한국마린엔지니어링학회지 제36권 제5호, pp. 618∼626, 2012. 7(ISSN 2234-8352 Online) / (ISSN 2234-7925 Print)
Journal of the Korean Society of Marine Engineering http://dx.doi.org/10.5916/jkosme.2012.36.5.618

Internal Flow Characteristics in the Draft Tube of a Francis Turbine

Qingsheng Wei[1] · Baoshan Zhu[2] · Young-Do Choi†

(Received June 14, 2012; Revised July 9, 2012; Accepted July 9, 2012)

Abstract: Suppression of abnormal flow phenomena in the Francis hydro turbine is very important to improve the turbine performance. Especially, as cavitation and cavitation surge makes serious problems when the turbine is operated in the range of partial flow rate, optimum method of suppressing the abnormal flow characteristics is required necessarily. Moreover, as swirl flow in the draft tube of the Francis turbine decreases pressure at the inlet of the draft tube, suppression of the swirl flow can be an useful method of suppressing the occurrence of cavitation. In order to clarifying the possibility of suppressing the swirl flow by J-Groove in the draft tube, a series of CFD analysis has been conducted in the range of partial load, designed condition and excessive flow rate of a Francis turbine. A kind of J-Groove is designed and applied to the draft tube of the Francis hydro turbine model. The pressure contours, circumferential velocity vectors and vortex core regions in the draft tube are compared by the conditions with or without J-Groove. In addition, a group of data about the velocity in the draft is presented to show the influence of J-Groove.

Key words: Francis hydro turbine, Swirl flow, J-Groove, Draft tube, Performance

Nomenclatur

D_e	Inlet diameter of draft tube [m]
D_{outlet}	Outlet diameter of draft tube [m]
L_d	Length of draft tube [m]
D	Depth of J-Groove [m]
W	Width of J-Groove [m]
L	Length of J-Groove [m]
R	Radius of cross section plane [m]
r	Local radius of test location [m]
N	Rotating speed [min^{-1}]
$Ave.v_\theta$	Circumferential averaged velocity [m/s]
H	Effective head [m]
η	Efficiency
θ	Expansion angle of draft tube [deg.]
Q	Flow rate [m^3/s]

1. Introduction

Draft tube surge, one-dimensional pressure pulsations in the draft tube of a Francis Turbine, causes electric power fluctuation. It is resulted from a rotating vortex structure, which caused by the unstable swirl flow from the runner outlet into the draft tube[1]. Some studies have been carried out to suppress the pressure pulsations. But conventional methods are complex and difficult to realize in the case of a large size turbine, Therefore, a common and simple method has been required.

To avoid the detrimental effect of swirl flow, Kurokawa et al[2, 3]. have conducted a series of experiments and proposed shallow grooves, which is named as “J-Groove”, mounted on a casing wall of turbomachinery, to suppress the cavitation to some extents. On the other hand, recent results of

---

† Corresponding author (Department of Mechanical Engineering, Mokpo National University,
E-mail: ydchoi@mokpo.ac.kr, Tel: 061-450-2419)

[1] Graduated School, Mokpo National University, E-mail: holyrain@mokpo.ac.kr, Tel: 061-450-2419

[2] Department of Thermal Engineering, Tsinghua University, E-mail: bszhu@mail.tsinghua.edu.cn

experiments verified that J-Groove can increase the pressure in the low pressure region by carrying the high pressure fluid to the low pressure region[4]. Previous studies show that the pressure in the draft tube increase from inlet to outlet, which indicated that if the characteristics of J-Groove can applied to the draft tube, suppression of the surge can be achieved.

Susan-Resiga et al. [5] took a deep sight into the vortex in the simplified discharge cone of a Francis Turbine. It is shown that both RSM and RKE (turbulence models they used) lead to similar shape and size of the recirculation bubble, which is developed as a result of the flow deceleration along the axis. In addition, the central recirculation bubble developed as a result of the vortex breakdown, which contributes to the draft tube surge.

