Anbar Journal of Engineering Science (AJES)
P-ISSN: 1997-9428; E-ISSN: 2706-7440

Vol. 11 , No. 2 (2020) , pp. 221 ~ 230
DOI: 10.37649/aengs.2020.171262

Unviersty of Anbar

Anbar Journal of Engineering Science©

journal homepage: http:// http://www.uoanbar.edu.iq/Evaluate/

Performance Study of Fluent-2D and Flow-3D Platforms in the CFD Modeling of a Flow Pattern Over Ogee Spillway

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

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

PAPER INFO

Paper history:
Received 7/ 8/ 2020
Received in revised form 17 /9/ 2020
Accepted 29/ 9/ 2020

Keywords:
Spillway
Numerical model
Physical model
Flow-3D
Ansys Fluent
Flow simulate

ABSTRACT

Recently, the investigations studies of simulating flow over spillways have increased using numerical models. Due to its important structure in the dams to pass flood wave to the downstream safely. Researches finding have shown that CFD (Computational fluid dynamics) models as the numerical method are a perfect alternative for laboratory tests. Performance analysis of the CFD platforms Ansys Fluent-2D and Flow-3D are presented, focus on finding the variations between the numerical results of the two programs to simulate the flow over ogee spillway. The present study treats the turbulence using RNG k-ε of RANS approach, and also use the Volume of Fluid (VOF) algorithm to track the water-air interaction. The Fluent-2D and Flow-3D accuracy are assessed by comparing representative flows variables (velocity; free surface profiles; pressure; and the turbulent kinetic energy). The results of both codes have been also compared with experimental data. The results of the analysis show an excellent agreement between the two platforms data, which could assist in the future by using both programs to calibrate each other, rather than traditionally relying on laboratory calibration models.

1. Introduction

The vastest driving force for dam construction worldwide are the need for flood control, reliable water supply, navigation, recreation and hydroelectric power generation [1]. The ogee is one of the most famous types of the spillway, that play a remarkable role in safety and stability of dams, which designed to safely transfer floods from the upstream of the dam to the watercourse downstream [2,3].

For more than 100 years, the laboratory modeling was the main tool to investigations of hydraulic characteristic for flow over spillways. Unfortunately, laboratory models have a lot of flaws, including construction expensive; time-consuming to develop and test; and also, many researchers have mentioned that there are some errors in results due to the scaling effects (i.e. reduce prototype scale according to law of similarly of Froude number can cause errors in the results) [4]. Therefore, recently, investigative studies have increased using numerical models, where the hydrodynamics behavior of spillway can be studied and simulated numerically with reasonable cost and time. The Computational fluid dynamics (CFD) models are a numerical process to solving the flow equations like conversation of mass and momentum (3-dimensional Navier-Stokes equations). Today, there are many CFD commercial program designed to represent fluid flow such as Ansys Fluent & CFX, Flow-3D, OpenFOAM, Power Flow, SimScale, COMSOL Multiphysics, Autodesk CFD, etc. Al-Zubaidy and Alhashimi; Dargahi; Dolon Banerjee; Hekmatzadeh et al.; Jahad et al.; Li et al.; Rahimzadeh et al.; Samadi-Boroujeni et al.; Sartaj et al.; and Yusuf and Micovic [5–14] conducted their studies using Ansys-Fluent software. The main aims of their studies are to investigations of overflow spillways and comparing results of a program with

Corresponding author: : **Ahmed Imad Rajaa** ; ahmed2000@gmail.com ; +964-7802367123

experiment models. Also, Serafeim et al., [15] conducted a numerical study using Fluent-3D with k-Ɛ turbulence model, to compare the water surface level of CFD model with the laboratory data, and their results indicate a reasonable agreement at different discharge values. On the other side, Flow-3D commercial software is most widely used for spillway modelling. In simulation flow pattern over spillway; many authors conducted their studies utilizing the Flow-3D program and concluded the results a good agreement with the experimental results in simulate flow over the spillway [16–25]. Another alternative program for simulation flow pattern over spillway is open source platforms OpenFOAM. By using this program, Imanian and Mohammadian, (2019) [26] conducted a study to investigate the performance of flow over spillway at water head significantly higher than the design head. The finding of this study shows a better agreement between the experimental and simulation model. In fact, choosing the most suitable of numerical codes is a difficult task. For this reason, Bayon et al., [27] using CFD codes of Flow-3D and OpenFOAM to compare and predict of the hydraulic jump characteristics at low Reynolds number. They reported that there is a superiority in some variables to the other for both platform codes. For example, the results accuracy can be expected for maximum velocity measured, in Flow-3D, with (99.7%) of determination coefficient (r^2). While the platform of OpenFOAM scored a better result than Flow-3D in the backward velocity evaluating with 88.2 % and 83.7 %, respectively. However, the Flow-3D platform appears to reproduce the interaction better between super-critical and sub-critical flow in the stilling basin.

The present paper focuses on the performance study of Fluent-2D and Flow-3D platforms to simulation flow over ogee spillway. The necessary hydraulic parameters like velocity, water depth and pressure have been comparing for both platforms. Add also, the pressure head of the Mandali dam (as a case study) has been selected to choose the better model. The definition of used mesh and the mesh sensitivity analysis have been performing for both codes. This paper gives explicit results to both platforms that help in choosing the right program in the future. This paper also examines the possibility of using the both models in calibration, instead of using physical models. Most of the time, numerical results must be verified, so the best alternative is using another numerical program as an alternative to laboratory models that are costly in time, cost, and effort.

