International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:9, No:8, 2015

# Numerical Simulation of Three-Dimensional Cavitating Turbulent Flow in Francis Turbines with ANSYS

Raza Abdulla Saeed

***Abstract*****—**In this study, the three-dimensional cavitating turbulent flow in a complete Francis turbine is simulated using mixture model for cavity/liquid two-phase flows. Numerical analysis is carried out using *ANSYS CFX* software release *12*, and standard k-ε turbulence model is adopted for this analysis. The computational fluid domain consist of spiral casing, stay vanes, guide vanes, runner and draft tube. The computational domain is discretized with a three-dimensional mesh system of unstructured tetrahedron mesh. The finite volume method (*FVM*) is used to solve the governing equations of the mixture model. Results of cavitation on the runner's blades under three different boundary conditions are presented and discussed. From the numerical results it has been found that the numerical method was successfully applied to simulate the cavitating two-phase turbulent flow through a Francis turbine, and also cavitation is clearly predicted in the form of water vapor formation inside the turbine. By comparison the numerical prediction results with a real runner; it's shown that the region of higher volume fraction obtained by simulation is consistent with the region of runner cavitation damage.

***Keywords*****—**Computational Fluid Dynamics, Hydraulic Francis Turbine, Numerical Simulation, Two-Phase Mixture Cavitation Model.

## I. Introduction

IN liquid flows, cavitation occurs when the static pressure in certain locations falls below the vapor pressure of the liquid. Consequently the liquid boils and large number of small individual bubbles of vapor-filled are formed. These vapor bubbles are carried by the flow to higher pressure regions where the vapors condensed and the bubbles suddenly collapse. This result in the streams of liquid coming from all directions collides at the center of cavity, as the vapors are condensed to liquid again. The process of cavity collapse takes place in a very short time of about several nanoseconds. Collapsing of the vapor bubbles close to a solid surface generates an extremely high impulse pressure. Recent studies evaluated the magnitude of this high impulse pressure to be approximately 700 *atm* (~70 MPa). If the amplitude of the resulting pressure pulse is larger than the limit of the material mechanical strength, an indentation of several micrometers called ‘‘pit’’ will be formed on the metallic surface. Formation of cavity and high pressure are repeated many thousand times a second. During the fluid flow through a Francis turbine, the formation of bubbles and their subsequent collapse causes pitting on the metallic surface of runner blades or draft tube walls and generates noise. If an accumulation of pits takes place in a narrow area, the material is eroded and mass loss occurs due to the repetitive action of the cavity collapses. After relatively short time of operation, hydraulic turbines get severely damaged and will need to be shut down; accordingly the damaged parts can be repaired or replaced. Its severity depends on design and operating regime of the hydraulic turbines [1]-[3], [4]. Frequent repairs of the cavitation damage in turbine runner destroyed regular flow passage; leading to significant reduction in turbine performance. As a result, these turbines do not generate their rated capacity [5].

In addition to the damage to material surface of blades, cavitation in a hydraulic turbine causes flow instabilities, noise and vibrations, and drastic deterioration of turbine performance. Furthermore, cavitation problem in hydraulic turbines increases the plant down-time and also poses a serious threat to the life of the operational and maintenance personnel. The propensity for cavitation in hydraulic turbines is being increased because of the following two main factors or the combination of both. First, the growth of the turbine output power is based on the reduction of dimensions to decrease the cost of its components. Hence, the speeds are being increased and the cavitation number is thereby decreased. Second, Francis turbines are more prone to cavitation where the turbines are operating in conditions far from their best efficiency point [2], [4].

Traditionally, the studies for cavitation is mainly depend on experimental model testing, which usually is very expensive, difficult and time consuming. Recently, due to the quick development in Computational Fluid Dynamics (*CFD*) with the growth of computer hardware, numerical simulations have been widely used to investigate the cavitation flow to rotating machinery such as pumps, marine propeller, and water turbines [3], [6].

Developments and application studies on *CFD* for cavitating flow is reported by [7]. The cavitation in Francis can be accurately predicted by *CFD* as illustrated by [6]. And also, [8] presents a numerical investigation of cavitation development and behavior in Francis turbine runners.