There are two main approaches. One is to act on the axial momentum of the flow, such as Falvey et al. [6] filling the stagnant of reverse flow region with a solid body of rotation, Thicke [7] using rather small conical or cylindrical extensions. The second approach is to reduce the swirl intensity in the cone. Nishi et al. [8] proposed fins to hinder the circumferential flow mainly in the neighborhood of the cone wall but it introduces additional losses and it cannot be adjusted with the operating point. The approach of this study belongs to the second approach based on the experiments of Kurokawa et al.

The purpose of this study is to examine 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. There are several factors to be concerned: vortex, circumferential velocity and efficiency et al. First, the performance characteristics of a Francis turbine model are studied. Then the effectiveness of J-Groove on the suppressing the swirl flow in the draft tube is investigated with the variation of guide vane opening.

2. Turbine model and J-Groove configuration

2.1 Turbine modele

Figure 1 shows the schematic view of the Francis turbine model which is used for CFD analysis in this study, whose runner outlet diameter D_e is 0.350m. The number of the blades is 17, and that of guide vanes is 16, whose opening varies from 6.9mm (10%) to 37.8mm (54.5%), while it fully opened to 69.4mm (100%).

The rotating speed of runner, effective head and flow rate at the design point are N=900min-1, H= 60.73m and Q= 0.46m3/s, respectively.

Moreover, **Table 1** shows cases of CFD analysis by the variation of guide vane angle.

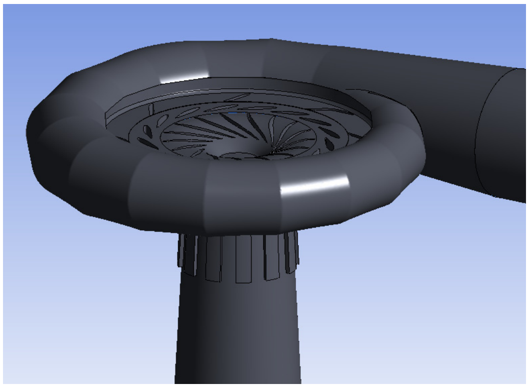


Figure 1: Schematic view of Francis turbine model

Table 1: Cases of CFD analysis by guide vane angle

Analysis case	Guide vane angle [%]	Mass flow rate [$\times 10^{-2} m^3/s$]
case 1	10.0	15.1
case 2	19.5	27.0
case 3	23.4	30.3
case 4	31.2	10.0
case 5	35.0	44.2
case 6	38.9	46.0
case 7	46.7	56.6
case 8	50.6	59.9
case 9	54.5	62.6

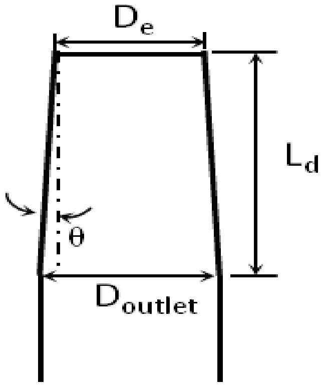


Figure 2: Dimensions of draft tube

2.2 Shape of draft tube

The shape of draft tube is a very 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. Kirschner e. al [9] has tried to control vortex with an axial jet in the draft tube. Maiwald e. al[10] has tried to use 13 segments instead of 8 segments in the design of elbow, and build the diffuser with 4 transition and 4 final diffuser segments while the base line is made of 3 transition segments and 1 final segment.

In this study, the shape of draft tube is determined by taking into account of combination with J-Groove. According to the following equation (1), the dimension of expansion angle of draft tube is determined.

$$L_d \geq 6(D_{outlet}-D_e) \qquad (1)$$

where D_{outlet} is diameter of cone base of the draft tube, and D_e is the diameter of the inlet of draft tube. And the L_d refers to the length of the cone. As shown in the **Figure 2**, the angle Θ should be less than 4.76 deg. to satisfy the equation (1). Moreover, previous study results point out that the ideal value of the core angle, which equals to 2Θ, is between 5deg. and 9deg. and thus, 3deg. is chosen to the value of Θ at last. From the cone base to the outlet of the draft tube, circular pipe with same diameter is attached.

