Shahrood University of Technology

Journal of Hydraulic and Water Engineering (JHWE)

Journal homepage: https://jhwe.shahroodut.ac.ir

Iranian Hydraulic Association (IHA)

The Effect of Suspended Load on the Lower Discharge of Large Dams using Flow-3D Numerical Model

Seyed Mahmoud Samavaki[1], Mohammad Amel Sadeghi[2,*], MohammadReza Asli Charandabi[3]

[1] MSc Student, Department of Water Engineering, Faculty of Agriculture, Takestan Branch, Islamic Azad University, Takestan, Iran Islamic Azad University, Takestan Branch, Takestan, Iran.
[2] Assistant Professor, Department of Water Engineering, Faculty of Agriculture, Takestan Branch, Islamic Azad University, Takestan, Iran.
[3] BSc Graduated, School of Engineering, Damghan University, Damghan, Iran.

Article Info

Article history:
Received: 01 Dec 2024
Received in revised form: 01 Nov 2025
Accepted: 6 Feb 2025
Published online: 8 Feb 2025

DOI:
10.22044/JHWE.2025.15424.1046

Keywords
Bottom Outlet
Suspended Sediment
Flow-3D
Numerical Modeling
Hydrodynamics

Abstract

Bottom outlets in dams are critical structures for regulating flow and releasing sediment, particularly during floods and emergency situations. These systems play a vital role in ensuring dam safety and effective water management. While previous studies have primarily focused on flow simulation without considering suspended sediments, this omission overlooks a significant factor influencing outlet performance under flood conditions. Suspended sediments increase the density of the flow, which can substantially alter the hydraulic characteristics of the outlet system. This study investigates the effect of suspended sediment concentration on the hydraulic efficiency of bottom outlets, using Flow-3D software to model the flow dynamics within the bottom outlet of Siazakh Dam. Siazakh Dam is located 7 km south of Diwandara and 95 km north of Sanandaj city in the Kurdistan province of Iran. Initial calibration and validation of the model were performed using laboratory data. Simulations were conducted with suspended sediment concentrations of 3000, 6000, 9000, and 12,000 ppm to examine the impacts on discharge and key hydraulic parameters such as flow velocity and pressure distribution. The results reveal that as sediment concentration increases, the discharge rate decreases significantly due to higher flow density, which alters both velocity profiles and pressure distributions. At higher concentrations, discharge reduction exceeded 20%, accompanied by notable variations in pressure and flow velocity across different sections of the outlet system. This study highlights the importance of accounting for sediment load in the design and operational management of dam outlet systems, as this factor can significantly influence performance. Future studies could further investigate the impact of varying sediment shapes and sizes on system efficiency.

1. Introduction

Dams have long been essential structures for water resource management, providing vital benefits such as drinking water supply, agricultural support, hydroelectric power generation, and flood control (Aminian et al., 2019a, 2019b; Aminian et al., 2023; Emamgholizadeh et al., 2013; Emamgholizadeh et al., 2018; Fathi-Moghadam et al., 2011; Khatsuria, 2004). As the global demand for water and energy grows, constructing large dams has become increasingly critical. In Iran, due to climatic conditions and the necessity for effective

* *Corresponding author:* aslimohammad77@gmail.com

water management, many dams have been constructed nationwide (Bureau, 1977). However, maintaining and operating dams optimally involves challenges such as sedimentation and the need for emergency water release. One of the most crucial components in this context is the bottom outlet system, responsible for discharging water and sediment during emergencies. These systems are designed to clear accumulated sediments and regulate flow discharge during both regular and emergency operations, ensuring dam safety and managing flood flows (Arman et al., 2009; Emamgholizadeh et al., 2006; Emamgholizadeh & Fathi-Moghdam, 2014; Emamgholizadeh & Samadi, 2008; Rajaratnam & Beltaos, 1977).

Typically, bottom outlet systems are located at the dam's base, where they allow water and suspended sediments to be conveyed downstream. They consist of service and emergency gates, which control the discharge of flow and sediments (Borodina, 1969). During flood events, significant amounts of river sediments can be transported into the reservoir, accumulating over time and reducing both the storage capacity and efficiency of the dam. Managing suspended sediments within the flow poses a primary challenge for bottom outlets. Due to increased water density and changes in hydraulic properties caused by suspended sediments, the system's efficiency may be compromised. Understanding the impact of sediments on flow and hydraulic parameters is crucial, as increased density can lead to reduced discharge rates and operational difficulties (Liu, 2014). Bottom outlet systems are a vital component of dams, playing an important role in managing outflow, controlling floods, and discharging accumulated sediments from behind dams. These systems are typically employed to control reservoir outflow and provide emergency discharge of water and sediment.

Sedimentation is a serious problem, reducing reservoir lifespan and storage capacity. Reports show that millions of cubic meters of usable reservoir volume are lost yearly due to sediment build-up worldwide. This issue is particularly pressing in arid and semi-arid regions like Iran, where seasonal floods and substantial sediment loads are common. Consequently, designing bottom outlet systems that can handle suspended sediments effectively is essential for the optimal operation of dams. In most prior studies, simulations of bottom outlet systems have focused on pure water flow, without considering suspended sediment effects. However, in real-world scenarios-particularly in dams prone to severe floods-substantial suspended sediments are present in the flow, which can significantly affect outlet performance. This study investigates how suspended sediments influence hydraulic flow parameters in bottom outlet systems using Flow-3D software, which can simulate turbulent, multiphase flows. The primary objective is to assess the impact of varying sediment concentrations on discharge and flow pressure in the bottom outlet system of Siazakh Dam. This research provides insights for engineers seeking to improve outlet system design and predict performance under high-flow, high-sediment conditions, ultimately helping to optimize dam operations in flood-prone areas. Bottom outlet systems, especially in large dams, are essential for ensuring dam safety and preventing risks associated with excessive water and sediment accumulation (Wilcox, 1998).

