Water Flow Stabilization Using Submerged Weir for Draft Tube Reaction Hydraulic Turbine

Ameen Mohammed Salih Ameen¹*, Zainah Ibrahim¹ and Faridah Othman¹, Zaher Mundher Yaseen², *

¹ Civil Engineering Department, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia

² Sustainable Developments in Civil Engineering Research Group, Faculty of Civil Engineering, Ton Duc Thang University, Ho Chi Minh City, Vietnam.

Corresponding authors: Ameen Mohammed Salih Ameen and Zaher Mundher Yaseen

*Corresponding email: ameenmsalih@siswa.um.edu.my and yaseen@tdt.edu.vn

Ameen Mohammed Salih Ameen No: 0060-1125112953
Zaher Mundher No: 00841634987030
Zainah Ibrahim No: 0060192781604
Faridah Othman No: 0060136061716
Abstract

In turbine practice engineering, draft tube downstream running under extreme water flow pressure and velocity. This is causing a vibrations and pressure variation during different operation frequencies. The practical challenge of obtaining a stabilized water flow is ongoing domain of research. In this paper, a proposition of initiating submerged weir in the downstream of draft tube reaction turbine is inspected. The main goal of this research is to reduce the water flow pressure variation, velocity and shear distribution in accordance to the upstream water level influence. Two types of turbines including vertical Kaplan and Francis turbine units are examined. ANSYS CFX software tool is used to build three-dimension (3D) numerical models for the Kaplan and Francis turbines with building a submerged weir at the outlet of the draft tubes at three deferent height suggestions. The influence of the proposed submerged weir is studied the flow through these turbines by considering the dimensions of their components including the penstock with inlets, spiral casing, shafts and blades, and the draft tube with outlets. The findings of this research were tremendous proposition to solve the problem of negative pressure pulsation in draft tube of Kaplan and Francis turbines types.

Keywords: Numerical fluid analysis; hydropower sustainability; water pressure and velocity; submerged weir; reaction turbine.

1. Introduction

Powerhouses are among the main parts of dams that are used to generate low-cost hydroelectric power. The identification of the hydraulic characteristics of Kaplan and Francis turbine units [1], which function as the main engine of a powerhouse. Kaplan and Francis turbines, which are classified as reaction turbines, are difficult to use under part-load operation because of pressure oscillation [2–4]. Studies on this topic have presented solutions
for the cavitation problem in draft tubes [5], the vibration effect in powerhouses caused by a running turbine, maximizing power generation, and generating low-cost power [1, 6].

In the last decade, there has been a very noticeable development in the computational fluid tools [7–9]. It has become very effectively to perform a robust and reliable analysis the flow pattern phenomenon inside the turbine structure. Based on the literature, numerous studies have been conducted utilizing those tools in simulate the flow behavior in the draft tube of turbine and inspect the critical condition such as vortex rope and vibration [2, 10]. Researchers have studied pressure pulsation in Francis hydraulic turbine units and discussed the cavitation phenomenon problem [11, 12]. Jošt and Lipej (2011) built a 3D numerical model for a Francis turbine unit to predict vortex rope in the draft tube based on numerical flow analyses [13]. Two analyses were performed, i.e., without and with cavitation effects. Another study performed a numerical analysis of cavitation turbulent flow in a Francis turbine under partial load operation using the k–ω shear stress transport turbulence model in the Reynolds-averaged Navier-Stokes equations [14]. Qian et al. (2007) simulated 3D multiphase flow in a Francis turbine to calculate pressure pulsation in the spiral casing, draft tube, runner front, and guide vanes using fast Fourier transform [15]. The investigation of hydrodynamic effects of pressure fluctuation in the draft tube was studied by [16]. The cause of rotor-stator interaction simulation under partial load operation using analyzing 3D transient state turbulence flow simulation in a Francis tube was investigated. The 3D Navier–Stokes computational fluid dynamics (CFD) solver ANSYS CFX was used to analyze flow through a vertical Francis turbine with different loads in situ. Most lately, Luna-Ramírez et al. (2016) calculated pressure on the blades of a 200 MW Francis hydraulic turbine to locate the failure on the blade surface based on CFD [17]. Recently, an attempt conducted on the investigation of the local wave speed and bulk flow viscosity in Francis turbine by [18].
There are also several other studies conducted on the Francis turbine analysis through the advantages of the computational features [19–23].

Other researchers have discussed pressure pulsation in Kaplan hydraulic turbine units and methods to reduce the cavitation phenomenon problem. Ko and Kurosawa (2014) evaluated and presented cavitation performance at a specific speed for a 400 MW Kaplan turbine using a finite volume method to solve the Reynolds-averaged Navier-Stokes equations combined with the Reynolds stress model [24]. The modified Rayleigh-Plesset equation was used to model the collapse and growth of cavitation bubbles. Javadi and Nilsson (2014) adopted the renormalization group k–ε turbulence model combined with the Reynolds-averaged Navier-Stokes equations to analyze unsteady turbulent flow in a U9 Kaplan turbine model [25]. Analyses were performed on the fluctuation of pressure in the draft tube, unsteady flow behavior, and cohesive flow structures. Another investigation was carried out on the runner outlet flow of a Francis turbine model using a two-component particle image velocimetry system by [26]. The finding of the research proposes a particular shape to provide suitable optical access across the draft tube elbow. The characteristics of the flow pattern in a Francis turbine runner with a small opening valve using the Reynolds-averaged Navier-Stokes equations and the continuity equation was inspected by [27]. The 3D unsteady turbulence flow throughout the entire passage of the turbine was simulated numerically based on the k-ε two-equation turbulence model using the CFD software ANSYS Fluent. The finding of the study showed that a low-pressure zone expanded around the blades of the runner when the valve was closed, and velocity increased throughout the runner area. On the other hand, the effect of hydraulic instabilities on increasing the service lifetime of Francis and Kaplan turbines was accomplished by [1]. In particular, (Cai shui, 2012) built a mathematical model to study pressure distribution in the flow pattern inside the powerhouse of a hydropower station using a fluid dynamics method (CFD) to
determine the velocity distribution and pressure pattern distribution under three operating conditions: one-unit load, two-unit load, and full-load rejection [28]. The results of this study outlined good flow pattern at the inlet with steady water level fluctuation. Based on the extensive state-of-the-art studies on the pressure pulsation in the draft tube of Kaplan and Francis turbines depend on the same methods and analysis but use different models. Based on the comprehensive review article published by [2], several researchers have suggested changes in turbine design to reduce the cavitation phenomenon and increase turbine efficiency.

