Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and
8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-31273

DEVELOPMENT OF A HIGH PERFORMANCE FRANCIS TURBINE FOR RUNNER REPLACEMENT USING A CFD-BASED DESIGN SYSTEM

Hyung-Woon Roh
IVAIGD3 Ltd.
Seoul, Republic of Korea
rohlee@ivai.co.kr
82-2-2093-2121

Sang-Ho Suh
Soongsil Univ.
Seoul, Republic of Korea
suhsh@ssu.ac.kr
82-2-820-0658

Jong-hyun Jung
K-water
Republic of Korea
Jungjong1001@hanmail.net
82-42-629-2776

SuegYoung Oh
K-water
Republic of Korea
ohs@kwater.or.kr
82-42-629-2776

ABSTRACT

Recently, Korean Government is encouraging to propagate the small hydro-electric power in renewable energy development policy as the「Low CO2 Green Growth」policy. However, turbines in the Korea Water Resources Corporation (below K-water) operated at an average of 10% less than well-designed turbines due to the design weakness or the inferior manufacturing techniques for the small hydropower facilities. Thus, maintenance fees increased because the cavitations had excessively occurred on the main parts of turbines such as the runner, guide vane and Stator, the life cycles of turbines were reduced and the frequency breakdowns were increased.

In order to improve the efficiency the CFD-based design system is applied to the Francis turbine replacement project with Korea Fluid Machinery Association (below KFMA) and K-water. Therefore, the inversed design technique and the fully turbulent 3-dimensional flow simulations are performed for both the existing and new turbines at design and off design conditions. As a result, the runner is optimized to the greatest extent with a possible minimum cost under the geometrical constraints of the existing machine. The performances of the new design are verified by extensive model tests and the guarantees have all been successfully met.

INTRODUCTION

The Korean government is pushing ahead with a renewable energy development as a [Low CO2 Green Growth] policy. Although they plan to raise a share of the alternative energy up to 5% until 2011 by encouraging hydropower, the average efficiency of the turbines for hydropower plants which is operating in Korea is less than 5% to 10% compared with the well designed turbines. In small-scale hydro-electric power plants Francis turbine is used preferentially. The Francis turbine is a type of reaction turbine, which means that during energy transfer in the runner. This is achieved in the stationary, but adjustable guide vanes, called wicked vanes. Due to the design weakness of turbine efficiency reduces and cavitations may also occur not only in draft tubes and but also on the runner. The occurrence of cavitations cause noise, erosion of metal surfaces, thus reducing power and lowering efficiency.

Therefore, in order to improve turbine performances and avoid cavitations a proper design is needed. In this study a high performance Francis turbine is designed and developed for runner replacement using a CFD-based design system.

TURBINE MODEL FOR REPLACEMENT

Heongsung hydro-power plant is the conduit type power plant located in Gangwon province shown in Fig. 1. Heongsung dam is a multipurpose dam with a scheduled flood with regular high water levels of 180 m, low water level of 160 m, an available area of 209 ㎢, and available capacity of reservoir water of 73.4 million tons.

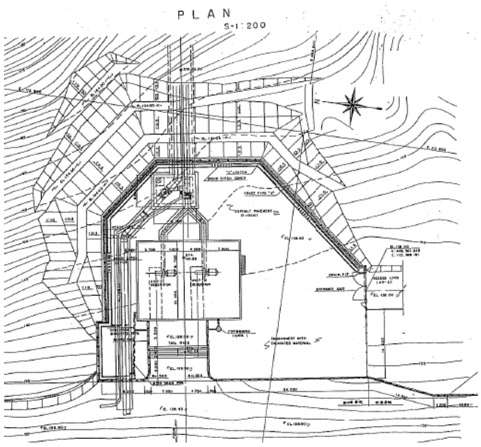
Fig. 1 Bird-eyed view of the small-scale hydroturbine

Heongsung hydro-power plant, located downstream from the intake tower, can provide maintenance and agricultural water through constant generation of electricity, ensuring the production of 1,000kW of hydro-power, even during a period of water shortage(Table 1). The plant consists of a Y-branched penstock with 800 mm in diameter, and from the penstock with 1,200 mm in diameter in a conduit pipe line that is 4.5 m in diameter and 382 m in length.