2. Numerical models

Numerical results have recently already become means of solving complicated problems that are difficult or expensive to obtain in the laboratory [28]. For representing the flow pattern over a spillway, the commercial software of Fluent-2D and Flow-3D are widely used to solving the continuity and unsteady 3-dimensional Reynold averaged Navier Stokes (RANS) equations. The classic formulations of continuity and RANS equations are depicted in equations (1) and (2):

$$\frac{\partial u_i}{\partial x_i} = \text{٠} \qquad (\text{١})$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\text{١}}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\text{٢}\nu S_{ij} + \tau_{ij}\right) \qquad (\text{٢})$$

Where u_j is the average velocity of cartesian components; x_j are cartesian-coordinates (j = 1, 2, 3); p is pressure; t is time; ρ is density; and ν is dynamic viscosity. The S_{ij} denote to the strain rate tensor; and τ_{ij} denote to the Reynolds stress tensor [29,30].

Both programs used Finite Volume Method (FVM) to solve the governing equations of flow over spillways [31,32]. Broadly, the FVM method is more popular than the Finite Difference Method (FDM) and Finite Element Method (FEM) in the hydraulic field [5]. The reason is the FVM requires less time-consuming in computational effort than the FEM. On the other hand, the used FDM method has required a structure meshes, whereas the FVM method contains various types of mesh to represent different computational domains [33]. Different turbulence closure models are available in both platforms including the standard k-ε, RNG k–ε, k–ω models. The RNG (k–ε) model is used in the present study for both models because it is more appropriate in the flows on surfaces with large curves [34]. The Flow-3d commercial software was developed by Flow Science Company. This program contains the Volume of Fluid (VOF) technique [35] for treating free surfaces. In the VOF algorithm, the interface of tracking between the water and air for treating free surfaces give value from one to zero for each cell in the computation domain [36]. The FAVOR (Fractional Area Volume Obstacle Representation) is an algorithm for precisely defining and inserting the model shape into the governing equations developed by Hirt and Sicilian, (1985) [37]. It should be noted that the effectiveness of this method increases with increasing mesh smoothness. The Fluent program which developed by Ansys Company is the second platform was used in the present study. The Fluent platform has been used in 2-dimensional (2D) domain, due to the enormous time typically spent in the simulation when modelling in 3-dimensional (3D). Although the comparison between the two platforms are unjust; Al-Hashimi et al., [38] conducted a numerical study using Fluent codes in 2 and 3D to simulation flow over weir; their result indicated that the comparison of 2D result gives reasonable agreements with 3D domain. Table 1 summarizes the settings for both platforms. It is worth mentioning, the Fluent software has two models for accurately representing the multiphase flow; the first namely a mixture-multiphase-flow model (MMF) and the second is the volume of fluid model (VOF). Al-Zubaidy and Alhashimi, [12] state that VOF model is more accurately than the MMF model, especially for simulated velocity profiles.

Table 1. Summary of used numerical models.

Information	Fluent-2D	Flow-3D
Turbulence model	RNG (k-e)	RNG (k-e)
Mesh type	2D- Triangular	3D-Hexahedral (uniform meshing)
Time step and total time	0.1; 30 second	Automatic; 30 second
Solving method	Finite volume methods	Finite volume methods
Free surface treatment	VOF method	VOF method

2.1. Mesh analysis

Meshing or grid generation consider the second essential step of preprocessing after geometry constructer for both platforms, and it is a crucial consideration for the success in attaining the numerical solutions to the governing equations of the CFD problem. It accounts for almost 60% of CFD works. A mesh can be classified according to various norm. Depended on the cell shape, the grid can classify into triangle or quadrilateral grids in a 2D-domain, and tetrahedral, pyramid, triangular prism or hexahedral meshes in a 3D-domain. Or grid can be categorized into the structured or unstructured (non-uniform) mesh, depending on the nature of the cell connection with the neighbouring cells [29].

The structured grids (also namely uniform mesh) are most commonly used within the hydraulic engineering field, due to it often be more efficient in terms of exactness, CPU time and memory requirement, whereas caution should be considered when using unstructured mesh because the rapid change in the cell sizes can reduce the numerical accuracy [27]. Add also, CFD programs using uniform meshes are usually faster and require less memory than programs using unstructured or non-uniform meshes. Daneshkhah and Vosoughifar, [39], mentioned that ogee spillway considered curved in shape, thus it is recommended to use the structural mesh to minimize the relative error of flow parameter and optimize the numerical solution time.

In Fluent software, there is a great difficulty to establishing a uniform mesh having the same cell sizes in all domain. The reason is the nature of the ogee spillway curved (see Fig. 1). Therefore, the grid size must be very small to obtain a uniform mesh, this means an increase in computational cost. Put differently, the Fluent program relies on the mesh to determine geometry boundaries and the computational field, thus, a uniform meshes cannot be easily obtained, due to the spillway curve. In contrast, Flow-3d codes have a cartesian mesh can be defined as uniform or non-uniform planes, where all cells are perfect cubes in a uniform plane with $\Delta x = \Delta y = \Delta z$ [32]. This is supported by the FAVOR method, which is used for determining the boundaries of geometry separately from the computational grid. Hirsch [40], declared uniform cartesian meshes represent the ideal solution from the accuracy view point and they should be used whenever possible. In summary, when curved surfaces are present as spillway surface, there are two options; either remain the grids in the cartesian structure (as used in Flow-3d program), or move away from the perfect and make a grid adjacent to the curved solid surface (as used in Fluent codes). The first option reduces the required number of cells to get a uniform mesh and consequently reduces the simulation time as opposed to the second option, it requires a larger number of cells to access the structure grids (see Fig. 1).