Accordingly, for simulating cavitating flows in a Francis turbine, the advanced commercial *CFD* software is used. Reference [3] employed *OpenFOAM* code for simulating the cavitating turbulent flow in a high head prototype Francis turbine at part load operation. As stated by [9], the *ANSYS CFX* software gives very good results for modelling cavitation in blade passages. Reference [6] presents numerical analysis of the flow in all types of water turbines and analysis is performed by *ANSYS CFX-11.0*. They explained that the cavitating two-phase turbulent flow through a Francis turbine

R. A. Saeed Engineering Faculty, University of Sulaimani, Sulaimani, Iraq (e-mail: raza.saeed@ univsul.edu.iq).

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:9, No:8, 2015

is successfully simulated and also they showed that the numerical results agreed well with the experimental results.

In this sense, it is noted that a few studies have been published related to cavitation within hydraulic turbines. The present paper is part of an ongoing effort to numerical investigation of *3D* cavitating turbulent flow in a Francis turbine. This study is focus numerical modelling of the turbulent cavitating flow in a Francis turbine using *CFD*. The study is based on the observations of cavitation damage on runner blade surfaces in Derbendikhan power station. The cavitation eroded zone on a runner blade of a Francis turbine in Derbendikhan power station is displayed in Fig. 1. From the figure, it was visually observed that severe cavitation damage was developed at the trailing edge suction side, close to the band.

Fig. 1 Cavitation damage on runner blade surface

Cavitation erosion can take place in hydraulic turbines in various forms and appear on different locations depending on the turbines design and the operating condition. The cavitation eroded zones on different locations on the blade surfaces of a Francis turbine in Derbendikhan power station is shown in Fig. 2.

## II. Cavitation in Hydraulic Turbines

The main types of cavitation phenomenon in Francis turbine are shown in Fig. 3, which demonstrates how leading edge cavitation, trailing edge cavitation, draft tube swirl and inter blade vortex cavitation can damage the Francis turbine. The general features of cavitation phenomenon in hydraulic turbines for the each type of cavitation occurring are given by [7]. A survey of different types of caviation featured by hydraulic machinery has been carried out by [10].

Cavitation near the leading edge of the forward face of the blade as shown in Fig. 3 (*a*) can be the serious problem that is likely to erode the blades deeply. It is on account of liquid flow obstruction and change in water pressure at the point of contact and subsequent water bubbles formation. Cavitation at the trailing edge of the blade, Fig. 3 (*b*), is a noisy type of cavitation that minimizes the machine performance and initiates blade erosion. Draft tube swirl, Fig. 3 (*c*) can produce low frequency pressure pulsation resulting into hydraulic resonance causing high amplitude vibration on the turbine components, which could potentially disturb the whole power house structure. Inter-blade vortex cavitation as depicted in Fig. 3 (*d*) is a phenomenon that can occur mostly on the trailing edge of the turbine components [1].

