Journal of the Korean Society of Marine Engineering, Vol. 33, No. 7, pp. 1052~1059, 2009(ISSN 1226-9549)

A Study on the Performance Analysis of Francis Hydraulic Turbine

Jin-ho Ha[1] · Chul-Ho Kim†

(Received August 25, 2009 ; Revised October 21, 2009 ; Accepted November 10, 2009)

Abstract : The effects of varying the inlet flow angle on the output power of a Francis hydraulic turbine were studied numerically and the result was compared to the experimental results conducted at Korea Institute of Energy Research to determine the brake power of the turbine for each set of operating conditions. The loss of mechanical power of the model turbine was determined by comparing the numerical and experimental results, and thus the turbine efficiency or energy conversion efficiency of the model turbine could be estimated. From the result, it was found that the maximum brake efficiency of the turbine is approximately 46% at an induced angle of 35 degrees. The maximum indicated mechanical efficiency of the turbine is approximately 93% at an induced angle of 25~30 degrees.

Key words : Francis hydraulic turbine, Indicated mechanical efficiency, CFD, Brake power

1. Introduction

Francis hydraulic turbine is classified as an impulse-type turbine because it uses the static pressure and kinetic energy of flowing water to generate power. Impulse-type turbines are useful in situations in which the hydraulic head is low but the mass flow-rate is high[1]. This study is a preliminary step in the development of a design algorithm for Francis hydraulic turbines. In order to optimize the design of the turbine, a detailed understanding of the flow phenomena of a blade-to-blade path and in the volute of the turbine is very important. An experimental approach to these phenomena can only return very limited information about the control volume. Thus, numerical simulation is a very useful tool for obtaining detailed information about flow characteristics, which makes it possible to develop an optimum design algorithm for hydraulic turbines.

When the mass flow-rate of water at the inlet of the turbine changes, the induced angle should be adjusted so as to maintain a smooth flow of water in the flow path, otherwise turbulent flow is generated in the blade-to-blade path of the turbine and the flow separation arises on the vane surface. These complicated flow phenomena convert useful energy to entropy in the flow field. Therefore a variable inlet guide vane system is used to adjust the flow induced angle according to changes in the mass flow rate in the turbine system. In this study, numerical simulations were conducted to determine

† Corresponding Author(Seoul National University of Technology E-mail : profchkim@snut.ac.kr, Tel: 02-970-6347)

1 Seoul National University of Technology, Graduate School of New Energy Engineering

the effects of varying the induced angle on the output power performance of a model Francis hydraulic turbine. These results were compared with experimental results obtained at the Korean Institute of Energy Research (KIER)[2] in order to estimate the energy conversion efficiency, the mechanical efficiency, and the mechanical loss power of the model turbine designed at KIER. The optimum operating condition for the designed model turbine was also estimated.

2. Flow field characteristics and geometry of the model turbine

Francis turbine system comprises two main components: the inlet guide vane and the rotor with volute. The inlet guide vane directs the inlet water into the rotor at a velocity with a tangential component. The water inlet angle is varied by the guide vanes and should be adjusted to the operating conditions, i.e., the flow rate and head characteristics, to produce the optimum performance of the hydraulic turbine. As water flows into the turbine rotor in the radial direction, the flow soon changes its direction to axial and exits into a diffuser, which acts to convert the kinetic energy of the water into a useable form.

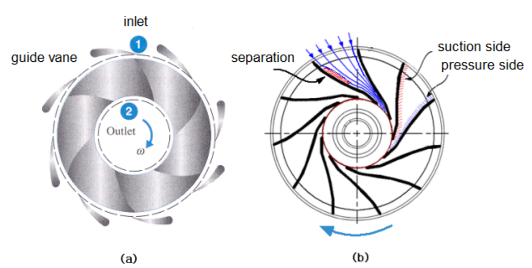
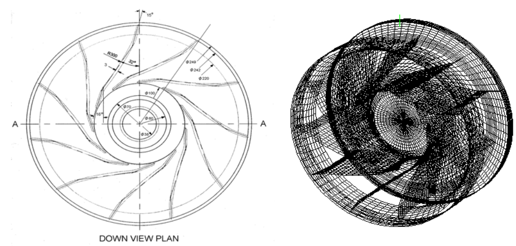


Figure 1: Schematic diagrams of a turbine with guide vanes and flow separation on the surface side of a runner vane

In this process, if the water does not smoothly drain into the diffuser because the flow in the flow path of the impeller is unstable, such as due to flow separation on the vane surface and casing wall (see Figure 1), the efficiency of the turbine is degraded and the life cycle of the system might be reduced by mechanical stress on the impeller. Figure 2 shows the physical and numerical domains of the model turbine that were used in this numerical and experimental study.