In the current work and for the best knowledge of the authors, constructing a submerged weir in three different heights (i.e., 1/6, 1/3, and 1/2 from the draft tube outlet height) was suggested to decrease the pressure variation in the draft tubes of turbine units. The main enthusiasm of establishing this research is owing to the main concept of broad-crested weir, that stabilize the flow through the open channel [29, 30]. Two reaction hydraulic turbines in two different embankment dams are selected as a real practical example to evaluate hydraulic performance. The hydraulic performances were including the pressure stability in the draft tubes of a vertical Kaplan turbine used in the Haditha Power Station and a vertical Francis turbine used in the Temenggor Power Station. A 3D numerical model with one turbine unit for each powerhouse is analyzed and simulated using ANSYS CFX software tool at different water levels (minimum to maximum). The discharge and hydraulic information obtained from sites. Flow velocity, pressure distribution, and shear wall distribution were determined under different loading cases using the k-ε turbulence model. The finite volume method was adopted, the physical properties and flow characteristics of water are defined. The simulation results of the models determined the characteristics of the turbines obtained by running the 3D turbine models in which changed according to the upstream, downstream water levels and discharge ranges. This study provides a foundation
for determining the hydraulic characteristic performance of reaction turbines to compare the two types of hydropower station: the Haditha powerhouse, which is an integral part of the dam body, and the Temenggor powerhouse, which is separate from the dam body. A safe and low-cost method for generating hydroelectric power can be identified.

2. Dams and Power Stations Description

2.1 Haditha Dam

The Haditha Dam is an earth-fill dam located on the Euphrates River, north of Haditha City in Iraq. This dam is over 9 km long and 57 m high. The Haditha Dam was built to generate hydroelectricity and regulate water for irrigation [31]. The power station in the Haditha Dam contains 6 vertical Kaplan turbines that can generate 660 MW of electricity. Figure 1a outlines the downstream flow of the Haditha Dam with 6 outlets opening into the spillway, 12 outlets opening into the power station, and 2 outlets opening into each turbine unit. Full details of the hydraulic characteristics belonging to the dam are tabulated in Table 1.

Figure 1.

Table 1.

2.2 Temenggor Dam

The Temenggor Dam is the third largest dam in Malaysia and the Temenggor Power Station is one of the largest hydroelectric power generation facilities in Malaysia. It is located on Sungai Perak, approximately 200 km northeast of Ipoh state [32]. This rock-fill dam has a height of 128 m and a crest length of 537 m. The Temenggor Power Station has four vertical Francis turbines with an installed capacity of 348 MW, it is considered a separate powerhouse. Figure 1b illustrates the Temenggor Dam with the eight outlets of the surface
downstream power station. Here also, Table 1 presents the hydraulic information required to build the models for the turbine units.

3. Methodology Overview

This Reaction turbines represent one of the largest hydraulic structures. Water pressure applies force on runner blades and this pressure decreases throughout a running turbine, this phenomenon presented in both Francis and Kaplan turbines. The runner and the blades of the turbines are fully immersed in water and they must be sufficiently strong to resist the operating pressure. The hydraulic power of the turbine units is given in formula (1) [33, 34]:

\[
P = \rho . Q . g . H . \eta
\]

where \( P \) is the water pressure, \( \rho \) is the water mass density, \( Q \) is water discharge, \( g \) is the gravity weight, \( H \) is the water head, and \( \eta \) is the efficiency.

The energy of the turbines \( E \) is defined as can be seen in equation (2) [35]:

\[
E = g . H_t = \left( \frac{p_1 - p_2}{\rho} \right) + \left( \frac{V_1^2 - V_2^2}{2} \right) + g . (z_1 - z_2) + \text{head loss}_{1-2}
\]

where \( H_t \) is the water head of turbine. Sections 1 and 2 are defined as the upstream and downstream measurements of the turbine, respectively. The determined behavior of the hydraulic turbine models is based on a dimensional analysis. Laboratory developments and model tests can guarantee hydraulic behavior and turbine efficiency [36]. The International Electrotechnical Commission standards 60193 and 60041 define all the simulation rules [37, 38].
The specific speed of a turbine based on these standards is defined in the following formula [34, 39]:

\[
n_{QE} = \frac{n \sqrt{Q}}{(gH_n)^{3/4}}
\]

(3)

The parameter \( n_{QE} \) is known as the specific speed of any type of turbines. Numerous statistical studies on reaction turbines have established a correlation between speed and net head for each type of turbine. The Schweiger and Gregory correlation formulas for Kaplan turbines are defined as [40]:

\[
\text{Kaplan } n_{QE} = \frac{2.294}{H_n^{0.486}}
\]

(4)

The Lugaresi and Mass correlation formulas for Francis turbines are presented as follows [41]:

\[
\text{Francis } n_{QE} = \frac{1.924}{H_n^{0.512}}
\]

(5)

The forgoing statistical formulas are used only for preliminary studies during the first trial to estimate the specific speed used to set the rotational speed of a turbine by applying Equation (3, 4 and 5) are used because there is no clear relationship exists among the head, flow rate, and the rotational speed of the turbine.

4. 3D Numerical Finite Element Turbine Model.

In this study, two different kind of turbines (e.g., Kaplan and Francis) were selected as a cases studies to be investigated. ANSYS CFX is used to simulate the 3D numerical finite
volume flow turbine models, including the runner with blades, and the shaft is defined as the submerged rotational body. The water field includes the two rectangular inlets for the Kaplan turbine and one circular inlet for the Francis turbine. The penstock, the spiral casing, the draft tube, and two rectangular outlets are defined. The boundary conditions include the discharge range, the operating head, the rotational speed of the turbines, and the effect of gravity. Latterly, the two models run with three submerged weirs suggestion at the downstream of turbine units. The Figure 2a and b display the dimensions of the units.

**Figure 2.**

The ANSYS-CFX software used for the simulation depending on the finite-volume method. The first step of calculation is study-state flow field, and the result of this step represent the initial condition for the next step. The flow simulation of the Francis turbine unit was employed by using several meshes to test the grid independence. After many iterations, the calculations reach to converge.

