Environment and Natural Resources Journal

Volume 23
Issue 2
2025

# Numerical Investigation of Hydraulic Performance in Makhool Spillway Dam

**Fatima A. Sadiq[1*], Haitham A. Hussein[2], and Mohd R. Rozainy Zainol[3]**

*[1]Department of Civil Engineering, Al-Nahrain University, Baghdad, Iraq*
*[2]Civil Engineering Department, College of Engineering, Al-Nahrain University, Baghdad, Iraq*
*[3]School of Civil Engineering, Engineering Campus, University Sains Malaysia, 14300 Nibong Tebal, Penang, Malaysia*

## ARTICLE INFO

Received: 22 Jun 2024
Received in revised: 11 Dec 2024
Accepted: 24 Dec 2024
Published online: 27 Feb 2025
DOI: 10.32526/ennrj/23/20240178

**Keywords:**
Spillway Dam/ Experimental Validation/ CFD Model/ Numerical modeling/ FLOW3D software/ Makhool Dam

*** Corresponding author:**
E-mail: fatima.mciv22@ced.nahrainuniv.edu.iq

## ABSTRACT

Dams are critical hydraulic structures, and analyzing their hydraulic properties is essential for ensuring their functionality. While experimental tests have traditionally been used to evaluate the performance of dams, but now (CFD) software, such as among this the Flow 3D, now offers a reliable alternative or complement to physical models, ensuring accuracy while reducing time and effort. This study investigates Makhools' Spillway Dam in Iraq, one of the country's most significant hydraulic projects. A 3D numerical model of the dam was developed to assess its operation and performance. Its properties were analyzed and validated by comparing the results with physical model data, focusing on key hydraulic parameters such as velocity and water flow depth. The velocity results closely align with the physical model data, with only a minor variation in flow depth, which remains within an acceptable range. The RMSE value for velocity was below 5%, while the difference in the flow depth was approximately 3.63%, indicating a strong correlation between the numerical and physical models. These findings confirm that advanced numerical modeling techniques can effectively complement or serve as a reliable alternative to real-world studies.

## 1. INTRODUCTION

The spillway is structure of the hydroelectric power plant destined to extravagant the exceptional floods to protect the dam against overtopping. In other words, it is the structure used to avoid exceeding the maximum water level of dam reservoir (Pereira, 2020). These structures, which are built with and without control, are one of the most important hydraulic elements of dams (Pataki and Cahill, 1985). The type of spillway depends on the general arrangement of the works of a hydroelectric power plant, which in turn depends on the topographical and geological characteristics of the site. The spillway generally has five distinct elements: approach channel, control structure, chute channel, energy dissipate, and exit channel (Pereira, 2020). It must be strong enough because it exposes the safety of the dam to danger if the amount of water is greater than its designed limit, especially in the event of a flood (Kocaer and Yarar, 2020). The United States Bureau of Reclamation (USBR) and the United States Army Crop of Engineers (USACE) conducted several model tests and investigations to develop a general formula for spillway hydraulic properties such as discharge coefficients of hydraulic pressure, crest, and energy-dispersion efficiency of the damping basin (Rajaa, 2020). The Physical models simulate and analyze the complexity of flow distribution over a spillway (5) (Erpicum et al., 2016). Creating a physical model is beneficial for understanding the characteristics of a dam and ogee spillway. Physical modeling allows for analyzing hydraulic behavior, such as discharge curves, flow elevation, flow velocity, flow direction patterns, and hydrostatic pressure (Manogaran et al., 2022). It helps in evaluating the effectiveness of energy-dissipating structures, such as stilling basins

**Citation:** Sadiq FA, Hussein HA, Zainol MRR. Numerical investigation of hydraulic performance in Makhool Spillway Dam. Environ. Nat. Resour. J. 2025;23(2):142-150. (https://doi.org/10.32526/ennrj/23/20240178)