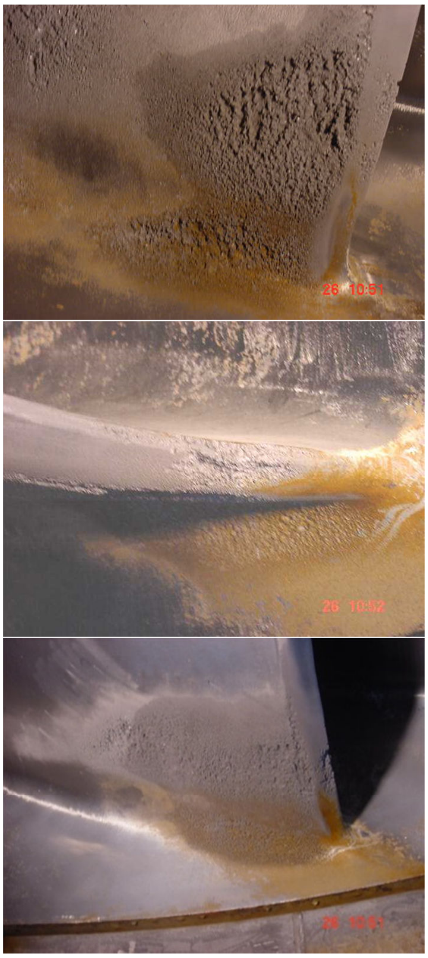

Fig. 2 Cavitation zones on runner blades in different locations

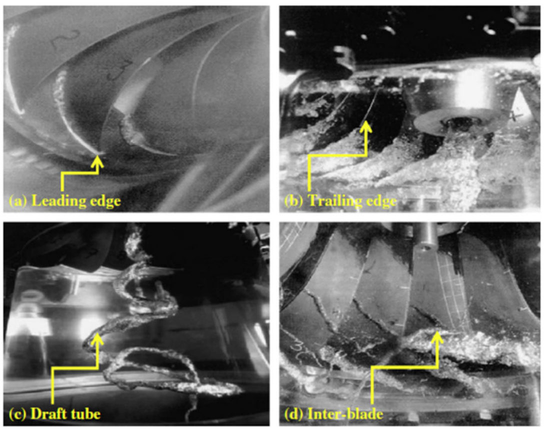


Fig. 3 Cavitation in Francis turbine (*a*) leading edge cavitation, (*b*) trailing edge cavitation, (*c*) draft tube swirling and (*d*) inter-blade vortex cavitation [1]

The areas most exposed to cavitation erosion in the runner are illustrated in Fig. 4. In general severe cavitation erosion damages are observed in Francis runners on the blade leading edge, shaded area at location (*A*) and (*B*), travelling bubble cavitation usually occurs at location (*C*) and inter-blades cavitation usually occurs at location (*D*). Leading edge cavity is the main type of cavitation development over a wide operating range of discharge [10]. On the other hand, [1] shows that the cavitation erosion in Francis turbine may be severe in the blade leading area and the trailing edge. Other associated components such as draft tube, guide vanes and wicket gates experience lesser cavitation damages compared to the blade.

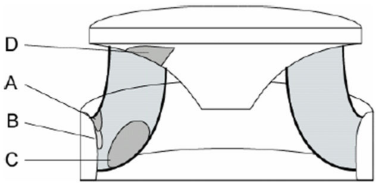


Fig. 4 Typical eroded areas of a Francis runner [10]

## III. Numerical Method

In this study, *ANSYS CFX* software Release 12 been used with *CV-FVM* to simulate the *3D* cavitating flow in Francis turbine. The computational flow domain is discretized with a three-dimensional mesh system of unstructured tetrahedron mesh. The software uses a control volume - based technique to convert the governing equations in algebraic equations that can be solved numerically. For this simulation, the *k-ε* turbulence model is considered because it is one of the most common turbulence models.

## IV. Governing Equations and Turbulence Model

The governing equations for two-phase media include the conservative form of the Reynolds Averaged Navier-Stokes (*RANS*) equations, plus a volume fraction transport equation. The governing equations for the continuity equation (1), momentum equation (2), and volume fraction transport equation (3) of a mixture are written in the Cartesian Coordinates, as follow [11]:

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial(\rho_m u_j)}{\partial x_j} = 0, \quad (1)$$

$$\frac{\partial}{\partial t}(\rho_m u_i) + \frac{\partial}{\partial x_j}(\rho_m u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \quad (2)$$

$$\frac{\partial \alpha_l}{\partial t} + \frac{\partial}{\partial x_j}(\alpha_l u_l) = (\dot{m}^- + \dot{m}^+), \quad (3)$$

The mixture density and the turbulent viscosity are defined as [11]:

$$\rho_m = \alpha_l \rho_l + \rho_v(1 - \alpha_l) \qquad \mu_t = \frac{\rho_m C_\mu k^2}{\varepsilon} \quad (4)$$

where $\rho_m$ and $\mu_m$ are the mixture density and dynamic viscosity, $u$ is the velocity, $p$ is the pressure, $\mu$ and $\mu_t$ stand for the laminar viscosity and turbulent viscosity, $\alpha$ is the volume fraction, $\dot{m}+$ and $\dot{m}-$ represent the source terms for evaporation and condensation. The subscripts *m, l, v* indicate the mixture, liquid and vapor, respectively. Subscripts *i, j, k* denote the axes directions.

