Effects of draft tube on the hydraulic performance of a Francis turbine

J H Jeon, S S Byeon and Y J Kim
Graduate School of Mechanical Engineering, Sungkyunkwan University
300 Cheoncheon-dong, Suwon 440-746, Korea
E-mail: yjkim@skku.edu

Abstract. The draft tube is an important component of a Francis turbine which influences the hydraulic performance. It is located just under the runner and allowed to decelerate the flow velocity exiting the runner, thereby converting the excess of kinetic energy into static pressure. In this study, we have numerically investigated the hydraulic performance of a Francis turbine on the 15MW hydropower generation with various design parameters (three types of draft tube, thickness of guide vane) through a three-dimensional numerical method with the SST turbulent model. The vortex rope characteristics of the draft tube were confirmed. The results of the vortex flow fields and flow characteristics were graphically depicted with different design parameters and operating conditions.

1. Introduction
Recently, hydropower is receiving attention because of its clean, renewable and abundant energy resources to develop. However, the suitable draft tube of a Francis turbine is not determined in the hydropower efficiency and it is necessary to study for the effective turbine. A draft tube is one important part of a hydro turbine, which is used to transform water into energy. Without a draft tube, the pressure could drop because of lack of water, and in turn, the entire turbine could fail to work and power could be lost. A draft tube can also be either straight or curved, depending on the general construction of the turbine. The extensive experimental investigation of the draft tube flow has been complemented with three-dimensional numerical flow simulation aimed at elucidating the swirling flow evolution up to the turbine outlet as well as the phenomena that led to the peculiar sudden drop in the turbine efficiency [1]. Choi et al. [2, 3] performed the possibility of suppressing the draft tube surge in the draft tube of a Francis turbine by using J-groove without decrease of the turbine efficiency. They showed that the change of draft tube significantly influenced on the performance of the hydraulic efficiency. Nishi et al. [4] investigated the water swirling flow in a 9.5° conical diffuser. They showed that the dimensionless peak-to-peak pressure fluctuation and the corresponding dimensionless fundamental frequency are constant at high cavitation parameter values, but decrease monotonically as the vortex cavitation develops. Ciocan et al. [5] presented a CFD (Computational Fluid Dynamics) methodology to study the unsteady rotating vortex flow in the draft tube of a Francis turbine at part load conditions. They performed unsteady Reynolds-Averaged Navier-Stokes (RANS) simulation for the flow and validated the same with experimental results. Qian et al. [6] studied the three-dimensional multiphase flow field in a Francis hydraulic turbine. They especially investigated the pressure
pulsation in the draft tube, the front runner, the guide vane and the spiral case via the FFT (Fast Fourier Transform) analysis.

In this study, the hydraulic performance of a Francis turbine is investigated with various design parameters (thickness of guide vane, configuration of draft tube) using the commercial code, ANSYS CFX ver. 13.0 [7]. Also, we confirmed the vortex rope characteristics of the draft tube and hydraulic efficiency of Francis turbine.

2. Numerical analysis

2.1. Numerical model
The Francis hydraulic turbine applied in this study consists of a spiral casing, runner blade, guide vane and draft tube, which is shown in Fig. 1. The modeled Francis turbine was generated through the 2-D plan of the Sunjin hydroelectric power plant in Korea. The details of guide vanes are shown in Fig. 2. The modeled Francis turbine is described in the following: the diameter of runner \( D = 1.4 \text{m} \), the head of Francis turbine \( H = 155 \text{m} \), and the rotational speed \( n = 514 \text{ rpm} \). It has a runner with 17 blades, a guide vane with 20 blades, and a stay vane with 9 blades. The shape of draft tube is an important factor to keep the stable flow condition as well as to suppress the occurrence of draft surge at the region. Therefore, many researchers have been tried to design optimum shape of the draft tube. Figure 3 shows the dimensions of different draft tube in the Francis turbine. Also, Table 1 shows the cases of CFD analysis with different values of the guide vane thickness.

Figure 1. 3-D modelling of the Francis turbine applied in this study.

Figure 2. Dimensions of a guide vane.

