methods were compared to get the final numerical methods. Steady state and transient analysis were carried out with both one and two phase flow. Refer to the paper generated by Kurokawa, the optimal procedure was accomplished.

2. Analysis Method Investigations
In this study, we adopted a commercial code of ANSYS CFX ver.12.0 [10] to conducted CFD analysis. As this software is dependent on the numerical grids, turbulent model and boundary condition, many cases are verified to get the final decision.

![Figure 1. Schematic view of the turbine model.](image1)

![Figure 2. Schematic view of the guide vane.](image2)

2.1. Controlling equation
We consider a turbine system consist of an inlet pipe of diameter $D_i$, a spiral casing with distributor (a cascade of stay vanes and guide vanes), a turbine runner of outlet $D_o$, a draft tube with J-groove and without groove at expansion angle $\beta$, and an outlet pipe of diameter $D_{out}$, as shown in Fig. 1. The guide vane angle is marked as $\alpha_g$ and turbine is operated from $\alpha_{gmin}$ to $\alpha_{gmax}$, as shown in Fig. 2. Groups of J-Groove are tested as shown in Table 1.

From the continuity, we have the governing equations

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho u) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\alpha_v \rho_v) + \nabla \cdot (\alpha_v \rho_v u) = \dot{m} \quad (2)$$

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \frac{1}{3} \nabla \left[ (\mu + \mu_t) \nabla \cdot u \right] + \nabla \cdot [(\mu + \mu_t) \nabla u] + \rho g \quad (3)$$

Where $\rho$ is the density of flow (water in one phase flow analysis and the mixture of the vapour and water flow in two phase flow analysis, the same in follow), $u$ is the Reynolds averaged relative velocity, $\alpha_v$ is the volume fraction of cavity phase, $\rho_v$ is the density of vapour cavity phase, $\dot{m}$ is the mass transfer between vapour and liquid phase. $\mu$ is the viscosity of mix-flow and $\mu_t$ is the eddy viscosity of mix-flow.
2.2. Turbulence model

Turbulence of Renormalization Group (RNG) $k-\varepsilon$ [11] and $k-\varepsilon$ SST are included in two phase flow simulations. Because the depth of J-Groove is quite shallow, so high accuracy boundary layer simulation is required. Thus SST turbulence model is included in this study. The SST combines advantage of $k-\omega$ model in the near wall region and standard $k-\varepsilon$ model away from the walls. The turbulent kinetic energy equation and the specific dissipation rate equation are given as follows:

The turbulence kinetic energy equation is:

$$
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho U_j k \right) = \frac{\partial}{\partial x_j} \left( \left( \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_t - C_m \rho k \omega + \frac{C_a \mu_t}{\sigma_k} \left( \frac{\partial (g_T)}{\partial x_j} \right)
$$

(4)

The dissipation rate equation is:

$$
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} \left( \rho U_j \varepsilon \right) = \frac{\partial}{\partial x_j} \left( \left( \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right) + \gamma \rho \varepsilon - \beta' \rho \omega^2 + \frac{(1-C_\omega) \beta \rho}{\sigma_t} \left( \frac{\partial (g_T)}{\partial x_j} \right) + \frac{(1-F_s) \rho \sigma_{\omega_2}}{\omega} \left[ \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \right]
$$

(5)

Where

$$
P_t = \min(\mu_t \phi, C_{int} \varepsilon)
$$

(6)

and $C_{int} = 10^{15}$, $\sigma_k = 1.176$, $\sigma_\omega = 2.0$, $\gamma = 0.5532$, $\beta' = 0.075$, $\sigma_{\omega_2} = 1.168$.

2.3. Others

According to the tutorial given by CFX help file, inlet of pressure base on reference pressure of 0 atm and bulk mass flow rate was attached to the outlet. The cavitation was activated at the pressure of 3574 Pa. transient rotor stator interface model was employed at the interfaces between distributor & runner and runner & draft tube. High resolution with second order backward euler was set in the solver and a maximum coefficient loops of 10 is adopted to get a smooth transaction between two adjacent time step.

3. Results and Discussion

| Group Number | L [mm] | W [mm] | D [mm] | J-Groove Quantity |
|-------------|--------|--------|--------|-------------------|
| case 1      | 140    | 46     | 14     | 12                |
| case 2      | 210    | 46     | 14     | 12                |
| case 3      | 280    | 46     | 14     | 12                |
| case 4      | 280    | 46     | 5      | 12                |
| case 5      | 280    | 23     | 14     | 12                |
| case 6      | 280    | 69     | 14     | 12                |
3.1. Performance of the Francis turbine
Though matured over past few decades, computational fluid dynamic (CFD) analysis, like most tradable instruments carry their own disadvantages, has its own drawback over simulating the actual fluid flow. That is, the analysis can be tailored to meet the needs of the researcher, despite of the careful selection of the grid geometry and resolution, as well as the computational technique used. In other words, the reliability of the CFD analysis results should be based on the experimental ones. Fig. 3 shows the experimental and computational efficiency curves versus guide vane angle. It can be easily pointed out that at the best efficiency point the computational and experimental efficiencies are almost the same. At partial load regime, the efficiencies deviation starts to get bigger while the flow rate decreases. That because on that condition guide vane angle gets smaller and water passes through the cascade of distributor in a higher rate. However, the flow characteristics near the wall is so complex that it hard to present it by a simple equation or a turbulence model in CFD analysis. The error relates to the Reynolds number which is obtained by the equation

