IRAQI
Academic Scientific Journals

UNIVERSITY OF ANBAR

Comparison among the Turbulence Models to Simulate Flow Pattern over ogee Spillway Case Study (Mandali dam in Iraq)

Ahmed Imad Rajaa[a], Ammar Hatem Kamel[a]

[a] *Dams & Water Resources Department, College of Engineering, University of Anbar, Ramadi-31001, Iraq*

ARTICLE INFO

ABSTRACT

The spillway is an important structure in the dams, used to pass the flood wave to the downstream safely. In the past decades, Computational fluid dynamics (CFD) has evolved. Research findings have shown the CFD models are a great alternative for laboratory models. According to it, the flow pattern over ogee spillways can be studied in a short time and without paying high expenses. Because the flow over the ogee spillway is turbulent and has a free surface, its properties are complex and often difficult to predict. Therefore, the present paper focuses on the study of turbulence closure models including the standard k-ε, RNG k–ε, k–ω, also, the large-eddy simulation (LES) models, to assess their performance to simulate flow over the spillway. The Flow-3d software with the volume-of-fluid (VOF) algorithm is applied to obtain the free surface for each turbulence model. The results of the analysis show that the LES model yielded better results when compared with laboratory results, while the turbulence closure models result of Reynold average Navier Stocks equations (RANS) was more stable, especially standard k-ε and RNG models.

1. Introduction

The basic purpose of spillways is conveying flood flow from the reservoir to downstream safely (Hong et al., 2018; Reese and Maynord, 1987). The ogee spillway has a curved-crest and is considered important species because of the ability to get rid of flood waves quickly and easily downstream (Ammar et al., 2016). Therefore, ogee spillways are one of the most investigated hydraulics structure studies. Numerical modeling has represented a revolution in most civil engineering disciplines (Tu et al., 2018). The available computer power and the continuous algorithm improvement helped to obtain accurate solutions for a wide range of problems including flow over the spillway.

Today, Numerical models provide hydraulic engineers with a tool to review the primary design of spillways to determine the problems of operation with lower cost, low time, and simple efforts (Salazar et al., 2020). Computation fluid dynamics (CFD) is a scientific approach used to study flow fluids by numerical simulations (Zeng et al., 2017). Runchal, (2012) reported that CFD is undergoing rapid evolution, and could be used in the future for designing spillway without conducting laboratory tests to study the overtopping of other hydraulic structure like dams (Al-Hashimi et al., 2017).

In CFD or numerical models, the continuity of mass and Navier Stocks equations are solved to represent the overflow spillway. In general, these equations are partial nonlinear differential equations and not easy to solve numerically or analytically. In other words, the Navier Stocks equations cannot be solved directly, because of the processing time limitations (its high computational expense) and memory of a computer. Therefore, there are several ways to solve the Navier Stocks equations. Unfortunately, all of these methods are an approximation. Furthermore, there are many models to represent the turbulence in Navier Stokes equations or so-called models of turbulence closure. The turbulence closure models are classified depending on: the differential equations number and the application of

their design to develop a relation between turbulence stresses and averaged rates. For example, Flow-3d software contains six turbulence closure models available, such as the one equation, standard k-ε, Renormalized group (RNG) k-ε, Prandtl mixing length, k–ω, and large-eddy simulation (LES) models (also called the problem of sub-grid scales (SGS) modeling).

The turbulence closure standard k-ε, RNG k–ε and k–ω models are utilized to solve the Reynold average Navier Stokes (RANS) equation when the number of unknown variables is quite than the number of the equations. In another word, when Reynolds average theory is placed in Navier Stokes equations, a turbulence analysis problem will occur, because the result is that the unknowns are more than equations. RANS models solve the Navier Stocks equations by taking the average properties of the flow (velocity and pressure). The alternative approach for RANS is called large-eddy simulations (LES) which was suggested as early as 1963 by Smagorinsk (Zhiyin, 2015). In this model, the motions of large eddies are computed directly in Navier Stocks equations and only small scale are modeled, resulting in a significant decrease in computational time.

After all, the turbulence closures models of the standard k-ε model (Aydin et al., 2020; Irzooki et al., 2016; Jahad et al., 2018; Tabbara et al., 2005; Wan et al., 2018; Yildiz et al., 2020) and RNG k-ε model (Al-Qadami et al., 2019; Bayon et al., 2016; Dastgheib et al., 2012; Dehdar-behbahani and Parsaie, 2016; Demeke et al., 2019; Fadaei and Barani, 2014; Ghanbari and Heidarnejad, 2020; Samadi-Boroujeni et al., 2019; Sarwar et al., 2020; Valero et al., 2018) had been used extensively by many researchers, and their results demonstrated high accuracy in simulating the overflow spillway. Tadayon and Ramamurthy, (2009) studied three different turbulence models like the standard k–ε, RNG k–ε, and Reynolds-stress-models (RSM) to analyze the characteristics of flow over the circular spillway. The RNG k–ε and RMS models enable us to obtain reasonably accurate properties for flow over the circular spillway. ALHASHIMI, (2013) conducted a numerical study on several models of

Corresponding author: Ahmed Imad Rajaa
E-mail address: eng.ahmed_ded@uoanbar.edu.iq

turbulence (standard k–ε, k–ω, RNG k–ε, Realizable k–ε) to simulate flow over ogee spillway by using 2D-Fluent software. This study was compared only turbulence closures of RANS equations and showed that the RNG k–ε turbulent modeled to increase the results accuracy for flow over ogee spillway. Hekmatzadeh et al., (2018) conducted an investigation study to evaluate several RANS turbulence models including (k–ε, RNG k–ε, k–ω, SST k–ω, and omega Reynolds-stress-model (RSM-ω) for simulation of flow on the stepped spillway. The comparison of numerical results with laboratory data indicated that RSM-ω and k–ω models, are superior to the rest of the turbulence models mentioned. Bayon et al., (2018) conducted studies on CFD codes of Flow-3D and OpenFOAM to simulate flow over nm stepped spillway, using diverse turbulence closures like the standard k–ε, RNG k–ε and Realizable k–ε, also the SST k–ω. According to results, the k–ε family has been confirmed to produce good results in the stepped spillway modeling. Imanian and Mohammadian,(2019) conducted a study using the OpenFOAM program to investigate the performance of flow over ogee spillway at water head significantly higher than the design head by using five turbulence closure models including standard k–ε, realizable k–ε, RNG k–ε, LRR and SST k–ω. The results indicate the agreement between the physical and numerical model is extremely good excluding for k–ε model. Macián-Pérez et al., (2020) compare the standard k–ε, RNG k–ε, and the k–ω turbulence models to accurately assess the hydraulic jump performance in the stilling basin. They reported, under certain conditions, the k–ω model mostly provides reliable approximations for specific flow conditions, like pressure gradients or flow near walls boundaries.

All the literature above discussed the use of RANS models to represent flow over the spillway, but the large-eddy simulation (LES) model usage was undiscussed. Therefore, In the present study, we discuss the difference between Reynold average Navier Stocks (RANS) (including slandered k-ε, RNG k–ε and k–ω models) and Large-eddy simulation (LES) models to simulate the flow pattern over the ogee spillway of Mandali dam. The present study uses the Flow-3d platform for comparison of velocity, free surface elevation, and pressure head among models.