(physical domain) (numerical domain)

Figure 2: Physical and numerical domains of the model Francis hydraulic turbine

3. Numerical Methods and Boundary Conditions

In this study, the numerical simulation of the three-dimensional flow field was conducted using a FVM code named PHOENICS (ver. 3.1)[3]. The control volume of the model turbine is reasonably defined as;

- Quasi-3D flow - Turbulent flow
- Incompressible flow - Steady state flow

3-dimensional Navier-Stokes equations[4] were solved with the standard (k-e) turbulence model[5]. The process was assumed to be steady state and adiabatic, and thus the energy equation was not required in these numerical calculations. The turbulent no-slip condition near the solid boundary was modelled with the logarithmic law. Fully implicit backward time differencing was used and the advection terms were hybrid differenced. Conjugate gradient techniques for pressure corrections in the transport equations were incorporated and the 'SIMPLE' algorithm[5] was employed for the velocity and pressure coupling.

3.1 Governing Equations

The basic equations describing the fluid dynamics in the control volume are based on the Navier-Stokes equations, which are comprised of equations for the conservation of mass and momentum.

1) Continuity equation

$$\frac{\partial U_i}{\partial u_i}+\frac{\partial U_j}{\partial y_i}+\frac{\partial U_k}{\partial z_i}=0 \qquad (1)$$

2) Momentum equation

$$\frac{\partial U_i}{\partial t}+\frac{\partial}{\partial x_i}(U_iU_j)=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+ \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\overline{u_iu_j}\right]-g_i \qquad (2)$$

3) Turbulent kinetic equation

$$\frac{\partial}{\partial x_i}(U_jk)=\frac{\partial}{\partial x_i}\left[\left(v+\frac{v_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G-\varepsilon \qquad (3)$$

4) Energy dissipation equation

$$\frac{\partial}{\partial x_i}(U_i\varepsilon)=\frac{\partial}{\partial x_i}\left[\left(v+\frac{v_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\frac{\varepsilon}{k}(C_{\varepsilon1}G-C_{\varepsilon2}\varepsilon) \qquad (4)$$

Where

$$G=-\overline{u_iu_j}\frac{\partial U_i}{\partial x_i}$$

$$-\overline{u_iu_j}=v_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}k\delta_{ij}\ , \qquad v_t=C_\mu\frac{k^2}{\varepsilon}$$

($C_\mu=0.09$, $C_{\varepsilon1}=1.44$, $C_{\varepsilon2}=1.92$, $\sigma_k=1.0$, $\sigma_\epsilon=1.0$)

3.2 The boundary and initial conditions

In this numerical study, Body Fitted Coordinates (BFC) grid generation method[6] was used in conjunction with non-orthogonal grids allowing irregular geometries to generate the numerical grid of the model turbine and the optimized grid size of the 3-D model was decided to 52x64x12 through the grid test. Figure 3 shows a perspective view of the three-dimensional numerical domain for the blade-to-blade path of the rotor that was used in this numerical study.

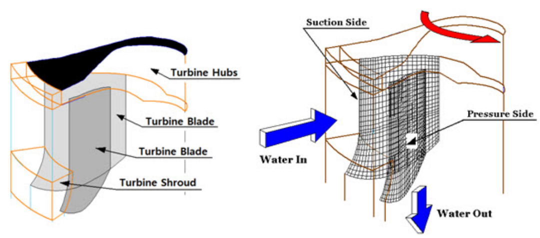


Figure 3: Perspective view of the 3-D numerical domain for the blade-to-blade path of the rotor (52x64x12)

The boundary and initial conditions of the calculations were as follows:

(1) Inlet : velocity boundary condition
(2) Outlet : Pressure boundary condition with an assumption of fully developed flow field
(3) No-Slip boundary condition on surface of the model impeller
(4) Symmetric boundary conditions on the surface of the control volume

3.3 Major Parameters and Their Ranges

In general the output power of the hydraulic turbine is controlled by the mass flow-rate of water and its head in a volute. The induced angle at the inlet of the turbine is a critical design parameter of turbine systems. In this study, the induced angle was controlled in six steps with inlet guide vanes from fully closed position to a position opened up partly to an angle of 45 degrees to determine the effects on the power performance of the turbine system.