## V. Computational Domain and Discretization Process

In order to investigate the cavitating turbulent flow through the entire Francis turbine, a three-dimensional geometrical model of the fluid domain has been created according to the provided specifications (see Fig. 6). The flow passageway between the spiral casing from the inlet side and the draft tube from the outlet side for the turbine is considered. The computational domain is divided into four domains namely spiral casing, vanes (Stay and Guide vanes), runner and draft tube domain (see Fig. 5), in which runner is rotating domain and others are stationary domains. The investigated turbine is described as following [12]: 13 runner blades, 24 guide vanes, rated head of 80 *m*, power output at rated head of 83*MW*, discharge at rated head of 113 $m^3/s$.

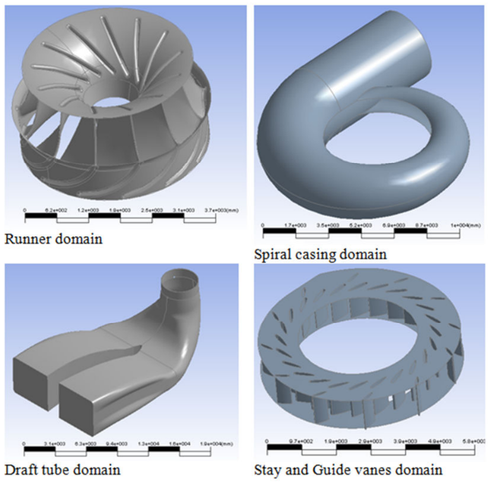


Fig. 5 The computational domains (a) spiral casing, (b) vanes (Stay and Guide vanes), (c) runner and (d) draft tube domain

Discretization process has been achieved to discretize the geometrical model with unstructured tetrahedral elements and prism layers near the walls. In this study, *3D* discretization has been utilized with the finite volume method (*FVM*) provided by the *ANSYS CFX* software. In order to accurately simulate the flow in a turbine passage, further mesh refinement around the vanes and runner blades' edges is required, which is locally zoomed up in Fig. 7 (b). With mesh refinement an improvement of the results can be expected. In numerical simulations, a mesh dependence test is important in order to check the convergence of the computation. It must ensure that the residuals of the computed equations and the continuity error are small. The test is performed by refining the mesh to its final configuration shown in Fig. 7 (a) and that has been selected for the analysis. The number of elements and the nodes generated are 3351396 and 652320 respectively.