The second step in turbine modeling involves selection of a suitable finite volume mesh. The grid of the turbine is made using tetrahedral elements after performing several trials to determine the smallest possible aspect ratio under 150 and the minimum orthogonal over 0.15 in accordance to ANSYS-CFX code recommendations. Whereas, hexahedral elements analysis performed for the walls boundary layers. The final mesh used satisfy $y^+ < 200$ around the boundary wall to obtain the required pressure fluctuation, this is following the previous research conducted by [42]. The runner, guide vanes, and draft tube interactions were counted by using slip meshes. This mesh slipping toured each other in the sides of interface. But it is important to ensure that velocity components, pressure, and flow flux are harmonious after interpolation. The meshing details used in the Haditha Kaplan
turbine model and Temenggor Francis turbine are shown in Table 2. The number of elements and nodes used in Temenggor turbine meshing is higher than the Haditha turbine model because the Temenggor turbine unit is longer than the Haditha turbine unit. The Francis runner with 12 blades has several fine details represented by small elements, whereas the Kaplan runner includes only 6 blades with details larger than those of the Temenggor runner turbine. The outlet boundary condition, the relative static pressure, turbulent kinetic energy and its diffusion rate are prescribed; no-slip boundary condition is applied to the wall, and standard wall functions are applied to the region near the wall [43]. Figure 3 illustrates examples of the meshing details for the two turbines model components.

**Table 2.**

**Figure 3.**

The water flow through the draft tube was modeled using the incompressible continuity formulation and Reynolds time average. The mathematical explanation can be presented as follows [42]. The water flow continuity formula is:

\[
\frac{\partial u_j}{\partial x_j} = 0
\]  

(6)

And the momentum formula is:

\[
\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = \rho F_i - \frac{\partial P}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_i} - \rho \frac{\partial (u'_i u'_j)}{\partial x_j}
\]  

(7)

where;
\[-\rho u'_i u'_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_e \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \] (8)

The double formula of the k-\(\epsilon\) model is:

\[\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + 2\mu_t S_{ij} \frac{\partial u_i}{\partial x_j} + \rho \epsilon \] (9)

\[\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left( \alpha_\epsilon \mu_{eff} \frac{\partial k}{\partial x_j} \right) + 2C_{1\epsilon} \frac{\epsilon}{k} \nu \delta_{ij} \frac{\partial u_i}{\partial x_j} - C_{2\epsilon} \frac{\epsilon^2}{k} - R \] (10)

where \( \delta_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \), \( \mu_{eff} = \mu_t + \mu \), and \( \mu_t = C_\mu \frac{k^2}{\epsilon} \). \( R \) can be determined:

\[ R = \frac{C_\mu \rho \eta^2 \left( 1 - \frac{\eta}{\eta_o} \right) \epsilon^2}{1 + \beta \eta^3} \] (11)

where \( \eta = \frac{Sk}{\epsilon} \), \( \eta_o = 4.38 \), \( C_\mu = 0.0845 \), \( \beta = 0.012 \), \( C_{1\epsilon} = 1.42 \) “originally in the model procedure”, \( C_{2\epsilon} = 1.68 \), \( \alpha_k = 1.0 \), and \( \alpha_\epsilon = 0.769 \). Among these constants used in the turbulence model, a properly chosen value of \( C_{1\epsilon} \) is essential for improving the prediction of the pressure variation. In the present simulation, \( C_{1\epsilon} = 1.45 \) was selected based on the preliminary computations.

5. Description of Hydraulic Simulation

The utilization of the k-\(\epsilon\) turbulence model is very essential for describing the motion of turbulent flow through the turbine unit and particularly due to streamlines exhibit random motion near the turbine runner [25, 27, 44]. The 3D numerical model based on the finite volume method is used to distinguish among the unsteady incompressible flow inside the
turbine unit running under varying head and discharge ranges and to solve the Reynolds-averaged Navier-Stokes equations. All simulation applied according to computational fluid dynamic (CFD) approach. The hydraulic data (i.e., upstream and downstream water levels with discharge ranges) required to operate the model were collected from engineering reports belong to the inspected cases studies. Table 3 provides the hydraulic data of Haditha and Temenggor powerhouses, and specific speed (column 3) is calculated using Equations (4) and (5) during the first trial, respectively. Equation (1) is used to calculate the hydraulic input power using the efficiency of the Haditha Kaplan and Temenggor Francis turbines, with results of 71.1% and 83.4%, respectively.

**Table 3.**

The inlet velocity calculated by applying the continuity equation and rotational speed are verified using Equation (3). The Kaplan and Francis numerical 3D models were running by defining the gravity weight of water and the turbine components including the runner and the shaft as the submerged rotational bodies. The boundary conditions include inlet velocity (column 7 in Table 3), outlet pressure (1 atm), and the rotational speed of the turbine (column 8 in Table 3). The central axis turbine is defined as the rotational axis, and the time step interval is 500 steps/0.01 s to show the rotational motion of the turbine around its axis [45]. Several runs were performed for the turbine models while changing the rotational speed of the turbine to calculate the pressure at the turbine inlets, which provide the total head that presents the summation of elevation head, velocity head, and pressure head, as tabulated in Table 4.

6. Application and findings analysis
As a fact, turbines are usually designed locally in accordance to the dam conditions such as upstream and downstream water level, water discharge and various other hydraulic and hydrological factors. Also, turbines efficiency usually determined based on off-design situation especially for example the draft tube turbine. Hence, there is a probability in its operation and reliability. Thus, it is necessary to maintain an optimal dimensional of individual component according to several properties (e.g., nature of flow and water level fluctuation). The motivation of this research is to optimize the draft tube outlet with minimize pressure fluctuation and fluent water velocity. Practically, two types of operated turbines including Kaplan and Francis were demonstrated as an example for the inspected application. According to Table 4, the calculation of the total head at the inlet of the Haditha and Temenggor turbine units are presented.

The phenomenon of fluid pattern is highly complex and thus the advantage of computational fluid dynamic (CFD) can be applied to analyze fluid flow through the hydraulic mechanic’s concept. The procedure of CFD govern by dividing the fluid flow into finite volumes that can solved numerically. In this work, the computational process was obtained by running the turbine models with different water levels, as shown in Table 3. The input rotational speed of a turbine is used, and several runs are performed to determine the inlet pressure that provides the total head at the inlet closest to the upstream water level.

