Journal of Electrical Power & Energy Systems, 2022, 6(1), 34-43
https://www.hillpublisher.com/journals/jepes/
ISSN Online: 2576-053X
ISSN Print: 2576-0521

Controlling Flow in Draft Tube of Francis Turbine by Vortex Preventing Element

Deniz Sarper Semerci*, Tahir Yavuz

Department of Mechanical Engineering, Başkent University, Ankara, Turkey.

How to cite this paper: Deniz Sarper Semerci, Tahir Yavuz. (2022) Controlling Flow in Draft Tube of Francis Turbine by Vortex Preventing Element. *Journal of Electrical Power & Energy Systems*, 6(1), 34-43.
DOI: 10.26855/jepes.2022.02.001

Received: December 30, 2021
Accepted: January 25, 2022
Published: February 25, 2022

***Corresponding author:** Deniz Sarper Semerci, Department of Mechanical Engineering, Başkent University, Ankara, Turkey.
Email: dssemerci@baskent.edu.tr

Abstract

As importance of renewable energy sources increase with respect to energy demands, turbine efficiencies become more important. Francis turbines are one of the most common turbine types in use at hydroelectric power plants. In some cases such as load rejection and high load operations, pressure fluctuations and vortex formation cause Francis turbine efficiency and endurance to decrease. To prevent these effects, a new component named Vortex Preventing Element (VPE) is designed. The main idea of this new design is damping the swirling slow with Vortex Preventing Element and improving the performance of suction side of runner blades. This new element is mounted between turbine runner and draft tube. CFD analyses are carried out with and without VPE. Preliminary results show that VPE with having only one spiral element provides more uniform flow through the draft tube. According to the preliminary results, the new design also provides an efficiency increment about 4%.

Keywords

Francis Turbine, Computational Fluid Dynamics, Efficiency, Vortex

1. Introduction

Energy demands of countries are increasing day by day due to technological developments and ever-growing needs of humanity. Renewable energy resources play quite important role on supplying these energy demands of countries without bringing out environmental pollution and greenhouse gas emission. Hydraulic power is one of the most important renewable energy resource type because of its high potential all over the world and its efficient convertibility to electrical power. In hydro-electrical power plants, potential energy of water is first transformed into kinetic energy by releasing it from a height, called head. Then, kinetic energy of water is transformed into mechanical energy by using a hydraulic turbine. And then, it is transformed into electric energy with generator, shaft and auxiliary equipment. In this process, turbine efficiency is the most effective parameter to obtain more electric energy.

Francisturbine is the most common hydraulic turbine type in use all around the world. In most studies of literature, it is seen that the efficiency values of Francis turbines are reached about 90% and above. Yet, this value can still be increased by preventing some undesirable events during the flow such as cavitation, pressure fluctuations etc. These events would not only damage the turbine's mechanic parts but also cause the turbine's efficiency to decrease. Therefore, some studies have been carried out in recent days to prevent pressure pulsations in draft tube. Anup et al. [1] showed that the runner outflow has a draft tube. Anup et al. [1] showed that the runner outflow has a swirling component at the middle section of turbine runner, when Francis turbines operate at partial loads or over loads. This swirling flow causes to pressure pulsations and flow irregularities which lead to pressure fluctuations in draft tube. In partialloads, these pressure fluctuations generate a vortex rope at inlet section of draft tube and this vortex structure causes disorders on torque, axial force and radial force. Also, this structure leads vibration, noise and wear on auxiliary equipment. The reasons of vortex formation are decreasing flow velocity at the draft tube inlet section and swirling component. The swirling component rotates the low-velocity flow and that causes the vortex formation to occur. Anup et al. investigated the vortex structure on the draft tube's inlet section by using different turbulence models.

Choi, Kurokawa, and Imamura [2] studied vortex structures in draft tube and they tried to prevent these structures by using J-grooves and inducer together. They obtained that the J-grooves and inducer provides runner blades' suction side to perform better at partial loads. As a consequence, they show that the hollows named J-grooves controlled the angular momentum of the flow and eliminated the swirling components. Wei, Choi, and Zu [3] also and Chen and Choi [4] also investigated the effects of J-grooves on vortex structures. They showed that the quantity of vortex structures would be decreased by using J-grooves. The jet effect of these hollows decreases circumferential velocity component and eliminates a fair amount of vortex structure at partial loads. Chen and Choi studied both numerical and experimental o J-grooves' effects on vortex formation. They determined four different points on draft tube wall and measured pressure pulsations. By using Root Mean Square method, they examined magnitude of pressure fluctuations. They pointed out that these hollow structures partially prevent the vortex rope generation but they would not affect turbine performance.