In studies related to bottom outlet systems, flow is generally categorized into two types: free surface flow and pressurized flow. Pressurized flow, a critical flow condition in these systems, has received particular attention. Given the specific design and operational requirements of these systems, understanding the hydraulic parameters governing flow within bottom outlets is essential. Many studies have explored the hydraulic conditions, pressure, velocity, and discharge through these systems.

Experimental and laboratory studies form the foundation for much of the research on the hydraulics of bottom outlet systems, measuring and analyzing flow parameters using small-scale physical models. One of the pioneering studies in this field was conducted by (Rajaratnam & Beltaos, 1977), who examined flow profiles through sluice gates in bottom outlets and calculated the flow contraction coefficient, providing empirical curves for determining discharge and showing the effects of factors such as gate opening height and flow head. By (Sharma et al., 2007) similarly investigated flow through sluice gates in open channels, deriving equations for the discharge coefficient that demonstrated the direct influence of gate opening height and effective flow head on discharge. The study also examined how flow profile formation and gate geometry affect discharge rate. Another significant study was conducted by (Heller et al., 2005), who used a physical model of a dam bottom outlet system to investigate the impact of factors such as channel width, cross-sectional height, and gate opening on discharge. This study demonstrated that increasing the channel cross-section and gate opening height leads to higher discharge rates. In other studies, such as that by (Razavi & Ahmadi, 2017), the aeration of flow in bottom outlet systems was examined. Aeration is an important hydraulic consideration in these systems to prevent cavitation, which can damage hydraulic structures. The study investigated how aeration reduces pressure and prevents cavitation. With advancements in technology and the emergence of powerful computational fluid dynamics (CFD) software, accurate hydraulic flow modeling has become possible. Numerical simulations using 3D models can analyze complex parameters such as velocity, pressure, and discharge under various conditions, particularly useful for studying the turbulent, multiphase flows typical of bottom outlet systems. Flow3D is one of the software tools used for numerical hydraulic simulations. It employs the Volume of Fluid (VOF) method and Navier-Stokes equations, enabling precise simulation of turbulent, multiphase flows. (Speerli & Hager, 2000) used the Flow3D model to simulate flow through the Sefidrud Dam outlet system, demonstrating the accuracy of numerical modeling in reproducing velocity and pressure profiles.

Another study by (Dargahi, 2010) examined cavitation phenomena in the Minab Dam bottom outlet using HEC-RAS and numerical simulations, showing that high flow velocities and low pressures in certain sections can cause cavitation and structural damage to the channel. An important study by (Taghavi & Ghodousi, 2015) examined the impact of suspended sediment load on discharge in bottom outlet systems. Using Flow3D, this study simulated the effects of increased sediment concentration on discharge and flow pressure, revealing that as suspended sediment concentration rises, discharge decreases significantly, and flow pressure undergoes major changes. Ahmadi and (Emamgholizadeh et al., 2020; Jabary et al., 2014; Razavi & Ahmadi, 2017) also investigated the impact of suspended sediments on discharge in bell-mouth spillways. This study found that suspended sediments can reduce discharge by up to 27%, underscoring the importance of considering suspended sediment effects on discharge and bottom outlet performance.

One major operational issue for bottom outlet systems is the effect of suspended sediments on flow. Increased flow density and changes in hydraulic properties due to suspended sediments can reduce discharge and increase energy losses. Studies indicate that with rising sediment concentrations, discharge decreases, impacting the effectiveness of bottom outlet systems. In a study by (Dargahi, 2010), the effects of suspended sediments on flow through bottom outlet systems were examined using Flow3D (Barnea et al., 1985). This study found that as suspended sediment concentration increases, discharge decreases, and flow velocity and pressure profiles undergo significant changes. Liu (2014) also explored flow profile variations in the presence of suspended sediments. These studies showed that suspended sediments can induce flow fluctuations and alter hydraulic parameters, with particular attention to discharge reduction under various flood conditions (Te Chow, 1959).

Previous research highlights the critical role of bottom outlet systems in controlling flood flows and discharging suspended sediments. Many of these studies have analyzed flow behavior without considering the impact of suspended sediments. However, recent research has shown that suspended sediments can significantly reduce the efficiency of these systems. The presence of suspended sediments increases flow density and decreases discharge, an important consideration for the design and operation of bottom outlet systems (Manual, 2011).

2. Materials and Methods

2.1. Study area and Data

Siazakh Dam, one of Iran's major dams, is equipped with a bottom outlet system specifically designed to handle flood flows and discharge suspended sediments. This system includes key hydraulic and geometric features, such as service and emergency gates, a discharge conduit, and a stilling basin. Laboratory data related to this dam, sourced from Iran’s Water Research Institute, were utilized for model calibration. The aim of this study is to present computational fluid dynamics (CFD) equations and numerical simulation methods for modeling flow in dam bottom outlet systems using Flow3D software. This section outlines not only the governing flow and continuity equations but also the computational methods and the 3D geometry construction of the system, created in SolidWorks. The Siazakh Reservoir Dam is located in Kurdistan Province, 7 kilometers south of Divandarreh and 95 kilometers north of Sanandaj, on the Qezel Owzan River. The project aims to supply irrigation water for 22,000 hectares on the left and right banks of the river along the Divandarreh-Bijar route, regulate and control irregular flows of the Qezel Owzan River, increase employment, and improve the region’s environmental conditions. Siazakh Dam is an earth-fill dam with a clay core, with a reservoir capacity of 265 million cubic meters at its maximum level (1825.5 meters). The dam’s height is designed to be 74 meters from the riverbed, and its crest length is 285 meters.