and stepped spillways, and also in minimizing kinetic energy and preventing scour and erosion (Zahari et al., 2022). For more than 100 years, laboratory modeling has been an effective tool for investigations of hydraulic characteristics (Rajaa, 2020). But from previous experiences with physical models, creating a physical model to understand the characteristics of a dam has several disadvantages. Firstly, physical modeling can be time-consuming and expensive, requiring the construction of scaled-down replicas of the structures (Manogaran et al., 2022). Secondly, there is a scale effect in physical modeling, which means that the results obtained from the model may not accurately represent the behavior of the actual structures (i.e., reduced prototype scale according to the law of similarly of Froude number can cause errors in the results) (Kote and Nangare, 2019). This can lead to discrepancies between the measured data and the prototype, affecting the accuracy of the design (Saneie et al., 2016). Errors that may happen in drainage ditches in any design mistake can lead to major and irreversible consequences, studies and experimental studies have recently been seen as supporting parts (Kocaer and Yarar, 2020). Therefore, there has been a surge in investigative studies employing numerical models. These models allow for the examination and simulation of the hydrodynamic performance of spillways with sensible cost and time, thus contributing to recent advances in the field. CFD (computational fluid dynamics) modeling is a numerical method for solving flow equations such as mass and momentum (3D Navier Stokes equations). Recently, there have been many CFD software designed for fluid flow modeling such as Autodesk CFD and CFX, Flow-3D, Sim Scale, Power Flow, Open FOAM, COMSOL Multiphysics, Ansys Fluent, etc. (Li et al., 2011; Samadi et al., 2020; Jahad et al., 2018; Yusuf and Micovic, 2020; Rahimzadeh et al., 2012; Banerjee, 2018; Dargahi, 2006; Sartaj et al., 2006; Hekmatzadeh et al., 2018). Flow-3D software system has been used to model and simulate flow patterns and the flow over dam spillways for many discharges with excellent accuracy, suitable cost, and less time to study more forecasts compared to physical models (Rajaa, 2020). It is considered the best for simulating a water drainage channel. Many studies have been conducted on it and the results have been very good. Several researchers have tried to predict flow characteristics on spillways using numerical and physical models (Rajaa, 2020). Savage and Johnson (2001) gave a numerical study in using the CFD systems of the Flow 3D program discretized with hexahedral cells to compute the flow in order to compare the numerical results with physical model data as presented in the experimental study.

In this research, a simulation was performed on the spillway of Makhool Dam, which was constructed on a scale of 1:50. The aim of the present work is to analyze the differences between experimental and numerical models. The velocity and flow depth results in the experimental model were compared with Makhool spillway dam 3D numerical model (case study) in order to determine the compatibility of results. The VOF method was used in a numerical model to predict two phase flows with turbulence closures, including standard K-e and RNG K-e model. The results are compared, and it was observed that there was a very excellent agreement. It was proven that Flow 3D software is effective in analyzing the hydraulic properties at a lower cost and in less time compared to the physical model.

## 2. METHODOLOGY

Makhool Dam is one of the under-construction projects on the Tigris River in Iraq. It is located in the northern part of the city of Baiji between the two longitudes (43°12'08"-43°38'48") east, and two latitudes (35°09'21"-35°41'19") north, on the Makhool mountain range in the Salah al-Din Governorate of Iraq as shown in (Figure 1). The Bottom Spillway Discharge (Gated) has a discharging capacity of 15297 cubic meters per second (MoWR, 2020).

### 2.1 Experimental study

The Experimental studies were acted on the physical model that was established inside the hydraulic laboratory, College of Engineering at Al-Mustansiriya University by (Abbas, 2002), as shown in (Figure 2). The model was created on a scale of 1:50 and the dimensions were represented according to the attached diagram (Figure 3). Hydraulic parameters of the dam such as downstream flow depth and Velocity are obtained by performing Eighty-one experiments on the models in specific places in the dam body (Abbas, 2002). The required results were measured at different times and operating conditions.