Table 4.

In addition, Table 4 outlines the error percentages between the total head at the turbine inlet estimated from running the numerical models with the upstream water levels. The results showed that the error percentages between the total head estimated at the turbine model inlets and upstream water level of Haditha and Temenggor dams equal to 2.16% and 0.3%, respectively. The reason of this minor percentages of errors due to the head loss varies
according to multiple forms of turbulent flow and the types of pipes (smooth or rough). In addition, the rotational speed of turbine varies according to two variables (the head and discharge) that obtained from data source which is not evidence by clear relationship. The comparison of total head at the turbine model inlets and the upstream water level indicates a practical procedure to find accuracy of the model results.

Figure 4a showed the velocity distribution in Haditha turbine model with the maximum upstream water level = 150.2 m from the entrance up to the spiral casing; it varies because of the change in the cross-sectional area based on the continuity equation. Maximum velocities occur around the turbine runner because of the contraction of the cross-sectional area, and flow is limited to the lower part of the draft tube and the outlet based on the amount of water flow. Figure 4b illustrated a constant velocity distribution at the penstock of the Temenggor turbine model with the maximum upstream water level = 248.42 m because of the constant cross-sectional area. Velocity gradually increases from the spiral casing to the turbine runner because of the contraction of the cross-sectional area. Velocity is consistently distributed across the draft tube because of the rotational motion and turbulent flow incident that occur after the turbine is running. The results showed that the maximum water velocity occurs at the location of the turbine runner, namely, 27.3 ms\(^{-1}\) for a discharge of 165.5 m\(^3\)s\(^{-1}\) in the Haditha turbine and 40 ms\(^{-1}\) for a discharge of 100 m\(^3\)s\(^{-1}\) in the Temenggor turbine. Although the discharge in the Haditha turbine is more than that in the Temenggor turbine, the cross-sectional area of the Haditha turbine is larger than that of the Temenggor turbine.

On the other hand, Figures 4c and d indicated the boundary pressure distributions in the Haditha and Temenggor turbine models, respectively. Here, the pressure distributions are proportional to the inverse of the velocity distribution based on the energy equation. The minimum pressure values are achieved after the turbines are running but do not reach
cavitation pressure [46]. The velocity and pressure results obtained in the current study are harmonized with the modeling results accomplished by [39, 44, 47].

**Figure 4.**

Figure 5a and b displayed the wall shear stress distributions of the Haditha and Temenggor turbine units running under the maximum head, respectively. The maximum wall shear stress values were 0.56 kPa and 1.5 kPa, which account for 0.1% and 0.25% of the maximum wall pressure value, respectively. Consequently, wall shear stress values are ignored in transporting boundary pressure from the turbine models to the dam models to determine the effect of a running turbine on the dynamic behavior of the embankment dams because their values are insignificant compared with the pressure values. Moreover, they depend on the pipe type (smooth or rough) and flow, which cannot be clearly identified.

**Figure 5.**

To have comprehensive details visualization of the attained results on the pressure variation, Table 5 presented. The maximum differences in pressure above and below the turbine runner exist at the following heads 144.9 m and 248.4 m for the Haditha and Temenggor turbine models, respectively. In which represents the best water level elevations to operate the turbines with the highest efficiency. Table outlined the pressure fluctuation results on both draft tube (i.e., left and right). The attained results were determined based on running Haditha and Temenggor turbine models (i) without a weir as a first case, and (ii) with three weir heights suggestions as a second case. The results showed a reasonable depth of the submerged weir that represent a 16.7% and 33.33% from the actual draft tube opening high for Haditha and Temenggor turbine. The optimal submerged weir heights determined for Haditha and Temenggor were 1.3 m and 1 m, in which reduced the variation of the pressure.
Table 5.

In more representable manner, the flow velocity (Figure 6 and 8) and flow pressure (Figure 7 and 9) on the left and right side of draft tubes were displayed graphically for Haditha and Temenggor turbine models, respectively. In figure 6, the flow velocity phenomenon of Haditha turbine was simulated in accordance two different components dam up-stream water level and submerged weir depth. Based on this Figure 6, it can be recognized that the velocity distribution in the both sides of the draft tube became more regular with increasing the submerged weir height. The maximum velocity range located in the turbine runner region effected by running turbine in minimum and maximum up-stream water levels with changing submerged weir height varies 7.6% and 4.1%, respectively. On other aspect, which is the water pressure value, Figure 7 indicated the water pressure fluctuation with the same regards dam up-stream water level and submerged weir depth. Based on the obtained results, it was evidenced that building 1.333 m submerged weirs in the downstream of Haditha turbine units with running the turbine model in minimum (129 m) and maximum upstream water level (150.2m) reduced of the pressure difference range between left and right side by 23% and 1% from the total head, respectively.

Figure 6

Figure 7

Figure 8
On the other case, Temenggor turbine was modeled to investigate the same two interesting measures water velocity and pressure. Figure 8 presented the influence of the up-stream water level and the proposed weir height to optimize the suitable steady water flow. The graphical visualization defined that the optimal velocity distribution attained when 1 m submerged weir built in the downstream of turbine outlet. The maximum velocity range located in the turbine runner region effected by running turbine in minimum (236.5m) and maximum (248.42m) upstream water levels with changing built submerged weir height varies 26.7% and 10.9%, respectively. Water flow pressure presentation was demonstrated in Figure 9. Revealing this figure conclude that running the turbine model in minimum (236.5m) and maximum upstream water level (248.42m) reduce of the range pressure difference between left and right side of 8.5% and 15.9% from the total head, respectively.

Based on the engineering prospective, cost of initiating a submerged weir is very important element that need to be considered by decision makers. Note that constructing the submerged weir is not an easy mission. It required carful investigation, inspection and optimized structure building. This is might be costly from the aspect of economic; yet, it is great proposition for the dam sustainability and stability, and hydropower operation.