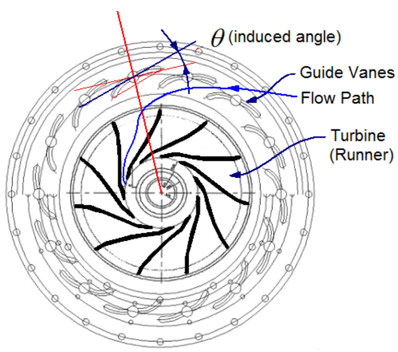


Figure 4: Top view of the model turbine and its induced angle

Figure 4 shows a top view of the model turbine assembly. If the induced angle of the guide vanes is low, the mass flow-rate is reduced because the inlet area is reduced and the rotational speed of the turbine is also affected.

Table 1 shows the variation of the experimental results such as the variation of the mass flow-rate and the rotational speed of the turbine, with the induced angle. These results were incorporated as boundary conditions in the numerical study.

Table 1: Variations of the experimental results with induced angle

Model No.	Induced Angle (degree)	Rotating Speed (rpm)	Head (mAq)	Flow rate (㎥/s)	Torque (N·m)
Model-25	11.3	617.5	6.5	0.052	13.58
Model-40	18.0	637.0	6.5	0.065	22.85
Model-55	24.8	655.0	5.5	0.089	30.46
Model-70	31.5	669.9	5.0	0.107	34.15
Model-85	40.5	706.3	5.0	0.126	38.91
Model-100	45.0	716.1	5.0	0.140	38.89

4. Performance analysis of the model turbine

The indicated power generated by the model turbine can be estimated from the results of the simulation. The static pressure distribution on the surface of the model impeller is the energy source for the torque generated on the turbine shaft.

The static pressure force can be calculated from the equations given below.[7]

① Hydrostatic pressure force on the impeller vane

$$F = \int_A P(x,y)dA \qquad (5)$$

② Indicated torque (τ_i) generated on the turbine shaft

$$\tau_i = \Delta F \times l_{cp} \qquad (6)$$

Where $\Delta F = F_{pressure} - F_{suction}$

③ Indicated power (P_i) produced on the model turbine

$$P_i = \tau_i \times \omega \qquad (7)$$

where ω is the rotational speed of the turbine.

④ Turbine efficiency

$$n_{t_i} = \frac{P_i}{P_{in}} \times 100(\%), \qquad n_{t_b} = \frac{P_b}{P_{in}} \times 100(\%) \tag{8}$$

where η_{t_i} is the turbine indicated efficiency.

η_{t_b} is the turbine brake efficiency

⑤ Mechanical efficiency of the model turbine

$$n_m = \frac{P_b}{P_i} \times 100(\%) \tag{9}$$

where P_b is the experimental brake power

⑥ Mechanical power loss of the model turbine

$$P_{loss} = P_i - P_b \tag{10}$$

5. Results and Discussion

The indicated power of the model Francis turbine was estimated numerically and compared to the brake power acquisitioned from the experiment by KIER to have an understanding of the general performance of the model Francis hydraulic turbine such as the indicated and brake efficiency, the mechanical efficiency and the mechanical power loss of the designed model turbine. The mechanical efficiency of the turbine was calculated by comparing the input power to the brake and indicated output power of the turbine. The efficiency of the turbine is very important performance parameter for estimating the energy conversion efficiency of the designed system. The mechanical power loss of the model turbine was determined by comparing to the indicated power to the experimental brake power.

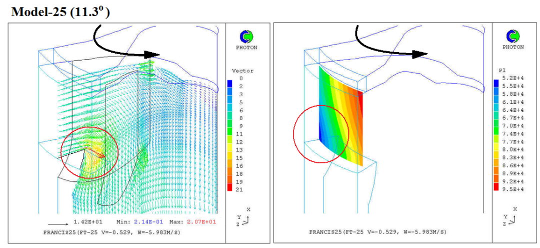


Figure 5: Velocity and pressure distributions at the entrance of the impeller; induced angle = 11.3 degree

Figure 5 shows the velocity and pressure distributions at the entrance of the impeller at 11.3 degrees of the induced angle of the guide vane. In the case of the velocity distribution, water accelerates at the lower side of the entrance because the flow direction of the fluid turns downwards abruptly by 90 degrees at the entrance. This flow phenomenon could be a critical reason of the cavitation in the flow path of the turbine impeller. The static pressure on the lower left comer of the entrance is the lowest in the area and as the value reaches to below the vapor pressure definitely forms air-bubbles in the region.