On the whole, the sensitivity analysis for both programs are performing to the determination of the suitable cell size and achieves the model results independence from the imposed cell size. Triangular and hexahedral grids have been imposing for Fluent-2D and Flow-3d, respectively. It is worth noting, Fluent-2D triangular grid produces a tremendous effective result from implementing a rectangular mesh for establishing a computational domain as more uniform mesh. A sensitivity analysis was run with five mesh sizes of 100, 50, 35, 20 and 10 cm for both platforms. The quantitative of data of velocity distribution at ogee spillway crest is present in Fig. 2. As shown, the 10 cm optimum size of the grid is select base on the sufficient accuracy and computational time for Fluent-2D platform, whereas 20 cm cell size has been imposing for Flow-3d.

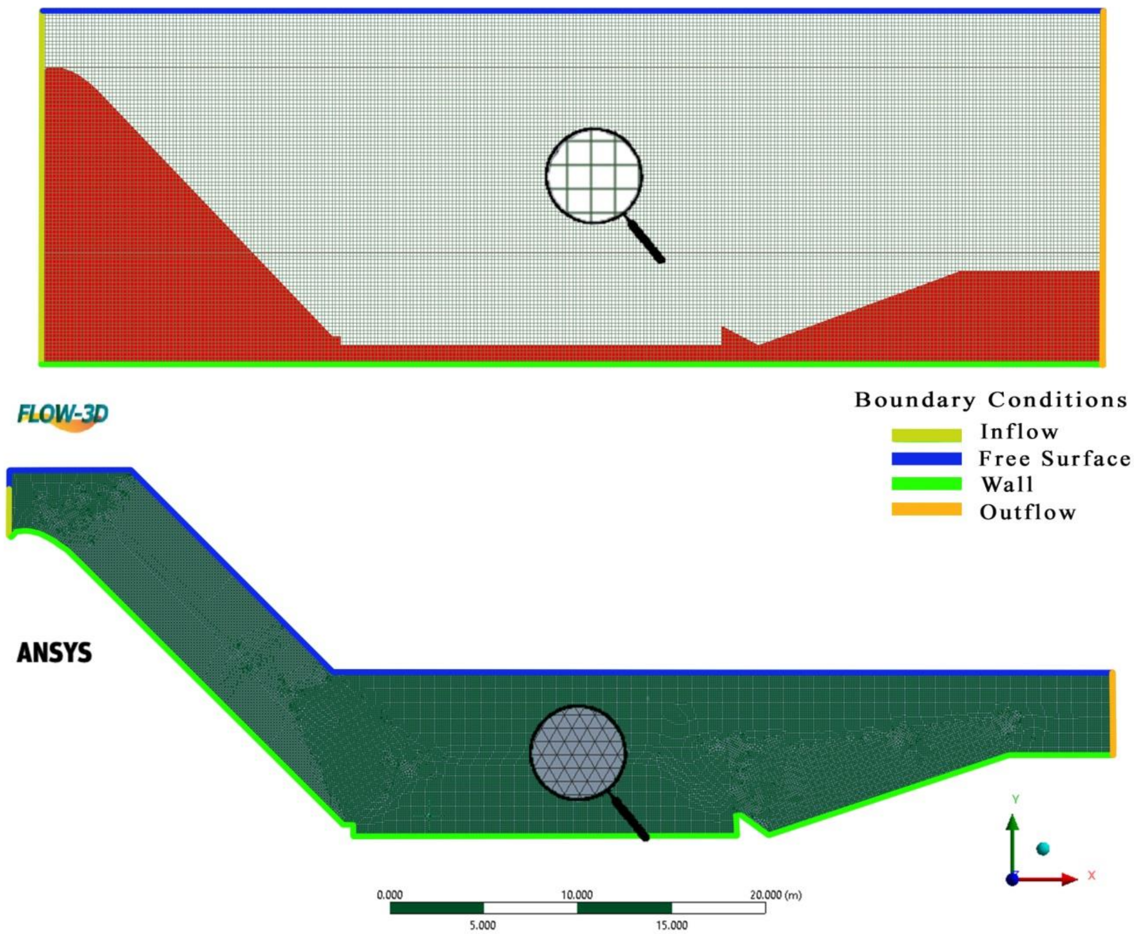

Fig. 1. Grid generation and boundary conditions for spillway model.