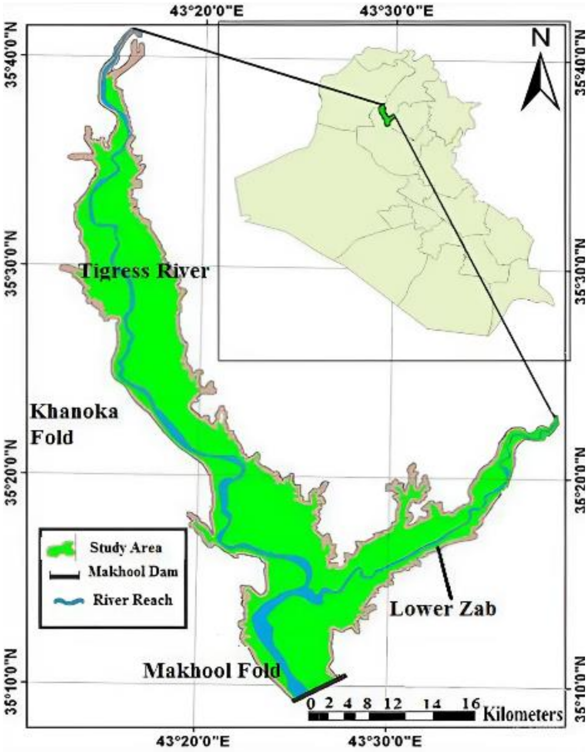


**Figure 1.** Makhool Dam location (Engineering Consulting Bureau of Anbar University, 2020)

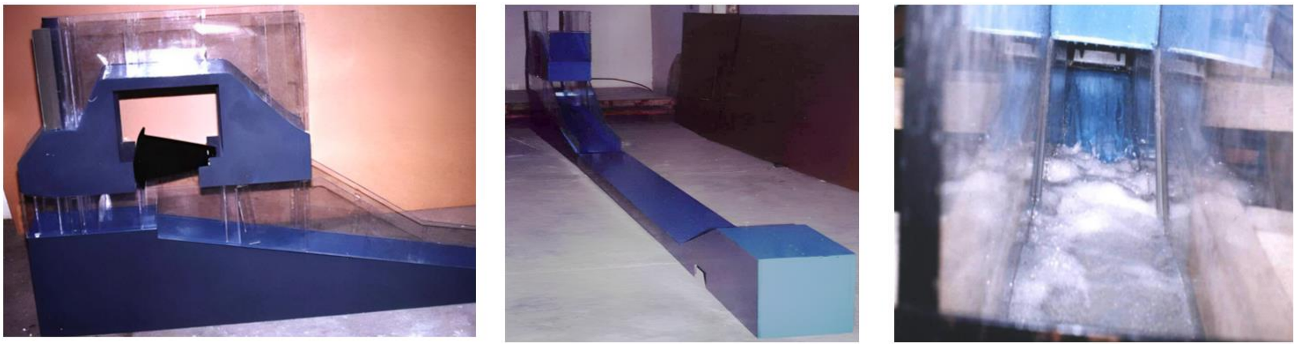

**Figure 2.** Physical model of for Makhool spillway (Abbas, 2002; Amen, 2002)

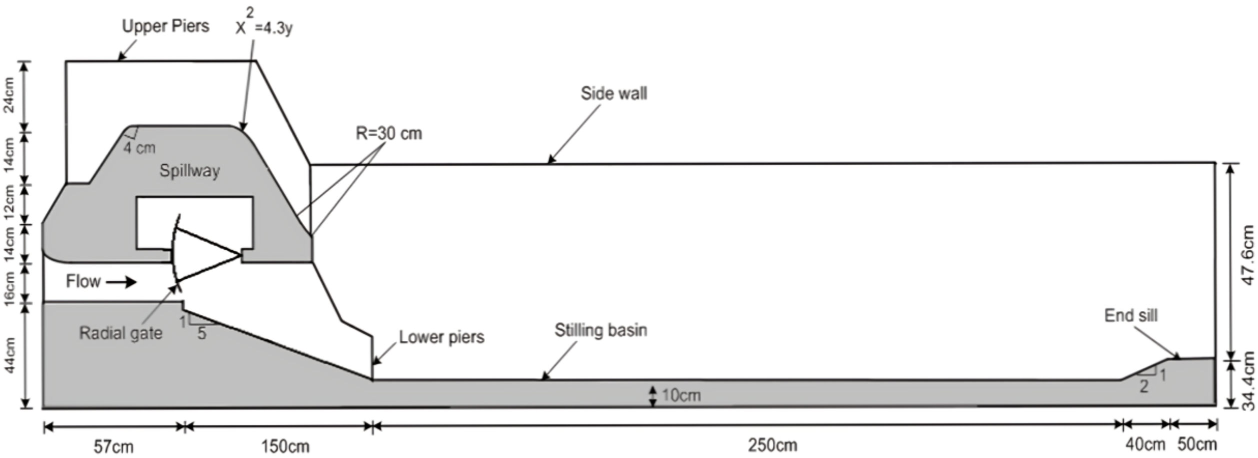


**Figure 3.** Cross section of Makhool spillway (Abbas, 2002)

## 2.2 Governing and flow equations

CFD system doing by the governing equations of fluid flow have been divided into a set of algebraic equations that are solved using computers. These equations define the conservation of mass, momentum, and energy within a fluid, accounting for factors like viscosity, turbulence, and thermal effects.

*2.2.1 Mass continuity equation*

It is expressed in differential form as equation (1) with its variables, which is also known as the equation for the conservation of mass within a fluid flow system.

$$\frac{V_F}{\rho c^2}\frac{\partial p}{\partial t} + \frac{\partial uA_s}{\partial x} + R\frac{\partial vA_y}{\partial y} + \frac{\partial wA_z}{\partial z} + \xi\frac{uA_x}{x} = \frac{R_{5ag}}{\rho} \qquad (1)$$

Where; $\frac{V_F}{\rho c^2}\frac{\partial p}{\partial t}$ This term likely represents acoustic wave propagation, where VF is a volume of the fluid, ρ refers to fluid density, p is the pressure, and c refers to the speed of sound in the fluid. This term accounts for the rate of change of pressure over time.

$\frac{\partial uA_s}{\partial x}$ The term refers to the rate of change of the mass flow rate (uAs) to the x direction, where u is a velocity in the x-direction and As is a cross-sectional area perpendicular to the flow. This term accounts for the advection of mass in the x-x-direction.

$R\frac{\partial vA_y}{\partial y}$ This term refers to the rate of change of mass flow rate (vAy) to y direction, Ay is the cross-sectional area perpendicular to the flow, and v is the velocity in the y-direction. R seems to represent some coefficient or term related to the flow.

(R=1, ξ= 0) applied when Cartesian coordinates are to be utilized.