Prof. Nishi [5] and his group determined pressure fluctuations have two parts: synchronous and asynchronous fluctuations. They carried out experimental studies and as a consequence, they obtained there is no synchronous pressure fluctuations in a straight draft tube so the elbow causes to synchronous fluctuations in a draft tube. But Stuparu and Susan-Resiga [6] claimed that pressure pulsations would always generate a vortex rope and they carried out numerical studies. Their numerical studies show that pressure fluctuations are not related with the interaction between vortex rope and draft tube elbow.

Muntean et al. [7] investigated pressure pulsations caused by swirling flow in a straight draft tube. Pressure sensors are located in several points on draft tube and Fourier spectrum of measurements is determined. They also carried out numerical simulations in different turbulence models. It's seen that RNG k-ε and SAS k-ω models are better turbulence models to reveal vortex formation at the draft tube inlet. But these two models are insufficient compared to RSM to model vortex development at the onward sections of draft tube.

Susan-Resiga [8] and Zhang [9] revealed the important topics on the vortex control technique. According to them, the control technique must show the reason of vortex formation, the vortex rope should be controlled at the inlet section of draft tube, the technique should aim the stagnant area located at the middle of draft tube and control technique must not decrease the turbine efficiency.

Foroutan and Yavuzkurt [10] used k- ε turbulence model to simulate the vortex rope under the effect of a water jet. They determined that unsteady RANS models are not appropriate to modelling vortex rope properties. Thus, they used Detached Eddy Simulation. As a consequence, they obtain that the water jet partially controlled the pressure pulsations in draft tube.

Dias and Riethmuller [11] injected air bubbles into the stagnant flow located in the inlet section of draft tube. They used Particle Image Velocimetry (PIV) to observe vortex formation.

Iliescu, Ciocan and Avellan [12] showed that runner blades with constant slope angle cause cavitation phenomena. They investigated the cavitating vortex rope formation by using PIV. They use image processing and filtering techniques to analyse the vortex rope development.

As it can be seen, swirling components and stagnant regions are occurring during flow because of Francis turbine's runner blade shapes. And these events are triggering pressure pulsations and vortex rope formation. These phenomena affect the Francis turbine in terms of both mechanical endurance and turbine efficiency.

Since the runner blades causes swirling flow, a vortex formation would occur at draft tube inlet as it is mentioned above. The aim of this study is designing a new component to prevent this stagnant area and vortex formation.

2. Methods

2.1. Design Methodology

Francis turbine is a reaction turbine which comprises of several components, such as spiral case, stay vanes, guide vanes, turbine runner, draft tube. Spiral case surrounds entirely the whole turbine components and converts the water to the stay vanes with equal volume rate and pressure in each section because of its decreasing cross sectional area. Stay vanes provide water to flow in a straight form against the guide vanes. They regulate the incoming flow from spiral case. Guide vanes lead water to flow towards the runner and they're also controlled by servo motors to adjust the flow rate of water by changing their directions. Turbine runner is the component which provides mechanical work to be done. With the rotational torque generated by the forces acting on the turbine wheel in the circumferential direction, the impeller starts to rotate and drives the generator through the connected shaft. Draft tube is the component where the water is released after mechanical work is done. The incoming high flow rate from the turbine runner is converted back to the pressure by increasing cross sectional area of draft tube. Then water discharged into the lower water channel. It is also important for adjusting cavitation number by adjusting the suction height of the turbine.

Designing a Francis turbine is a complex process due to several components and complex geometries of these components. Thus, a design methodology is generated and Francis turbine is designed based on these steps.

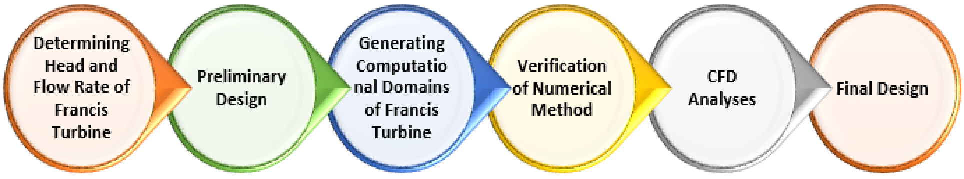

Figure 1. Design Methodology.