2. Theory and governing equations

2.1. Flow equation

Recently, numerical models have become a tool for solving complex problems that are expensive in time and cost in the laboratory. (Dargahi, 2006). The CFD depends on the continuity and Navier Stocks equations to solve complicated flow problems over the spillway. These equations can be expressed for incompressible as follows:

$$\nabla U = 0 \tag{1}$$

$$\frac{\partial U}{\partial t} + U.\nabla U = -\frac{1}{\rho}\nabla P + v\nabla^2 U \dots \dots \dots \tag{2}$$

Where t is the time; U is the velocity; ρ is the density; p is the pressure; and ν is the dynamic viscosity (Liu and Zhang, 2019). It is worth mentioning that equations of Navier Stocks are set for four coupled; pressure and nonlinear-partial-differential equation of velocity for three components in 3d-space. directly solving the Navier Stocks equations is difficult, if not impossible (Ho and Riddette, 2010). To solve the Navier Stocks equations, the turbulence must be modeled to obtain approximate solutions.

2.2. Turbulent models

Turbulence modeling allows one to simulate the flow over spillways. Turbulence is defined as the unstable fluid motion that occurs when water collides with the walls of the spillway at high Reynolds numbers and appeared in the eddy formation with various sizes (Hajimirzaie and Ansar, 2020). All turbulent flows contain an enormous range of temporal and spatial vortices or scales with the range increasing with increasing Reynolds number. The turbulent flows in spillway modeling involve great difficulties, related to the instantaneous position determination, the accurate computation of the eddy dynamics, and configuration of the free surface (Carvalho et al., 2008).

The direct numerical simulations (DNS) is the most forceful way to solve turbulently in the unsteady 3d Navier Stokes directly, where all

motion scales are simulated. DNS model is the most natural approach, where almost 100% of the turbulence flows are resolved, additionally does not require turbulence closure or modeling as RANS. Unfortunate, the DNS is inappropriate for most flows, due to its high computational cost, where the domain of computation must be large enough to capture large scales and the mesh or grid must be sufficiently accurate to capture the smallest scales or eddies (Unnikrishnan et al., 2017). Moreover, the time step should be short to capture the quick time scales related to the smallest eddies. solving the DNS model at a high Reynolds number is difficult. Even if these could be applied to modeling for engineering applications, the amount of data would be overwhelming. To solve this problem, many models are developed to filter the equations by taken average properties of flow or resolving only intermediate to large scales, thus, another approach was created (Viti et al., 2018).

The LES model was developed by considering large scales are energetic more than small eddies (scales) in the computational domain, thus, the smallest scales in Navier Stocks equations are modeled, whilst the large eddies resolved directly (all details available in Liu and Zhang (2019) book). This technicality has helped to reduce computational expenses.

The most commonly applied approach to simulation the flows of turbulent is the Reynolds-averaged Navier Stocks (RANS) model. In the RANS statistical approach, the instantaneous values of pressure and velocity are described as the sum of an average and component of fluctuating (Rodi, 1993). The RANS models are taken as time-averaged to remove the small eddies, the result of that is a set of less costly equations containing extra variables (Hewakandamby, 2012).

2.2.1. Reynolds-averaged Navier Stocks (RANS) equations

Turbulent flows have the characteristic of presenting random changes in the pressure and velocity both in time and space. In 1895, a statistical approach was developed by Osborne Reynolds to depict the dynamics of the average flow. In the RANS model, the average of fluctuating velocity and pressure is taken as shown in Fig. 1.
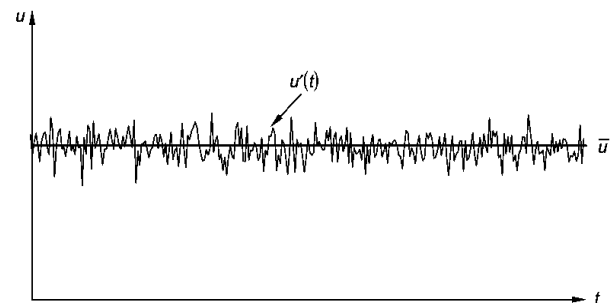

Fig. 1. Velocity against time for turbulent flows (Tu et al., 2018).

When terms of this average flow variables are substituted into continuity and Navier Stokes (momentum) equations and a time average is taken, the RANS equation is yielded (Andersson et al., 2011). They can be written as:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \tag{3}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2vS_{ij} + \tau_{ij}\right)\dots\dots \qquad \dots \text{(4)}$$

where x_j (j = 1, 2, 3) are cartesian-coordinates; u_j are the average velocity of cartesian components; S_{ij} denote to the strain rate tensor; and τ_{ij} denote to the Reynolds stress tensor.

The Reynolds stress (τ_{ij}) terms in RANS equation must be modeled to close system because the numbers of variables are more than number of equations (Davidson, 2015). These unknowns appear as correlations between the fluctuating quantities. Traditionally, called a closure problem, where turbulence closure models must be applied to approximate the Reynolds stress tensor (τ_{ij}). As a result, many models have appeared to lock the system and solve the RANS equations (Fleit et al., 2017). Based on the Boussinesq assumption, the Reynolds stress tensor (τ_{ij}) and strain rate tensor (S_{ij}) can be obtained as follows:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \text{ ....} \qquad \text{... (5)}$$

$$\tau_{ij} = 2v_T S_{ij} - \frac{2}{3}k\delta_{ij} \text{ ....} \qquad (6)$$

Here, v_T is the kinematic eddy viscosity; δ_{ij} is the Kronecker delta (δ_{ij}=1 for i=j; and δ_{ij}=0 for i≠j); and k= $\tau_{ij}/2$ is the turbulent kinetic energy. The kinematic eddy viscosity (v_T) can be found depending on the type of turbulence closure models:

2.2.1.1. Standard k-ε model

The k-ε model is one of the strongest turbulence closure models which is widely used in CFD modeling. The model was firstly suggested by (Jones and Launder, 1972) and later by (Launder and Sharma, 1974) supposing that rate between Reynolds effort and the average strain rate is the same in all directions. The k-ε model is considered a semi-empirical model for modeling the kinetic energy (k) transport equation of turbulence and turbulence dissipation rate (ε). The turbulence kinetic energy (k) and dissipation rate (ε) of turbulence for an incompressible fluid, are computed from equations (Eqs.7, 8), respectively.

$$\frac{\partial k}{\partial t} + \overline{u}_j\frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v+\frac{v_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - \varepsilon \text{ ....} \qquad (7)$$

$$\frac{\partial \epsilon}{\partial t} + \overline{u}_j\frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v+\frac{v_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1}\tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - C_{\varepsilon 2}(\varepsilon)\right) \qquad (8)$$

Thus; the turbulent viscosity is modeled as:

$$v_T = C_\mu \frac{k^2}{\varepsilon} \text{ ....} \qquad \text{... (9)}$$

Where $C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_μ are constants and having the following values 1.44, 1.92 ,and 0.09, respectively. σ_k and σ_ε are the Prandtl number of turbulent for k and ε and having the following values 1.0 and 1.3, respectively.