*2.2.2 Momentum equations*

The equations of motion for the fluid velocity components (u, v, w) in the three coordinate directions are the Navier-Stokes equations with some additional terms:

$$\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x} + vA_yR\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right\} - \xi\frac{A_y v^2}{xV_F} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x - b_x - \frac{R_{SCE}}{\rho V_F}(u - u_w - \delta u_x) \quad (2)$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x} + vA_yR\frac{\partial v}{\partial y} + wA_z\frac{\partial v}{\partial z}\right\} + \xi\frac{A_y uv}{xV_F} = -\frac{1}{\rho}\left(R\frac{\partial p}{\partial y}\right) + G_y + f_y - b_y - \frac{R_{sov}}{\rho V_F}(v - v_z - \delta v_s) \quad (3)$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x} + vA_yR\frac{\partial w}{\partial y} + wA_z\frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z - b_z - \frac{R_{sog}}{\rho V_F}(w - w_w - \delta w_s) \quad (4)$$

Where; ($G_x$, $G_y$, $G_z$) refer to accelerations, ($b_x$, $b_y$, $b_z$) are flow losses across porous baffle plates, and ($f_x$, $f_y$, $f_z$) are viscous accelerations (Hirt and Sicilian,1985; Hirt and Nichols, 1981).

*2.2.3 Turbulence models*

Turbulence models are mathematical representations, employed in computational fluid dynamics (CFD) for the simulation of turbulent flow phenomena. Turbulence is represented by chaotic, unsteady motion and is present in many practical fluid flow situations, such as air flowing over an airplane wing or water flowing in a river. The available turbulence models vary in complexity and suitability, depending on the properties of the flow being simulated as well as the computational resources available. There are some common types:

(1) RANS (Reynolds Averaged Navier-Stokes) Models: These models are based on the Reynolds-averaged Navier-Stokes equations, where the turbulent quantities are averaged over time, RANS models include the following models (Wilcox, 1998):

(2) k-e model: This is one of the most widely used RANS models and can solve transport equations for turbulent kinetic energy (k) and its dissipation rate (ε).

(3) Spalart-Allmaras model: It is a one-equation model that solves a turbulence variable related to turbulent viscosity (Pope, 2000). As shown in equations 5, 6, and 7.

(4) The RNG Ke model (Renormalization Group): RNG Ke is according to transport equations of the Ke model equations (5, 6, 7) with different coefficients and replaced variables in equation (6) and extracted from equation (8).

(5) LES (Large Eddy Simulation): LES resolves large-scale turbulent structures explicitly while modeling the effects of small-scale turbulence. It’s computationally more intensive compared to RANS but provides more accurate predictions for flows with large turbulent structures (Caughey, 2005).

(6) DNS (Direct Numerical Simulation): DNS solves the entire range of turbulent scales without any modeling. It’s highly accurate but computationally

very expensive and is typically limited to academic research or small-scale simulations (Davidson, 2015).

(7) Hybrid Models: These combine elements of both RANS and LES to achieve accurate results at a reduced computational cost. Examples include Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) (Versteeg and Malalasekera, 2007).

(8) Reynolds Stress Models (RSM): RSM directly models the Reynolds stresses in the flow and is more suitable for complex flows with significant streamline curvature or rotation.

(9) Select the suitable turbine model based on elements such as the characteristics of the flow (e.g., boundary layer, separated flow), the required level of accuracy, and the available computational resources. Engineers perform sensitivity analyses using different turbine models to evaluate their impact on simulation results. According to previous studies, accurate results can be obtained from k-e and The RNG Ke models is higher than the LES turbulence model result (Daneshfaraz et al., 2021).

$$\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - \varepsilon \ldots\ldots\ldots\ldots \quad (5)$$

$$\frac{\partial \epsilon}{\partial t} + \bar{u}_J \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(v + \frac{v_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1}\tau_{ij}\frac{\partial \bar{u}_i}{\partial x_j} - C_{\varepsilon 2}(\varepsilon)\right) \quad (6)$$

Therefore, the turbulent viscosity is given as:

$$v_T = C_\mu \frac{k^2}{\varepsilon} \quad (7)$$

Where; ($C_{13}$, $C_{23}$, $C_\mu$) are constant and have the following values (1.44, 1.92, 0.09) Consecutive, ($\sigma_k$, $\sigma_\varepsilon$) are represent the Prandtl number of turbulent K and e, are have the following values (1, 1.3) respectively (Shaheed et al., 2019).

$$C^*_{\varepsilon 2} = C_{2\varepsilon} + \frac{C_\mu \mu^3\left(1-\frac{\mu}{\mu_0}\right)}{1+\beta\mu^3};\ \mu = \frac{k}{\varepsilon}\left(2S_{ij}S_{ij}\right)^{\frac{1}{2}} \quad (8)$$

## 2.3 Numerical model

The Makhool Dam Spillway was simulated using the FLOW 3D program. FLOW-3D is accurate, fast, and proven CFD software that solves the toughest free-surface flow problems (Flow Science, 2020).