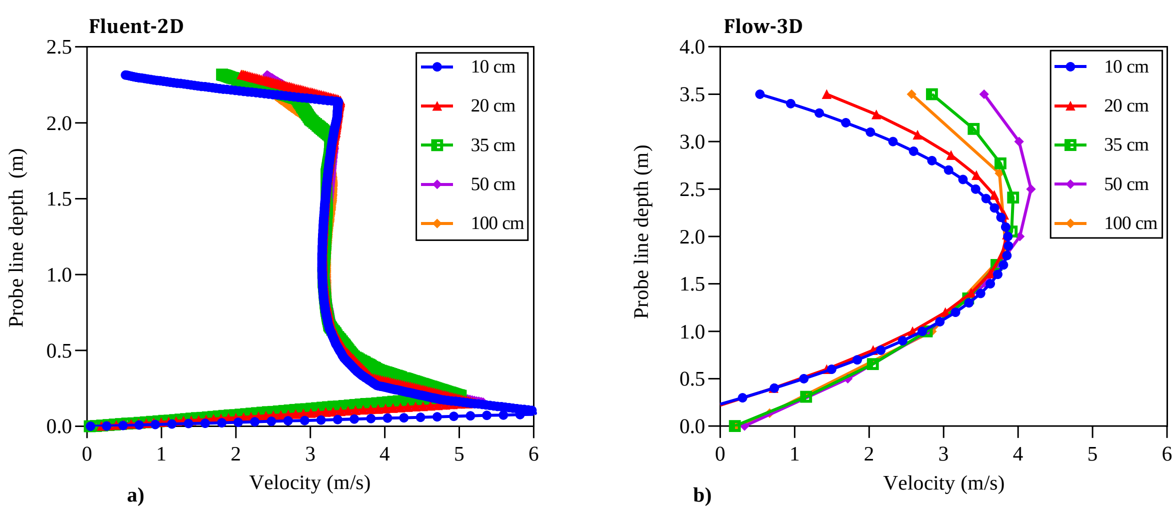

Fig. 2. Mesh sensitivity analysis: a) velocity distribution for Fluent-2D b) velocity fit-curve for Flow-3D.

2.2 Boundary conditions

As shown in Fig. 1, there are four different boundaries (X(min), X(max), Y(min) and Y(max)) in 2D-domain and add two boundaries (Z(min) and Z(max)) at 3D-domain, which must be precisely defined. Where x (min) means the spillway upstream; x (max) is downstream of spillway; and y (min), y (max), z (min), and z (max) are bottom, top, right and left of spillway indirect the fluid flow, respectively. In Present study, the boundary condition defines as indicate in Table 2 for both platforms. Where wall boundary denotes a non-slip condition and the outflow represents the amount of water out from spillway (determines automatically). The top of the domain is air; therefore, it was traditionally defined as specified pressure with the fluid fraction equal zero (free surface). The upstream boundary condition (x (min)) has been selected as inflow rate (depending on water head and discharge values) with four cases as illustrates in Table 3.

Table 2. Boundary conditions that using for both numerical models.

Platform	X(min)	X(max)	Y(min)	Y(max)	Z(min)	Z(max)
Fluent-2D	Inflow	Outflow	wall	Specified Pressure	wall	wall
Flow-3D	Inflow	Outflow	wall	Specified Pressure	-	-

Table 3. Upstream boundary conditions values.

Run	1	2	3	4
Flow rate (m^3/s)	666	813	1360	1803
Flow depth (m)	1.25	1.4	1.94	2.32

3. Results and Discussion

The simulation has been performed for Thirty seconds for four cases and all the hydraulic information including velocity, pressure and water depth are available. The main objective of the simulation is to accurately compare the results of both codes to determine the convergence of data between them.

Fig. 3 demonstrates that both platforms values provided similar velocity and pressure values at the spillway crest. As mentioned earlier, the Flow-3d program used a Cartesian structure of the grids and depended on FAVOR method to determine the curve of the spillway. The use of this method led to a contraction in the size of the model. For example, the ogee spillway was 10 meters in height. After the simulation, the height of the spillway has been reduced to 9.85 meters (see the lowest value of Flow-3d in Fig. 3a). By contrast, the Fluent-2D program instantly recognizes geometry with high accurately, against the significant increase in simulation time. Therefore, it can be said that the curves for both velocity and pressure are exactly the same for both platforms.

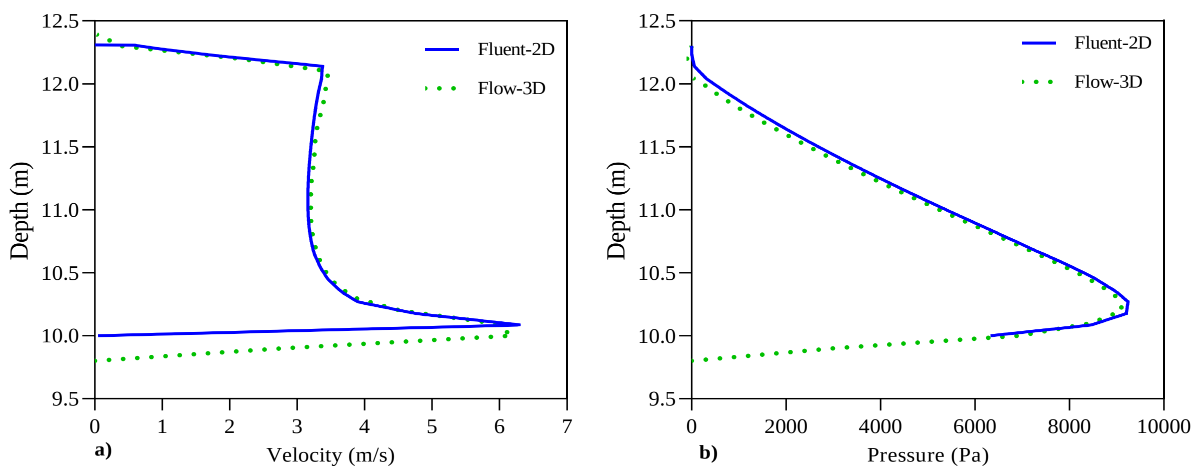

Fig. 3. Comparison between Fluent-2D and Flow-3D software at spillway crest for a) velocity distribution b) pressure distribution

To investigate of both model's accuracy in predicts flow pressure. The Mandali dam was selected as a case study. The Mandali dam is one of the small dams that was built for irrigation purposes. It is located in the east of Iraq in the governorate of Diyala (33°47'4.98"N, 45°35'34.51"E) [41]. Table 4, clearly summarizes the accuracy of both platforms according to physical model data of Mandali dam. The Flow-3D yields better accuracy results than the Fluent-2D in the estimations of pressure head with 2.17 and 6.23% maximum errors, respectively.