2.3 Shape of J-Groove

Figure 3 shows the dimensions of J-Groove installed in a draft tube of Francis turbine. In order to suppress the surge caused by the swirl flow in the draft tube, a series of J-Groove are mounted on the wall of the inlet of draft tube with the size 280mm × 45.8mm × 14mm (length L × width W × depth D) while the diameter of inlet of draft tube D_e is 350mm.

Measuring points in the draft tube are fixed in the directions both by vertical location from the draft tube inlet and radial non-dimensional location from the center of draft tube at each vertical location. Three planes are setup to measure the related data in the draft tube. Y is the vertical axis to measure the locations of the planes from the inlet of the draft tube.

As shown in **Figure 4**, the vertical measuring locations of the planes (Layer 1, Layer 2 and Layer 3) are Y = - 0.100m, -0.175m and -0.250m, while the value of y axis of the inlet of draft tube is Y = - 0.075m.

Radial locations are determined by r/R, r is the local radial distance from the center axis to an arbitrary location on the planes, and R is the radius of each section plane cross the draft tube. Three kinds of non-dimensional radial locations are chosen by r/R = 0.58, 0.78 and 0.98.

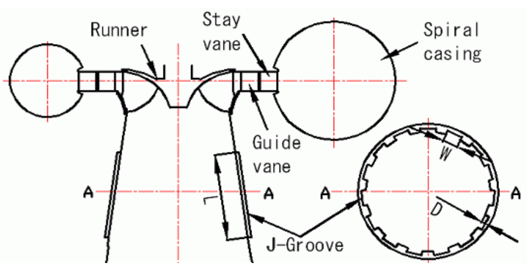


Figure 3: Dimensions of J-Groove installed in a draft tube of Francis turbine

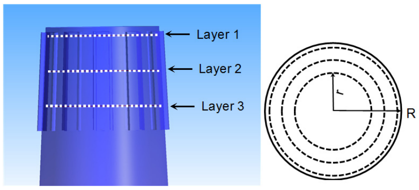


Figure 4: Measurement layers and local radius

3. Numerical Methods

This study employs a commercial code of ANSYS CFX ver.12.0 to conduct CFD analysis [11] and Table 2 shows numerical methods and boundary condition applied to this study.

Table 2: Numerical methods and boundary condition

Numerical methods	Mesh type	Tetra-hedral (turbine) & Hexa-hedral (draft tube)
	Mesh number	19.8×10^{6}
	y^{+}	below 15(runner) below 50(casing)
	Turbulence model	*SST*
	Calculation type	Steady state
Boundary condition	Rotor-stator interface	Frozen rotor
	Inlet of turbine	Constant pressure
	Outlet of draft tube	Averaged outflow
	Wall	No-slip

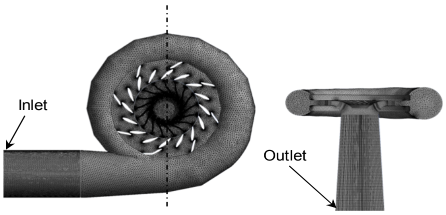


Figure 5: Numerical mesh of Francis turbine model

As shown in **Figure 5**, numerical grids of about 19.8×10^{6} are adopted for the analysis of the calculation domain including inlet pipe, casing, runner and draft tube. Relatively dense tetra-hedral grids are applied to the runner. Fine hexahedral numerical grids are employed for the draft tube of the turbine to ensure the high accuracy of calculated results. The value of y+, which means non-dimensional distance from wall, is determined to below 15 for runner and below 50 for the other parts of the turbine in consideration of the shape of turbine passage. *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 [12]. Constant pressure at the inlet and averaged outflow at the outlet of the calculation domain are the used boundary conditions. All the calculations are conducted under the conditions of steady state.

