Comparison Between Numerical Flow3d Software and Laboratory Data, For Sediment Incipient Motion.

Rasool Kosaj¹, Rafid S Alboresha¹, Sadeq O Sulaiman¹
¹Dams and Water Resources Engineering Department, College of Engineering, University of Anbar.
E-mail: Ras19e4002@uoanbar.edu.iq, rafid.alboresha@uoanbar.edu.iq, sadeq.sulaiman@uoanbar.edu.iq

Abstract: In this paper, the laboratory data were compared with computational fluid dynamics (CFD) Flow3D for predicting the beginning of sediment incipient motion in rigid boundary channel for two types of sands, irrigation, and sewer types, in rectangular flume (0.5*0.5) m cross-section. Tests were made for soil samples with different diameters, specific weights. The testing was performed in slopes ranging from 0.001-0.003 for irrigation types and 0.0025-0.025 for sewer types depending on the original parameter. The Flow-3D software has simulated the laboratory work using scouring models MPM and Nielsen. The relation between sediment incipient motion velocity, particle size, and channel bed slope was predicted. The results were relatively more than laboratory data for the MPM model, while grating convergence for Nielsen model, especially for small diameter sediment. Also, the laboratory results are more close to the results of Flow3D using the Nielsen model when the value of bed slope of the channel is greater, and vice versa when the slope decreases.

Keywords: Incipient motion, Flow3D, Flow velocity, sediment transport

1. Introduction
The movement of sediment in open channels is a complicated mechanism. Sediment incipient motion in a rigid rectangular open channel threshold condition is the state at which a few sediment particles started to move [1]. The movement of sediment in open channels is a completed mechanism, and the mechanics of this phenomenon has not yet been fully understood. To this extent, determination of sediment threshold incipient motion of sediment, become quite an important issue in the drainage system design as a Hydraulic Engineering practice [2]; [3]. There are several studies in the field of initial motion for sediments in rigid boundary channels. This study aims to determine the initial sediment motion for the rectangular channel using Flow3D software, then comparing with laboratory results by R.kosaj et.al 2021 [4].

The Numerical Computation for flow is known as the Computational fluid dynamics (CFD) program, is a numerical tool, used a computer technique to evaluate, describe and predict fluid motion within a domain of interest [5]; [6]. More recently, numerical models in most civil engineering disciplines have become a tool for solving complex problems that are expensive in time and cost in the laboratory [7]; [8]; [9]. CFD has proven to be a valuable and sometimes indispensable tool for designing hydraulic structures [10]; [11].
Generally, CFD science includes three main branches: The first branch is fluid mechanics to represent physical phenomena in the form of differential equations; the second branch is mathematics or the mechanism of converting the equations that represent the physical phenomena of the fluid transition to a series of algebraic equations; and the last branch is computer science to convert algebraic equations into a code understood by a computer [8]. The Flow-3D platform [12]; [13]; [14]; [15]; [16]; [17]; [18]; [19]; [20]; [21], are widely used by many research and they have proven their effectiveness in flow modeling. In the recent decade, numerical modeling has become one of the most important methods for studying flow properties [3]. The 3D CFD model Flow-3D v11.2 is used as the default software in this work to simulate laboratory analysis for sediment incipient motion in a channel and evaluated the results. Flow-3D is a multi-purpose program developed by Flow Science Company to portray turbulence and free surface flow.

2. Numerical Model
As a basic, the CFD model in an open channel depends on the continuity and Navier Stocks equations to solve complicated flow problems in boundary channels. These equations can be expressed for compressible and incompressible flow. In general, the continuity of mass and Navier Stocks equations are partial nonlinear differential equations and not easy to solve numerically or analytically, if not impossible [22]. It also offers different calculation models that the user can activate to fulfill hydraulic requirements [23]. Flow-3D employs a self-correcting approach as well as an autonomous convergence criterion setting that adapts to whatever occurs during the numerical solution process. It solves the continuity and Navier-Stokes equations that characterize fluid flow using a Finite Volume Method (FVM). The Flow-3D discretizes the computational domain into a hexahedral mesh with different sizes using Cartesian coordinates [24]. Cell simulation in computing domain grids for channel models could be one of five scenarios: totally fluid, partially solid and fluid, completely solid, partially fluid, and empty cell [20].