Table 4. Comparison of piezometer reading for both models with physical model at 1823 m^3/s discharge.

Point	laboratory data	Fluent-2D		Flow-3D	
		Value (m)	Error (%)	Value (m)	Error (%)
1	20.63	20.76	0.78	20.6	0.15
2	20.62	20.73	0.53	20.62	0
3	20.25	20.26	0.05	20.25	0
4	19.9	19.8	0.2	19.76	0.7
5	19.42	19.03	0.52	19.13	1.49
6	17.69	17.34	0.4	17.41	1.58
7	14.76	14.33	0.76	14.44	2.17
8	11.79	12.28	6.23	11.56	1.95

Note: The zero reading of the piezometer was 160 m a.s.l.

The relative error percent (REP%) and square error of root mean (RMSE) using Eqs. (3) and (4), respectively, are present in Fig. 4 for different discharge values. Where P_m and P_s are the pressure head of the experiment and simulation results of both codes, respectively [42].

$$\mathrm{RMSE\%} = ١٠٠ * \sqrt{\frac{١}{N}\sum_{i=١}^{N}\left(\frac{P_m - P_s}{P_m}\right)^{٢}} \qquad (٣)$$

$$\mathrm{REP\%} = \frac{١٠٠}{N} * \sum_{i=١}^{N}\left|\frac{P_m - P_s}{P_m}\right| \qquad (٤)$$

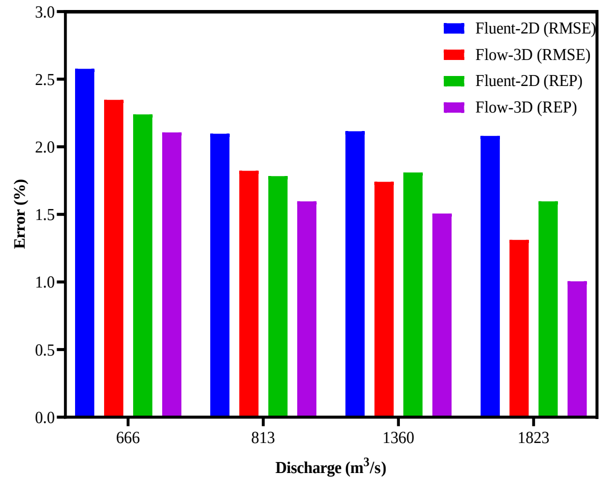

Fig. 4. RMSE and REP values for both numerical models with laboratory data.

According to Fig. 4, the Flow-3D platform is better numerical models when comparing with Fluent-2D. Regardless, these differences in both used platforms are small, and the results are very close to each other.

In turbulence simulation over stilling basin of Mandali dam, the maximum turbulent kinetic energy value is almost equal in both codes (range from 6 - 6.9 J/kg). As seen in Fig. 5, both models succeed in accurately predicting of the immediate location of the kinetic energy of turbulent (at the toe of spillway).

In the attempt to extract the water level over spillways, the researchers (Al-Qadami et al.; Dolon Banerjee; Kumcu; Shahheydari et al.; Shojaeian et al.; and Yildiz et al.) [11,18,20,43–45] stated that an excellent result could be obtained from numerical models when compared with laboratory results. Fig. 6. illustrates the results accuracy of free water surface simulation for different discharge values. At all discharge values, the Fluent-2D program gives a higher head level values than Flow-3d at the stilling basin, whereas the similar results have presented on the overflow surface. It is due to the severe turbulence in the stilling basin. A singular point of head measurement in the stilling basin of the physical model is added to accurately compare with both numerical results. Both models predicted results with reasonable accuracy in water level simulation. It should be noted that an exact head level of both models can be obtained based on the determination of the outflow boundary conditions as pressure specified (by set the water original level of the laboratory channel downstream).