First step of design methodology is determining head and flow rate values. Since all other geometric parameters depend on these two parameters, these parameters should be determined. Second step is specified as preliminary design. Preliminary design is a stage that all the geometric parameters of turbine components obtained from experimental curves and theoretical data in literature. In this study, geometric parameter calculations are carried out by using empirical equations given by Siervo and Leva [13]. But when all the parameters are calculated, it is seen that these turbine parameters are too big to compute in terms of time and CPU since base parameters are taken from an actual turbine. Thus, a scaling process is applied due to affinity laws. Some of the model turbine parameters are given in Table 1.

Table 1. Preliminary Design Parameters

Parameter	Value
Head	4.07 m
Flow Rate	4.16 m^3/s
Spiral Case Inlet Diameter	1.493 m
Stay Vanes Inlet Diameter	1.984 m
Guide Vanes Inlet Diameter	1.663 m
Runner Inlet Diameter	1.003 m
Draft Tube Inlet Diameter	1.326 m

Next step is generating computational domains of Francis turbine. Ansys BladeGen is used to generate blades meridional profiles and Design Modeler is used to generate turbine components' computational domains.

Besides of these components, a new component—which is named as Vortex Preventing Element, is designed to prevent the vortex formations which cause the lower endurance and turbine efficiency. It is spiral structure in a cylinder which has the same diameter with runner outlet and draft tube inlet. It is aimed to damp the swirling effect of the outflow of runner and diminish the vortex formation. The computational domain of vortex preventing element is given in Figure 2.

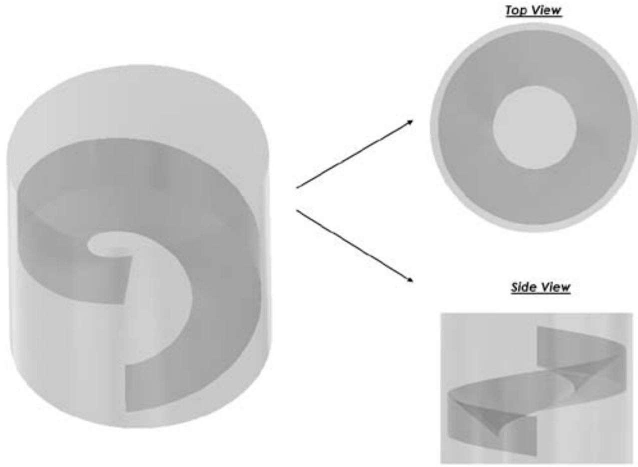

Figure 2. Vortex Preventing Element.

In Figure 3, the full view of computational domain is given. All components are mounted to each other respectively. Water enters into the spiral case firstly. Then stay vanes regulates the flow towards guide vanes. Flow rate is controlled by guide vanes and water will flow towards turbine runner. Runner is rotated by kinetic energy of water and turned into mechanical work. Then water leaves the runner and enters the Vortex Preventing Element and it is thought to be prevented pressure pulsations in the draft tube. Then water will be discharged by draft tube at atmospheric pressure.

Figure 3. Francis Turbine Computational Domain.

In Figure 4, the position of Vortex Preventing Element is shown.

Remaining steps of design methodology are about Computational Fluid Dynamics. They are detailed in following sections.

2.2. Numerical Method

After computational domains are generated, numerical model is developed. Mesh structure of model is generated by ANSYS Mesher. 2x107 tetrahedral elements are used. This is a large number of elements needed to analyse the whole system. In Figure 5, mesh structures of Francis turbine's components are given.

ANSYS CFX is used for generating numerical model. The turbulent flow in the Francis turbine is modelled with a Reynolds Averaged Navier-Stokes (RANS) model, RNG k-ε, because of its reliability. The equations are used is given below. Since there are rotating and stationary parts of Francis turbine, some approaches should be applied between mesh interfaces. By using Multiple Reference Frame approach, mesh interfaces are settled. General Grid Interface is used for stationary interfaces. Frozen rotor is used for runner-draft tube and runner-guide vanes interfaces. With this method, runner is specified as rotating but its interfaces between stationary parts such as guide vanes and draft tube remain stationary.