4. Results and Discussion

4.1 Performance curves

The flow field is investigated when the swirl enters the draft tube both in the cases of no J-Groove installation and cases of J-Groove installation. **Figure 6** shows the performance curves of the Francis Turbine model, including efficiency curves and mass flow rate curve. The curve without mark is obtained from experiments without J-Groove [13]. Efficiency curve of no J-Groove cases by CFD analysis is marked with ▲, while the efficiency curve with J-Groove mounted on the wall of draft tube is marked with △. The experimental efficiency curve with a little decrement when the guide vane angle deviates from designed point (case 6) indicates the Francis turbine working in good condition. Moreover, two efficiency curves by experiment and CFD analysis in case of no J-Groove installation are almost coincident with small deviation in the ranges of the partial and excessive flow rates, which makes the computational results reliable. Moreover, when the

J-Groove is installed, little difference of efficiency is shown in comparison with the case of no J-Groove installation.

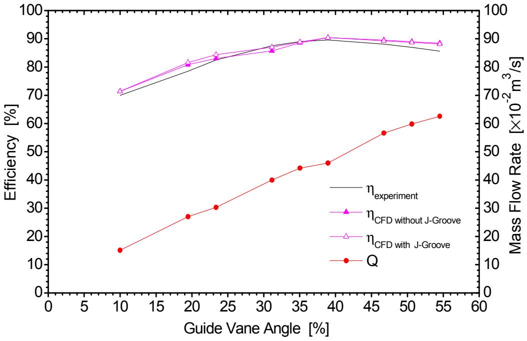
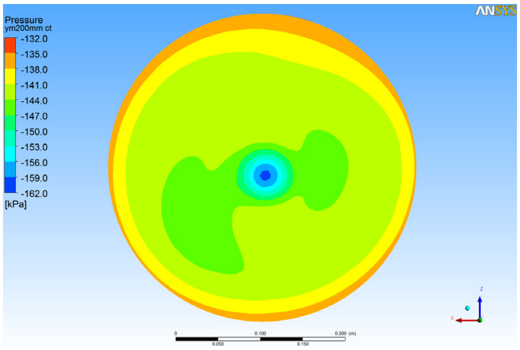
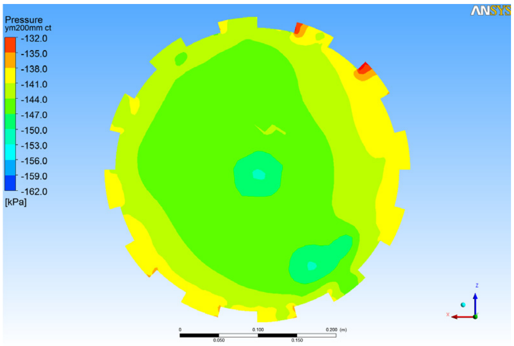


Figure 6: Performance curves of the turbine model



(a) Original draft tube



(b) J-Groove installation

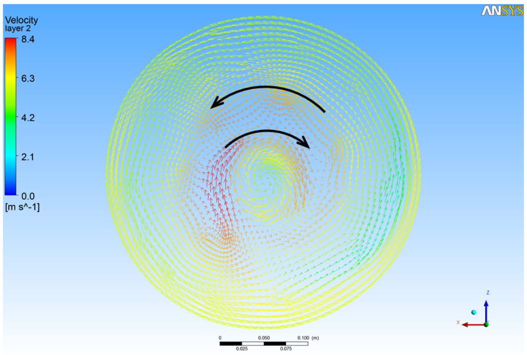
Figure 7: Pressure contours at Layer 2 of the draft tube in case 6

4.2 Pressure contours and velocity vectors

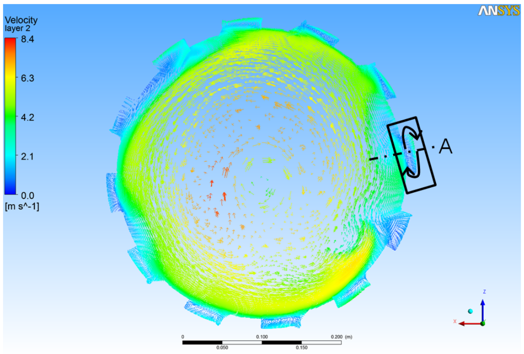
Figure 7 shows the static pressure contours at Layer 2 of draft tube in the case 6. Its location can be seen in **Figure 4**. **Figure** 7(b) shows the results of the same location by installing a group of J-Groove. The pressure difference is decreased in the plane. As a result, the stability of fluid is enhanced, which may attribute to the suppression of draft surge.