The static pressure on the pressure side of the vane is much higher than on the suction side. The pressure difference between two sides of the vane is the energy source of the torque rotating the turbine impeller.

Figure 6 shows the static pressure distributions on the pressure and suction sides of the vane. The pressure on the pressure side of the vane is higher than on the suction side. This information is very important for the estimation of the indicated power of the designed impeller.

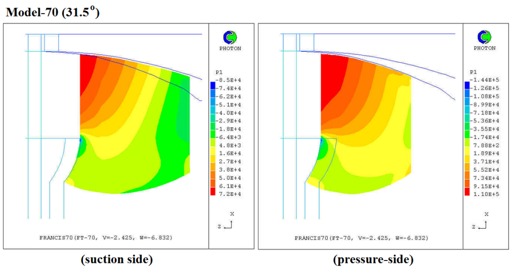


Figure 6: Distributions of the hydrostatic pressure on the pressure and suction surfaces of a vane of the model turbine at an induced angle of 31.5 degree

Figure 7 shows the variations of the power input and output of the model turbine. The rotational speed of the turbine is directly related to the mass flow-rate of the inlet water and the induced angle. As shown in the figure, all the powers are continuously increasing along with the impeller speed and the induced angle. In particular, the difference between the output brake and indicated powers is minimized in the middle speed range, which means that the mechanical energy loss of the impeller is minimized in this range.

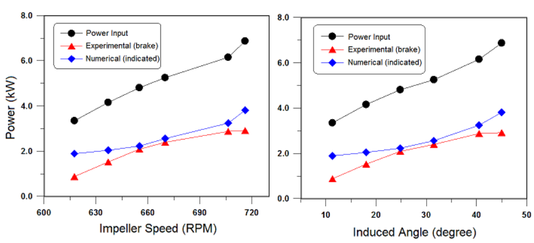


Figure 7: Variations of the power input and output with the rotational speed and induced angle

Figure 8 shows the variation in the mechanical efficiency of the turbine with the induced angle. As shown in this figure, the mechanical efficiency of the turbine is maximized in the middle range of the induced angle. In general, it is known that the mechanical efficiency of Francis hydraulic turbine is approximately 85~90%. It is estimated that the optimum conditions of operation of this model turbine arise for an induced angle of 25~30 degrees, resulting in an optimum efficiency of approx. 93%. The mean value of the mechanical efficiency of this turbine model is approx. 79% at the rated operating conditions

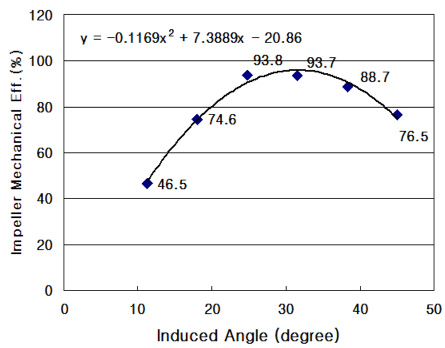


Figure 8: Variation of the mechanical efficiency of the model turbine with the induced angle

The brake and indicated efficiencies of the model turbine are shown in Figure 9.

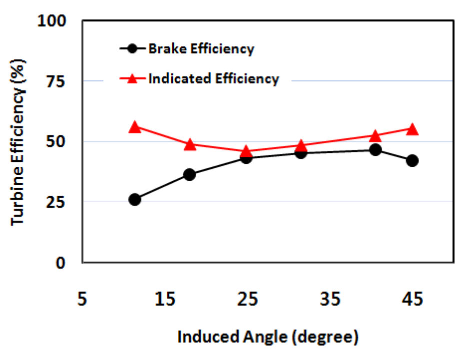


Figure 9: Variations of the brake and indicated efficiencies with the induced angle