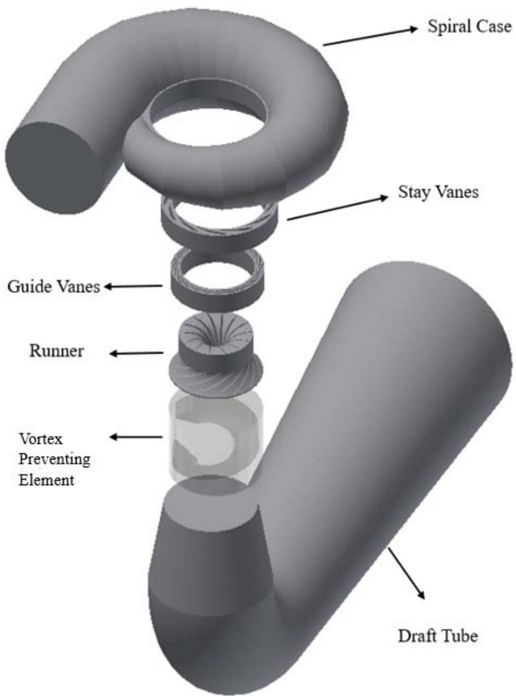

Figure 4. Francis Turbine Components.

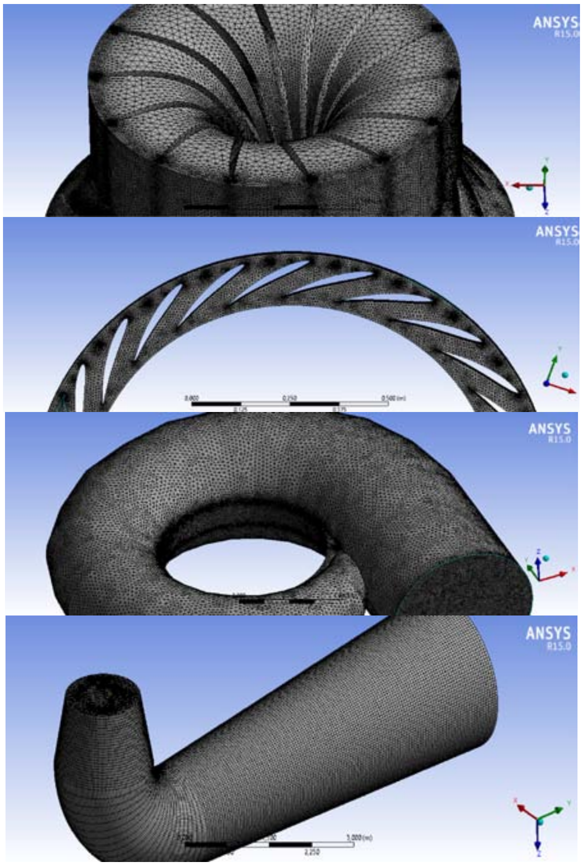

Figure 5. Mesh Structures of Computational Domains.

When scaling process is applied, a discharge value is obtained for model Francis turbine. This value is specified as inlet boundary condition for spiral case inlet section. Since the water flows from draft tube to water channel, the pressure is assumed to be equal the atmospheric pressure, 1 atm. Thus, pressure outlet is defined as outlet boundary condition at draft tube outlet section.

$$\rho\left(\frac{\partial \bar{u}}{\partial t}+\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}\right)=F_x-\frac{\partial \bar{p}}{\partial x}+\mu\Delta\bar{u}-\rho\left(\frac{\partial \overline{u'u'}}{\partial x}+\frac{\partial \overline{u'v'}}{\partial y}+\frac{\partial \overline{u'w'}}{\partial z}\right) \tag{1}$$

$$\rho\left(\frac{\partial \bar{v}}{\partial t}+\bar{u}\frac{\partial \bar{v}}{\partial x}+\bar{v}\frac{\partial \bar{v}}{\partial y}+\bar{w}\frac{\partial \bar{v}}{\partial z}\right)=F_y-\frac{\partial \bar{p}}{\partial y}+\mu\Delta\bar{v}-\rho\left(\frac{\partial \overline{u'v'}}{\partial x}+\frac{\partial \overline{v'v'}}{\partial y}+\frac{\partial \overline{v'w'}}{\partial z}\right) \tag{2}$$

$$\rho\left(\frac{\partial \bar{w}}{\partial t}+\bar{u}\frac{\partial \bar{w}}{\partial x}+\bar{v}\frac{\partial \bar{w}}{\partial y}+\bar{w}\frac{\partial \bar{w}}{\partial z}\right)=F_z-\frac{\partial \bar{p}}{\partial z}+\mu\Delta\bar{w}-\rho\left(\frac{\partial \overline{u'w'}}{\partial x}+\frac{\partial \overline{v'w'}}{\partial y}+\frac{\partial \overline{w'w'}}{\partial z}\right) \tag{3}$$