2.1 Volume of Fluid (VOF)
The volume of Fluid is one of the most extensively used ways to capture the position of the free surface [25]. It handles the difficult problem of variable flow and properly tracks the free surface [26]. The VOF technology in Flow-3D identifies whether cells are filled, partially filled, or partially filled with fluids or solids. Where F denotes the fluid percentage in each cell, which can range from 0 to 1. When the value of F equals 0, the cells are filled with air. F equal 1 denotes cells that are filled with water. For each cell in the computing domain, the interface of tracking between water and air for treating free surfaces gives a value ranging from one to zero [27]. (See the figure 1).

![Figure 1. Cases for water volume friction.](image)

With the use of the volume of fluid method, the continuity and RANS equations for incompressible flow has been applied at the 3D Cartesian coordinate system as follow:
The volume fraction of or two type eri sewer) for each type range of slopes 0.001 though lim

Therefore, it was relied upon to determine it thr other CFD programs in general, there is no option to specify the initial velocity of the sediment.

Numerically for the score Model, sediment can exist as the packed bed. The model calculates packed sediment transport in each mesh cell containing the bed interface, using the equation of

2.1.1 Meyer-Peter Müller (1948).

The Meyer-Peter and Müller (MPM) equation (1948) [29] was one of the earliest equations developed and is still one of the most widely used. It is a simple excess shear relationship.

\[
\left( \frac{k}{k_r} \right)^{3/2} \gamma RS = 0.047(\gamma s - \gamma)d^2 \left( \frac{B_s}{B} \right)^{2/3} \left( \frac{Y_s - Y}{Y} \right)^{2/3} \]

(5)

Where \( g \), Unit sediment transport rate in weight/time/unit width, \( k_r \) is a roughness coefficient, \( k' \) is a roughness coefficient based on the grains, \( d \) is the sediment median diameter, and \( \gamma, \gamma_s \) is the unit weight of water and sediment respectively.

2.1.2 Nielsen (1992)

In (1992) Nielsen proposed an empirical formula that sets the Shields parameter to be a sum of two terms [30]. One term is a function of the near-bed (free-stream) velocity and the other is a function of the near-bed acceleration. The weighting of each term is determined by an adjustable model parameter.

\[
q_s \sqrt{g d^2(s-1)} = \left( \frac{12}{9} \frac{\tau_{im}}{\rho g d^2(s-1)} - 0.05 \right) \frac{\tau_{im}}{\rho g d^2(s-1)}
\]

(6)

Where \( q_s \) is volumetric sediment discharge and \( s \) is sediment-specific wight. The suspended sediment is represented as a scalar mass concentration in the fluid. For each species, the suspended sediment concentration is calculated by solving a transport equation.

3. Methodology

The simulation was carried out for laboratory work using Flow3D software. To determine the initial movement of the sediments. To perform this simulation, models were made as in the laboratory experiments, two types of sediments (irrigation and sewer) for each type range of slopes 0.001 - 0.003 for irrigation and 0.0025-0.02 for sewer. During the simulation, a velocity range was imposed starting from 0 and ending with 0.55 meters per second, for 240 seconds as shown in the figure 2, and a variable depth proportional to the change in velocity. There is one thing to mention that in Flow3D as well as in other CFD programs in general, there is no option to specify the initial velocity of the sediment. Therefore, it was relied upon to determine it through limiting the time for the beginning of the concentration of suspended sediments, then evaluating the velocity using the figure 2, this methodology is given as an indicator of the beginning of the sediment movement.

The results of Flow3D simulations for two types of sediment scour equations (Meyer Peter & Muller (MPM) and Nielsen) is a comparison with experimental laboratory data. The results of the Flow-3D software such as incipient motion velocity for each diameter, and effect of channel slope, have been
compared with the laboratory data. The main comparison objective is to evaluate the success of the numerical model in the simulation of the sediment motion in channels.