Figure 3. Various draft tube models in the Francis turbine.
Table 1. Cases of CFD analysis with different values of the guide vane thickness (refer to Fig. 2).

| Guide vane | Thickness (mm) |
|------------|----------------|
| Type A     | 15             |
| Type B     | 14             |
| Type C     | 13             |
| Type D     | 12             |

Table 2. Numerical methods and boundary conditions.

| Numerical methods | Mesh type                | Tetrahedral & hexahedral |
|-------------------|--------------------------|--------------------------|
| Mesh number       | 1,300,000                |                          |
| Turbulence model  | SST                      |                          |
| Calculation type  | Steady state             |                          |
| Boundary conditions |                       |                          |
| Rotor stator interface | Frozen rotor             |                          |
| Inlet of turbine  | 10.97 [m³/s]             |                          |
| Outlet of draft tube | 101.325 [kPa]           |                          |
| Wall              | No-slip wall             |                          |

2.2. Grid systems
In this study, three-dimensional discretization has been used with a finite volume method (FVM). Due to its flexibility when solving complex geometries, unstructured 3-D tetrahedral and hexahedron meshing systems have been employed for the computational domain. Table 2 shows numerical methods and boundary conditions applied in this study. As shown in Fig. 4, numerical grids of about 1,300,000 are adopted for the analysis of the calculation domain including the spiral casing, the guide vane, the stay vane and the draft tube.

Fine hexahedral numerical grids are employed for the draft tube of the turbine to ensure the high accuracy of the calculated results. Moreover, in order to investigating the rotating effect of runner blades, we applied the MRF (Multiple Frame Model) method that is a steady state approximation where the fluid zone is modeled in a rotating frame of reference and the surrounding zones are modeled in a stationary frame. The prism-layer methods are applied to the draft tube and guide vane in order to improve the convergence of numerical calculation (see Fig. 4(b)). The SST model is adopted as turbulence model because of its relatively good convergence in the complicated flow field of turbomachinery in comparison with the other models [8]. In order to obtain the accuracy of numerical results, the relative tolerance of calculations on the analysis model in this study was determined as $10^{-4}$.

![Figure 4](image)  
(a) Runner blade  
(b) Draft tube  
(c) Spiral casing  
(d) Guide vane  

**Figure 4.** Grid systems of the modeled Francis turbine.
2.3. Governing equations

The governing equations for conservation of mass and momentum can be written as follows:

i) Continuity:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \]  

(1)

ii) Momentum:

\[ \frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \cdot \mathbf{U}) = -\nabla p + \nabla \cdot \mathbf{\tau} \]

\[ \mathbf{\tau} = \mu \left( \nabla \mathbf{U} + (\nabla \mathbf{U})^\top - \frac{2}{3} \nabla \cdot \mathbf{U} \right) \]  

(3)

where \( \mathbf{U}, \rho, p \) and \( \nabla \cdot (\mathbf{U} \cdot \mathbf{\tau}) \) are denoted as the velocity, density, pressure and viscous force, respectively. To consider the turbulent flow in the analysis model, a shear stress transport (SST) turbulence model was employed. The turbulent velocity scale is computed from the turbulent kinetic energy \( (k) \). The turbulent length scale is estimated from the two properties of the turbulence field, usually the turbulence kinetic energy and dissipation rate \( (\varepsilon) \). The turbulence transport term, \( \rho u_i u_j \), can be written as [9]:

\[ \rho u_i u_j = -\mu \left[ \frac{2}{3} \delta_{ij} \frac{\partial U_i}{\partial x_j} + \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] \]

(4)

where \( \mathbf{U} \) is the velocity vector, \( \rho \) is the fluid density, and \( \mu \) is the turbulent viscosity, which is denoted as:

\[ \mu = \rho C_\mu \sqrt{\frac{k}{\varepsilon}} \]

(5)

The velocity and length scales are written as:

\[ V_t = \sqrt{k} \], \( l_t = \frac{k^{\frac{3}{2}}}{\varepsilon} C_\mu \]

(6)

The turbulence kinetic energy \( (k) \) and the dissipation rate \( (\varepsilon) \) are written from the transport equations:

\[ \frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + P_k - \rho \varepsilon \]

(7)

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \varepsilon) = \frac{\partial}{\partial x_j} \left( \Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} \left( C_{\varepsilon} P_k - \rho C_{\varepsilon} \varepsilon \right) \]