As shown in **Figure** 8, the directions of the vectors are counterclockwise when r/R is beyond about 0.6. On the contrary, when r/R is less than 0.4, the directions turn to clockwise.

Figure 8(b) shows that, with the installation of J-Groove on the wall of draft tube, the strength of swirl flow decreases to a low level, especially in the region near the wall.



(a) Original draft tube



(b) J-Groove installation

Figure 8: Velocity vectors contours at Layer 2 of the draft tube in case 6

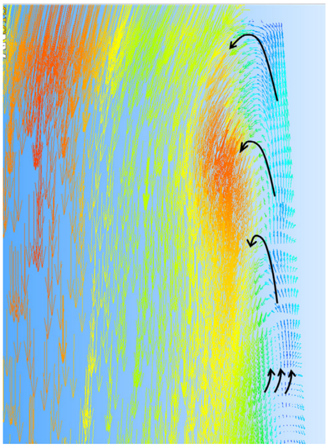


Figure 9: Velocity vectors through the J-Groove passage A in Fig. 8(cross-sectional view, case 6)

Figure 9 shows the magnified view of velocity vectors in the passage of J-Groove. On the cross-sectional plane in the draft tube, flow direction in the passage of J-Groove shows upward which is reverse to that of main flow in the draft tube. The reverse flow in the passage of J-Groove goes out to the center region of draft tube. Therefore, it is conjectured that the strong jet flow from the J-Groove to the center region of draft tube performs a role of decreasing circumferential velocity component.

4.3 Vortex core

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. When the swirl flow enters to the flow passage, dead water region including reverse flow region is formed in the center region of the flow passage because of the pressure drop in the vicinity of the flow passage center due to the swirl flow, and the gradual pressure increase in the axial direction by the draft tube. Main flow is pushed to the region nearby wall surface on the draft tube passage. In the **Figure 10**, a level of swirling strength is applied for the observation of vortex core region. **Figure 10**(a) shows that in partial flow rate range (case 1), strong swirl flow exists, which may cause draft tube surge. The Francis turbine works in good condition in the design point (case 6) with small rotating flow in the center of draft tube. In the case of excessive flow rate (case 9), vortex core region occurs near the central axis. When fluid passes through the guide vane passage and goes into the runner passage, its circumferential velocity becomes high. However, in the case of partial flow rate, axial velocity is relatively low when the fluid runs into draft tube. As a result, the swirl flow exists far away from the center of draft tube, and the depth of vortex core region is shallow. In the case of excessive flow rate, to the contrary, fluid rotates around the axis of draft tube.

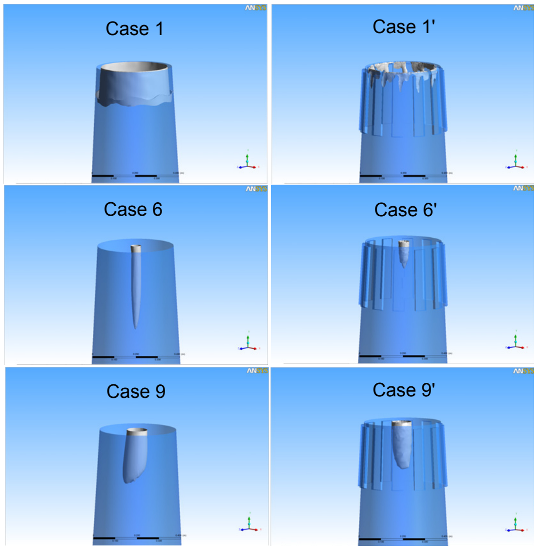


Figure 10: Vortex core region in the draft tube (left: original draft tube; right: J-Groove installation)

Compared with Figure 10(a), it is clear that vortex core region diminishes with the installation of J-Groove on the wall of draft tube (case 1’, 6’ and 9’), as shown in Figure 10(b). Especially, in the case 1’ and case 6’, vortex core region diminishes considerably.