2.2.1.2. Renormalized group (RNG) k-ε model

To solve the stress tensor of Reynolds for equations (6), the RNG k-ε model was evolved based on transport equations of k-ε model (Eqs.7, 8, 9) with different coefficients. Plus, the variable ($C_{1\varepsilon}$) in equation (Eq.8) replaced by the variable ($C_{\varepsilon 2}^*$); and extracted from equation (Eq.10). the constants in equations (Eqs.7, 8, 10) $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ , μ_0 , β, σ_k and σ_ε have the following values 1.42, 1.68, 0.0845, 4.38, 0.012, 0.7194 and 0.7194, respectively (Yakhot et al., 1992).

$$C_{\varepsilon 2}^* = C_{2\varepsilon} + \frac{C_\mu \mu^3(1-\frac{\mu}{\mu_\circ})}{1+\beta\mu^3}; \quad \mu = \frac{k}{\varepsilon}\left(2S_{ij}S_{ij}\right)^{\frac{1}{2}} \qquad (10)$$

2.2.1.3. The k–ω model

Another communal two equations model to solve RANS is the k–ω model. The k–ω model was first established by (Kolmogorov, 1942), and later the (Saffman, 1970) and (Wilcox, 1988) have improved the model during the past four decades. It worth noting, the result of this model demonstrated high accuracy for an ample range of turbulent flows. The characteristic of k–ω model is compared with k–ε model is the high performance in regions with low turbulence when k and ε approach zero. Also, this model has good behavior in gradients of adverse pressure and separating of flow (Lee, 2018). The modeled k equation is

$$\frac{\partial k}{\partial t} + \overline{u}_j\frac{\partial k}{\partial x_j} = \tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\left(v+\frac{v_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] - \beta k\omega \text{ ...} \qquad \text{...... ....(11)}$$

; dissipation rate of specific ω is

$$\frac{\partial \omega}{\partial t} + \overline{u}_j\frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v+\frac{v_T}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] - \beta^*\omega^2 + \alpha\frac{\omega}{k}v_T\left(\tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j}\right) \text{ ...} \qquad (12)$$

; and Turbulent viscosity is

$$v_T = \frac{k}{\omega} \qquad (13)$$

Where σ_k, σ_ω are constants and having same values 0.5; α is constant 0.52; $\beta = \beta_0 f_\beta$ and $\beta^* = \beta_0^* f_\beta^*$. β_0, β^* are constants having the values 0.072, 0.09, respectively (Wilcox, 2006). Other constants can be found from the equations below (Wilcox, 2008):

$$f_\beta = \frac{1+70x_w}{1+80x_w}; x_w = \left|\frac{\tau_{ij}\tau_{jk}S_{ki}}{(\beta_0^*\omega)^3}\right|;$$

$$f_\beta^* = \begin{cases} 1, & x \leq 0 \\ \frac{1+680x_k^2}{1+80x_k^2}, & x > 0 \end{cases}; \text{and } x_k \equiv \frac{1}{\omega^3}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \qquad (14)$$

2.2.2. Large-eddy simulation (LES)

The LES model of turbulent flows is derived by performed filtering process on the Navier Stokes equations (Ghosal and Moin, 1995). LES model application to representation and calculation the turbulent flows consists of two detach steps (Zhiyin, 2015). First step, a filtering process applied on the Navier Stokes equations to eject small spatial eddies or scales. Fig. 2 illustrates the explicit filtering for LES model, which is different from the velocity average of RANS.
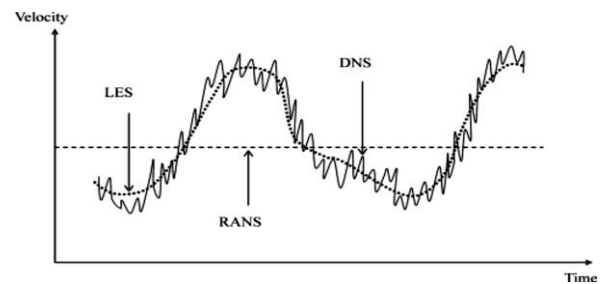

Fig. 2. Representation of resolved turbulence scales in a steady turbulent flow (Andersson et al., 2011)**.**

The filtered equations of the continuity and momentum (Navier Stokes equations) for incompressible flows can be expressed as follows:

$$\frac{\partial \overline{\overline{u}}_i}{\partial x_i} = 0 \text{ ....} \qquad \text{... ...(15)}$$

$$\frac{\partial \overline{\overline{u}}_i}{\partial t} + \overline{\overline{u}}_j\frac{\partial \overline{\overline{u}}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{\overline{P}}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2vS_{ij} + \tau_{ij}\right) \text{ ......} \qquad (16)$$

where $\overline{\overline{u}}_i$ is filtered velocity; p is filtered pressure ; S_{ij} is the resolved, or filtered tensor of scale-strain rate; and τ_{ij} is the unknown stress tensor of SGS (scale of sub-gird), also representing the scales that are smaller than the filter.

The second step is modelling the SGS stress tensor (unresolved scales) to solved closure problem in LES arises from the residual stress tensor, τ_{ij} as following:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \overline{\overline{u}}_i}{\partial x_j} + \frac{\partial \overline{\overline{u}}_j}{\partial x_i}\right) \text{ ....} \qquad \text{... (17)}$$

$$\tau_{ij} = \left(\overline{u_i u_j} - \overline{u}_i\overline{u}_j\right) = 2v_T S_{ij} + \frac{1}{3}\delta_{ij}\tau_{jj} \text{ ......} \qquad \text{....} \qquad \text{... (18)}$$

$$\overline{\overline{P}} = \overline{\overline{p}} + \frac{1}{3}\tau_{jj} \qquad (19)$$

Here, v_T is the kinematic eddy viscosity (Sagaut, 2006). The remaining problem in solving LES models, is determinizing the eddy viscosity (v_T) of SGS. Therefore, Smagorinsk was proposed model to determine the eddy viscosity of SGS as present in equation 20. where C_S is the constant of Smagorinsk which based on the type of a flow.

$$\begin{cases} v_T = (C_s\Delta)^2 S \\ S = \left(2S_{ij}S_{ij}\right)^{\frac{1}{2}} \\ \Delta = (\Delta x\Delta y\Delta z)^{\frac{1}{3}} \end{cases} \qquad (20)$$

3. Experiment data

In order to compare the models of turbulence and select the closest model for the laboratory results, the Mandali dam physical model has been selected as a case study. The Mandali dam is located in the east of Iraq in Harran wadi, at the Diyala governorate (33°47'4.98"N, 45°35'34.51"E).

The dam spillway is ogee type with 250 m long, 10 m height and 180 m.a.s.l crest level.

The physical model was tested by the Engineering Consultancy Bureau, University of Al-Mustansiriya (Al-Zubaidi et al., 2010). The concrete spillway was designed and constructed with scale 1:50 to investigate the hydraulic performance of Mandali spillway (see Fig. 3). It was 5 m in width and total height 0.3 m above stilling basin. Twenty-six-test applied on the model. The water level and pressure head had been measured for several discharges. Piezometer head gage was installed along the spillway surface with three-group. Each group consists eight points with accuracy nearest to 1mm.

Fig. 3. Physical model of Mandali spillway.