In conclusion, results showed that reducing pressure fluctuation that provides a uniform velocity distribution according to the Reynolds-averaged Navier-Stokes equations at the draft tubes, and especially for the high discharges rates. This fluctuation is more visible in
the Temenggor Francis turbine than the Haditha Kaplan turbine due to the operation under high water level of the upstream (Table 1). The recommended submerged weir construction at the outlet draft tube was owing the uneven distribution of water flow and particularly in diffuser section. The findings indicated an essential solution that can be implemented practically in the powerhouse system operation in order to maintain a steady draft tube water flow with balanced water pressures on the both side outlet. The proposed submerged weir can be further extended to envisage the turbine outlet components instabilities of prototype off-design operation.

7. Conclusions

Draft tube is an essential component of the powerhouse system that located in the downstream part of any type of reaction turbines. The main goal of this draft tube is to convert the high impacted kinetic energy at the turbine runner outlet into pressure energy as possible gradually. However, the off-design of draft tube initiate a turbulent and fluctuated pressures during the online operation and hence solving this problem is extremely significant for dam body sustainability. This research provides a detailed analysis of the characteristics of the vertical Kaplan and Francis turbines influenced by changing water levels and discharge ranges. The actual pressure of the water flowing through the draft tube is computed qualitatively and presented numerically to verify the operation and performance of the turbines. The main determined pressures were obtained based on proposing a regulated submerged weir with different depth. The pressure distribution of unsteady flow was predicted through time-dependent turbine running. The change in upstream water level head provides a guideline for flow characteristics in the turbine units. The increase in head per unit discharge is the main guideline for generating hydroelectric power. The construction of the
power station far from the dam body increases the length of the waterway, which leads to increased energy loss. However, dam is protected from powerhouse vibration. The results of the proposed constructed submerged weir showed a very reasonable and reliable draft tube turbine operation with very regular pressures. The Kaplan draft tube turbine was suited with 1.33 m submerged weir height that comprises a 16.7% from the total opening. On the other hand, Francis draft tube turbine designed optimally with 1 m submerged weir height that represents 33.3% from the total tube open depth. The intended suggestion was very sufficient and feasible for minimizing pressure fluctuation in both sides of the draft tubes of the investigated cases studies. This research can be further extended to inspect the stresses component on the draft tube walls and propose a systematic hydropower operation based on the computed stresses.

**Conflicts of Interest:** The authors declare no conflict of interest.

**References**


7. Glatzel, T., Litterst, C., Cupelli, C., Lindemann, T., Moosmann, C., Niekrawietz, R.,


19. Trivedi, C., Cervantes, M.J., Gandhi, B.K., Dahlhaug, O.G. “Experimental and Numerical Studies for a High Head Francis Turbine at Several Operating Points”, *J.


42. Liu, S., Li, S., Wu, Y. “Pressure fluctuation prediction of a model Kaplan turbine by unsteady turbulent flow simulation”, *J. Fluids Eng.*, 131, 101102 (2009).


**Figures titles.**

**Figure 1:** a) Haditha dam with spillway and of power-station outlets, b) Downstream of Temenggor dam with outlets of the surface power-station.

**Figure 2:** a) 3D model of vertical Kaplan unit with dimensions, b) 3D model of vertical Francis unit with dimensions.

**Figure 3:** a) 3D model of vertical Kaplan unit with mesh details, b) FE-Meshing of Haditha turbine runner model, c) 3D model of vertical Francis unit with mesh details, and d) FE-Meshing of Temenggor turbine runner model.

**Figure 4:** a) Velocity vector of fluid in the flow passage at \( t = 4.80 \) s in Haditha turbine unit, b) Velocity flow lines at \( t = 4.80 \) s in Temenggor turbine unit, c) Pressure distribution of fluid in flow passage at \( t = 4.80 \) s in Haditha turbine unit, d) Pressure distribution of fluid in flow passage at \( t = 4.80 \) s in Temenggor turbine unit.

**Figure 5:** a) Wall shear distribution of Haditha turbine unit, b) Wall shear distribution of Temenggor turbine unit.

**Figure 6:** Flow velocity magnitude at Haditha turbine unit.
**Figure 7:** Flow pressure magnitude at Haditha turbine unit.
**Figure 8:** Flow velocity magnitude at Temenggor turbine unit.
**Figure 9:** Flow pressure magnitude at Temenggor turbine unit.

**Tables titles**

**Table 1:** The hydraulic information’s of Hadith and Temenggor Power Stations.
**Table 2:** The meshing details of Haditha Kaplan turbine unit and Temenggor Francis turbine unit.
**Table 3:** Hydraulic calculations for Haditha turbine and Temenggor turbine units.
**Table 4:** Total Head and percent of error calculation at the inlet of Haditha turbine unit and Temenggor turbine unit.
**Table 5:** Draft tube pressure results in different weir height suggestion in Haditha Kaplan turbine units and Temenggor Francis turbine unit.
List of figures

(a)

(b)
Figure 1

(a) Two inlets = 9.5m x 7m
Total length on unit = 63.35 m
Kaplan turbine
Runner diameter = 6.6 m
Draft tubes
Two outlets = 8m x 7 m

(b) Spiral case
Frances turbine
runner diameter = 4.4 m
Penstock
Two outlets = 3m x 3m
Total length on unit = 260 m

The two outlets with downstream submerged weir

Turbine unit dimensions without weir

Figure 2
Figure 3
Figure 4
Figure 5

(a) Weir depth = 4 m & U/S.W.L. = 129 m

(b) Weir depth = 4 m & U/S.W.L. = 150.2 m

(c) Weir depth = 2.667 m & U/S.W.L. = 129 m

(d) Weir depth = 2.667 m & U/S.W.L. = 150.2 m

(e) Weir depth = 1.333 m & U/S.W.L. = 129 m

(f) Weir depth = 1.333 m & U/S.W.L. = 150.2 m
(g) Weir depth = 0 m & U/S.W.L. = 129 m

(h) Weir depth = 0 m & U/S.W.L. = 150.2 m.

Figure 6
(a) Weir depth = 4 m & U/S.W.L. = 129 m
(b) Weir depth = 4 m & U/S.W.L. = 150.2 m
(c) Weir depth = 2.667 m & U/S.W.L. = 129 m
(d) Weir depth = 2.667 m & U/S.W.L. = 150.2 m
(e) Weir depth = 1.333 m & U/S.W.L. = 129 m
(f) Weir depth = 1.333 m & U/S.W.L. = 150.2 m
(g) Weir depth = 0 m & U/S.W.L. = 129 m

(h) Weir depth = 0 m & U/S.W.L. = 150.2 m.