2.2. Flow 3D Model and Governing Equations

The Flow3D model operates based on the Navier-Stokes equations for turbulent, multiphase flows. These equations include the continuity and momentum equations in three coordinate directions. The continuity equation for flow is as follows:

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + \frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) = 0 \quad (1)$$

In this equation, v_f represents the open volume fraction of the flow, ρ is the fluid density, and the velocity components (u, v, w) are in the directions $(\mathrm{x}, \mathrm{y}, \mathrm{z})$. A_x denotes the open surface fraction in the x direction, while A_{y} and A_z similarly represent the surface fractions in the y and z directions (Flow-3D Help). The momentum equation in fluids is as follows:

$$\begin{aligned}
\frac{\partial u}{\partial t} + \frac{1}{V_F}\left(uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right) &= -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x \\
\frac{\partial v}{\partial t} + \frac{1}{V_F}\left(uA_x\frac{\partial v}{\partial x} + vA_y\frac{\partial v}{\partial y} + wA_z\frac{\partial v}{\partial z}\right) &= -\frac{1}{\rho}\frac{\partial p}{\partial y} + G_y + f_y \\
\frac{\partial w}{\partial t} + \frac{1}{V_F}\left(uA_x\frac{\partial w}{\partial x} + vA_y\frac{\partial w}{\partial y} + wA_z\frac{\partial w}{\partial z}\right) &= -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z
\end{aligned} \quad (2)$$

In these equations, (G_x, G_y, G_z) represent the body accelerations, and (f_x, f_y, f_z) are the viscous accelerations. For the dynamic viscosity variable μ, the viscous accelerations are:

$$\rho V_F f_x = wsx - \left\{ \frac{\partial}{\partial x}\left(A_x \tau_{xx}\right) + \frac{\partial}{\partial y}\left(A_y \tau_{xy}\right) + \frac{\partial}{\partial z}\left(A_z \tau_{xz}\right) \right\}$$
$$\rho V_F f_y = wsy - \left\{ \frac{\partial}{\partial x}\left(A_x \tau_{xy}\right) + \frac{\partial}{\partial y}\left(A_y \tau_{yy}\right) + \frac{\partial}{\partial z}\left(A_z \tau_{yz}\right) \right\} \quad (3)$$
$$\rho V_F f_z = wsz - \left\{ \frac{\partial}{\partial x}\left(A_x \tau_{xz}\right) + \frac{\partial}{\partial y}\left(A_y \tau_{yz}\right) + \frac{\partial}{\partial z}\left(A_z \tau_{zz}\right) \right\}$$

In this formula:

$$\tau_{xx} = -2\mu\left\{\frac{\partial u}{\partial x} - \frac{1}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right\} \qquad \tau_{xy} = -\mu\left\{\frac{\partial v}{\partial y} + \frac{\partial u}{\partial x}\right\}$$
$$\tau_{yy} = -2\mu\left\{\frac{\partial v}{\partial y} - \frac{1}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right\} \qquad \tau_{xz} = -\mu\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\} \quad (4)$$
$$\tau_{zz} = -2\mu\left\{\frac{\partial w}{\partial z} - \frac{1}{3}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right\} \qquad \tau_{yz} = -\mu\left\{\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right\}$$

2.3. Shear Stress Parameters and Turbulence Modeling

In the presented equations, parameters wsx, wsy and wsz represent shear stresses at the walls. Removing these parameters eliminates wall shear stress effects. To accurately simulate flow, the RNG turbulence model, known for effectively modeling turbulent flows, was used. For turbulent flow simulation, various models, including two-equation models (k-ε), RNG, and large eddy simulations (LES), are commonly applied due to their capacity to capture turbulence under complex hydrodynamic conditions.

Numerical simulations employed computational fluid dynamics (CFD) methods, governed by the continuity and Navier-Stokes equations. With advances in computing, numerical methods have become more applicable than experimental and theoretical approaches. Numerical methods require domain meshing, followed by temporal and spatial discretization to convert differential equations into algebraic equations, which are solved across all mesh points using computers. The accuracy of these methods largely depends on the discretization approach. Factors causing variations in results include: (1) simplifications and assumptions, (2) discretization errors, (3) iterative and prolonged computational methods, and (4) simplifications in geometric conditions.

2.4. Simulation Steps

First, the geometry of the bottom outlet system was designed in software, and initial and boundary flow conditions were defined. Simulations were then conducted for four different suspended sediment concentrations: 3000, 6000, 9000, and 12000 ppm. These simulations included various gate openings and head conditions, with output data on flow rate, pressure, and velocity at different system sections. The 3D geometry of the Siazakh Dam's bottom outlet system, including service and emergency gates and the downstream tunnel, was created in SolidWorks, then imported into Flow-3D for simulation and meshing. The model includes service gates, emergency gates, and a downstream tunnel. Simulation steps in Flow-3D included defining the bottom outlet geometry, applying non-uniform adaptive meshing, setting boundary and initial conditions on inlet and outlet boundaries with specified inflow rates and fluid type (with and without suspended sediments), running the simulation for a set time, and analyzing results for flow rate, pressure, and velocity at different sections, with graphical representations for velocity, pressure, and outflow rate fields. The VOF (Volume of Fluid) method was used to detect free surfaces, and the FAVOR method to identify solid boundaries. The VOF approach in Flow-3D, based on the donor-acceptor cell approximation by Hirt and Nichols, calculates fluxes across each direction, taking into account partially filled elements when a free surface is present.

The major advantage of the VOF (Volume of Fluid) method is that the fluid moves within a fixed grid, eliminating the need for grid reshaping or reconstruction. Fluid masses can combine or separate based on dynamic laws

without needing a specific interface-tracking algorithm. The VOF method is particularly suitable for processes that undergo repeated wetting and drying, such as coastal waves or tidal flows. The new VOF transfer method involves three steps: (1) estimating the interface level with the horizontal surface, (2) transferring fluid volume according to local velocity, and (3) calculating new fluid fraction values in computational cells using the overlay method. The FAVOR (Fractional Area/Volume Obstacle Representation) method provides simplicity and accuracy in solving equations by assigning fractional areas and volumes to cells that contain fluid-solid interfaces. Its main advantage is the use of structured Cartesian grids, which facilitates meshing for complex geometries.
The FAVOR method works by using numerical algorithms that only require one set of values-such as pressure, velocity, or temperature-for each control volume. This contrasts with the need for large amounts of data to define geometry. FAVOR retains simple rectangular elements but can represent complex shapes in alignment with the averaged flow values for each element (Flow-3D Manual).