### *2.3.1 Model geometry*

The geometry of the physical model consists of spillway and stilling basin type 1. The numerical model was drawn in Auto-CAD 3D then exported as a STL (stereo lithography) in the Flow-3D program as shown in (Figure 4(a)).

### *2.3.2 Boundary and initial condition*

There are ten boundary conditions which are categorized according to flow condition and spillway design, in this simulation, the boundary conditions applied to the numerical model are shown in the (Table 1) and (Figure 4(b)).

### *2.3.3 Physics and fluid properties*

In Flow-3D many physical properties can be applied in simulation as appropriate to the design. In this research, these options enabled the gravitational non-inertial reference frame and viscosity and turbulence (Saneie et al., 2016). In gravitational non-inertial reference frame option entered into cartesian coordinates (X=0, Y=0, Z= 9.81), As for the viscosity option has been selected viscous flow with the RNG model once and with k-e in other runs. These options were used because they can Predict vortices and give greater accuracy to rotary flow. The fluid properties entered into the modeling are based on the International System of Units (SI) unit (Yusuf and Micovic, 2020), where water at a temperature of 20°C (Rahimzadeh et al., 2012), a density of 1,000 kg/m$^3$ (Banerjee, 2018), and a dynamic viscosity of 0.001 pa.s were chosen in the modeling (Hekmatzadeh et al., 2018).

**Table 1.** The boundary conditions of model

| (X-min) Upstream boundary | (X-max) Downstream boundary | (Z-min, Z-max) Side boundary | (Y-min) Bottom boundary | (Y-max) Top boundary |
|---|---|---|---|---|
| inflow(Q) | outflow(O) | wall condition (W) | wall condition (W) | symmetry (S) |

### *2.3.4 Mesh analysis and grid generation*

Determined the Grid generation because the solutions accuracy depends on the mesh quality, using Cartesian coordinates, and entering the suitable mesh size which is equal to 0.01. The size of cells was the same in all tests as shown in (Figure 4 (c)-(d)). After selecting the size of mesh size, the Favorizer icon is rendered to view how the grid generated resolves the geometry in addition to the view of the initial fluid region as shown in (Figure 4(e)).

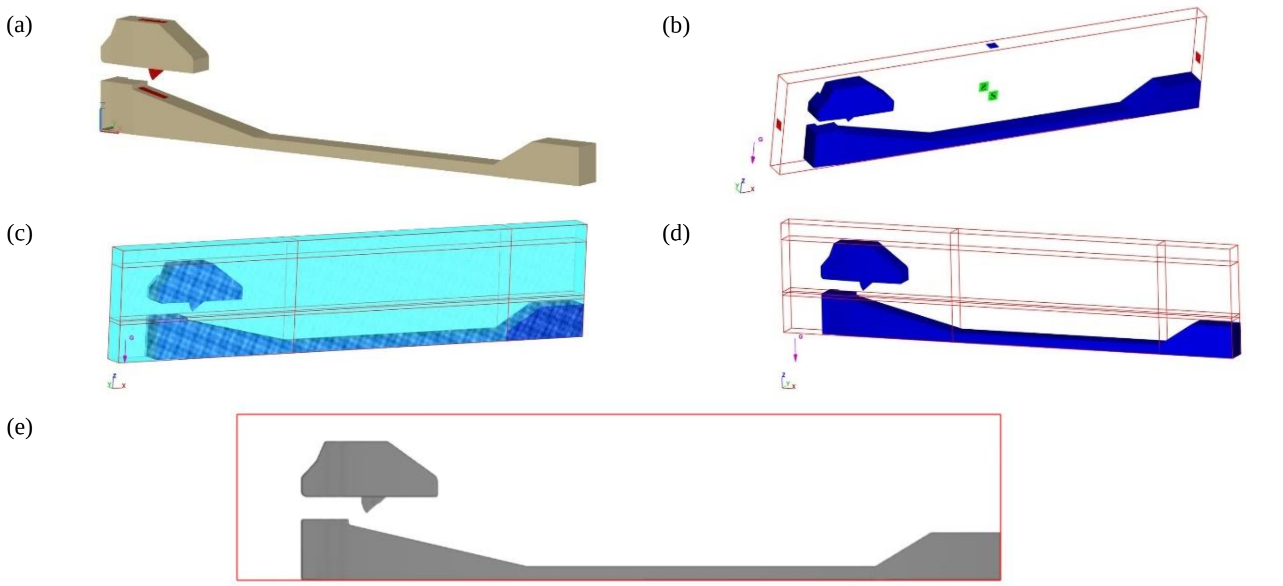


**Figure 4.** Numerical model: (a) model Geometry; (b) boundary condition; (c) the computational cells; (d) mesh block; (e) favorizer render

## 3. RESULTS AND DISCUSSION

The hydraulic models were simulated in different scenarios such as different discharges with different gate openings of the spillway based on the data of a physical model in the experimental study and then simulated by using two different equations in Flow 3D software RNG equation and k-e equation because the RNG turbulence and standard k-e models give more stable values than other models according to previous studies. As shown in Supplementary data, which shows the run of the dam for all cases with the velocity and flow water depth parameters. The results were selected after achieving steady state conditions to obtain exact values of a numerical model so the results were close to the results of the laboratory study with some slight differences that are considered acceptable. As shown in (Table 2) it shows the difference in the results and the accuracy of the program.