4. Numerical setup

Flow-3d is one of the popular commercial software for solving complex fluid modeling problems, which has been developed by Flow Science Company. The Flow-3d platform solves the 3D equations of Navier Stokes with the volume of fluid (VOF) method (Hirt and Nichols, 1981) to determine free surfaces. Additionally, the Flow-3d contains the fractional area volume obstacle representation (FAVOR) algorithm for exemplifying curved boundaries (or defining and inserting the model shape into the governing equations) (Hirt and Sicilian, 1985) and six turbulence models. In the present study, the Sketch-up (2019) program has been used to design the spillway model, with the constant concrete roughness of 1mm (Kim and Park, 2005) (see Fig. 4).
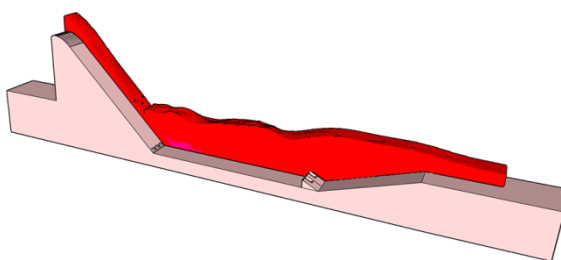

Fig. 4. Spillway model of Mandali dam (Red color indicates to water after simulation).

In Flow-3d, the mesh block can be defined as uniform or non-uniform (Flow-3d, 2016). Jacobsen and Olsen (2010) stated that the uniform mesh very well suits for the computation of the complex shapes models and free-surface flows, even for hydraulic jump, in addtion to that, it is an accurate and stable with time (Uçar and Kumcu, 2018). The present study, uniform mesh was imposed with 20 cm cells size in all directions of control volume, this size enable to increasing the stability and accuracy of results.

As mentioned earlier, the computation domain was assigned as a hexahedral mesh block in the cartesian-coordinates and there are six different boundaries (x_{min}, x_{max}, y_{min}, y_{max}, z_{min} and z_{max}) for mesh block

must be defined. Where x_{min} means the spillway upstream; x_{max} the downstream of spillway; and y_{min}, y_{max}, z_{min}, and z_{max} are right, left, bottom and top of spillway indirect the flow, respectively. The boundary conditions defined as presented in Table 1. Where, wall boundary denotes a non-slip condition in Limits (non-porous); and the outflow represents the amount of water out from spillway. The specified pressure condition is able to set at one or more boundaries of control volume, and it is an important and useful numerical tool when giving water levels and no discharge data available. The top of mesh block is air; therefore, it was defined as pressure specifier with the fluid fraction (F) = 0 (no water). The process of comparing pressure between physical and numerical models are extraordinarily difficult. It is also difficult to control the fluctuation of pressure on the surface of the spillway. Therefore, the upstream boundary condition (X-min) has been selected as water head and discharge values of Mandali physical model with four cases shown in Table 2.

Table 1. Boundary conditions that using in Flow-3d.

X-min	X-max	Y-min	Y-max	Z-min	Z-max
Inflow rate	Outflow	wall	wall	wall	Specified pressure

Table 2. Upstream boundary conditions values.

Run	1	2	3	4
Discharge (m³/s)	436	1179	1473	1803
Water Head (m)	10.98	11.78	12.04	12.32

5. Results and Discussion

Flow-3d V11.2 Version was used on MSI-GF63 computer. The simulation was performed for Twenty-five seconds for four cases and all the hydraulic information including velocity, pressure and water depth are available. Researchers suggest that a very smooth-grid should be used when simulation using the LES model, unlike RANS models which rely on the type of closure models selected, when implementing an appropriate mesh (Viti et al., 2018). However, when employing this model with the same grid size of RANS models (20 cm mesh size), provided positive performance especially in comparison with a laboratory model results. For velocity distribution at spillway crest, no difference was observed among the model values (see Fig. 5). The pressure and velocity results along the spillway can be seen in Fig. 6 for different turbulence models (RANS package and LES) and at 1802 m³/s discharge. Fig. 6 demonstrates that RANS models values provided similar pressure and velocity values, with a slight difference with k–ω closure model. Otherwise, the LES model displays lower values in terms of velocity, especially at the end part of chute spillway.
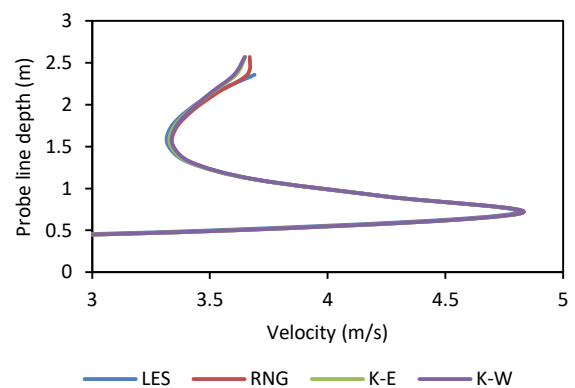

Fig. 5. Velocity distribution at crest spillway among turbulence models.

Pressure analysis is an important factor in knowing the probability of cavitation occurs over the spillway (Yusuf and Micovic, 2020). Therefore, the pressure head was analyzed and observed over time as shown in Fig. 7. As depicted Fig. 7, the LES model gives values higher than other models in all different discharge. On the other hand, the sharp change in the values of pressure head can be observed about 8 meters from the peak of the overflow (see Fig. 8). Sometimes, the high-frequency fluctuation of flows makes it difficult to choose values for comparison and therefore

decision-making. The standard k-ε and RNG turbulence models provide more stable values than other models (as results seen), helping to get correct approach values easily.
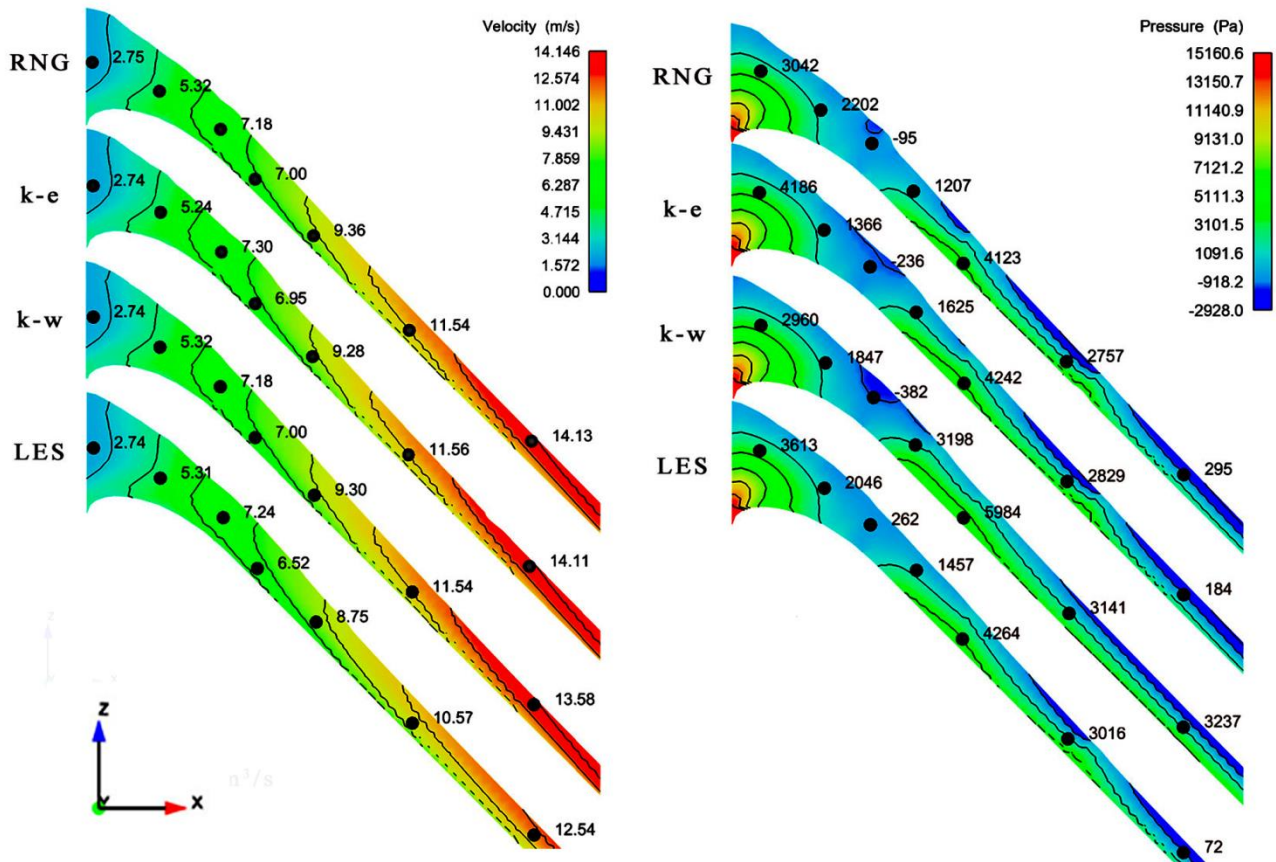