![Figure 2](image.png)

**Figure 2.** Change of velocity with time in the inlet boundary.

4. **Flow-3D Model setup**

Using computational fluid dynamic (CFD) flow-3D, the simulation steps in the Flow-3D platform can be summarized as follows:

4.1 **General**

The first step is to define the simulation duration (240 seconds), the type of flow (incompressible or compressible), the number of fluids (one), and the units (SI). This is referred to as a general step.

4.2 **Physics models**

This step includes several calculation models that the user can choose from, such as gravity (X=0, Y=0, Z= -9.81), turbulence models, cavitation models, surface tension models, air entrainment models, shallow water, sediment scour, and so on. The choice of models is determined by the study's goal, and more than one model can be determined (increasing computational time). As shown in the figure 3, the chosen model in this study includes gravity, sediment scour, density, and viscosity.

For scour model, we analyzed using to equations Meyer Peter & Muller (MPM) and Nielsen.
4.3 Fluid
Specify the material properties of the fluid (water) like density (998 kg/m³), viscosity (0.001 kg/s/m) and Temperature (20 °C).

4.4 Mashing & Geometry
The fourth step is the important one, that made of:

4.4.1 Geometry
In this study the geometry is simple, made of two-component, one of these components is defined as a solid, refers to the channel base and the second is defined as packed sediment refers to sediment sample as shown in the figure 4.
Grid generation is the second essential step of preprocessing, and it is a critical consideration for obtaining numerical solutions for the governing equations of the CFD problem. A mesh can be classified in a variety of ways in general. The grid can be divided into tetrahedral, pyramid, triangular prism, or hexahedral meshes in the 3D domain, depending on the cell shape. The Flow-3D platform used Cartesian coordinates to create hexahedral mesh planes that are uniform or non-uniform (see the figure 5). The structured mesh (also known as uniform mesh) is more accurate than the unstructured mesh in most cases. the fixed mesh is ideal for the computation of geometrically complex free-surface flows, including hydraulic jumps, and it is also curate and stable over time [31].

Figure 5. a) Grid generation for channel (flume) model. b) uniform mesh is automatically defined. c) uniform mesh plain at sediment edges after making the plain lines parallel to the model.

As we can see in Fig 5a, the mesh plane is automatically defined as a uniform mesh. There is an aspect to be paid attention to, the plane lines should be as touched as possible with the model to achieve a well-defined shape (See the figure 5c).

4.4.3 Boundary Conditions
With Flow-3D, six boundary conditions are representing the boundary of hexahedral mesh block (X-min, X-max, Y-min, Y-max, Z-min, Z-max) (see the figure 6). In the present study, the X-min denote the channel inlet; X-max is the channel outlet take it as continuity; Y-min, Y-max is channel sides; Z-min is the channel base, and Z-max is the top of the computational domain take it as free surface.
4.4.4 Initial Conditions
The initial conditions are a useful tool to decrease the computational time required for a simulation to reach a steady state. The initial conditions are used always to determine an initial channel water level by using a water level value of available information or by the water levels recorded on the experimental model. Observed from the experimental model, to make the sediment samples fixed at begin of flow against the soil erosion by initial wave, so we take initial depth is 8 cm.

4.4.5 Monitoring Tools
The monitoring tools (probes and flux surface) were added. Most CFD software includes them, and they can be used to calculate global flow parameters such as discharge, flow depth, velocity, suspended load, and pressure.

4.5 Output
Define simulation parameters such as time step size, result output intervals, or history data (2 seconds is set to reduce output file size), as well as hydraulic parameters extracted.

4.6 Numerical Options
The Generalized Minimum Residual (GMRES) algorithm was used by Flow-3D software to solve non-linear physical phenomena because of its rapid convergence and parallel efficiency [32]. The default selections are used in the numerical options in most cases; however, the default momentum advection algorithm is a first-order upwind differencing method in the case of momentum advection. In both space and time, this option is first-order accurate. A second-order method can be used when greater accuracy is required. In both space and time, this option is second-order accurate [33]. The second order was chosen for this study.