4.4 Swirl velocity

In order to examine the effect of J-Groove quantitatively, a comparison of circumferential velocity in the draft tube is made between no J-Groove cases and J-Groove installation cases. **Figure 11** is shown to identify the effectiveness of J-Groove to the axial direction in the draft tube. The lines with the same mark show the results of no J-Groove cases and J-Groove installation cases with the same guide vane angle. The curves with solid mark represent for the CFD results of no J-Groove cases, and others are J-Groove installation cases (Case 1’, 6’ and 9’). It is found that the J-Groove can suppress the rotating velocity up to 75% in Case 1. The suppression shows a big difference in different vertical locations and in different cases. It is due to asymmetrical internal flow pattern in the draft tube passage. In others words, the design of J-Groove remains improvement. Therefore, shape optimization of J-Groove is needed for the further improvement of its suppression performance.

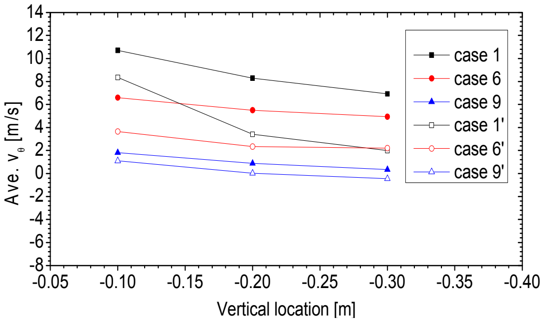


Figure 11: Averaged circumferential velocity near the wall of draft tube

Figure 12 shows the averaged circumferential velocity at each layer by the radius ratio of r/R in different cases. The results show that the J-Groove can give considerable effect in the region at the r/R is beyond about 0.85. In the partial flow rate case, the influence of J-Groove is more obvious. However, in the excessive flow rate case, the effectiveness is a little, especially near the inlet of draft tube.

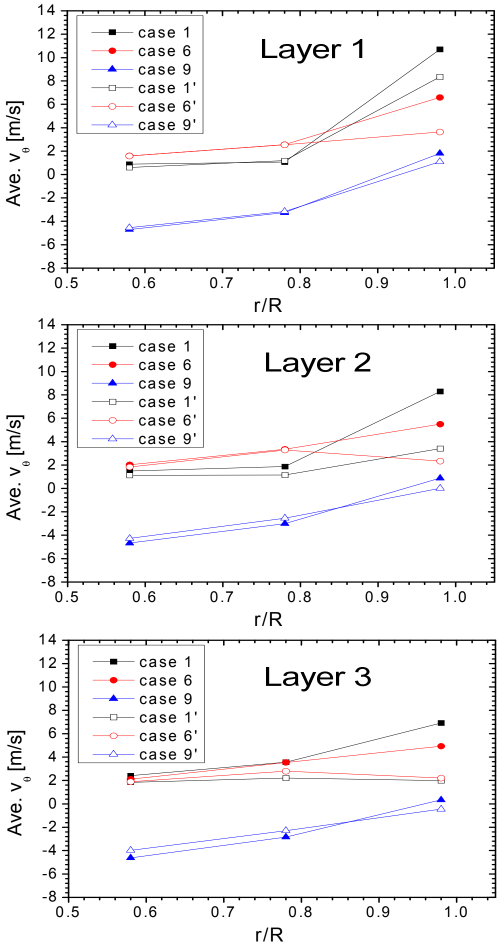


Figure 12: Averaged circumferential velocity at each layer by the radius ratio of r/R in different cases

5. Conclusions

In order to examine the validity of J-Groove in controlling and suppressing the swirl flow in a draft tube of Francis hydro turbine which may cause draft surge, CFD analysis has been performed using the draft tube of a Francis hydro turbine model and the following conclusions are achieved.