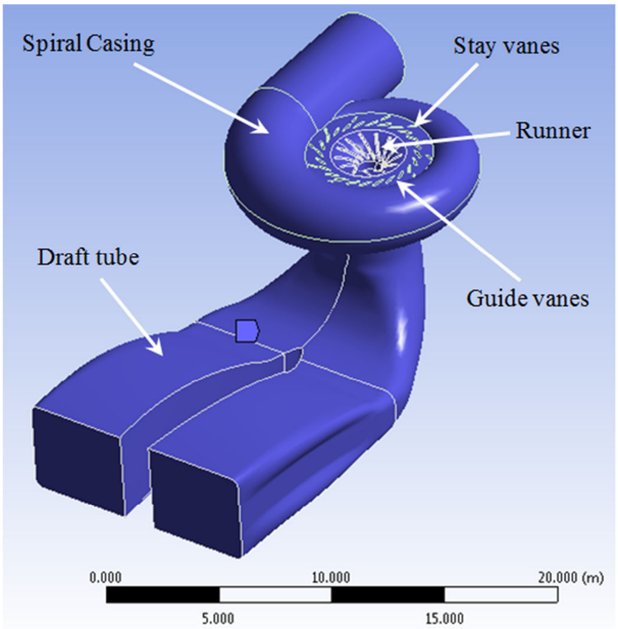


Fig. 6 3D computational flow domain for the Francis turbine

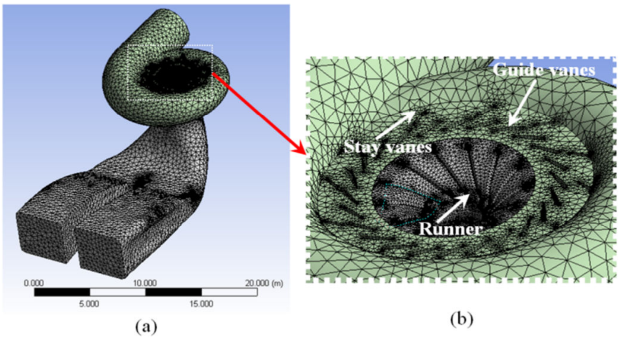


Fig. 7 (a) *3D* unstructured tetrahedral mesh of the computational domain (b) mesh refinement on several locations (runner's blades and vanes)

## VI. Boundary Conditions

Flow in water turbines is turbulent and unsteady, while the efficiency and cavitation in Francis turbines can be predicted by a steady state flow analysis and the results are usually accurate enough [13]. Therefore, in this study the computations assume steady state incompressible uniform fluid flow. A frozen rotor condition is set at the interface between the rotating and stationary frames. With frozen rotor model the two frames of reference (stationary and rotating) are connected in such a way the position of runner is fixed according to the stationary parts throughout the calculation [6].

At the inlet and outlet of the computational domain the normal velocity and pressure boundary conditions are prescribed, respectively. The simulations are carried out for three different operation points, the inlet velocity is set to 2 *m/s*, 2.5 *m/s*, and 3 *m/s*, along with the outlet used a constant pressure boundary condition (zero). On the solid walls of the domain, no-slip boundary condition is applied. All the calculations are firstly carried out under non-cavitation condition to obtain a steady solution subsequently the cavitation model is activated. Being the governing equations non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. The iterative solvers are considered converged when the residuals have been reduced by a factor of $10^{-4}$.

## VII. Results and Discussion

Three-dimensional numerical analyses of the cavitating flow in a complete Francis turbine have been carried out and the computational results under three different operation conditions are presented and discussed. The simulations have been performed to predict cavitation in the form of water vapor inside the turbine. The volume fraction of water vapor is an important feature for cavitating flow within the computational domain; it varies between 0 and 1. The distributions of volume fraction of water vapor generated with the *ANSYS CFX* postprocessor are presented in the form of contour.

The simulation results show that the cavitation can occurs at a number of different locations on turbine runner, especially at the blade leading and trailing edges on the suction side, in the runner either at crown or at band (see Fig. 8). As illuminated in the figure, the volume fraction of water vapor at the blade leading and trailing edges on the suction side is higher than others.

Fig. 9 shows the volume fraction of water vapor distributions at the top of the blade on the suction side under three operation conditions. Remarkably, the volume fraction of water vapor distribution on the blades shows asymmetry because of the existence of the spiral casing, which breaks the symmetrical characteristic of the runner passageway. On the other hand, the coupling effects between the runner and spiral casing, making volume fraction distribution on the blade surface asymmetric.