\[ Re = \frac{VL}{\nu} \]  

where \( V \) is the mean velocity of the object relative to the fluid (m/s); \( L \) is a characteristic linear dimension, (travelled length of the fluid; hydraulic diameter when dealing with river systems) (m); \( \nu \) is the kinematic viscosity (m²/s). On the condition of partial load and overload condition, relatively high velocity exists near the hub and cover that the deviation is enlarged especially in partial load cases. Nevertheless, the maximum deviation is severely restricted within 5%. In other words, the CFD simulation with the employed method is reliable. Fig. 4 presents the efficiency by different kinds of J-Groove shapes and the dash line is that of non-groove case. It can be easily found that the efficiency can be maintained.

3.2. Suppression of swirl flow in the draft tube
Swirl flow plays an important role in causing the draft surge of the turbine. When turbine operated at partial load, strong circumferential flow occurs and vortex will be formed. Fig. 5 shows the vortex in non-groove case in the draft tube. Fig. 6 presents the relative areas of the vortex surface \( V_{vor}/V_{non} \) from case 4 to 6 are below 1 that the vortices in those cases are diminished. Fig. 7 and Fig. 8 show the axial and tangential velocity along the radius at the region near the draft tube inlet. From the figures we can find that the J-Groove do have suppressed the swirl flow in the swirl flow both in axial and tangential direction and among that case 6 and case 2 show a significant effect.

3.3. Suppression of cavitation in the draft tube
The occurrence of cavitation can be predicted by the analysis of Cavitation Number which is calculated by the equation of

$$\sigma_{cav} = \frac{p_{test} - p_{sat}}{\frac{1}{2} \rho V^2}$$ (8)

Where $p_{test}$ is the absolute pressure of test location and $p_{sat}$ is the saturation pressure of water. $\rho$ is the density of water and $V$ is the upstream water velocity. As a result, when cavitation number is less than 0, which means pressure is less than saturation pressure, water is likely to be vaporized and cavitation may occurs. Study results shown that cavitation number near the draft tube considerable suppressed and in case 6 the J-Groove shows the best effect.

Figure 5. Vortex core region in the draft tube.

Figure 6. Relative area of vortex surface $V_{vor}/V_{non}$.

Figure 7. Axial velocity distributions along the radius.

Figure 8. Tangential velocity along the radius.
4. Conclusions
Conclusions can be concluded by the above discussions as follows:
1. The efficiency curves present that the efficiency can be maintained by different kinds of J-Groove shapes.
2. The J-Groove can suppress the swirl flow both in axial and tangential components.
3. The J-Groove can reduce the possibility of cavitation occurrence in the draft tube.
4. The optimal J-Groove in case 6 which have a dimension of 280mm×69mm×14mm is obtained.

5. Reference
[1] Gorla R S and Khan A A 2003 Turbomachinery Design and Theory (New York: N. Y. Marcel Dekker Inc.)
[2] Jacob T and Prenat J E 1996 Francis Turbine Surge: Discussion and Data Base Proceedings of the 18th IAHR Symposium (Valencia, Spain, 16-19 September 1996)
[3] Nishi M, 1984, Surging Characteristics of Conical and Elbow Type Draft Tubes, Proceedings of the 12th IAHR Symposium on Hydraulic Machinery and System (Stirling, U.K., August 1984)pp272–283.
[4] Prenat J-E and Jacob T, 1986, Investigating the Behavior at High Load of a Francis Turbine Model Proceedings of the 13th IAHR Symposium (Montreal, Canada, 2-5 September 1986)
[5] Susan-Resiga R F, Muntean S, Avellan F and Anton I 2011 Applied Mathematical Modeling 35 4759–73.
[6] Tridon S, Barre S, Ciocan G D and Tomasz L 2010 European Journal of Mechanics B/Fluids 29 321-35.
[7] Kumar P and Saini R P. 2010 Renewable and Sustainable Energy Reviews 14 374-83
[8] Choi H J, Zullah M A, Roh H W, Ha P S, Oh S Y and Lee Y H 2013 Renewable Energy 54 111-23
[9] Zhang H M and Zhang L X 2012 Procedia Engineering 31 156 –65
[10] ANSYS Inc., ANSYS CFX Documentation, ver. 12, http://www.ansys.com, 2010.
[11] Escaler X, Egusquiza E, Farhat M, Avellan F and Coussirat M 2006 Mechanical Systems and Signal Processing 20 983-1007