2.5. Modeling of the Siazakh Dam Bottom Outlet System

In bottom outlet systems of dams, ensuring adequate discharge capacity and flow rate, as well as reliable performance of the outlet system and related hydraulic and hydro-mechanical components-such as service and emergency gates, valves, and branches-is crucial for designers and operators. For accurately estimating and verifying these hydraulic functions, a scaled hydraulic model of the dam’s bottom outlet must be built and designed. In this study, a hydraulic model of the Siazakh Dam, available at the Water Research Institute of Iran's Ministry of Energy, will be used for numerical simulation purposes.

2.6. Laboratory Model for the Siazakh Dam Bottom Outlet System

In laboratory studies focusing on hydraulic and hydrodynamic principles, as well as dimensional analysis and scale effects, a 1:12 scale laboratory model was constructed and designed. The emergency and service gates of the Siazakh Dam bottom outlet system are simple sluice gates made of transparent Plexiglass. The emergency gate has an upstream-facing flow-resistant plate. Gate reinforcements, the angle of the bottom edge, sealing rubbers, and an upright shaft for gate movement have all been precisely constructed and installed. An appropriate scale is marked on the outer shaft for various gate openings, and the gate groove is marked accordingly. To measure the pressures acting on the gate, eight piezometers were installed, each securely connected to hoses that route through the open spaces within the gate to external manometers, allowing pressure readings from these points. The laboratory model of the Siazakh Dam’s bottom outlet system has eight piezometers on the service gate for pressure measurements. The outlet section of the laboratory model control room includes a tunnel with a diameter of approximately 3.5 m, connected to a stilling basin by two specific slopes and a curve. The model's inlet level is set at 1754.75 meters, guiding the flow through a 3.64% slope channel and then into the downstream stilling basin with a 10 % slope. Table1 (a), (b) and (c) present the variations in Froude numbers for different gate openings, head levels, and flow rates based on the laboratory model of the Siazakh Dam.

Table 1 (a). Changes of the Froude number for different gate openings and head levels for the laboratory model of the Siazakh Dam bottom outlet system

Gate Opening (%) service	Gate Opening (%) Emergency	Q (cms)	Fr
10	6	2.57	21.20
20	17	7.55	13.06
40	38	17.34	8.98
50	50	21.14	7.25
60	60	25.42	6.63
80	77	34.05	5.99

Table 1(b). Changes of the Froude number for different gate openings and head levels for the laboratory model of the Siazakh Dam bottom outlet system

Gate Opening (%) service	Gate Opening (%) Emergency	Q (cms)	Fr
10	5	3.16	34.31
20	16	9.89	18.74
40	35	22.89	13.41
50	49	31.69	11.20
60	58	37.96	10.42
80	79	53.42	9.05

Table 1(c). Changes in the Froude number for different gate openings and head levels for the laboratory model of the Siazakh Dam bottom outlet system

Gate Opening (%) service	Gate Opening (%) Emergency	Q (cms)	Fr
10	5	2.76	29.93
20	15	9.89	20.64
40	37	20.46	11.03
50	47	25.42	9.57
60	58	32.16	8.83
80	78	44.64	7.81
100	100	58.95	8.24

Figure 1 presents the variations in the Froude number within the discharge channel of the Siazakh Dam bottom outlet system.

Figure 2 shows the variations in pressure characteristics for a 100% gate opening at different head levels.

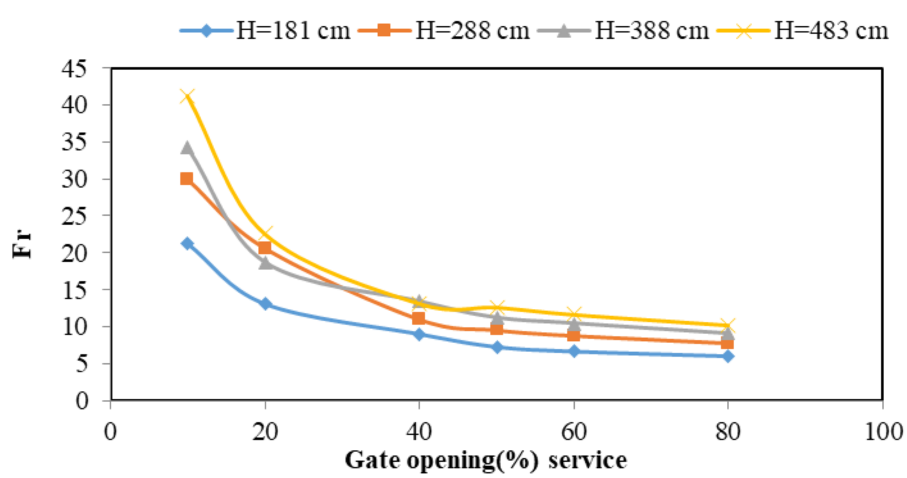


Figure 1. Changes in the Froude number within the discharge channel of the bottom outlet system.

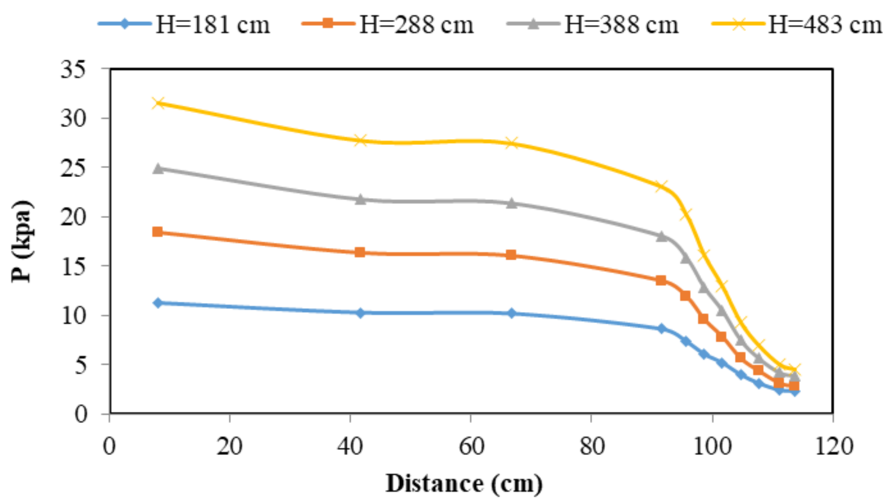


Figure 2. The pressure variations on the floor for a 100% gate opening.