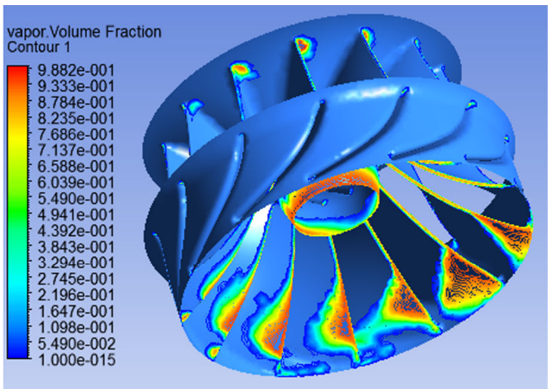


Fig. 8 Different locations of cavitation on turbine runner

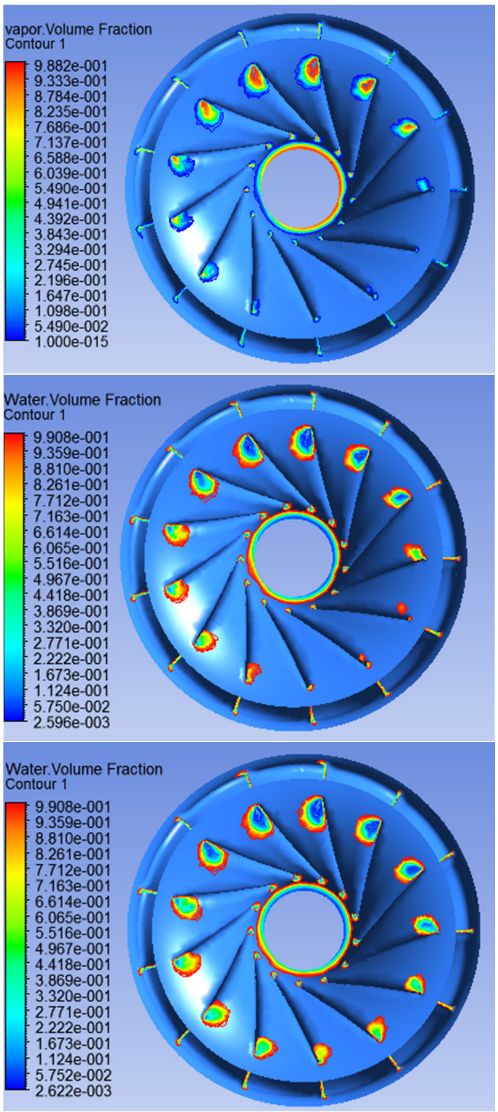


Fig. 9 The volume fraction of vapour distribution at the top of the blade on the suction side

A significant vapor volume can be seen in Fig. 10, on the suction side near the blade trailing edge which leads to barely visible erosion. The simulation results show that the cavitation is clearly predicted in the form of water vapor formation under three different operation conditions.

The simulation results in Figs. 9 and 10, demonstrate that the cavitation area and its locations are change by changing the operation conditions. When the inlet velocity is increase the cavitation occurred in the area of the suction side close to the leading edge is increases, concurrently, the cavitation region at the runner blade trailing edge suction side, close to the band is decrease.

By comparison the numerical prediction results with cavitation damage in real runner (see Fig. 1) one can conclude that the region of runner cavitation damage and the region of higher volume fraction in the simulation are consistent. This indicates that the *ANSYS* software has ability to capture the cavitation phenomenon in a Francis turbine.

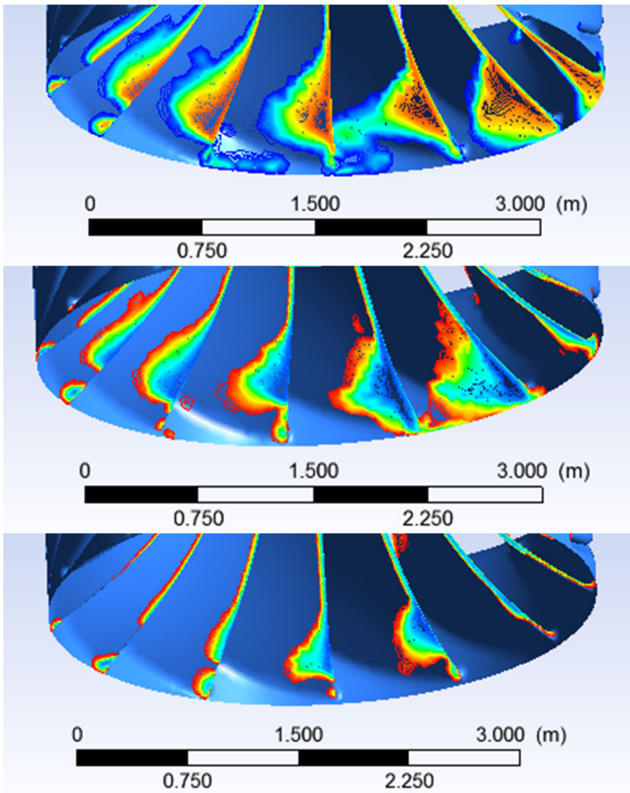


Fig. 10 The volume fraction of water vapour distribution on the suction side near the blade leading edge