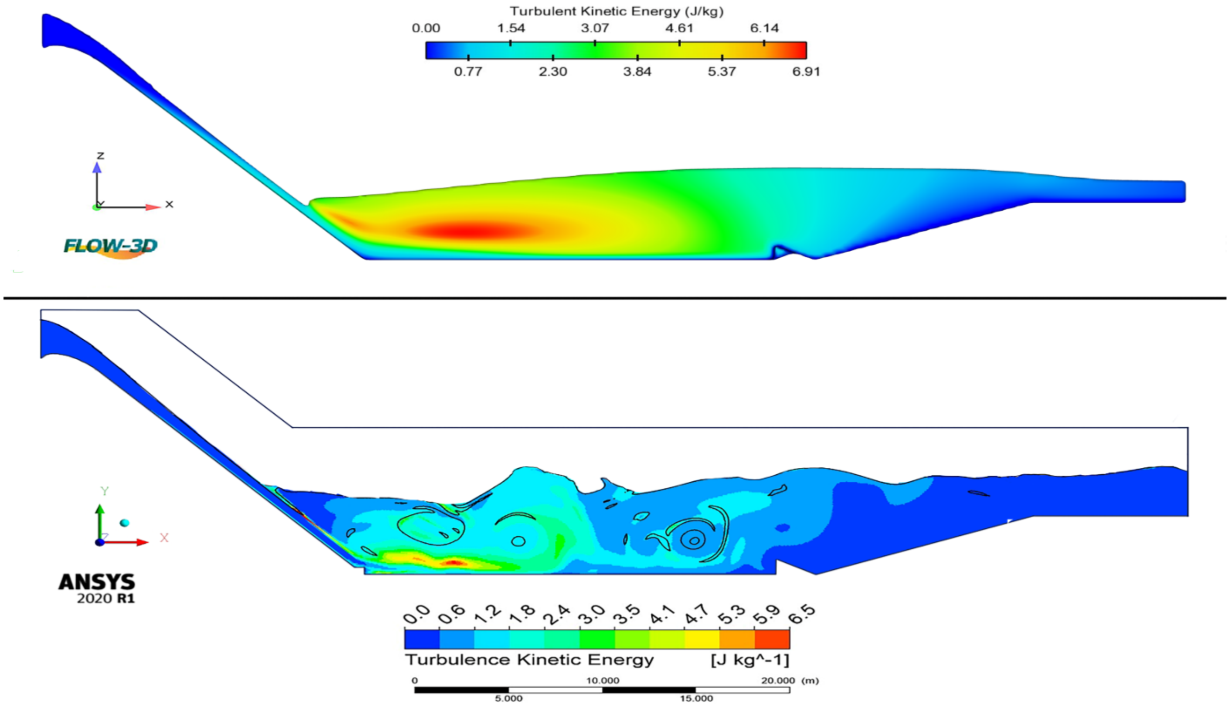

Fig. 5. Comparison of the turbulent kinetic energy changes between two platforms.

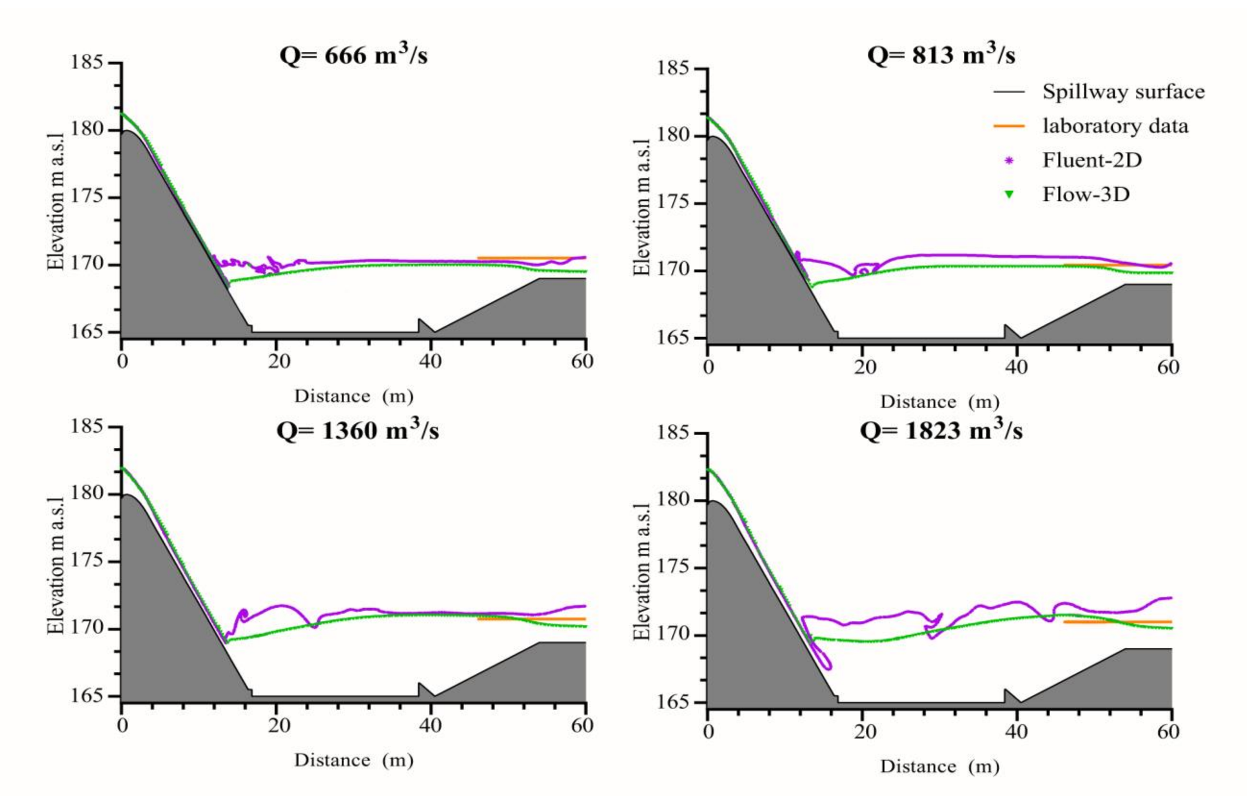

Fig. 6. Comparison of changes in the water head level between two platforms.