Figure 7
Weir depth = 1.5 m & U/S.W.L. = 236.5 m

Weir depth = 1.5 m & U/S.W.L. = 248.42 m

Weir depth = 1 m & U/S.W.L. = 236.5 m

Weir depth = 1 m & U/S.W.L. = 248.42 m

Weir depth = 0.5 m & U/S.W.L. = 236.5 m

Weir depth = 0.5 m & U/S.W.L. = 248.42 m
(g) Weir depth = 0 m & U/S.W.L. = 236.5 m
(h) Weir depth = 0 m & U/S.W.L. = 248.42 m

Figure 8
(a) Weir depth = 1.5 m & U/S.W.L. = 236.5 m

(b) Weir depth = 1.5 m & U/S.W.L. = 248.42 m

(c) Weir depth = 1 m & U/S.W.L. = 236.5 m

(d) Weir depth = 1 m & U/S.W.L. = 248.42 m

(e) Weir depth = 0.5m & U/S.W.L. = 236.5 m

(f) Weir depth = 0.5 m & U/S.W.L. = 248.42 m
(g) Weir depth = 0 m & U/S.W.L. = 236.5 m

(h) Weir depth = 0 m & U/S.W.L. = 248.42 m

Figure 9
<table>
<thead>
<tr>
<th>Properties</th>
<th>Haditha Dam</th>
<th>Temenggor Dam</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of the dam</td>
<td>Earthfill</td>
<td>Rockfill</td>
</tr>
<tr>
<td>location</td>
<td>34° 12′ 25″ N, 42° 21′ 18″ E</td>
<td>5° 24′ 24″ N, 101° 18′ 4″ E</td>
</tr>
<tr>
<td>Power house type</td>
<td>Integral part of dam body</td>
<td>surface power house</td>
</tr>
<tr>
<td>Type of turbines</td>
<td>Vertical Kaplan</td>
<td>Vertical Francis</td>
</tr>
<tr>
<td>Number of units</td>
<td>6</td>
<td>4</td>
</tr>
<tr>
<td>Install capacity (MW)</td>
<td>6×110 = 600</td>
<td>4×87 = 348</td>
</tr>
<tr>
<td>Length of unit (m)</td>
<td>67.35</td>
<td>260</td>
</tr>
<tr>
<td>Penstock diameter (m)</td>
<td>The details shown in Figure</td>
<td>5.5</td>
</tr>
<tr>
<td>Maximum U/S.W. L (m)</td>
<td>150.2</td>
<td>248.42</td>
</tr>
<tr>
<td>Minimum U/S.W. L (m)</td>
<td>129</td>
<td>236.5</td>
</tr>
<tr>
<td>D/S.W.L. (m)</td>
<td>107.3</td>
<td>142</td>
</tr>
<tr>
<td>Maximum power</td>
<td>6×339 = 2034</td>
<td>4×100 = 400</td>
</tr>
<tr>
<td>mesh details</td>
<td>nodes</td>
<td>elements</td>
</tr>
<tr>
<td>-------------</td>
<td>-------</td>
<td>----------</td>
</tr>
<tr>
<td><strong>haditha kaplan</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>without weir</td>
<td>water</td>
<td>978583</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>1808946</td>
</tr>
<tr>
<td>weir height = 1.333 m</td>
<td>water</td>
<td>10371281</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>1835460</td>
</tr>
<tr>
<td>weir height = 2.667 m</td>
<td>water</td>
<td>10480064</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>1843668</td>
</tr>
<tr>
<td>weir height = 4 m</td>
<td>water</td>
<td>105932812</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>1844775</td>
</tr>
<tr>
<td><strong>temenggor francis</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>without weir</td>
<td>water</td>
<td>97242978</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>2278165</td>
</tr>
<tr>
<td>weir height = 0.5 m</td>
<td>water</td>
<td>10099238</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>2276995</td>
</tr>
<tr>
<td>weir height = 1 m</td>
<td>water</td>
<td>10099465</td>
</tr>
<tr>
<td></td>
<td>turbine</td>
<td>2275416</td>
</tr>
<tr>
<td>weir height</td>
<td>water</td>
<td>10137060</td>
</tr>
</tbody>
</table>
\[
\begin{aligned}
  &= 1.5 \text{ m} \\
  \text{turbine} &\quad 2277756 \\
  &\quad 506168 \\
  &\quad 13.94 \\
  &\quad 0.19626
\end{aligned}
\]

<table>
<thead>
<tr>
<th>No.</th>
<th>U/S.W. L (m)</th>
<th>Net head (m)</th>
<th>(N_{QE})</th>
<th>(Q) (m(^3)/s)</th>
<th>(P) (KW)</th>
<th>(V_{inlet}) (m/s)</th>
<th>(N) (rad/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Haditha turbine</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>129</td>
<td>18.5</td>
<td>0.6779</td>
<td>100</td>
<td>25807</td>
<td>1.5038</td>
<td>3.3520</td>
</tr>
<tr>
<td>2</td>
<td>134.3</td>
<td>25.5</td>
<td>0.5800</td>
<td>118</td>
<td>41975</td>
<td>1.7744</td>
<td>3.3586</td>
</tr>
<tr>
<td>3</td>
<td>139.6</td>
<td>32.5</td>
<td>0.5155</td>
<td>136</td>
<td>61658</td>
<td>2.0451</td>
<td>3.3353</td>
</tr>
<tr>
<td>4</td>
<td>144.9</td>
<td>39.5</td>
<td>0.4689</td>
<td>151</td>
<td>83204</td>
<td>2.2707</td>
<td>3.3326</td>
</tr>
<tr>
<td>5</td>
<td>150.2</td>
<td>46.5</td>
<td>0.4331</td>
<td>169.5</td>
<td>109949</td>
<td>2.5489</td>
<td>3.2839</td>
</tr>
<tr>
<td><strong>Temenggor turbine</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>236.5</td>
<td>94.50</td>
<td>0.2058</td>
<td>50.0</td>
<td>38658</td>
<td>2.1045</td>
<td>4.2671</td>
</tr>
<tr>
<td>2</td>
<td>239.48</td>
<td>97.48</td>
<td>0.2068</td>
<td>65.0</td>
<td>51840</td>
<td>2.7359</td>
<td>3.8491</td>
</tr>
<tr>
<td>3</td>
<td>242.46</td>
<td>100.46</td>
<td>0.2036</td>
<td>75.0</td>
<td>61644</td>
<td>3.1568</td>
<td>3.6091</td>
</tr>
<tr>
<td>4</td>
<td>245.44</td>
<td>103.44</td>
<td>0.2006</td>
<td>88.0</td>
<td>74474</td>
<td>3.7040</td>
<td>3.3551</td>
</tr>
<tr>
<td>5</td>
<td>248.42</td>
<td>106.42</td>
<td>0.1977</td>
<td>100.0</td>
<td>87068</td>
<td>4.2090</td>
<td>3.1687</td>
</tr>
<tr>
<td>1</td>
<td>236.5</td>
<td>94.50</td>
<td>0.2058</td>
<td>50.0</td>
<td>38658</td>
<td>2.1045</td>
<td>4.2671</td>
</tr>
</tbody>
</table>