1. Strong jet flow from the J-Groove passage to

the center region of draft tube performs a role of decreasing circumferential velocity component. Therefore, vortex core region diminishes considerably with the installation of J-Groove on the wall of draft tube, especially, in the cases of partial and designed flow rate range.

2. J-Groove can give considerable effect on the suppression of swirl velocity in the region at the r/R is beyond about 0.85. In the range of partial flow rate, the influence of J-Groove is more obvious. However, in the excessive flow rate case, the effectiveness is a little, especially near the inlet of draft tube.

3. From the continued following study for the shape optimization of J-Groove, further improvement of suppression performance for the swirl flow in draft tube will be expected.

Acknowledgments

This research was financially supported by the Ministry of Education, Science and Technology (MEST) and National Research Foundation of Korea (NRF) and Jeonnam Science & Technology Promotion Center (JNSP) through the research & development project of Jeonnam Science Park. Moreover, the authors also wish to express their thanks to State Key Laboratory of Hydroscience and Engineering, Tsinghua University for the support by Open Research Fund Program.

References

[1] J. Kurokawa, H. Imamura and Y.-D. Choi, "Effect of groove on the suppression of swirl flow in a conical diffuser," ASME J. Fluids Eng., vol. 132, pp. 071101-1-071101-8, 2010.

[2] J. Kurokawa, S. L. Saha, J. Mstsui and T. Kitahora, "Passive control of rotating stall in a Parallel-Wall vaneless diffuser by radial grooves," ASME Trans. vol. 122, pp. 90-96, 2000.

[3] S. L. Saha, J. Kurokawa, J. Matsui and H. Imamura, "Suppression of performance curve instability of a mixed flow pump by use of J-Groove," ASME J. Fluids Eng., vol. 122, pp. 592-597, 2000.

[4] Y.-D. Choi, J. Kurokawa and H. Imamura, "Suppression of cavitation in inducers by J-Grooves," ASME J. Fluids Eng., vol. 129, pp. 15-22, 2007.

[5] R. S. Resiga, S. Muntean, V. Hasmatuchi, I. Anton and F. Avellan, "Analysis and prevention of vortex breakdown in the simplified dischargr cone of a francis turbine," ASME J. Fluids Eng., vol. 132, pp. 051102-1-051102-25, 2010.

[6] H. T. Flavey, Draft Tube Surges-A Review of Present Knowledge and an Annotated Bibliograpfy, U.S. Bureau of Reclamation, Report No. REC-ERC-71-42, 1971.

[7] R. H. Thicke, "Pratical solutions for draft tube instability," Int. Water Power and Dam Construction, vol. 33, no. 2, pp. 31-37, 1981.

[8] M. Nishi, X. M. Wang, K. Yoshida, T. Takahashi and T. Tsukamoto, "An experimental study on fins, their role in control of the draft tube surging," Proceedings of the XVIII IAHR Symposium on Hydraulic Machinery and Cavitation, vol. 1, pp. 905-914, 1996.

[9] O. Kirschner, H. Schmidt, A. Ruprecht, R. Mader and P. Meuburger, "Experimental investigation of vortex control with an axial jet in the draft tube of a model Pump-Turbine," IOP Conf. Earth and Environmental Science, No. 012092, 2010.

[10] M. Maiwald, R. Jester-Zürker and N. A. D. Agostini, "Numerical investigation of francis turbine draft tubes with respect to geometry modification and turbulence treatment," IOP Conf. Earth and Environmental Science, No.

012012, 2010.

[11] ANSYS Inc., “ANSYS CFX Documentation,” ver. 12, http://www.ansys.com, 2010.

[12] F. R. Menter, "Two-Equation Eddy-Viscosity turbulence models for engineering applications," AIAA Journal, vol. 32, no. 8, pp. 1598-1605, 1994.

[13] Tanaka Suiryoku, Technical Design Book of Francis Hydro Turbine, 2012.