Fig. 6. Compression among turbulence models for velocity and pressure along chute spillway.
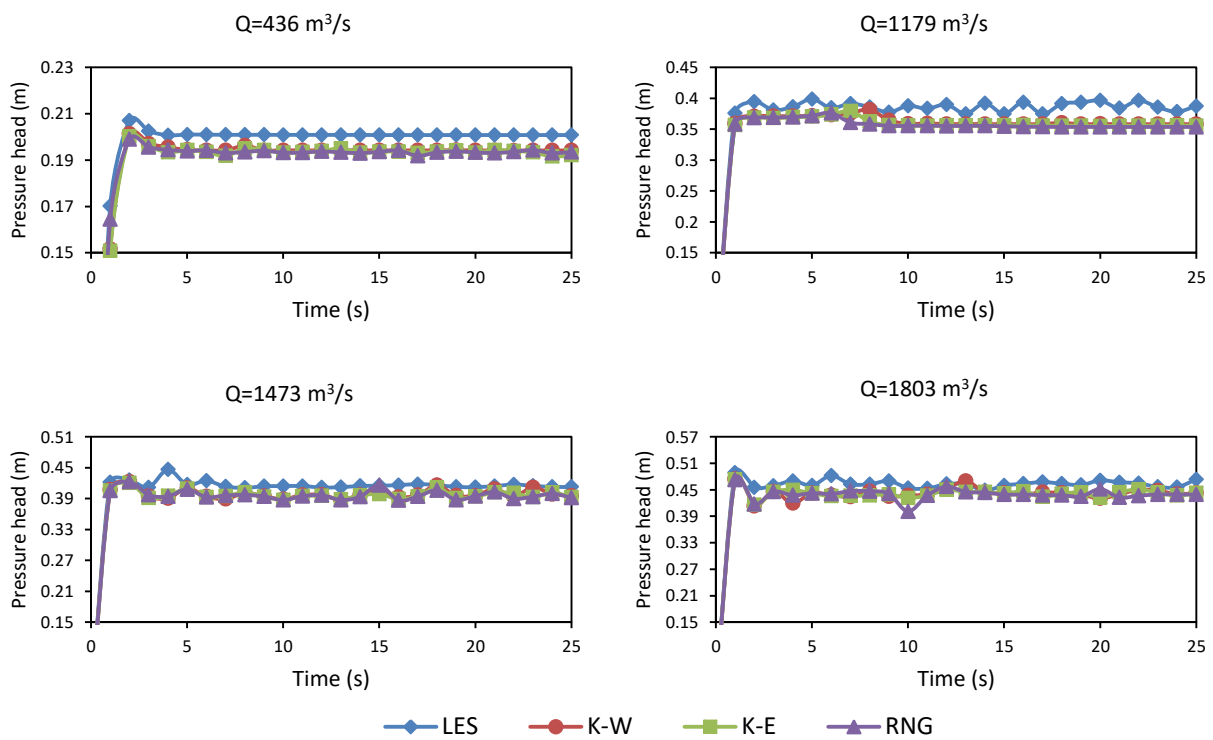

Fig. 7. Turbulence models over time for gauge point 3 (after 2 m from crest)
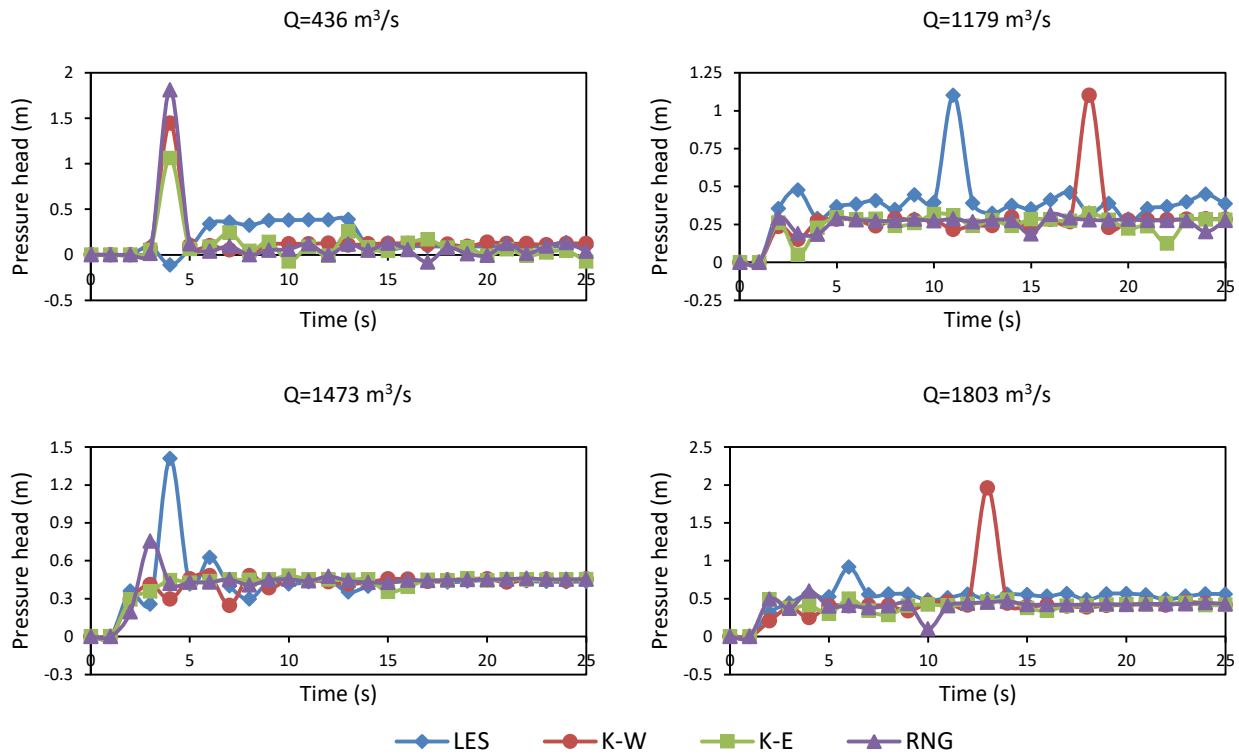
Q=436 m³/s   Q=1179 m³/s

Q=1473 m³/s   Q=1803 m³/s

LES   K-W   K-E   RNG

Fig. 8. Turbulence models over time for gauge point 8 (after 8 m from crest).