RMSE was calculated by:

$$\text{RMSE} = \sqrt{\frac{1}{n}\sum_{1}^{n}(y\text{ experimental - }y\text{ numerical })^2} \tag{9}$$

The values of RMSE as presented in (Table 2) indicate accuracy in CFD simulation. The simulation was run for all the indicated points, as shown in Supplementary data, which shows the accuracy of the models in the Flow 3D by the compatibility between the flow state in the physical and numerical models through the shape of the hydraulic jump.

**Table 2.** Numerical results of simulating Makhool dam by using Flow3D with experimental results

| | | | | RNG | | | KE | | |
|---|---|---|---|---|---|---|---|---|---|
| $G_0$ | Q | $V_{exp}$ | $Y_{exp}$ | $V_{num}$ | Diff% | $Y_{num}$ | $V_{num}$ | Diff% | $Y_{num}$ |
| 0.15 | 0.083 | 3.28 | 0.11 | 3.12 | 4.878 | 0.115 | 3.21 | 2.134 | 0.112 |
| | 0.07866 | 3.1975 | 0.1025 | 3.1 | 3.049 | 0.106 | 3.125 | 2.267 | 0.105 |
| | 0.057 | 2.7142 | 0.10 | 2.56 | 5.681 | 0.106 | 2.567 | 5.423 | 0.106 |
| 0.155 | 0.0405 | 1.7743 | 0.11 | 1.6563 | 6.651 | 0.111 | 1.66 | 6.442 | 0.114 |
| 0.16 | 0.1 | 3.3064 | 0.135 | 3.01 | 8.964 | 0.145 | 3.019 | 8.692 | 0.145 |
| | 0.083 | 2.8614 | 0.1275 | 2.8107 | 1.772 | 0.130 | 2.8142 | 1.650 | 0.130 |
| | 0.07866 | 2.6773 | 0.13 | 2.8429 | -6.185 | 0.122 | 2.71 | -1.221 | 0.128 |
| | 0.057 | 2.0348 | 0.135 | 2.01 | 1.219 | 0.137 | 2.1 | -3.204 | 0.131 |
| RMSE | | | | 0.005547 | | | 0.004854 | | |

$G_0$ is Gate opening (m), Q is Discharges ($m^3/s$), $V_{exp}$ is Velocity in the physical model (m/s), $Y_{exp}$ (m) is downstream flow depth in the physical model, $V_{num}$ Velocity in the numerical model (m/s), $Y_{num}$ (m) is downstream flow depth in the numerical model and RMSE is root mean square error (Lafta,2020; Hodson,2022), was used to measure the error in predicting simulated results by comparing them with the experimental results.

### 3.1 Comparison of velocity

Through Table 2, it can be seen the comparison between the results of the numerical model with experimental tests carried out by (Abbas, 2002). Three gate openings were selected for the physical model, each operated at specific discharges. The run provided positive performance especially when using the Ke turbulence model as shown in (Figure 5).

The velocity magnitudes demonstrate a high level of concordance. The difference rate percentage between the laboratory data and numerical results stands at 8.96% and 8.69% for RNG and Ke turbulence models respectively. Although there is a slight disparity between the numerical velocities and those observed in the laboratory model, this difference is considered acceptable. The levels of difference rate observed in the recorded measurements are considered to be within an acceptable range. It is postulated that the turbulent fluctuations induced by the incoming flood discharge may impact the measurement outcomes, thus serving as a plausible source of difference.

### 3.2 Comparison of Water Depth

Experimental results of water depth at the specified point (y1) were compared with the simulation results numerical model in Flow-3D. The simulation was run in the following cases illustrating the size of the open gates (0.16-fully gated,0.155,0.15) m, as shown in (Figure 6).