4.7 Calculation Begins
After completing the previous steps, click on the run calculation icon to get the required results.
4.8 FlowSight
Postprocessing is the final step in the modeling. The FlowSight application is the main element for analyzing the required output in Postprocessing. Flow-3D, in conjunction with Flow Sight, offers a powerful and straightforward method for comprehending and sharing simulation results.

5. Results
The main purpose of this part is to display and analyze CFD flow3D results. Then, the results are discussed including comparing the experimental data with numerical simulation data for the incipient motion velocity and effect of bed slope. Fig 8 shows some numerical profile results.

5.1 Velocity, channel bed slope, and particle diameter
To determine the initial velocity of the sediments as mentioned in paragraph 3 depending on the concentration of suspended sediments. The Figure 9, show an example of the relationship between the concentration of suspended sediments over time for each of the irrigation and sewers canals using MPM and Nielsen methods. The results for each type of channel with each software method are shown in Tables 1, and 2, the relation between sediment incipient motion velocity with particle size and channel bed slope. the results show that the velocity required for incipient motion in software using Meyer Peter and Muller PMP model is more than the Nielsen model.

![Figure 7](image_url)

**Figure 7.** Numerical flow3D profiles for, a) Sediment suspension Concentration profiles at initial motion, b) Sediment suspension Concentration profiles after initial motion, c) Shear stress, d) Flow velocity.
Figure 8. Numerical results for suspended sediment concentration for slope 0.002 for sewer channel. The values of scl(1,2,3,4,5) represent the sediment gradients (1.18, 0.6, 0.15 and 0.075) mm respectively, a) Nielsen, b) Meyer Peter and Muller PMP.

Table 1. Numerical results for irrigation channel using Flow3D.

| Slope (mm) | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen | Nielsen |
|------------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|---------|
| 1.18       | 0.44    | 0.43    | 0.36    | 0.33    | 0.28    | 0.5253  | 0.5     | 0.48    | 0.44    | 0.38    |
| 0.6        | 0.39    | 0.36    | 0.32    | 0.27    | 0.25    | 0.44    | 0.43    | 0.4     | 0.37    | 33      |
| 0.15       | 0.32    | 0.32    | 0.27    | 0.25    | 0.21    | 0.4     | 0.39    | 0.365   | 0.34    | 0.3     |
| 0.076      | 0.3     | 0.29    | 0.26    | 0.23    | 0.19    | 0.39    | 0.38    | 0.36    | 0.34    | 0.29    |
Table 2. Numerical results for sewer channel using Flow3D.

| slope d mm | 0.0025 | 0.005 | 0.01 | 0.015 | 0.02 | 0.0025 | 0.005 | 0.01 | 0.015 | 0.02 |
|------------|--------|-------|------|-------|------|--------|-------|------|-------|------|
| 1.18       | 0.38   | 0.35  | 0.34 | 0.29  | 0.27 | 0.46   | 0.4   | 0.37 | 0.35  | 0.33 |
| 0.6        | 0.33   | 0.31  | 0.27 | 0.26  | 0.24 | 0.38   | 0.34  | 0.32 | 0.3   | 0.28 |
| 0.15       | 0.27   | 0.26  | 0.24 | 0.22  | 0.2  | 0.33   | 0.31  | 0.28 | 0.27  | 0.26 |
| 0.076      | 0.25   | 0.23  | 0.22 | 0.2   | 0.18 | 0.3    | 0.28  | 0.27 | 0.23  | 0.23 |
|            | 0.24   | 0.22  | 0.21 | 0.2   | 0.18 | 0.29   | 0.27  | 0.26 | 0.23  | 0.22 |

5.2 Comparison Between Flow-3D And Laboratory Results.

Using CFD Flow3D software, the figure 10, and 11 show the relation between sediment incipient motion velocity, particle size with specific channel bed slope, (0.001-0.003) for irrigation, and (0.0025-0.02) for sewer. The results show that the velocity required for incipient motion in Flow3D using MPM equation is more than laboratory data, while the results using Flow3D with Nielsen equation show that the velocity required for incipient motion is relatively more than laboratory data with somewhat similar. Also, the laboratory results for the sewer channel are given more convergence with the flow3D Nielsen model more than other.