To investigation the turbulence models accuracy in predict pressure head of flow, the relative error percent (REP%) and root mean square error (RMSE) are using Eqs. (21) and (22), respectively (Azimi et al., 2019).

$$RMSE\% = 100 * \sqrt{\frac{1}{N}\sum_{i=1}^{N}\left(\frac{P_{measured} - P_{numerical}}{P_{measured}}\right)^2} \qquad (21)$$

$$REP\% = \frac{100}{N} * \sum_{i=1}^{N}\left|\frac{P_{measured} - P_{numerical}}{P_{measured}}\right| \qquad (22)$$

where $P_{(measured)}$ and $P_{(numerical)}$ are the pressure head of the laboratory and simulation results, respectively. The REP and RMSE of the experimental and simulated values are presented. According to Table 3, the LES model is the best model when compare with other models. The RMSE and REP of LES models for pressure head over spillway are equal to 1.92% and 1.59%, respectively. The RANS models provide nearly the same prognostication for pressure head with maximum difference average smaller than 1.7%.

Table 3. RMSE and REP values for the changes of the pressure head of physical and numerical models for RANS package and LES model.

Run	k–ω		RNG		k–ε		LES	
	RMSE	REP	RMSE	REP	RMSE	REP	RMSE	REP
1	3.57	2.96	3.73	3.06	3.71	3.04	3.49	2.91
2	1.73	1.53	1.73	1.54	1.73	1.54	1.52	1.32
3	1.19	0.95	1.18	0.96	1.19	0.96	1.12	0.89
4	1.61	1.3	1.52	1.25	1.56	1.26	1.53	1.25
Average	2.03	1.69	2.04	1.7	2.05	1.7	1.92	1.59

It is worth noting, that the k–ε and RNG models are an extremely close approach with the relative error percent (REP%) arrived at 1.7 for the laboratory and simulation results. This can be seen clearly through Fig. 9, which shows the distribution of free surface head along the spillway channel. Farhadi et al., (2018) stated that the k–ω model uses the same turbulence frequency of the LES model. Therefore, As shown in Fig. 10, the k–ω and LES models can be described as more disordered than k–ε and RNG models. The RNG model uses equations analogous to the equations for the standard k-ε model (Alfonsi, 2009). However, equation constants found through empirical observation in the standard k- ε model are derived explicitly in the RNG model (Babaali et al., 2015). In general, the standard k-ε and RNG turbulence models give very close values in all cases as results shown.
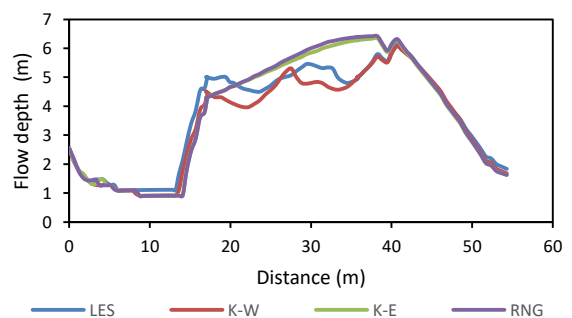
LES   K-W   K-E   RNG

Fig. 9. Comparison of the flow free surface changes among turbulence closure models.
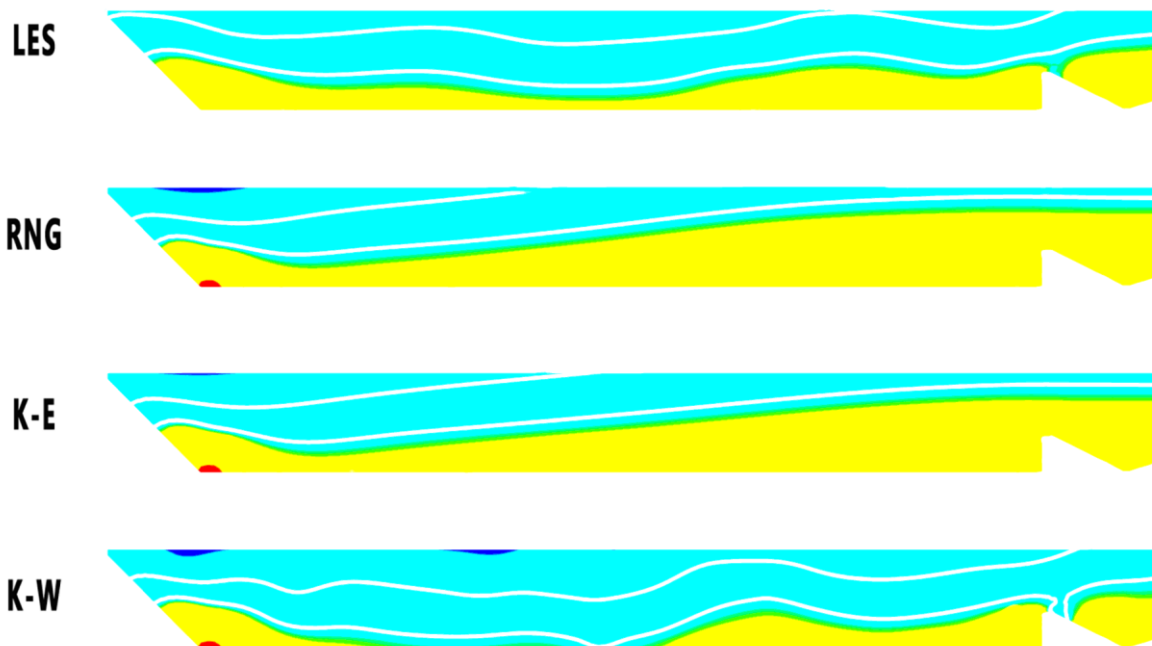
Fig. 10. Pressure distribution over stilling basin.

6. Conclusions

CFD modeling have evolved in the last two decades, and its results have become satisfactory compared to the physical model data. Within Eulerian methods, the RANS turbulence closure models are the most widely used in turbulence modeling over spillway from LES model, especially RNG k-ε model. Therefore, the present study focuses on studying the difference between LES and RANS of turbulence closure models in simulation flow pattern over spillways. Flow-3d has been used to modeling flow over spillway with four discharge values. The analysis of the results indicates that LES turbulence model is more accurate than the RANS turbulence closure models; where LES models capture eddies or scales in whole detail, whereas they are modeled in the RANS model. There is a slight difference when comparing the velocity and depth of water among the models over chute spillway. Otherwise, the pressure distribution and the flow depth values were unstable over a stilling basin for the k–ω and LES models. Finally, the results of the modelling show that LES model is a suitable tool to simulating the flow over ogee spillway.