\[H_t = \frac{v^2}{2g} + \frac{p}{\gamma} + Z\]

<table>
<thead>
<tr>
<th>No.</th>
<th>U/S.W. L (m)</th>
<th>(Q) (m(^3)/s)</th>
<th>(V_{inlet}) (m/s)</th>
<th>(P_{inlet}) (KPa)</th>
<th>(\frac{v^2}{2g}) (m)</th>
<th>(\frac{p}{\gamma}) (m)</th>
<th>(Z) (m)</th>
<th>(H_t = \frac{v^2}{2g} + \frac{p}{\gamma} + Z)</th>
<th>error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>129</td>
<td>100</td>
<td>1.504</td>
<td>232.61</td>
<td>0.115</td>
<td>23.71</td>
<td>105</td>
<td>129.08</td>
<td>0.06</td>
</tr>
</tbody>
</table>

Table 3

Table 4

**Haditha turbine**

<table>
<thead>
<tr>
<th>No.</th>
<th>U/S.W. L (m)</th>
<th>(Q) (m(^3)/s)</th>
<th>(V_{inlet}) (m/s)</th>
<th>(P_{inlet}) (KPa)</th>
<th>(\frac{v^2}{2g}) (m)</th>
<th>(\frac{p}{\gamma}) (m)</th>
<th>(Z) (m)</th>
<th>(H_t = \frac{v^2}{2g} + \frac{p}{\gamma} + Z)</th>
<th>error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>129</td>
<td>100</td>
<td>1.504</td>
<td>232.61</td>
<td>0.115</td>
<td>23.71</td>
<td>105</td>
<td>129.08</td>
<td>0.06</td>
</tr>
</tbody>
</table>
### Temenggor turbine

<table>
<thead>
<tr>
<th>No.</th>
<th>U/S.W. L (m)</th>
<th>Q (m³/s)</th>
<th>V_inlet (m/s)</th>
<th>P_inlet (KPa)</th>
<th>v²/2g (m)</th>
<th>p/γ (m)</th>
<th>Z (m)</th>
<th>H_t = v²/2g + p/γ + Z</th>
<th>error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>236.5</td>
<td>50</td>
<td>2.105</td>
<td>208.98</td>
<td>0.226</td>
<td>21.30</td>
<td>215</td>
<td>236.53</td>
<td>0.01</td>
</tr>
<tr>
<td>2</td>
<td>239.48</td>
<td>65</td>
<td>2.736</td>
<td>236.04</td>
<td>0.382</td>
<td>24.06</td>
<td>215</td>
<td>239.44</td>
<td>0.02</td>
</tr>
<tr>
<td>3</td>
<td>242.46</td>
<td>75</td>
<td>3.157</td>
<td>261.53</td>
<td>0.508</td>
<td>26.66</td>
<td>215</td>
<td>242.17</td>
<td>0.12</td>
</tr>
<tr>
<td>4</td>
<td>245.44</td>
<td>88</td>
<td>3.704</td>
<td>293.85</td>
<td>0.699</td>
<td>29.95</td>
<td>215</td>
<td>245.65</td>
<td>0.08</td>
</tr>
<tr>
<td>5</td>
<td>248.42</td>
<td>100</td>
<td>4.209</td>
<td>326.38</td>
<td>0.903</td>
<td>33.27</td>
<td>215</td>
<td>249.17</td>
<td>0.30</td>
</tr>
</tbody>
</table>

### Table 5

#### Haditha Kaplan turbine

the top surface of cone and draft tube connection

<table>
<thead>
<tr>
<th>without weir</th>
<th>U/S.W. L (m)</th>
<th>Head (m)</th>
<th>p (Pa)</th>
<th>p/γ (m)</th>
<th>p (Pa)</th>
<th>p/γ (m)</th>
<th>difference in pressure head between L &amp; R sides (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>150.2</td>
<td>46.5</td>
<td>-37526</td>
<td>-3.8253</td>
<td>-18781</td>
<td>-1.9145</td>
<td>1.9108</td>
</tr>
<tr>
<td></td>
<td>139.6</td>
<td>32.5</td>
<td>-14033</td>
<td>-1.4305</td>
<td>-2298</td>
<td>-0.2343</td>
<td>1.1962</td>
</tr>
<tr>
<td></td>
<td>129</td>
<td>18.5</td>
<td>-5679</td>
<td>-0.5789</td>
<td>-699</td>
<td>-0.0713</td>
<td>0.5076</td>
</tr>
<tr>
<td>Weir height = 1.333 m*</td>
<td>150.2</td>
<td>46.5</td>
<td>-12411</td>
<td>-1.2651</td>
<td>-5916</td>
<td>-0.6031</td>
<td>0.6621</td>
</tr>
<tr>
<td></td>
<td>139.6</td>
<td>32.5</td>
<td>-20255</td>
<td>-2.0647</td>
<td>-15899</td>
<td>-1.6207</td>
<td>0.4440</td>
</tr>
<tr>
<td></td>
<td>129</td>
<td>18.5</td>
<td>-5905</td>
<td>-0.6019</td>
<td>-6039</td>
<td>-0.6156</td>
<td>0.0137</td>
</tr>
<tr>
<td>Weir height = 2.667 m</td>
<td>150.2</td>
<td>46.5</td>
<td>33013</td>
<td>3.36524</td>
<td>-39305</td>
<td>-4.0066</td>
<td>7.3719</td>
</tr>
<tr>
<td></td>
<td>139.6</td>
<td>32.5</td>
<td>11481</td>
<td>1.17034</td>
<td>-23374</td>
<td>-2.3827</td>
<td>3.5530</td>
</tr>
<tr>
<td></td>
<td>129</td>
<td>18.5</td>
<td>-12293</td>
<td>-1.2531</td>
<td>-1850</td>
<td>-0.1886</td>
<td>1.0645</td>
</tr>
<tr>
<td>Weir height = 4 m</td>
<td>150.2</td>
<td>46.5</td>
<td>-42998</td>
<td>-4.3831</td>
<td>60855</td>
<td>6.20363</td>
<td>10.5864</td>
</tr>
<tr>
<td></td>
<td>139.6</td>
<td>32.5</td>
<td>-46978</td>
<td>-4.7888</td>
<td>66184</td>
<td>6.74659</td>
<td>11.5354</td>
</tr>
<tr>
<td></td>
<td>129</td>
<td>18.5</td>
<td>-22010</td>
<td>-2.2436</td>
<td>35709</td>
<td>3.64006</td>
<td>5.8837</td>
</tr>
</tbody>
</table>