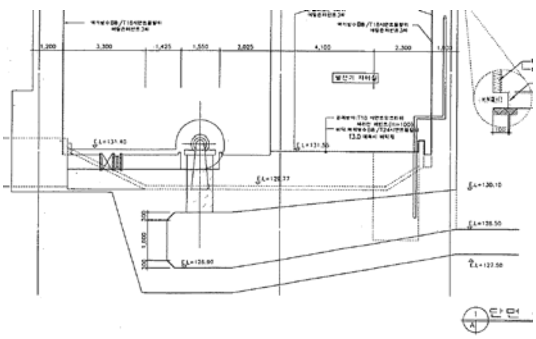
The Heongsung hydro-electric power turbine can operate between 60% and 76% of optimum efficiency, as shown in Fig. 3, which is much less than the designed efficiency. Although most of the of the general turbine efficiency depends on specific velocity, according to the EPRI report, the efficiency of the turbine is 92% at the specific number of Ns=180. The reason can be noticed from Fig 4.

Table 1 Specifications of the small-scale hydroturbine

Specification	Value	Remark
Angular velocity	900rpm	
Rated head	43m	
Flow rate	1.37CMS	
Target eff. of turbine	92.0%	
Target eff. of generator	95.0%	
Power	500kW	



(a) Plan drawing



(b) Section drawing

Fig. 2 Drawings of the hydro-electric power plant

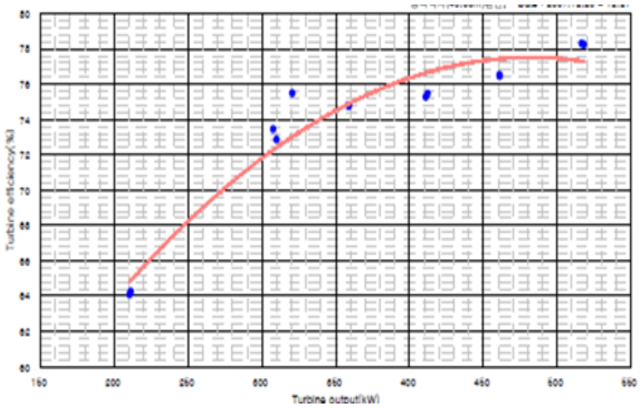


Fig. 3 Measured turbine efficiency as a function of power output

Fig. 4 Runner of the small-scale hydroturbine

Because the shape of the runner is a semi-circle, all kinds of head energy cannot be transferred to the turning force and the shape of the manufactured runner vane is not constant. Cavitation occurs so excessively at the end of the runner to which is accompanied by vibration and noise.

The effect of cavitations is erosion which occurs on the surface of the end of the runner vanes. Besides vibration and erosion the main effect on turbines is the possibility of performance failure. Therefore, improper design and manufacturing of the runner lead to cavitations and vibrations during the operation resulting in lowered efficiency. Applying the renewal runner's shape and process of manufacturing using new technology can improve the efficiency.

REVERSE DESIGN

3D scanning

Francis turbines including the runner, spiral casing, wicket gates, stay vane and draft tube should be designed hydraulically. Because the purpose of this study is to increase the efficiency by changing only the runner of the model turbine which has low efficiency, there are a lot of difficulties and constraints. As the dimensions indicated in blue color cannot be changed to design in accordance with the existing spiral casing, only the dashed lines indicated in red color should be changed to produce a highly efficient runner like Fig. 5. Because the only changeable part in the shape of the runner is a path line from the vane as shown in Fig. 5, the shape of the vane is referenced from the high efficient runner of 500kW such as Fig. 6 measured in three dimensions. Heongsung hydro-power utilizes a highly efficient 500kW runner that has operated at a 90% efficiency rating since the dam was constructed in 1990.

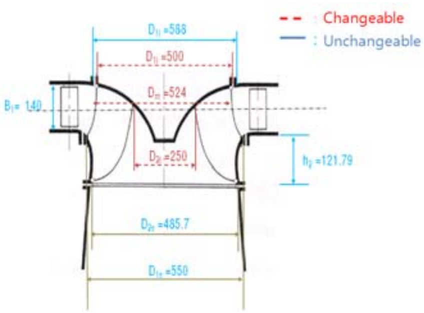


Fig. 5 Schematic diagram of a new designed runner

(a) Reference runner (b) 3D scanning

(c) 3D scanned runner

Fig. 6 3D scanning working and the output

Reverse engineering

The algorithm for the reverse design is developed in this study as described in Fig. 7 to renew improper runner shape of Heongsung hydro-power in operation with low efficiency[2]. The design factors in the algorithm of Fig.7 are similar to that of Fig. 8. The shape of the vane can be calculated in three dimensions from Fig. 8