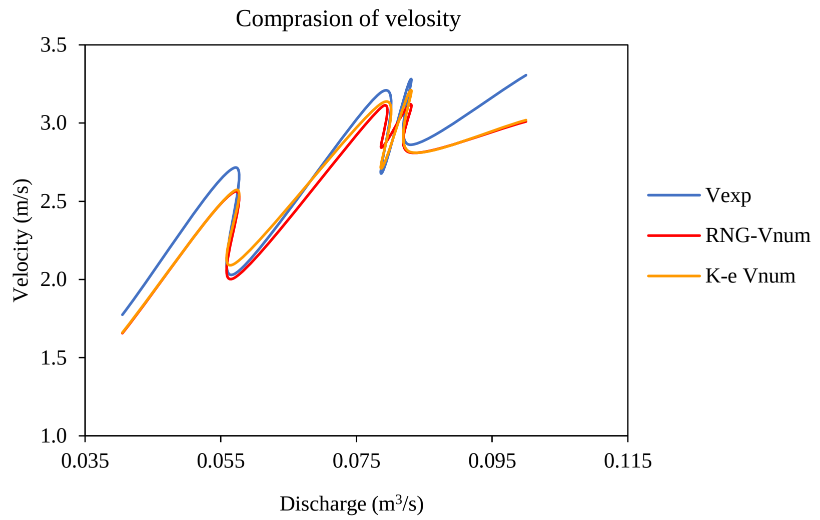


**Figure 5.** The result of the velocity and validation of experimental and numerical models

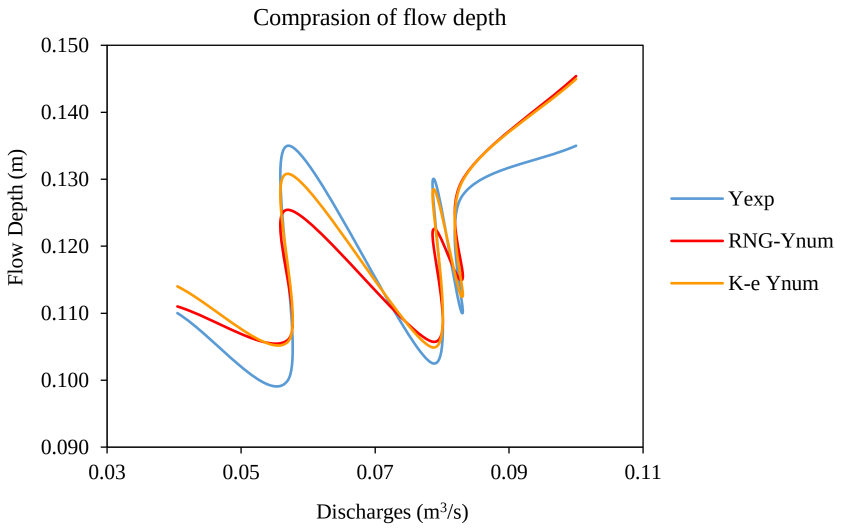


**Figure 6.** The result of flow depth and validation of experimental and numerical data using standard k-e and RNG turbulence models equation

# 4. CONCLUSION

The results derived from the research are enumerated as follows:

1. As a result of the studies in the research, it was observed that the difference rate was acceptable between the experimental and numerical analysis velocity and depth results, whether operating by RNG or Ke equation. This agreement the capability of Flow 3D software for numerical simulation of flow over a spillway. It may be more economical in many cases from physical model measurements in a hydraulic laboratory.

2. Turbulence equations effects in the flow usually caused differences in the measurements of the model. It showed that the difference in results between the turbulence models used depends on the type of flow, as the RNG equation is used for complex and unsteady flows, while the KE equation is used for stable flows, so its results are closer to reality in this type of flow.

3. The flow depth of water and velocity were measured at the location where the hydraulic jump is initiated and at a certain distance like the physical model and it was observed that there is some difference in the results but the difference in RMSE was shown to be within the permissible limit.

# REFERENCES


Abbas MM. Experimental Investigation on the Hydraulic Performance of the Combined Stilling Basin at Makhool Dam [dissertation]. Baghdad, Iraq: Al-Mustansiriya University; 2002.

Amen RF. Hydraulic Performance of Large Radial Gates Installed in Closed Conduits under High Head for Makhool Dam [dissertation]. Baghdad, Iraq: Al-Mustansiriya University; 2002.

Banerjee D, Jhamnani B. CFD analysis of ogee spillway hydraulics. International Journal of Modern Trends in Engineering Research 2018;5:Article No. 79.

Caughey DA. Turbulence Modeling for Engineers: A Primer. New York, NY: Wiley; 2005.

Daneshfaraz R, Minaei O, Abraham J, Dadashi S, Ghaderi A. 3-D numerical simulation of water flow over a broad-crested weir with openings. ISH Journal of Hydraulic Engineering 2021;25(2):183-9.

Dargahi B. Experimental study and 3D numerical simulations for a free-overflow spillway. Journal of Hydraulic Engineering 2006;132(9):899-907.

Davidson PA. Turbulence: An Introduction for Scientists and Engineers. USA: Oxford University Press; 2015.

Engineering Consulting Bureau of Anbar University. Makhool Dam Project Hydraulic Study. Anbar, Iraq: Engineering Consulting Bureau of Anbar University; 2020.