The numerical model was calibrated using laboratory data. The outflow rates from the laboratory model were compared with the results of the numerical model, yielding an average relative error of less than 5%, indicating the high accuracy of the Flow-3D model in simulating hydrodynamic flows in bottom outlet systems.

The simulation results showed that with an increase in suspended sediment concentration, the outflow rate significantly decreased. This reduction in flow rate is due to the increased flow density and its impact on the hydraulic parameters of the bottom outlet system. To model hydrodynamic flow in bottom outlet systems within the Flow-3D environment (ver. 10.1), users must follow five steps: Navigator, Model Setup, Simulate, Analyze, and Display. Additionally, all necessary data for modeling can be entered as text files, with options for icon-based and graphical data entry for user convenience.

2.7. Effect of Sediment Concentration on Outflow Rate

As sediment concentration increased from 3000 ppm to 12000 ppm, the outflow rate gradually declined. At a concentration of 12000 ppm, the flow rate dropped by over 20% compared to flow without suspended sediments. This reduction was particularly noticeable in the early stages of the simulation. The results indicate that as sediment concentration increases, both the flow rate and discharge coefficient decrease, with a significantly higher reduction observed at high concentrations (12000 ppm). Simulations demonstrate that in pure water flow, the outflow rate is substantially higher than in flows with suspended sediments. These findings are presented numerically and graphically, and the reduction in the discharge coefficient has also been calculated. Figures 3(a) and (b) illustrate, respectively, the mesh block of the Siazakh Dam’s bottom outlet in the Flow-3D numerical model and the computational cells on the mesh block in the Flow-3D model.

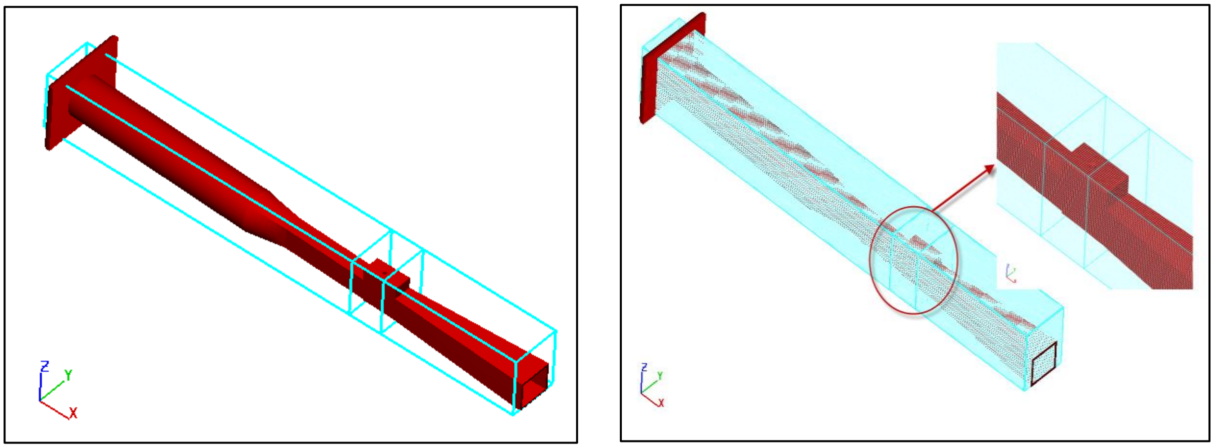


Figure 3(a) and (b). Mesh block of the Siazakh Dam's bottom outlet in the Flow-3D numerical model, along with computational cells on the mesh block in the Flow-3D model.

Using the boundary conditions in the Flow3D numerical model, the pressure should be applied as stagnation pressure to ensure accuracy in calibration and validation. As flow passes through the bottom outlet system, it should transfer toward the outlet boundary and ultimately exit the channel and mesh block, so the Outflow boundary condition must be applied to this boundary. The boundary conditions applied in the numerical model are shown in Figure 4.

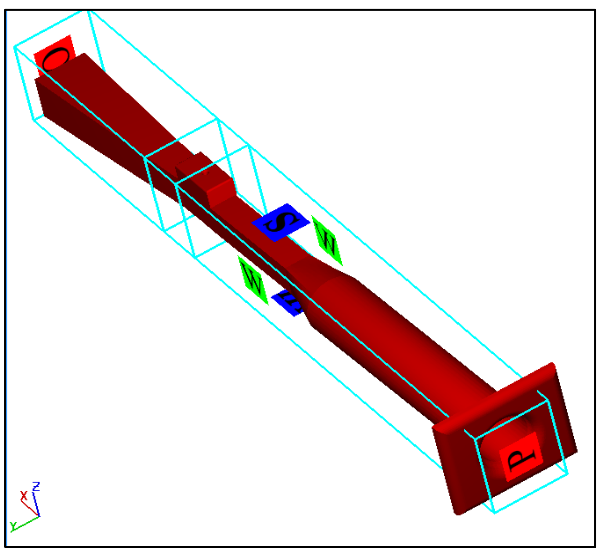


Figure 4. boundary condition in the numerical model.

The water reservoir is defined as the input fluid at the entrance to the channel, aligning with the conditions in the main dam reservoir. At the inlet and in the initial condition block (defining a water reservoir), this setup is applied in the numerical model. Figure 6 illustrates the application of initial conditions in the numerical model.