2.3. Validation of Numerical Models

For the developed computational model, after 2x107 elements, the torque value of turbine runner remains almost at a constant value. That means, numerical model is mesh-independent. Also, independency from iteration number is checked by simulating in different iteration numbers. Figure 6 shows that the solution with developed numerical model gains its independency against iteration number at about 1250 iterations.

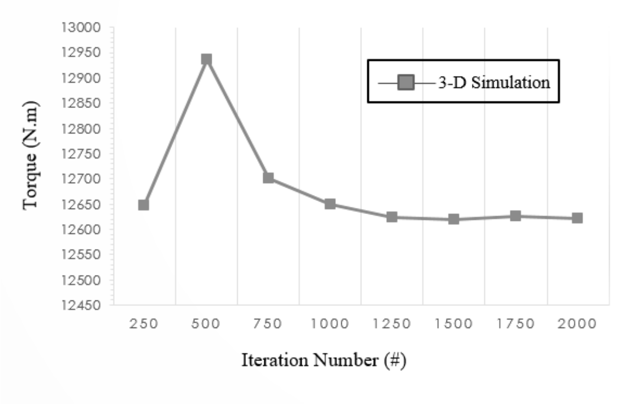

Figure 6. Independency.

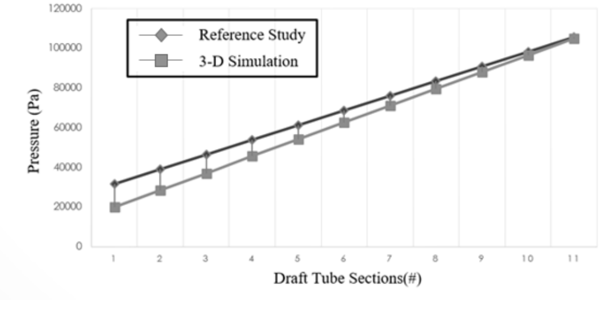

Figure 7. Validation of Numerical Method.

To validate the numerical method, the work [14] is used. From reference study, eleven sections of draft tube are chosen. Pressure values of these sections are compared. It is seen that pressure values are intersecting at the end of the tube. Yet, there are some differences between these two methods but the differences correspond to 4% after first a few sections of draft tube, which is a negligible difference.

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{4}$$

3. Results and Discussion

With developed model, CFD analyses are carried out for a standard Francis turbine and Francis turbine with Vortex Preventing Element. It is aimed to obtain the difference between the cases that Vortex Preventing Element exists and the case it does not exist.

3.1. Analysis without vortex preventing element

First, the design without Vortex Preventive Element is examined. A Francis turbine is designed based on design methodology mentioned above and torque and efficiency values are taken from CFX. A power curve is generated for model Francis turbine.

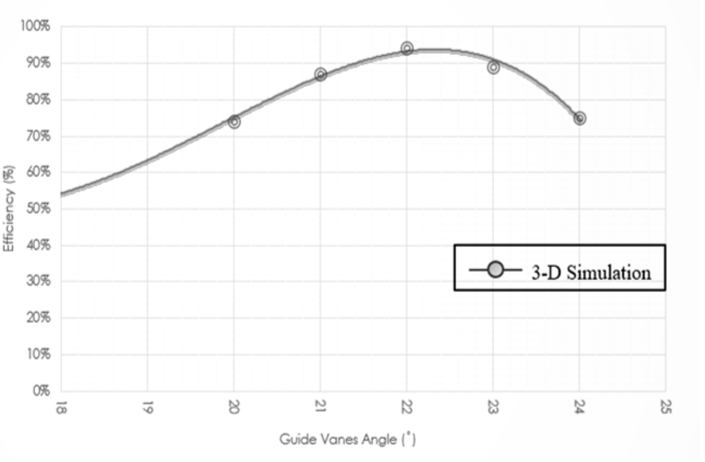

Figure 8. Power Curve of Francis Turbine.