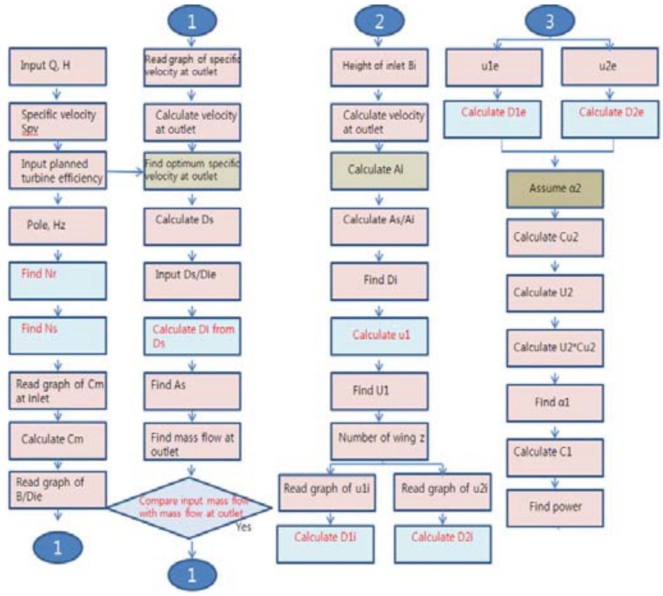


Fig. 7 Algorithm for runner design procedure

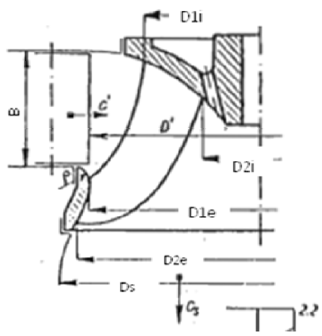


Fig. 8 Side view of the designed runner

NUMERICAL ANALYSIS

The Finite Volume Method (FVM) is used to simulate flow characteristics in the turbine in three dimensions. The governing equations for this simulation is expressed in equations (1) and (2).

ANSYS-CFX 11[3] is used as an analysis tool and the high resolution scheme is adopted for the discretization of the convective terms. Accuracy and safety is guaranteed by using the fully implicit scheme for discretization of unsteady terms. The solution is converged by limiting the residual of 10^{-5} for velocity and pressure, and thus the non-linear equation is calculated iteratively.

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

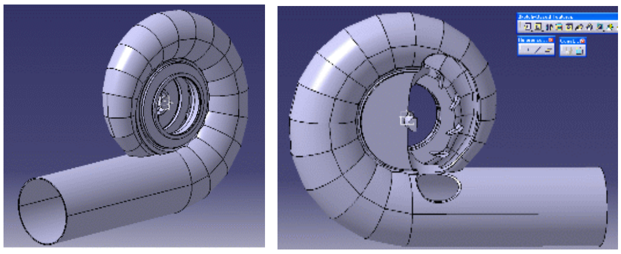
$$\rho\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_j} + \eta \frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{2}$$

In case of a turbulence model, applying the optimized turbulence model is generally needed to cover viscous sub-layer safely to acquire accurate numerical results. The K-ω(omega) SST(Shear Stress Transport) model, suggested by Menter in 1994, is used as the turbulence model to simulate accurately secondary flow by separation and influence in a wall boundary layer with a low Reynolds number.

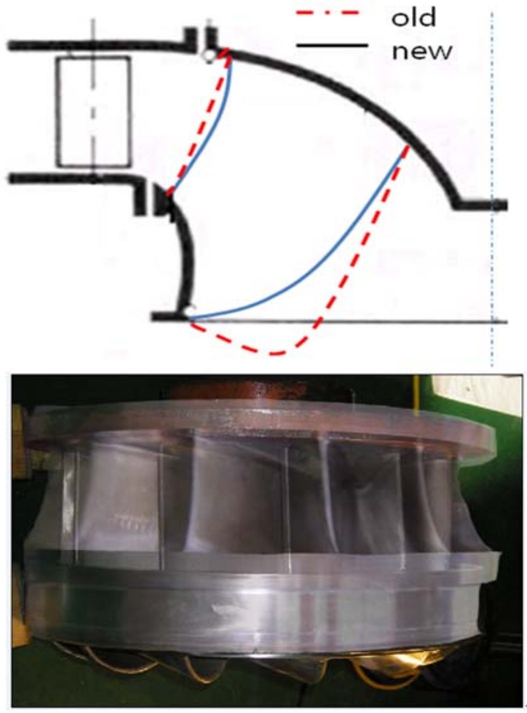
Three dimensional modeling for the runner is acieved such as Fig. 10 and Fig. 11 by using Fig. 7 and Fig. 9 for the numerical analysis. Semi-circular type runner is avoided and the number of runner blades increase from 11 to 13 for the new design to increase dynamic pressure within the runner as shown in Fig. 10.