7. Acknowledgement

8. Reference

[1] Al-Hashimi, S.A.M., Huda M. Madhloom, Rasul M. Khalaf, Thameen N. Nahi, Nadhir A. Al-Ansari, 2017. "Flow over Broad Crested Weirs: Comparison of 2D and 3D Models". J. Civ. Eng. Archit. 11, 769–779. https://doi.org/10.17265/1934-7359/2017.08.005

[2] Al-Qadami, E.H.H., Abdurrasheed, A.S., Mustaffa, Z., Yusof, K.W., Malek, M.A., Ghani, A.A., 2019. "Numerical modelling of flow characteristics over sharp crested triangular hump". Results Eng. 4, 100052. https://doi.org/10.1016/j.rineng.2019.100052

[3] Al-Zubaidi, R.Z., Khalaf, R.M., Salman, S., 2010. "Hydraulic Performance Of Mandali Dam Spillway In Iraq". J. Environ. Stud. [JES] 5, 35–48.

[4] Alfonsi, G., 2009. "Reynolds-Averaged Navier–Stokes Equations for Turbulence Modeling". Appl. Mech. Rev. 62. https://doi.org/10.1115/1.3124648

[5] Alhashimi, D.R.S.A.M., 2013. "CFD Modeling of Flow over Ogee Spillway by Using Different Turbulence Models".

[6] Ammar, Isam, Zainab, 2016. "Study the Effect of Spillway Locations on the Hydraulic Properties of Spillway". Ciência e Técnica Vitivinícola.

[7] Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., Van Wachem, B., 2011. "Computational fluid dynamics for engineers". Cambridge University Press.

[8] Aydin, M.C., Isik, E., Ulu, A.E., 2020. "Numerical modeling of spillway aerators in high-head dams". Appl. Water Sci. 10, 42. https://doi.org/10.1007/s13201-019-1126-2

[9] Azimi, H., Shabanlou, S., Kardar, S., 2019. "Flow field within rectangular lateral intakes in the subcritical flow regimes". Model. Earth Syst. Environ. 5, 421–430. https://doi.org/10.1007/s40808-018-0548-4

[10] Babaali, H., Shamsai, A., Vosoughifar, H., 2015. "Computational Modeling of the Hydraulic Jump in the Stilling Basin with Convergence Walls Using CFD Codes". Arab. J. Sci. Eng. 40, 381–395. https://doi.org/10.1007/s13369-014-1466-z

[11] Bayon, A., Toro, J.P., Bombardelli, F.A., Matos, J., López-Jiménez, P.A., 2018. "Influence of VOF technique, turbulence model and discretization scheme on the numerical simulation of the non-aerated, skimming flow in stepped spillways". J. Hydro-environment Res. 19, 137–149. https://doi.org/10.1016/j.jher.2017.10.002

[12] Bayon, A., Valero, D., García-Bartual, R., Vallés-Morán, F. José, López-Jiménez, P.A., 2016. "Performance assessment of OpenFOAM and FLOW-3D in the numerical modeling of a low Reynolds number hydraulic jump". Environ. Model. Softw. 80, 322–335. https://doi.org/10.1016/j.envsoft.2016.02.018

[13] Carvalho, R.F., Lemos, C.M., Ramos, C.M., 2008. "Numerical computation of the flow in hydraulic jump stilling basins". J. Hydraul. Res. 46, 739–752.

[14] Dargahi, B., 2006. "Experimental Study and 3D Numerical Simulations for a Free-Overflow Spillway". J. Hydraul. Eng. 132, 899–907. https://doi.org/10.1061/(ASCE)0733-9429(2006)132:9(899)

[15] Dastgheib, A., Niksokhan, M.H., Nowroozpour, A.R., 2012. "Comparing of flow pattern and energy dissipation over different forms of stepped spillway", in: World Environmental and Water Resources Congress 2012: Crossing Boundaries. pp. 1246–1259.

[16] Davidson, P.A., 2015. "Turbulence: an introduction for scientists and engineers. Oxford university press".

[17] Dehdar-behbahani, S., Parsaie, A., 2016. "Numerical modeling of flow pattern in dam spillway's guide wall". Case study: Balaroud dam, Iran. Alexandria Eng. J. 55, 467–473. https://doi.org/10.1016/j.aej.2016.01.006

[18] Demeke, G.K., Asfaw, D.H., Shiferaw, Y.S., 2019. "3D Hydrodynamic Modelling Enhances the Design of Tendaho Dam Spillway", Ethiopia. Water 11, 82. https://doi.org/10.3390/w11010082

[19] Fadaei, K.E., Barani, G.A., 2014. "Numerical simulation of flow over spillway based on the CFD method".

[20] Farhadi, A., Mayrhofer, A., Tritthart, M., Glas, M., Habersack, H., 2018. "Accuracy and comparison of standard k-ϵ with two variants of k-ω turbulence models in fluvial applications". Eng. Appl. Comput. Fluid Mech. 12, 216–235. https://doi.org/10.1080/19942060.2017.1393006

[21] Fleit, G., Baranya, S., Bihs, H., 2017. "CFD Modeling of Varied Flow Conditions Over an Ogee-Weir". Period. Polytech. Civ. Eng. https://doi.org/10.3311/PPci.10821

[22] Flow-3d, M., 2016. User Manual.

[23] Ghanbari, R., Heidarnejad, M., 2020." Experimental and numerical analysis of flow hydraulics in triangular and rectangular piano key weirs". Water Sci. 1–7. https://doi.org/10.1080/11104929.2020.1724649

[24] Ghosal, S., Moin, P., 1995. "The basic equations for the large eddy simulation of turbulent flows in complex geometry". J. Comput. Phys. 118, 24–37.

[25] Hajimirzaie, S.M., Ansar, M., 2020. "Review of Computational Fluid Dynamics: Applications in Water, Wastewater, and Stormwater Treatment" edited by Xiaofeng Liu and Jie ZhangASCE Press, Reston, VA; 2019; ISBN 9780784415313; 224 pp.;. Hydraul. Eng. 146, 07520001. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001724

[26] Hekmatzadeh, A.A., Papari, S., Amiri, S.M., 2018. "Investigation of Energy Dissipation on Various Configurations of Stepped Spillways Considering Several RANS Turbulence Models". Iran. J. Sci. Technol. Trans. Civ. Eng. 42, 97–109. https://doi.org/10.1007/s40996-017-0085-9

[27] Hewakandamby, B.N., 2012. "A first course in Fluid Mechanics for Engineers". Bookboon.

[28] Hirt, C.., Nichols, B.., 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries". J. Comput. Phys. 39, 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

[29] Hirt, C.W., Sicilian, J.M., 1985. "A porosity technique for the definition of obstacles in rectangular cell meshes", in: International Conference on Numerical Ship Hydrodynamics, 4th. Washington, DC: The National Academies Press, pp. 450–468.

[30] Ho, D.K.H., Riddette, K.M., 2010. "Application of computational fluid dynamics to evaluate hydraulic performance of spillways in australia". Aust. J. Civ. Eng. 6, 81–104. https://doi.org/10.1080/14488353.2010.11463946

[31] Hong, S.H., Sturm, T.W., González-Castro, J.A., 2018. Transitional Flow at Low-Head Ogee Spillway. J. Hydraul. Eng. 144, 04017062. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001398

[32] Imanian, H., Mohammadian, A., 2019. "Numerical simulation of flow over ogee crested spillways under high hydraulic head ratio". Eng. Appl. Comput. Fluid Mech. 13, 983–1000. https://doi.org/10.1080/19942060.2019.1661014

[33] Irzooki, R.H., Mohammed, J.R., Ameen, A.S., 2016. "Computational fluid dynamics modeling of flow over stepped spillway". Tikrit J. Eng. Sci. 23, 1–11.