Through the figure 10, and 11, it can be seen that the laboratory results are more close to the results of Flow3D using the Nielsen model when the value of the bed slope of the channel is greater, and vice versa when the slope decreases. Also, the convergence between the laboratory results with the Flow3D Nielsen equation is greater when the sediment size is decreasing. Finally, it can be said that the results of Flow3D using the Scouring Nielsen model give results that are closer to the laboratory work. The reason for this can be attributed to the fact that The MPM experiments mostly examined uniform gravel, making the transport function MPM most applicable in gravel systems. MPM tends to underpredict the transport of finer materials [29].
Figure 9. Relation between laboratory data with Numerical data using Nielsen and MPM equations for irrigation channel for slope, a) 0.002, b) 0.001, c) 0.003.

Figure 11. Relation between laboratory data with Numerical data using Nielsen and MPM equations, for sewer channel for slope, a) 0.01, b) 0.0025, c) 0.02.
6. Conclusions

Computational fluid dynamics (CFD) have developed in recent years and have given satisfactory results compared to the laboratory model results. Today, the CFD numerical models can be used to simulate the sediment motion in rigid channels with considerable accuracy, low cost, and less time to study more scenarios in comparison with the laboratory models. The Flow-3D software has been used as the laboratory parameter to modeling sediment incipient motion in the rigid rectangular channel using scouring models MPM and Nielsen. tests were made for soil samples with a median diameter (0.3, 0.8) mm for irrigation and sewer channel respectively, where the specific weight (s) is ranging from 2.59-2.67 for irrigation sample and (2.53-2.63) for sewer sample. The granular gradient is ranging (0.075-1.18) mm for the irrigation samples and (0.075-2) mm for the sewer sample. For irrigation canals, the slope was taken (0.001-0.003), while for sewer canals were taken (0.0025-0.02) depending on original data. The two main objectives including verifying the accuracy of numerical models compare with laboratory results; choose the most successful model to represent the sediment motion.

The numerical simulation using CFD Flow3D software shows the relation between sediment incipient motion velocity, particle size, and channel bed slope is relatively more than laboratory data for MPM model, while for Nielsen model the grate convergence especially for small diameter sediment. Also, the laboratory results are closer to the results of Flow3D using the Nielsen model when the value of bed slope of the channel is greater, and vice versa when the slope decreases. Finally, Flow3D can be used as an alternative tool to give an indicator for laboratory data to reduce time and cost.

Reference

[1] Dey S and Debnath K, (2000). Influence of streamwise bed slope on sediment threshold under streamflow. J. Irrig. Drain. Eng. 126(4), 255-263.
[2] Sulaiman S O, Al-Ansari, N., Shahadha, A. et al. Evaluation of sediment transport empirical equations: case study of the Euphrates River West Iraq. Arab J Geosci 14, 825 (2021). https://doi.org/10.1007/s12517-021-07177-1.
[3] Yaseen Z M, Sulaiman S O, Sharif A. (2021) The Nature of Tigris–Euphrates Rivers Flow: Current Status and Future Prospective. In: Jawad L.A. (eds) Tigris a Environment from Headwaters to Mouth. Aquatic Ecology Series, vol 11. Springer, Cham. https://doi.org/10.1007/978-3-030-57570-0_8
[4] Kosaj R, Alboresha R, and Sulaiman S, University of Anbar, sediment incipient motion in the rigid boundary open channel of the city of Ramadi, Iraq. Design Engineering, 2021, p.1474-1484.
[5] Liu N, Zhang J, 2019. Computational Fluid Dynamics, ASCE Publications. American Society of Civil Engineers, Reston, VA. https://doi.org/10.1061/9780784415313.
[6] Mustafa A S, Sulaiman S O and Hussein O M. (2016) “Application of SWAT Model for Sediment Loads from Valleys Transmitted to Haditha Reservoir”, Journal of Engineering, 22(1), pp. 184–197.
[7] Dargahi J, 2006. Physical model Study and 3D Numerical Simulations. Hydraul. Eng. 132, 899–907. https://doi.org/10.1061/(ASCE)0733-9429(2006)132:9(899)
[8] Tu J, Yeoh G H, Liu C, 2018. Computational fluid dynamics: a practical approach. Butterworth-Heinemann.
[9] Sulaiman S O, Abdullah H H, Al-Ansari, N., Laue, J., Yaseen, Z.M. (2021). Simulation model for optimal operation of Dokan Dam reservoir northern of Iraq. International Journal of Design & Nature and Ecodynamics, Vol. 16, No. 3, pp. 301-306. https://doi.org/10.18280/ijdne.160308
[10] Sarwar M K, Ahmad I, Chaudary Z A, Mughal H U R, 2020. Physical model 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.
[11] 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