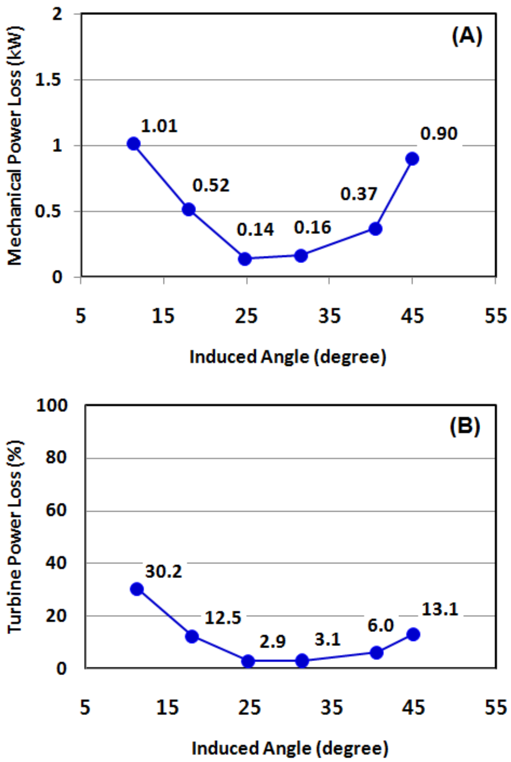


Figure 10: Variations of the mechanical power loss(A) and its loss rate with induced angle(B)

The indicated efficiency is minimized in the middle range of the induced angle; however, the opposite trend is found for the brake efficiency. The indicated efficiency is strongly affected by the flow phenomena in the flow path of the turbine, which means that the flow is quite stable at lower and higher indicated angles. The brake efficiency is related to the mechanical loss of the turbine. As shown in Figure 9, mechanical loss is minimized in the middle range of the induced angle.

Figure 10 shows the variation of the mechanical power loss and its rate of the model turbine with the induced angle. The power loss is minimized in the middle range of the induced angle, which means that the optimum operating range of the model turbine arises for an induced angle of 25~35degrees. The lowest loss rate of the turbine is approximately 2.9% in this rated range.

6. Conclusion

In this study, numerical simulations were conducted to determine the effects of varying the induced angle on the output power performance of a model Francis hydraulic turbine designed at KIER and the results were compared with experimental results in order to estimate the energy conversion efficiency, the mechanical efficiency, and the mechanical power loss of the model turbine. The optimum operating conditions for the designed model turbine were also estimated.

From the study, it was found that induced angle at the inlet of a Francis hydraulic turbine controlled by an inlet guide vane system significantly affects the output power generated by the hydraulic turbine; that is; the induced angle is a very important parameter for the design of hydraulic turbines.

In the case of this model Francis hydraulic turbine, the induced angle should be adjusted to 25 to 30 degrees at a given hydraulic head (5.5~6.5m), mass flow-rate (0.065~0.089m^3/s)and rotational speed (655~670rpm) to obtain its optimum performance.

References

[1] W.W.Peng, Fundamentals of Turbo -Machinery, John Wileys & Sons Inc.,

pp. 275-293, 2008.

[2] C.H.Lee and W.S.Park, Technaical Report, Korea Institute of Energy Research, Vol.1, (2004).

[3] PHOENICS PIL Manual, Version 3.1, CHAM Ltd., (2002).

[4] Y.A. Cengel and J.M.Cimbala, Fluid Mechnaics (Fundamentals and Applications), McGraw-Hill International, 1st edition, pp. 472-476, 2006.

[5] S. V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp., 1980.

[6] R. L. Thompson, Body Fitted Coordinate, John Wiley & Sons, Inc., 1991.

[7] M. Potter and D. Wiggert, Mechanics of Fluids, Brooks/Cole, 3rd edition, 2002, pp. 1-60, 2002.

Author Profile

Chul-Ho Kim

He received his B.E., M.E. degree from Inha University (Korea) in 1980 and 1982 and Ph.D degree from The Univ. of New South Wales, Australia in 1995. He is currently a professor at the department of automotive engineering in Seoul National University of Technology.
His research interests are Power Train Design of an Electric Vehicle, Turbo-machine Design and Performance Analysis, Automotive Aerodynamics and CFD Applications

Jin-Ho Ha

He received his B.E. degree from Seoul National University of Technology in 1998 and M.S. degree from Korea University in 2002. He is currently a Ph.D candidate at the Graduate School of New Energy Engineering in Seoul National University of Technology.
His research topic for his Ph.D degree is "A Study on the Optimum Design of CPT(Cross- flow Power Turbine) system for the Electric Power Generation on a Running Vehicle."