Pressure and velocity distributions are obtained and examined for each component of Francis Turbine at best efficiency point [15]. Pressure decreases gradually towards the outlet section of spiral case, as it can be seen in Figure 9. Flow seems uniform according to the pressure contours.

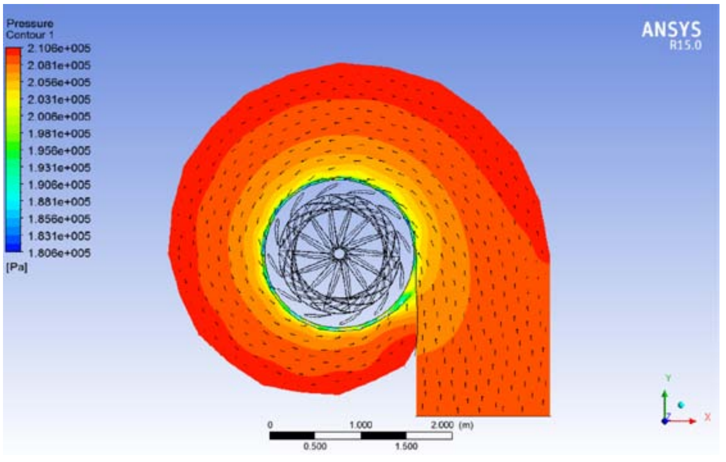

Figure 9. Pressure Distribution in Spiral Case.

Figure 10 shows the pressure and velocity distribution around the guide vane blades. Since guide vanes are the most important parameter on turbine efficiency [16], it is quite essential to obtain straight flow here. The stagnation point is located on the symmetry point of leading edge of guide vanes. Therefore, the flow between stay vanes-guide vanes and guide vanes-runner are fixed and blade angles seem to be selected correctly.

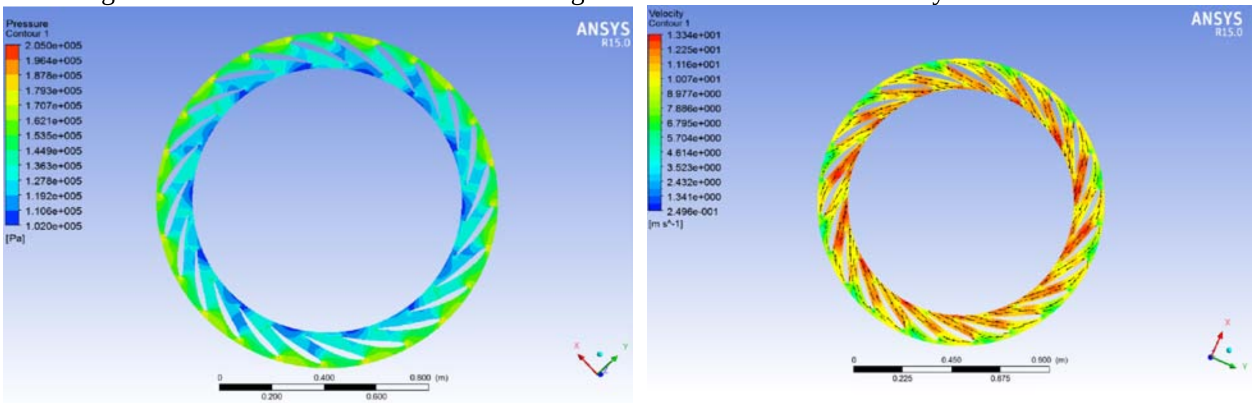

Figure 10. Pressure and Velocity Distributions around the Guide Vanes.