## VIII. Conclusions

The paper presents a numerical investigation of the *3D* cavitating flow in a Francis turbine runner using the mixture model. Numerical simulation is implemented in the *ANSYS CFX* software and standard k-ε turbulence model is adopted. The simulation results show that the cavitating flows in Francis turbine can be accurately predicted. Zones of cavitation damage which are situated at the runner blade trailing edge suction side, close to the band are consistent with the region of higher volume fraction obtained from the simulation.

## References

[1] U. Dorji, R. Ghomashchi, "Hydro turbine failure mechanisms: An overview," Engineering Failure Analysis, vol. 44, pp. 136-147, 2014.

[2] P., Kumar, R. P. Saini, "Study of Cavitation in Hydro Turbines a Review," Renewable and Sustainable Energy Reviews, vol. 14, no. 1, pp. 374–83, 2010.

[3] L., Zhang, J. T. Liu, Y. L. Wu and S. H. Liu, "Numerical simulation of cavitating turbulent flow through a Francis turbine," 26th *IAHR* Symposium on Hydraulic Machinery and Systems, pp. 1-7, 2012.

[4] X. Escaler, E. Egusquiza, M. Farhat, F. Avellan, "Detection of cavitation in hydraulic turbines". Mechanical Systems and Signal Processing 20, pp. 983-1007, 2006.

[5] H.-J. Choi, M. A. Zullah, H.-W. Roh, P.-S. Ha, S.-Y. Oh, and Y.-H. Lee, "*CFD* validation of performance improvement of a 500 kW Francis turbine," Renewable Energy, vol. 54, pp. 111–123, 2013.

[6] D. Jošt, A. Lipej, P. Mežnar, "Numerical prediction of efficiency, cavitation and unsteady phenomena in water turbines," ASME 2008 9th biennial conference on engineering systems design and analysis, pp.157-166, 2008.
[7] S. Bernad, S. Muntean, R. Resiga, I. Anton, "Numerical analysis of the cavitating flows," Proceedings of the Romanian Academy A, vol. 7, no. 1, pp. 33-45, 2006.
[8] R. Resiga, S. Muntean, S. Bernad, I. Anton, "Numerical Investigation of 3D Cavitating Flow in Francis Turbines." Proceedings of the Conference on Modelling Fluid Flow 2, pp. 950-957, 2003.
[9] M. Sedlar, P. Zima, M. Muller, "*CFD* Analysis of Cavitation Erosion Potential in Hydraulic Machinery," Proc. 3rd IAHR WG Meeting, pp. 205-21, 2009.
[10] F. Avellan, "Introduction to cavitation in hydraulic machinery", Proceedings of 6th International Conference on Hydraulic Machinery and Hydrodynamics, Timisoara, Romania, pp.11-22, 2004.
[11] G. Wang, I Senocak, W. Shyy, "Dynamics of attached turbulent cavitating flows," Progress in Aerospace Sciences, pp. vol. 37, pp.551–581, 2001.
[12] R. A. Saeed, A. N. Galybin, V. Popov, "*3D* fluid-structure modelling and vibration analysis for fault diagnosis of Francis turbine using multiple ANN and multiple ANFIS," Mechanical Systems and Signal Processing, Vol. 34, no. 1-2, pp. 259-276, 2013.
[13] D. Jost, A. Lipej, "Numerical prediction of non-cavitating and cavitating vortex rope in a Francis turbine draft tube," Strojniski vestnik-Journal of Mechanical Engineering, Vol. 57, no. 6, pp. 445-456, 2011.

**Dr. Saeed** was born in 1972. He started his academic carrier in the University of Sulaimani, Kurdistan Region-Iraq in 1995. He received his Ph.D. degree in 2012 and he is currently a lecturer at University of Sulaimani. His research interests include: the Artificial Intelligent techniques in addition to modeling and simulation.