[12] Akintorinwa O J, Ajayi C A, Oyedele A A, 2020. Determination and establishment of the empirical relationship between magnetic susceptibility and mechanical properties of typical basement rocks in Southwestern Nigeria. Int. J. Phys. Sci. 15, 70 – 89. https://doi.org/10.5897/IJPS2020.4870.

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

[14] 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.

[15] 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.

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

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

[18] Ghanbari R, Heidarnejad M, 2020. Physical model and numerical analysis of flow hydraulics in triangular and rectangular piano key weirs. Water Sci. 1– 7. https://doi.org/10.1080/11014929.2020.1724649.

[19] 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 Stillling Basin through a Numerical and Physical Modeling Approach. Water 12, 227. https://doi.org/10.3390/w12010227.

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

[21] Yildiz A, Yarar A, Kumcu S Y, Marti, A.L., 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.

[22] Ho D K H, Boyes K M, Donohoo S M, 2001. Investigation of behavior under increased maximum flood by computational fluid dynamics technique, in 14th Australasian Fluid Mechanics Conference, Adelaide University, Adelaide, Australia.

[23] Morais V H P, Gireli T, Vatavuk P, 2020. Numerical and physical model models applied to an ogee crest spillway and roller bucket stilling basin. RBRH 25. https://doi.org/10.1590/2318-0331.252020190005.

[24] Yusuf F, Micovic Z, 2020. Prototype-Scale Investigation of Spillway Cavitation Damage and Flow3D Numerical Modeling of Mitigation Options. J. Hydraul. Eng. 146, 04019057. https://doi.org/10.1061/(ASCE)HY.1943- 7900.0001671.

[25] Hirt C W, Nichols B D, 1981. Volume of fluid (VOF) method for the dynamics of free boundaries, Journal of Computational Physics, Volume 39, Issue 1, January 1981, Pages 201-225, https://doi.org/10.1016/0021-9991(81)90145-5.

[26] Simsek O, Akoz M S, Soydan N G, 2016. Numerical validation of open channel flow over a curvilinear broad-crested weir. Prog. Comput. fluid Dyn. an Int. J. 16, 364–378.

[27] 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.

[28] Parsaie A, Haghiahi A H, Moradinejad A, 2015. CFD modeling of flow pattern in the approach channel. Sustain. Water Resour. Manag. 1, 245–251. https://doi.org/10.1007/s40899-015-0020-9.

[29] Meyer-Peter E and Müller R, 1948. Formulas for bed-load transport, Proceedings of the 2nd Meeting of the International Association for Hydraulic Structures Research. pp. 39–64.

13
[30] Nielsen P, 1992, Coastal bottom boundary layers and sediment transport (Vol. 4). World scientific.

[31] Jacobsen J, Olsen N R B, 2010. Numerical modeling of the capacity for a complex spillway, January 2010 Water Management 163(6):283-288, DOI:10.1680/wama.2010.163.6.283.b

[32] Chanel P G, Doering J C, 2008. Assessment of spillway modeling using computational fluid dynamics. Can. J. Civ. Eng. 35, 1481–1485.

[33] 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.