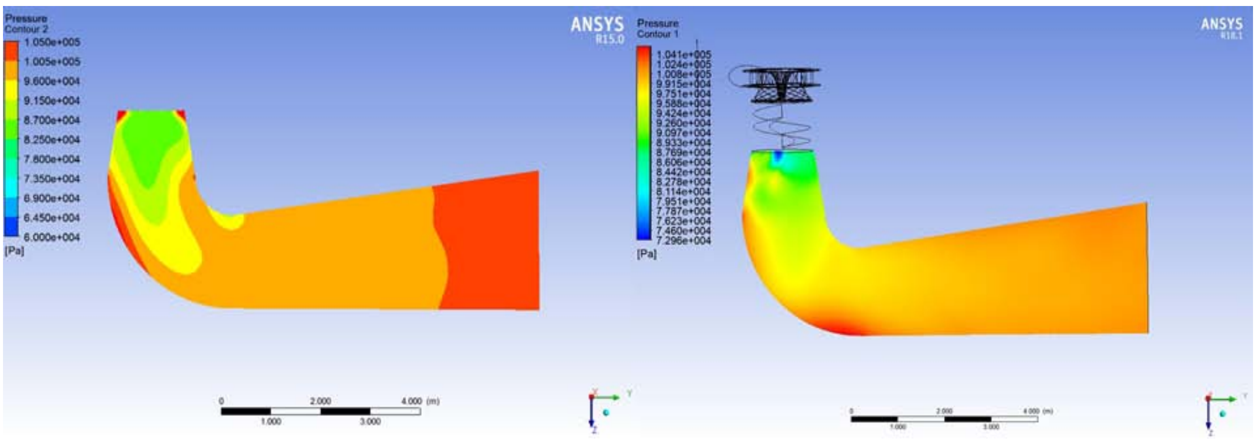

Figure 11. Pressure distribution of Francis Turbine draft tubes without and with Vortex Preventing Element respectively.

3.2. Analysis with vortex preventing element

Vortex Preventing Element is designed and mounted between runner and draft tube. A CFD analysis is carried out for the best efficiency point and it is compared with standard Francis turbine. The left hand-side of Figure 11 shows the pressure distribution of standard Francis turbine draft tube. It is seen that there are some pressure fluctuations in the entrance sections. There are symmetric high-pressure regions on the sharp edges located at the entrance sections. It is seen that pressure fluctuations still exist until water passes from the draft tube elbow.

Then it flows uniformly. On the other hand, the right hand-side of the Figure 11 shows pressure distribution of Francis turbine draft tube with Vortex Preventing Element. It is seen that pressure fluctuations are partially diminished. The uniform flow begins more earlier than the other case. Also, the symmetric high-pressure regions located at entrance of draft tube are no longer existed. Pressure transitions are smoother than first case as it can be understood from contours.

Figure 12 represents the velocity vectors of turbine runner and Vortex Preventing Element (VPE). Water enters radially to the runner and leaves axially. As it is seen, the outflow has swirling component and makes flow to rotate around its axis. When Vortex Preventing Element is used, it is seen that, the rotating flow turned into axial flow and the swirling component is eliminated at the outlet of component. The spiral form of VPE, leads water to turn opposite direction of swirling component of velocity and makes a resistance against flow.

This motion causes water to flow with minimum swirling velocity component at the outlet of VPE. The efficiencies with and without the VPE are given in Table 2. The efficiency of turbine is increased about 4 percent by using the VPE. This is the effect of the VPE having only one spiral. The optimization on the number of spirals leading maximum efficiency and uniform flow in the draft tube is needed.

To summarize, it is seen that Vortex Preventing Element provides more uniform flow and higher efficiency at the first glance.

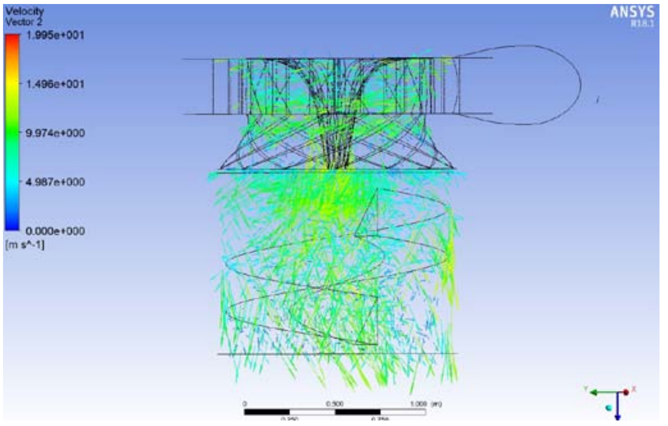

Figure 12. Velocity Vectors of Runner and Vortex Preventive Element.

Table 1. Comparison of Efficiency

Case	Efficiency
Francis Turbine without VPE	89%
Francis Turbine with VPE	92.8%

4. Conclusion

In this study, the vortex preventing element is designed and mounted in the inlet of the draft tube of the Francis turbine.

A preliminary study is carried out on the Francis turbine without and with the vortex preventing element. Conclusions from the study are:

- The Francis turbine without the VPE has pressure fluctuations in its draft tube and an efficiency about 88%.
- The Francis turbine with the VPE the pressure fluctuations are partially diminished and efficiency value reaches to 92.8%.
- The VPE in Francis turbine provides more uniform flow in draft tube and creates higher efficiency by increasing turbine blades suction sides performance.