2.8. Viscous Stress, Thermal Conductivity, and Shallow Water Options

For viscous stress, thermal conductivity, and shallow water, users can choose between explicit or implicit solutions. In the momentum advection section, first-order or second-order solution options are available. The flow solver options include various settings, such as "Solve All Fluids Transport Equipment." Finally, all model input settings must be saved, preparing the program for hydrodynamic modeling. After execution, results can be viewed in the "Results" option. If "Existing" is selected, results are displayed with overall charts; if "Custom" is selected, results can be customized.

2.9. Analysis of Velocity and Pressure Profiles

Velocity and pressure profiles at different sections of the bottom outlet system indicated significant changes due to suspended sediments. Flow velocity decreased near the gates, and pressure in the discharge channel, especially at high sediment concentrations, increased.

3. Results and Discussions

Numerical modeling results were compared with experimental data. The flow rate variation charts showed good agreement between laboratory and numerical results, with differences below 5%, indicating high modeling accuracy. In the Flow3D model, both RNG and k-ε turbulence models can be used. The RNG model generally has advantages over k-ε, particularly in handling rapidly strained flows and geometrically complex surfaces (Yakhot & Orszag, 1986). (Razavi & Ahmadi, 2017) study also demonstrated that the RNG turbulence model provided more accurate results on spillways and hydraulic structures than other models. In this study, both models were tested under the same conditions, such as cell count, computation time, and mesh configuration. Results showed that the RNG turbulence model achieved better alignment with the physical model (pressure, water head, and flow rates) and produced lower numerical error. Calculation time was also reduced by 17% with the RNG model. Therefore, the RNG model was used for all main analyses and simulations. Table 2 presents error values for both RNG and k-ε turbulence models.

Table 2. Evaluation and Comparison of Numerical Flow Modeling Results Using RNG and k-ε Turbulence Models

Turbulence Model	Mean Relative Error (%)	Maximum Relative Error (%)	Minimum Relative Error (%)
k-ε	8	23	1
RNG	5	15	0.7

To ensure the numerical model's quantitative accuracy, parameters such as pressure, flow rate, and flow head were analyzed with different mesh sizes. Modeling was conducted under identical conditions for two mesh sizes: 1 cm and 0.5 cm. The impact of using finer meshes on computation time and parameter accuracy was assessed. Table 3 presents the maximum, minimum, and average relative error percentages for these two mesh sizes, along with the simulation time as a determining factor.

Table 3. Comparison and Results of Numerical Modeling Using Different Numerical Model Cell Counts

Grid Condition	Simulation Time	Mean Relative Error (%)	Maximum Relative Error (%)	Minimum Relative Error (%)
1×1×1 cm	66783	7	14	0.9
0.5×0.5×0.5 cm	109251	6.3	12	0.8

In the Table 3, it is observed that doubling the cell count does not significantly impact simulation accuracy, but it increases simulation time by nearly 90%, leading to a substantial extension of numerical model runtime. Given that the simulation results and errors are acceptable with the 1 cm mesh size, this mesh size will be used for all simulations.

It is concluded that the required cell count for the present numerical model is 800,000 cells. Additionally, to establish an appropriate profile, a 0.1-meter length of the channel behind the spillway should be set as the initial condition in the numerical model. In the hydrodynamic modeling of flow in the Siazakh Dam's bottom outlet system, boundary conditions were applied as follows: at the inlet boundary, the flow is set as a pressure head, while the outlet boundary is set to "Outflow." The side walls of the mesh block were designated as "wall" boundaries, and a "symmetry" boundary condition was applied to the top of the mesh. A 3D hydrodynamic simulation of flow in the Siazakh Dam bottom outlet system with a 50% gate opening is shown in Figure 5.

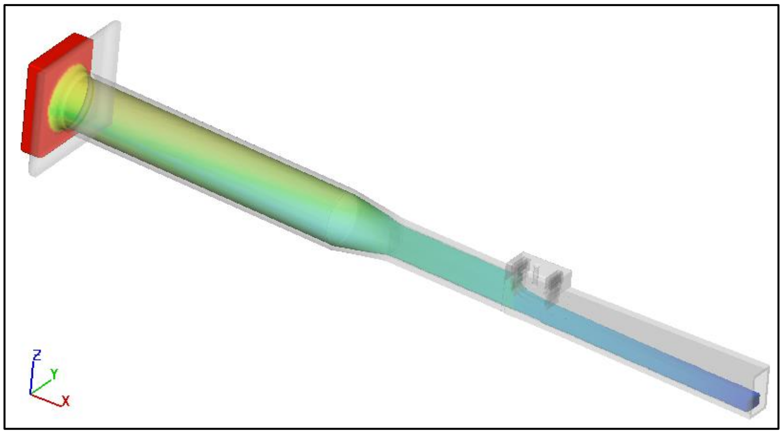
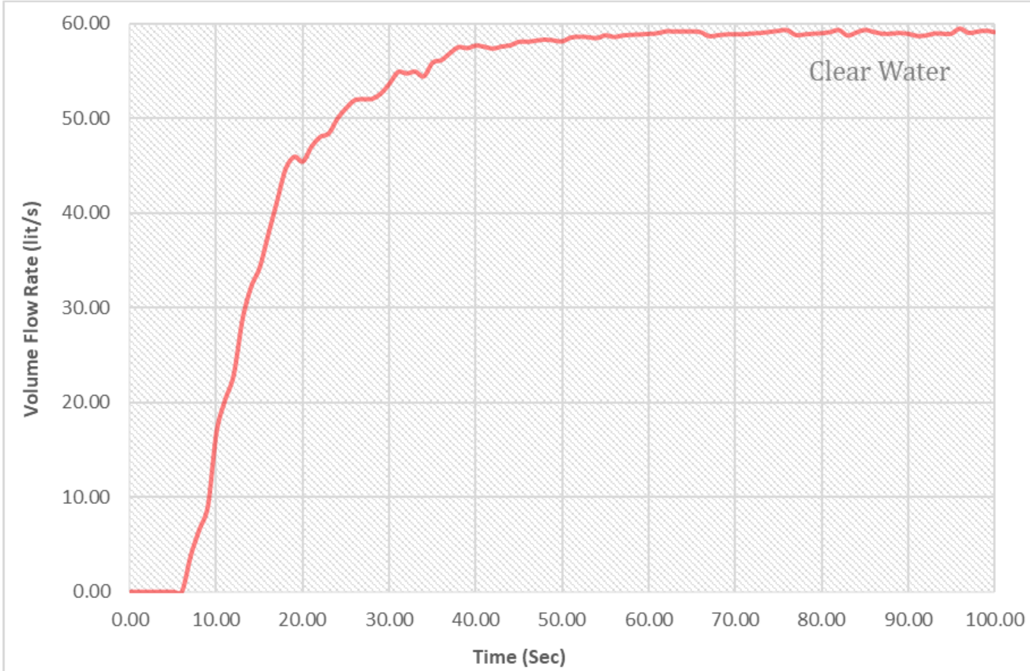