#### Temenggor Francis turbine

the top surface of cone and draft tube connection

<table>
<thead>
<tr>
<th>U/S.W. L</th>
<th>Head</th>
<th>p (Pa)</th>
<th>p/γ (m)</th>
<th>p (Pa)</th>
<th>p/γ (m)</th>
<th>difference in pressure head between L &amp; R sides (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Without weir</td>
<td>(m)</td>
<td>(m)</td>
<td>-13295</td>
<td>-13.553</td>
<td>-26501</td>
<td>-2.7014</td>
</tr>
<tr>
<td>--------------</td>
<td>-----</td>
<td>-----</td>
<td>---------</td>
<td>---------</td>
<td>--------</td>
<td>---------</td>
</tr>
<tr>
<td></td>
<td>248.42</td>
<td>106.42</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>242.46</td>
<td>100.46</td>
<td></td>
<td>-79352</td>
<td>-8.0889</td>
<td>-69172</td>
</tr>
<tr>
<td></td>
<td>236.5</td>
<td>94.5</td>
<td>-34499</td>
<td>-3.5167</td>
<td>-2722</td>
<td>-0.2775</td>
</tr>
<tr>
<td>= 0.5 m</td>
<td>242.46</td>
<td>100.46</td>
<td></td>
<td>-49423</td>
<td>-5.038</td>
<td>-14575</td>
</tr>
<tr>
<td></td>
<td>236.5</td>
<td>94.5</td>
<td>-24250</td>
<td>-2.472</td>
<td>-7019</td>
<td>-0.7155</td>
</tr>
<tr>
<td>Weir height</td>
<td>248.42</td>
<td>106.42</td>
<td>-39805</td>
<td>-4.0576</td>
<td>-43601</td>
<td>-4.4445</td>
</tr>
<tr>
<td>= 1 m*</td>
<td>242.46</td>
<td>100.46</td>
<td>-22171</td>
<td>-2.26</td>
<td>-23700</td>
<td>-2.4159</td>
</tr>
<tr>
<td></td>
<td>236.5</td>
<td>94.5</td>
<td>-12194</td>
<td>-1.243</td>
<td>-5728</td>
<td>-0.5839</td>
</tr>
<tr>
<td>= 1.5m</td>
<td>242.46</td>
<td>100.46</td>
<td>10345</td>
<td>1.05454</td>
<td>-7566</td>
<td>-0.7713</td>
</tr>
<tr>
<td></td>
<td>236.5</td>
<td>94.5</td>
<td>2867</td>
<td>0.29225</td>
<td>-1572</td>
<td>-0.1602</td>
</tr>
</tbody>
</table>

* Indicate the optimal weir height.

**Ameen Mohammed Salih Ameen** is currently a Ph.D candidate at University of Malaya and assistance lecturer at the water resources Engineering Department, Faculty of Engineering, University of Baghdad. He obtained his Bachelor of Science in Irrigation and Drainage Engineering and Masters in Hydraulic Engineering from the University of Baghdad, Iraq. His field of specialization is water resources, hydraulics, and hydraulic structures. Ameen has authored and co-authored technical papers and reports in national and international publications and has served as a referee for papers published in local and international journals.

**Zainah Ibrahim** received her Bachelor degree in Civil Engineering, in 1990, from Middlesex Polytechnic, UK, Master degree in Structural Engineering, in 1994, from University of Liverpool, UK, and a Ph.D. degree in Structural Engineering, in 2007, from University of Sheffield, UK. She joined the University of Malaya, Malaysia, as Lecturer in 1995, where she is currently Associate Professor in the Department of Civil Engineering. She is the author or co-author of 50 peer refereed articles, and her research interests include structural health monitoring and system identification of civil structures, FEM, composite materials & structures, damage detection and rehabilitation of structures.

Associate Professor Dr. **Faridah Othman** is currently a lecturer at the Civil Engineering Dept, Faculty of Engineering, University of Malaya. She obtained her Bachelor of Science in Civil Engineering from the University of Missouri Kansas City, USA, Masters and Ph.D from the University of Newcastle Upon Tyne, UK. Her field of specialization is water resources, environmental hydraulics, river and water quality modeling, water distribution and sewerage network. She has been involved in modelling exercise for almost 20 years, and is actively involved in the study of the water resources management, river and surface water quality studies, performance and quality of water supply, sewerage network, flood mitigation works, and GIS application in water engineering. Dr Faridah has supervised a number of Ph.Ds, Masters as well as undergraduates students in their final year theses. She has authored and
co-authored several technical papers and reports in national and international publications, and has served as a referee for papers published in local and international journals. Dr. Zaher Mundher Yaseen is a lecturer in the faculty of Civil Engineering, Ton Duc Thang University, Ho Chi Minh City, Vietnam. Earned his master and Ph.D degrees from the National University of Malaysia. After obtaining the doctorate degree in the civil and structural engineering with major in water resources management, Dr. Yaseen had joined TDTU in 2017. His fields of specialization and interest include: practical application of soft computing in the field of water resources engineering that covers “i.e., hydrology, environment, climate, morphology, hydraulic”. He is authored and co-authored over 20 scientific researches published in high impacted international journals. In addition, he has performed as a reviewer for several international journals.