In oncoming studies, experimental and numerical studies combining with the optimization analysis will be carried out, to obtain optimum spiral form and spiral number leading more uniform flow in the draft tube and high efficiency.

References

[1] Anup, K. C., Young Hoo, L., and Bhola, T. (2016). “CFD study on prediction of vortex shedding in draft tube of Francis turbine and vortex control technique”. Renewable Energy, 86, 2016, 1406-1421.

[2] Choi, Y. D., Kurokawa, J., and Imamura, H. (2007). “Suppression of Cavitation in Inducers by J-Grooves”. Journal of Fluids Engineering, 129, 2007, 15-22.

[3] Wei, Q. S., Choi, Y. D., and Zhu B.S. (2012). “Application of J-Groove to the suppression of swirl flow in the draft tube of a Francis hydro turbine”, 26th IAHR Symposium on Hydraulic Machinery and Systems, 2012.

[4] Chen, Z. and Choi, Y. D. (2018). “Suppression of cavitation in the draft tube of Francis turbine model by J-Groove”. Journal of Mechanical Engineering Science, 2018, 1-11.

[5] Nishi, M., Matsunaga, S., Okamoto, M., Uno, M., and Nishitani, K. (1988). “Measurement of threedimensional periodic flow

on a conical draft tube at surging condition", (eds.) Flows in Non-Rotating Turbomachinery Components FED, 1988, 69, 81-88.

[6] Stuparu, A. and Susan-Resiga, R. "The origin of the plunging pressure fluctuations for a swirling flow with precessing vortex rope in a straight diffuser", in Proc. of the 6th IAHR Int. Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems (Ljubljana, Slovenia), 1-8.

[7] Muntean, S., Tanasa, C., Bosioc, A. I., and Moş, D. C. (2016). "Investigation of the Plunging Pressure Pulsation in a Swirling Flow with Precessing Vortex Rope in a Straight Diffuser", 28th IAHR Symposium on Hydraulic Machinery and Systems, 49, 2016, 1-10.

[8] Susan-Resiga, R., Vu, T. C., Muntean, S., Ciocan, G. D., and Nennemann, B. (2006). "Jet Control of the Draft Tube Vortex Rope in Francis Turbines at Partial Discharge," 23rd IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan, 2006, October 17-21.

[9] Zhang, R. K., Mao, F., Wu, J. Z., Chen, S. Y., Wu, Y. L., and Liu, S. H. "Characteristics and Control of the Draft-Tube Flow in Part-Load Francis Turbine," ASME J. Fluids Eng., 131(2), p. 021101.

[10] Foroutan, H. and Yavuzkurt, S. "Flow in the Simplified Draft Tube of a Francis Turbine Operating at Partial Load—Part II: Control of the Vortex Rope", Journal of Applied Mechanics, 81, 061011-1.

[11] Dias, I. and Riethmuller, M. L. (1998). "Visualisation of the Forming Bubble and PIV Measurement of Surrounding Liquid Using Fluorescent Particles," Proceedings of Eighth International Symposium of Flow Visualisation, Sorrento, Italy, 1998.

[12] Iliescu, M. S., Ciocan, G. D., Avellan, F. "Analysis of the Cavitating Draft Tube Vortex in a Francis Turbine Using Particle Image Velocimetry Measurements in Two-Phase Flow", Journal of Fluids Engineering, 130, 021102-1.

[13] Siervo, J., Leva, F. (1976). "Modern trends in selecting and designing francis turbines", Water Power and Dam cont., Aug., 1976, 28-35.

[14] Akin, H., Çelebioğlu, K., and Aradağ, S. (2014). A CFD-Based Design Metodology For Hydraulic Turbines Applied to A Case Study in Turkey, 10th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, Orlando-Floriad-USA, July 2014.

[15] Semerci, D. S. and Yavuz, T. (2016). "Increasing Efficiency of an Exising Francis Turbine by Rehabilitation Process", The 5th IEEE International Conference on Renewable Energy Research and Applications, Birmingham/UK, 2016.

[16] Semerci, D. S., Yavuz, T. (2018). "Optimization of a Model Francis Turbine Paramaters for the Most Efficient Case", International Conference on Clean Energy, Famagurta/Cyprus, 2018.