The meridional section of the new runner is first optimized by the adopted reverse design technique and the CFD-based design system and it is then compared to the old runner in the Fig, 10. A Hybrid grid is also used for the new design as described in Fig. 11. The global number of nodes and elements are approximately 382,039 and 1,448,827 respectively for computational domain with a combination of tetra and prism grid.

Fig. 9 Developed Software for runner design



(a) Models



(b) Real shape of the runner

Fig. 10 Three dimensional modeling for the numerical analyses

RESULTS AND DISCUSSION

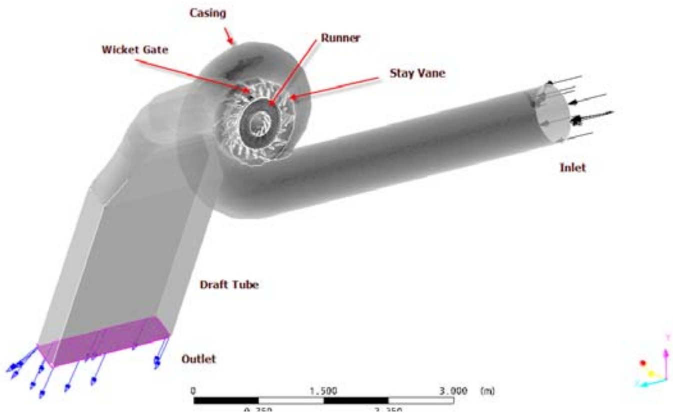
The CFD-based design system has been applied to the optimization of the Francis turbine runner. The full turbulent flow simulations have been conducted for both the existing and new runners at design and off design conditions. The numerical prediction including head loss calculation for the runner at off design conditions are very important to evaluate the turbine performances related to cavitation phenomena and pressure pulsations.

According to the numerical analysis by using the renewed design model of the runner, available flow rate of 1.37 CMS is satisfied near 30 degrees at the wicked gate as expressed in Fig. 12. As can be seen from this figure, when wicket gate opens about 30 degrees, the efficiency of the turbine is 85%. This result is more enhanced than the efficiency of the old model.

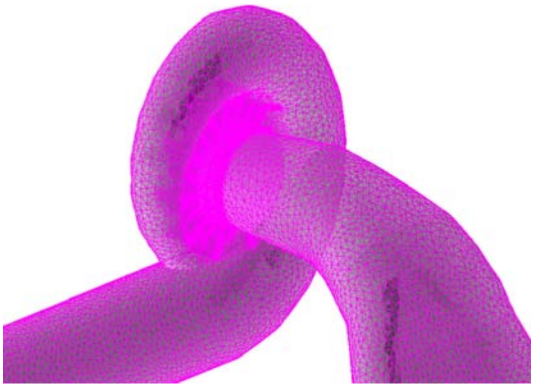
When a specific velocity is 180, the efficiency of the turbine should be around 92% according to the EPRI report[4]. However, this reduction of efficiency happens because the wicket gate is nearly closed in the H small hydraulic turbine plant. There is a huge difference of pressure that leads to losses between inlet and outlet at the wicket gate as shown in Fig. 12. Although the angle β_1 should be 90 degrees in the design stage, the angle might be less than the 90 degrees because of the outer diameter, D_{1i}, of the existing runner was an oversized. Consequently, these reasons cannot be exactly underestimated and be induced the losses.

Actually as you know, the CFD results are recently less than the efficiency of the turbine in the EPRI report. Thus, in order to enhance the calculated efficiency of newly designed runners in the study, the flow phenomena in the casing would be investigated using the CFD analyses. Another reason for the efficiency reduction would be obtain by a CFD analysis. From the discussions, it is realized that these reduced tendency is caused by the unbalanced distribution of pressure and the wrong casing shape as the shown the Fig, 13.