4. Conclusions

With the development of computational fluid dynamics, it has become straightforward to simulate large hydraulic installations at a reasonable time and cost. The Ansys Fluent and Flow-3D platforms are most widely used in the hydraulic engineering field to simulation flow patterns over spillways. The present study attempts to get important hydraulic parameters like velocity profile, head of water, pressure distribution, and turbulent kinetic energy to show the difference between Fluent-2D and Flow-3D. Moreover, laboratory data have been used to validate the results. The mesh sensitivity analysis for both programs has been performing to the determination of the suitable cell size. Both models agreed on the results of the velocity and flow depth distribution with a slight difference in the values of the pressure distribution over chute spillway. Both models also succeed in locating the immediate location of the turbulent kinetic energy as well as a slight difference in the water level in the stilling basin. This paper encourages engineers to conduct calibration using both programs to confirm their results, instead of relying on laboratory models.

٥. Acknowledgement

The authors would like to thank the Engineering Consultancy Bureau (ECB) of the College of Engineering, University of Al- Mustansiriya for construction of physical model.

٦. Reference

[1] Demeke, G. K.; Asfaw, D. H.; Shiferaw, Y. S. 3D Hydrodynamic Modelling Enhances the Design of Tendaho Dam Spillway, Ethiopia. Water 2019, 11 (1), 82. https://doi.org/10.3390/w11010082.

[2] Ammar; Isam; Zainab. Study the Effect of Spillway Locations on the Hydraulic Properties of Spillway. Ciência e Técnica Vitivinícola 2016, No. 0254–0223.

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

[4] Ho, D. K. H.; Riddette, K. M. Application of Computational Fluid Dynamics to Evaluate Hydraulic Performance of Spillways in Australia. Aust. J. Civ. Eng. 2010, 6 (1), 81–104. https://doi.org/10.1080/14488353.2010.1146 1946.

[5] Li, S.; Cain, S.; Wosnik, M.; Miller, C.; Kocahan, H.; Wyckoff, R. Numerical Modeling of Probable Maximum Flood Flowing through a System of Spillways. J. Hydraul. Eng. 2011, 137 (1), 66–74.

[6] 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. Int. J. Civ. Eng. 2019, 1–12.

[7] 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; Department of Civil and Environmental Engineering, Faculty of Engineering …, 2018; pp 289–296.

[8] Hekmatzadeh, A. A.; Papari, S.; Amiri, S. M. Investigation of Energy Dissipation on Various Configurations of Stepped Spillways Considering Several RANS Turbulence Models. Iran. J. Sci. Technol. Trans. Civ. Eng. 2018, 42 (2), 97–109. https://doi.org/10.1007/s40996-017-0085-9.

[9] Yusuf, F.; Micovic, Z. Prototype-Scale Investigation of Spillway Cavitation Damage and Numerical Modeling of Mitigation Options. J. Hydraul. Eng. 2020, 146 (2), 04019057. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001671.

[10] Rahimzadeh, H.; Maghsoodi, R.; Sarkardeh, H.; Tavakkol, S. Simulating Flow Over Circular Spillways by Using Different Turbulence Models. Eng. Appl. Comput. Fluid Mech. 2012, 6 (1), 100–109. https://doi.org/10.1080/19942060.2012.1101 5406.

[11] Dolon Banerjee, B. J. CFD ANALYSIS OF OGEE SPILLWAY HYDRUALICS. Int. J. Mod. Trends Eng. Res. 2018, 5 (07).

[12] Al-Zubaidy; Alhashimi. Numerical Simulation of Two-Phase Flow Over Mandali Dam Ogee Spillway. 2013, 2.

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

[14] Sartaj, M.; Beirami, K.; Fooladgar, A. M. Analysis of Twodimentional Flow over Standard Ogee Spillway Using RNG Turbulence Model. In 7th International Congress on Civil Engineering, Tehran, Iran; 2006.

[15] Serafeim, A.; Avgeris, L.; Hrissanthou, V.; Bellos, K. Experimental and Numerical Simulation of the Flow Over a Spillway. Eur. Water 2017, 57, 253–260.

[16] Savage, B. M.; Johnson, M. C. Flow over Ogee Spillway: Physical and Numerical Model Case Study. J. Hydraul. Eng. 2001, 127 (8), 640–649. https://doi.org/10.1061/(ASCE)0733-9429(2001)127:8(640).

[17] Ghanbari, R.; Heidarnejad, M. Experimental and Numerical Analysis of Flow Hydraulics in Triangular and Rectangular Piano Key Weirs. Water Sci. 2020, 1–7. https://doi.org/10.1080/11104929.2020.1724649.

[18] Yildiz, A.; Yarar, A.; Kumcu, S. Y.; Marti, A. I. Numerical and ANFIS Modeling of Flow over an Ogee-Crested Spillway. Appl. Water Sci. 2020, 10 (4), 90. https://doi.org/10.1007/s13201-020-1177-4.

[19] Akintorinwa, O. J.; Ajayi, C. A.; Oyedele, A. A. Determination and Establishment of Empirical Relationship between Magnetic Susceptibility and Mechanical Properties of Typical Basement Rocks in Southwestern Nigeria. Int. J. Phys. Sci. 2020, 15 (2), 70–89. https://doi.org/10.5897/IJPS2020.4870.

[20] Al-Qadami, E. H. H.; Abdurrasheed, A. S.; Mustaffa, Z.; Yusof, K. W.; Malek, M. A.; Ghani, A. A. Numerical Modelling of Flow Characteristics over Sharp Crested Triangular Hump. Results Eng. 2019, 4, 100052. https://doi.org/10.1016/j.rineng.2019.100052

[21] .Dong; Wang; Vetsch; Boes; Tan. Numerical Simulation of Air–Water Two-Phase Flow on Stepped Spillways Behind X-Shaped Flaring Gate Piers under Very High Unit Discharge. Water 2019, 11 (10), 1956. https://doi.org/10.3390/w11101956.