[34] Jahad, U., Al-Ameri, R., Chua, L., Das, S., 2018. "Investigating the effects of geometry on the flow characteristics and energy dissipation of stepped spillway using two-dimensional flow modelling". Department of Civil and Environmental Engineering, Faculty of Engineering …, pp. 289–296.

[35] Jones, W.P., Launder, B.E., 1972. The prediction of laminarization with a two-equation model of turbulence. Int. J. Heat Mass Transf. 15, 301–314.

[36] Kim, D.G., Park, J.H., 2005. "Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model". KSCE J. Civ. Eng. 9, 161–169. https://doi.org/10.1007/BF02829067

[37] Kolmogorov, A.N., 1942. "Equations of motion of an incompressible turbulent fluid". Izv Akad Nauk SSSR Ser Phys 6, 56–58.

[38] Launder, B.E., Sharma, B.I., 1974. "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc". Lett. heat mass Transf. 1, 131–137.

[39] Lee, C.-H., 2018. "Rough boundary treatment method for the shear-stress transport k - ω model". Eng. Appl. Comput. Fluid Mech. 12, 261–269. https://doi.org/10.1080/19942060.2017.1410497

[40] Liu, X., Zhang, J., 2019. "Computational Fluid Dynamics, ASCE Publications". American Society of Civil Engineers, Reston, VA. https://doi.org/10.1061/9780784415313

[41] Macián-Pérez, J.F., García-Bartual, R., Huber, B., Bayon, A., Vallés-Morán, F.J., 2020. "Analysis of the Flow in a Typified USBR II Stilling Basin through a Numerical and Physical Modeling Approach". Water 12, 227. https://doi.org/10.3390/w12010227

[42] Reese, A.J., Maynord, S.T., 1987. "Design of Spillway Crests". J. Hydraul. Eng. 113, 476–490. https://doi.org/10.1061/(ASCE)0733-9429(1987)113:4(476)

[43] Rodi, W., 1993. "Turbulence models and their application in hydraulics", IAHR Monograph. Balkema, Rotterdam, Brookf.

[44] Runchal, A.K., 2012. The future of CFD and the CFD of the future. Comput. Therm. Sci. An Int. J. 4.

[45] Saffman, P.G., 1970. "A model for inhomogeneous turbulent flow". Proc. R. Soc. London. A. Math. Phys. Sci. 317, 417–433. https://doi.org/10.1098/rspa.1970.0125

[46] Sagaut, P., 2006. "Large eddy simulation for incompressible flows: an introduction". Springer Science & Business Media.

[47] Salazar, F., San-Mauro, J., Celigueta, M.Á., Oñate, E., 2020. "Shockwaves in spillways with the particle finite element method". Comput. Part. Mech. 7, 87–99. https://doi.org/10.1007/s40571-019-00252-1

[48] Samadi-Boroujeni, H., Abbasi, S., Altaee, A., Fattahi-Nafchi, R., 2019. "Numerical and Physical Modeling of the Effect of Roughness Height on Cavitation Index in Chute Spillways". Int. J. Civ. Eng. 1–12.

[49] Sarwar, M.K., Ahmad, I., Chaudary, Z.A., Mughal, H.-U.-R., 2020. "Experimental and numerical studies on orifice spillway aerator of Bunji Dam". J. Chinese Inst. Eng. 43, 27–36. https://doi.org/10.1080/02533839.2019.1676652

[50] Tabbara, M., Chatila, J., Awwad, R., 2005. "Computational simulation of flow over stepped spillways". Comput. Struct. 83, 2215–2224. https://doi.org/10.1016/j.compstruc.2005.04.005

[51] Tadayon, R., Ramamurthy, A.S., 2009. "Turbulence Modeling of Flows over Circular Spillways". J. Irrig. Drain. Eng. 135, 493–498. https://doi.org/10.1061/(ASCE)IR.1943-4774.0000012

[52] Tu, J., Yeoh, G.H., Liu, C., 2018. "Computational fluid dynamics: a practical approach". Butterworth-Heinemann.

[53] Uçar, M., Kumcu, Ş.Y., 2018. "Comparison Of Physical Modeling And Cfd Simulation Of Flow Over Spillway In The Arkun Dam".

[54] Unnikrishnan, U., Wang, X., Yang, S., Yang, V., 2017. "Subgrid Scale Modeling of the Equation of State for Turbulent Flows under Supercritical Conditions", in: 53rd AIAA/SAE/ASEE Joint Propulsion Conference. American Institute of Aeronautics and Astronautics, Reston, Virginia, p. 4855. https://doi.org/10.2514/6.2017-4855

[55] Valero, D., Bung, D.B., Crookston, B.M., 2018. "Energy Dissipation of a Type III Basin under Design and Adverse Conditions for Stepped and Smooth Spillways". J. Hydraul. Eng. 144, 04018036. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001482

[56] Viti, N., Valero, D., Gualtieri, C., 2018. "Numerical Simulation of Hydraulic Jumps. Part 2: Recent Results and Future Outlook". Water 11, 28. https://doi.org/10.3390/w11010028

[57] Wan, W., Liu, B., Raza, A., 2018. "Numerical Prediction and Risk Analysis of Hydraulic Cavitation Damage in a High-Speed-Flow Spillway". Shock Vib. 2018, 1–11. https://doi.org/10.1155/2018/1817307

[58] Wilcox, D.C., 2008. "Formulation of the k-w Turbulence Model Revisited". AIAA J. 46, 2823–2838. https://doi.org/10.2514/1.36541

[59] Wilcox, D.C., 2006. "Reynolds Averaging. Turbul. Model. CFD", Third Ed. DCW Ind. Inc., La Cañada, Calif. 34–38.

[60] Wilcox, D.C., 1988. "Reassessment of the scale-determining equation for advanced turbulence models". AIAA J. 26, 1299–1310. https://doi.org/10.2514/3.10041

[61] Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., Speziale, C.G., 1992. Development of turbulence models for shear flows by a double expansion technique. Phys. Fluids A Fluid Dyn. 4, 1510–1520. https://doi.org/10.1063/1.858424

[62] Yildiz, A., Yarar, A., Kumcu, S.Y., Marti, A.I., 2020. "Numerical and ANFIS modeling of flow over an ogee-crested spillway". Appl. Water Sci. 10, 90. https://doi.org/10.1007/s13201-020-1177-4

[63] Yusuf, F., Micovic, Z., 2020. "Prototype-Scale Investigation of Spillway Cavitation Damage and Numerical Modeling of Mitigation Options". J. Hydraul. Eng. 146, 04019057. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001671

[64] Zeng, J., Zhang, L., Ansar, M., Damisse, E., González-Castro, J.A., 2017. "Applications of Computational Fluid Dynamics to Flow Ratings at Prototype Spillways and Weirs. II: Framework for Planning, Data Assessment", and Flow Rating. J. Irrig. Drain. Eng. 143, 04016073. https://doi.org/10.1061/(ASCE)IR.1943-4774.0001113

[65] Zhiyin, Y., 2015. "Large-eddy simulation: Past, present and the future". Chinese J. Aeronaut. 28, 11–24.