In the Fig. 14, the casing is modified which is expanded to 75mm towards outer regions at cut-off sections and is smoothly changed with less pressure loss. As a result, we would be expected to stabilize pressure distribution near the cut-off region and then obtain higher efficiency after the casing modification. Finally, the perfect combination of the CFD analysis has finally resulted in a new model which can provide about a 15% increase in peak efficiency

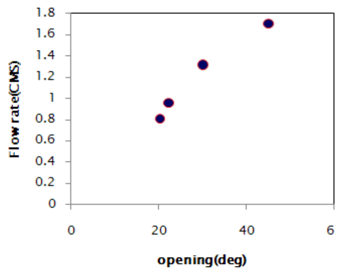


(a) Computational domain

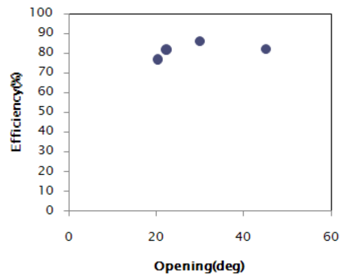


(b) 3D grid

Fig. 11 Computational domain and 3D grid

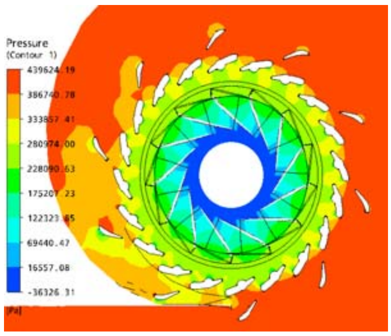


(a) Flow rate

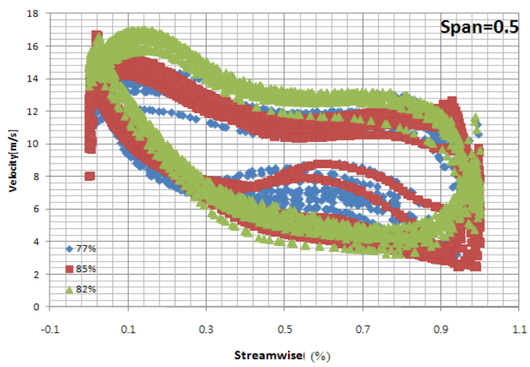


(b) Efficiency

Fig. 12 Performance curves as a function of wicket gate angle**.**

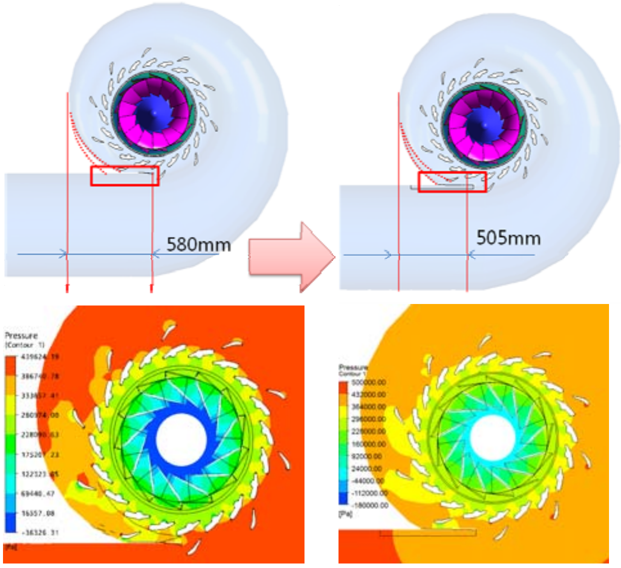


(a) Pressure contour



(b) Velocity variation

Fig. 13 Pressure contour plot and velocity variation of the new designed runner



(a) Old medel (b) New model

Fig. 14 Pressure contour plots for the old and new runner models

ACKNOWLEDGE

This work is the result of "A study for improvement of design techniques and performance through turbine CFD (1st year)" by the K-water research fund.

REFERENCES

(1) Miroslav Nechleba, 1957, Hydraulic Turbines: Their design and Equipment, Constable & Co Ltd.

(2) Daniel Balint, Sebastian Munean, Romeo Susan-Resina, Ioan Anton, 2004 A numerical investigation of the full 3D turbulent flow in Kaplan hydraulic turbines, The 6th International Conference on Hydraulic Machinery and Hydrodynamics Timisoara, Romania.

(3) ANSYS-CFX11 Manual, ANSYS

(4) EPRI Report TR-112350