[22] Macián-Pérez, J. F.; García-Bartual, R.; Huber, B.; Bayon, A.; Vallés-Morán, F. J. Analysis of the Flow in a Typified USBR II Stilling Basin through a Numerical and Physical Modeling Approach. Water 2020, 12 (1), 227. https://doi.org/10.3390/w12010227.

[23] Fadaei, K. E.; Barani, G. A. Numerical Simulation of Flow over Spillway Based on the CFD Method. 2014.

[24] Dehdar-behbahani, S.; Parsaie, A. Numerical Modeling of Flow Pattern in Dam Spillway's Guide Wall. Case Study: Balaroud Dam, Iran. Alexandria Eng. J. 2016, 55 (1), 467–473. https://doi.org/10.1016/j.aej.2016.01.006.

[25] Aydin, M. C.; Isik, E.; Ulu, A. E. Numerical Modeling of Spillway Aerators in High-Head Dams. Appl. Water Sci. 2020, 10 (1), 42. https://doi.org/10.1007/s13201-019-1126-2.

[26] Imanian, H.; Mohammadian, A. Numerical Simulation of Flow over Ogee Crested Spillways under High Hydraulic Head Ratio. Eng. Appl. Comput. Fluid Mech. 2019, 13 (1), 983–1000. https://doi.org/10.1080/19942060.2019.1661014.

[27] Bayon, A.; Valero, D.; García-Bartual, R.; Vallés-Morán, F. José; López-Jiménez, P. A. Performance Assessment of OpenFOAM and FLOW-3D in the Numerical Modeling of a Low Reynolds Number Hydraulic Jump. Environ. Model. Softw. 2016, 80, 322–335. https://doi.org/10.1016/j.envsoft.2016.02.018

[28] .Aydin, M. C.; Ozturk, M. Verification and Validation of a Computational Fluid Dynamics (CFD) Model for Air Entrainment at Spillway Aerators. Can. J. Civ. Eng. 2009, 36 (5), 826–836. https://doi.org/10.1139/L09-017.

[29] Liu, X.; Zhang, J. Computational Fluid Dynamics; Liu, X., Zhang, J., Eds.; American Society of Civil Engineers: Reston, VA, 2019. https://doi.org/10.1061/9780784415313.

[30] Versteeg, H. K.; Malalasekera, W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method; Pearson education, 2007.

[31] ANSYS, User Manual, 2020.

[32] Flow-3d, User Manual. 2016.

[33] Andersson, B.; Andersson, R.; Håkansson, L.; Mortensen, M.; Sudiyo, R.; Van Wachem, B. Computational Fluid Dynamics for Engineers; Cambridge University Press, 2011.

[34] Sarwar, M. K.; Ahmad, I.; Chaudary, Z. A.; Mughal, H.-U.-R. Experimental and Numerical Studies on Orifice Spillway Aerator of Bunji Dam. J. Chinese Inst. Eng. 2020, 43 (1), 27–36. https://doi.org/10.1080/02533839.2019.1676452

[35] Hirt, C. .; Nichols, B. . Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. J. Comput. Phys. 1981, 39 (1), 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

[36] Bayon, A.; Toro, J. P.; Bombardelli, F. A.; Matos,

J.; López-Jiménez, P. A. 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. 2018, 19, 137–149. https://doi.org/10.1016/j.jher.2017.10.002.

[37] Hirt, C. W.; Sicilian, J. M. 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, 1985; pp 450–468.

[38] Al-Hashimi, S. A. M.; Huda M. Madhloom; Rasul M. Khalaf; Thameen N. Nahi; Nadhir A. Al-Ansari. Flow over Broad Crested Weirs: Comparison of 2D and 3D Models. J. Civ. Eng. Archit. 2017, 11 (8), 769–779. https://doi.org/10.17265/1934-7359/2017.08.005.

[39] Daneshkhah, A. R.; Vosoughifar, H. R. A Mesh Convergence Study for the Flow over Ogee Spillways. 2010.

[40] Hirsch, C. Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics; Elsevier, 2007.

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

[42] Azimi, H.; Shabanlou, S.; Kardar, S. Flow Field within Rectangular Lateral Intakes in the Subcritical Flow Regimes. Model. Earth Syst. Environ. 2019, 5 (2), 421–430. https://doi.org/10.1007/s40808-018-0548-4.

[43] Kumcu, S. Y. Investigation of Flow over Spillway Modeling and Comparison between Experimental Data and CFD Analysis. KSCE J. Civ. Eng. 2017, 21 (3), 994–1003. https://doi.org/10.1007/s12205-016-1257-z.

[44] Shojaeian, Z.; Dalir, A. H.; Farsadizadeh, D.; Salmasi, F. Investigation of Hydraulic Jump Characteristics in Divergent Rectangular Sections on Inverse Slope. 2011.

[45] Shahheydari, H.; Nodoshan, E. J.; Barati, R.; Moghadam, M. A. Discharge Coefficient and Energy Dissipation over Stepped Spillway under Skimming Flow Regime. KSCE J. Civ. Eng. 2015, 19 (4), 1174–1182. https://doi.org/10.1007/s12205-013-0749-3.