Figure 5. 3D hydrodynamic simulation of flow in the Siazakh Dam bottom outlet system with a 50% gate opening.

Using the output results, the outflow rate for each model with suspended sediment load was extracted over time. Figures 6 (a), (b), (c) and (d) show the outflow rate over time for different sediment concentrations of 3000, 6000, 9000, and 12000 ppm.

(a)



(b)

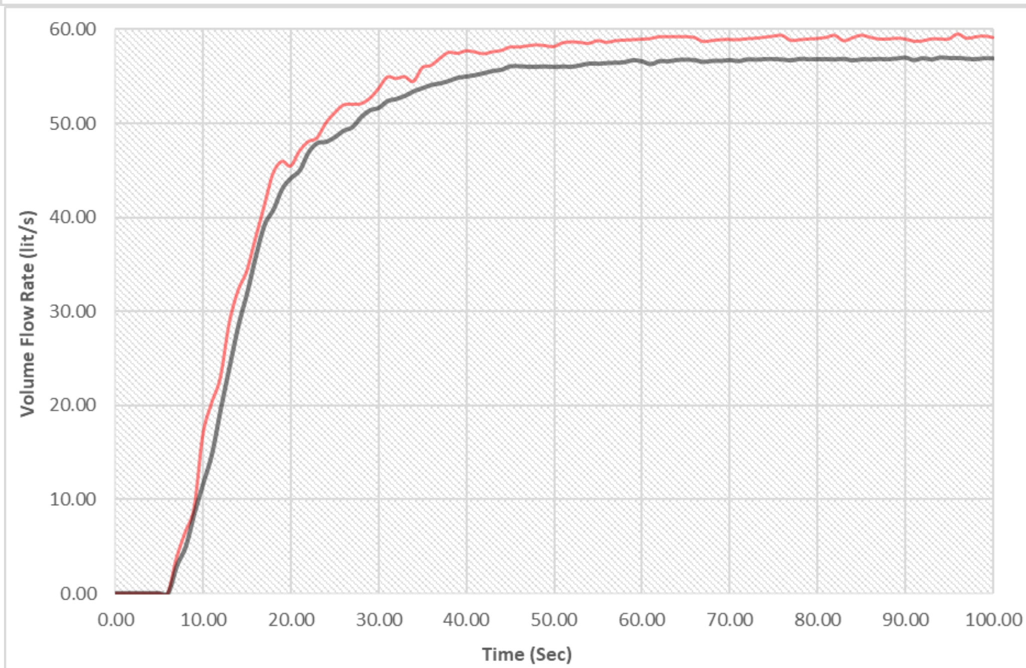
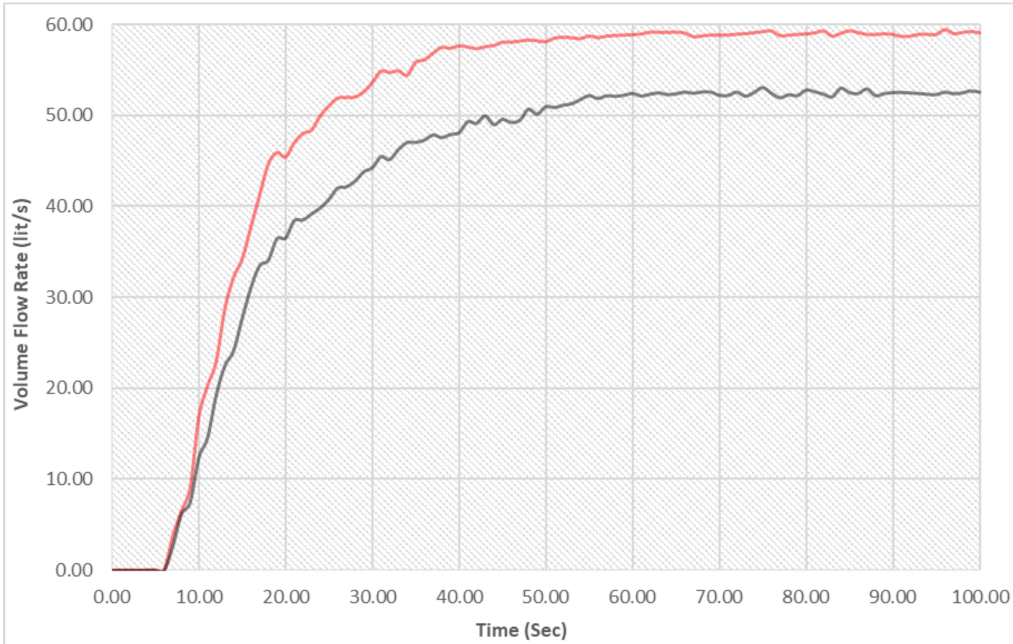


Figure 6 (a), (b), (c) and (d). Outflow rate over time for sediment concentrations of 3000, 6000, 9000, and 12000 ppm.