Erpicum S, Tullis BP, Lodomez M, Archambeau P, Dewals B, Pirotton M. Scale effects in physical piano key weirs models. Journal of Hydraulic Research 2016;54(6):692-8.

Flow Science Inc. FLOW-3D Version 10.0 User Manual [manual]. Santa Fe, NM: Flow Science Inc.; 2020.

Hekmatzadeh AA, Papari S, Amiri SM. Investigation of energy dissipation on various configurations of stepped spillways considering several RANS turbulence models. Iranian Journal of Science and Technology, Transactions of Civil Engineering 2018;42:97-109.

Hodson TO. Root mean square error (RMSE) or mean absolute error (MAE): When to use them or not. Geoscientific Model Development 2022;15:5481-8.

Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics 1981;39:201-25.

Hirt C, Sicilian J. A porosity technique for the definition of obstacles in rectangular cell meshes. Proceedings of the 4th International Conference on Numerical Ship Hydrodynamics; 1985 Sep 24-27; Washington, DC; 1985.

Jahad U, Al-Ameri R, Chua L, Das S. Investigating the effects of geometry on the flow characteristics and energy dissipation of stepped spillway using two-dimensional flow modelling. Proceedings of the 21st IAHR Asia and Pacific Division Conogress; 2018 September 2-5; Yogyakarta, Indonesia; 2018. p. 289-96.

Kocaer Ö, Yarar A. Experimental and numerical investigation of flow over ogee spillway. Water Resources Management 2020;34(13):3949-65.

Kote AS, Nangare PB. Hydraulic model investigation on stepped spillway’s plain and slotted roller bucket. Engineering, Technology and Applied Science Research 2019;9(4):4419-22.

Lafta BS. Numerical Study of the Effect of Stepped Spillway on Distribution of Flow Parameters [dissertation]. Wasit, Iraq: Wasit University, College of Engineering; 2020.

Li S, Cain S, Wosnik M, Miller C, Kocahan H, Wyckoff R. Numerical modeling of probable maximum flood flowing through a system of spillways. Journal of Hydraulic Engineering 2011;137(1):66-74.

Manogaran T, Zainol MMA, Wahab MKA, Aziz MA, Zahari NM. Assessment of flow characteristics along the hydraulic physical model of a dam spillway. Journal of Civil Engineering, Science and Technology 2022;13:69-79.

Ministry of Water Resources (MoWR). The Center for Studies and Designs. Feasibility Study of Makhool Dam; 2020.

Pataki GE, Cahill JP. Guidelines for Design of Dams. Albany, New York: Division of Water, Department of Environmental Conservation; 1985.

Pereira GM. Spillway Design-Step by Step. United States: CRC Press; 2020.

Pope SB. Turbulent Flows. United Kingdom: Cambridge University Press; 2000.

Rahimzadeh H, Maghsoodi R, Sarkardeh H, Tavakkol S. Simulating flow over circular spillways by using different turbulence models. Engineering Applications of Computational Fluid Mechanics 2012;6(1):100-9.

Rajaa AI. Numerical Modeling of Flow Patterns over Spillway [dissertation]. Anbar, Iraq: University of Anbar, College of Engineering; 2020.

Shaheed R, Mohammadian A, Kheirkhah Gildeh H. A comparison of standard k-ε and realizable k-ε turbulence models in curved and confluent channels. Environmental Fluid Mechanics 2019;19(2):543-68.

Samadi-Boroujeni H, Abbasi S, Altaee A, Fattahi-Nafchi R. Numerical and physical modeling of the effect of roughness height on cavitation index in Chute Spillways. International Journal of Civil Engineering 2020;18(5):539-50.

Saneie M, SheikhKazemi J, Azhdary Moghaddam M. Scale effects on the discharge coefficient of ogee spillway with an arc in plan and converging training walls. Civil Engineering Infrastructures Journal 2016;49(2):361-74.

Sartaj M, Beirami K, Fooladgar A. Analysis of two-dimentional flow over standard ogee spillway using RNG turbulence model. Proceedings of the 7th International Congress on Civil Engineering; Tehran, Iran: Tarbiat Modares University; 2006.

Savage BM, Johnson MC. Flow over ogee spillway: Physical and numerical model case study. Journal of Hydraulic Engineering 2001;127(8):640-9.

Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics the Finite Volume Method. 2nd ed. India: Pearson Education; 2007.

Wilcox DC. Turbulence Modeling for CFD. La Canada, California: DCW industries, Inc.; 1998.

Yusuf F, Micovic Z. Prototype-scale investigation of spillway cavitation damage and numerical modeling of mitigation options. Journal of Hydraulic Engineering 2020;146(2):Article No. 04019057.

Zahari NM, Zawawi MH, Sidek LM, Ng FC, Abas MA. Discrete particle method numerical simulation on the distributions of suspended particles in the flow of ogee spillway structure. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 2022;98(1):42-55.