(8)

where the diffusion coefficients are given by

\[ \Gamma_k = \mu + \frac{\mu_k}{\sigma_k}, \quad \Gamma_\varepsilon = \mu + \frac{\mu_\varepsilon}{\sigma_\varepsilon} \]

(9)

The diffusion rate of the turbulence kinetic energy \( P_k \) is given by

\[ P_k = -\rho u_i u_j \frac{\partial U_j}{\partial x_i} \]

(10)

where the \( k - \varepsilon \) turbulence model constant \( C_\mu, C_{\varepsilon 1}, C_{\varepsilon 2} \) and \( \sigma_\varepsilon \) are 0.09, 1.44, 1.92 and 1.3, respectively. The turbulence model constant for the \( k \) equation \( \sigma_k \) is 1. These equations are solved simultaneously by using a commercial CFD code, ANSYS CFX.
3. Results and discussion
In order to confirm the vortex rope characteristics and hydraulic performance of the Francis turbine, the effects of various parameters (three different types of draft tube and four different values of the thickness of guide vane) were investigated.

Figure 6. Efficiency and head of the Francis turbine for different values of the guide vane thickness.

Figure 6 shows the hydraulic efficiency and head of the modeled Francis turbine with different values of the guide vane thickness. From the viewpoint of guide vane, it is noted that the efficiency and mass flow are increased as the guide vane thickness decreases due to its least drag force.

Figure 7. Pressure distributions at the cross section of the draft tube.

Figure 7 shows the static pressure distribution at the cross section of draft tube. Results showed that the Case B has relatively uniform distribution of pressure field compared with other cases. In hence, the stability of flow is enhanced, which may attribute to the suppression of draft surge.

A vortex core region in the draft tube also shows that there is high possibility of suppressing draft surge to some extent. The draft surge phenomenon is a result of unstable fluid flow. Main flow is pushed to the region nearby wall surface on the draft tube passage. In the Fig. 8, a level of swirling strength is applied for the observation of vortex core region. Figure 8(a) shows that in the one gorge draft tube, strong whirl flow exists, which may cause draft tube surge. Results also showed that the Case B (having two gorge draft tube) works good condition with small rotating flow in the draft tube. When fluid passes through the guide vane passage, its circumferential velocity becomes high. A strong flow at the center region of draft tube performs a role of decreasing circumferential vortex core. The swirl flow exists far away from the center of draft tube, and the depth of vortex core region is shallow.
4. Conclusions

In this study, the flow characteristics of a Francis turbine were simulated with three different guide vanes and draft tube profiles. In particular, the vortex core and pressure distributions in the considered hydraulic turbine were investigated. Based on the data from the CFD model, the following conclusions are obtained:

The guide vane of Type D showed the highest efficiency value among the others, although this model improves the efficiency 1.3% more than the worst model. For the case of two gorge draft tube (Case B), a vortex core region was diminished at the wall of draft tube.

For better understanding of this fascinating flow problem, however, it may be necessary to perform the experimental works. In the near future we would be glad to compare these numerical results with those obtained by anyone in the same field.

References

[1] Avellan F 2000, Flow investigation in a Francis draft tube: the FLINDT project Proc. of the 20th IAHR Symp. (North Carolina, USA, 6-9 August 2000)
[2] Wei Q, Zhu B and Choi Y D 2012 J. KSME 36(5) 618-26
[3] Kurokawa J, Imamura H and Choi Y D 2010 ASME J. Fluids Engng 132 071101-1-071101-8
[4] Nishi M, Matsunaga S, Okamoto M, Uno M and Nishitani K. 1988 Flows in Non-Rotating Turbomachinery Components (FED) 69 81-88
[5] Ciocan G D, Iliescu M S, Vu T C, Nennemann B and Avellan F 2007 ASME J. Fluids Engng. 129 146-58
[6] Qian Z D, Yang J D and Huai W X 2007 J. Hydrodynamics Ser. B 19(4) 467-72
[7] ANSYS Corporation CFX ver. 13 Manual, 2009.
[8] Menter F R 2012 J. AIAA 32(8) 1598-605
[9] Orieux S, Rossi C and Esteve D 2002 Rev. Sci. Instrum. 73(7) 2694-98