(c)



(d)

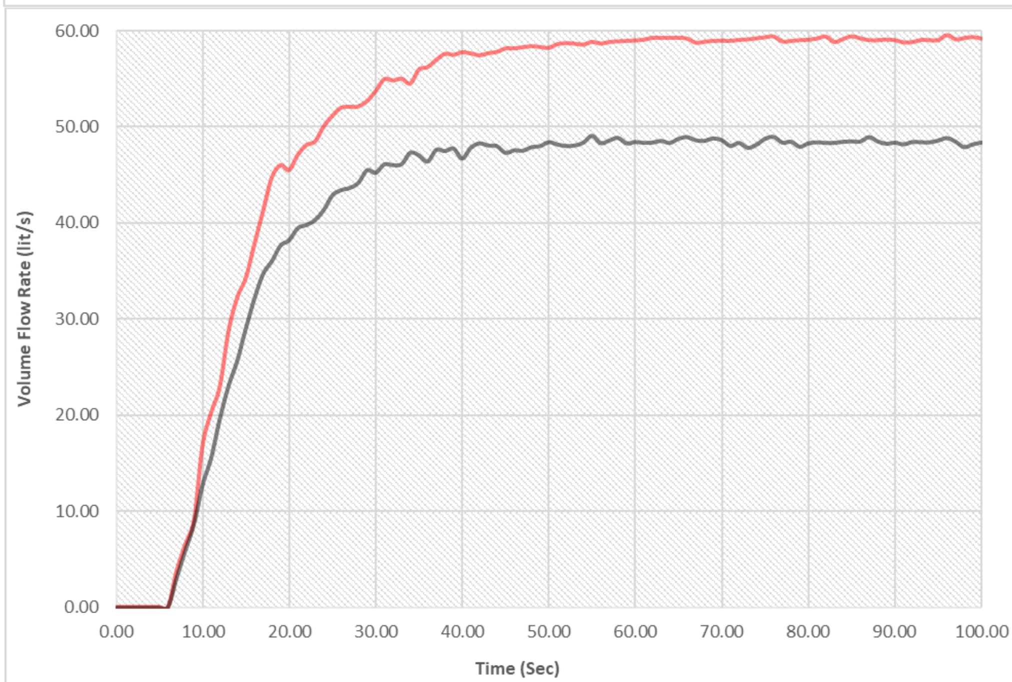


Figure 6 (a), (b), (c) and (d). (Continued) Outflow rate over time for sediment concentrations of 3000, 6000, 9000, and 12000 ppm.

To analyze the discharge coefficient of the bottom outlet channels more accurately, the discharge coefficient (C_0) for each model was calculated under downstream channel conditions based on the outflow rate of each model. The discharge coefficient for the bottom outlet channels can be calculated using the outflow rate (Q) in cubic meters per second, the cross-sectional area of the channel (A = 0.0165 m^2), and the fluid height in the reservoir (H). Table 4 presents the various models in comparison to the baseline model, which is calibrated with pure water.

Table 4. Comparison of various models to the baseline model calibrated with pure water

	Q (lit/s)	C0	Flow Rate Change	Changing the Flow Coeffient
Main Model	59.14	0.484	Clear Water	%
3000 PPM	56.4	0.461	-3.9%	4.633%
6000 PPM	52.3	0.428	-11.56%	7.270%
9000 PPM	48.13	0.394	-18.61%	7.973%
12000 PPM	42.81	0.350	-27.60%	11.053%

As shown in the Table 4, the discharge coefficient is influenced by changes in outflow rates from the bottom outlet channels due to suspended sediment concentrations. For example, a suspended load concentration of approximately 12000 ppm results in an 11.05% reduction in the discharge coefficient of the bottom outlet channels. Using the available data, a proposed relationship for changes in the discharge coefficient in bottom outlet channels, as affected by suspended sediment load, can be developed. Figure 7 illustrates the variation of discharge coefficients with suspended load concentrations ranging from 0 to 12000 ppm. By fitting the existing data, the desired relationship can be established.

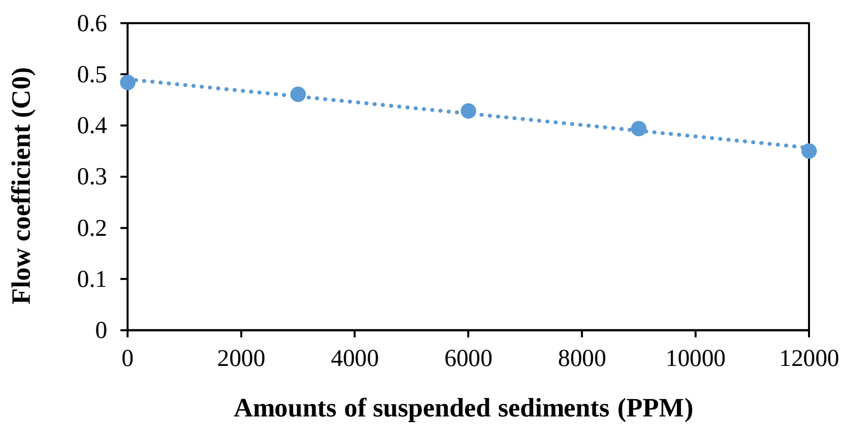


Figure 7. Variation of discharge coefficients with suspended load concentrations from 0 to 12000 ppm, illustrating the fitted relationship based on existing data.

4. Conclusions

The results of this study indicate that suspended sediments in the flow through bottom outlet systems can significantly reduce the outflow rate. This reduction is due to changes in flow density and its impact on the hydraulic parameters of the channel. Under flood conditions with high suspended sediment concentrations, the performance of the outlet system is substantially affected, potentially decreasing its efficiency. The Flow3D numerical model, with its strong capabilities in simulating turbulent and multiphase flows, proves to be an effective tool for analyzing bottom outlet systems. The findings of this research can aid in optimizing the design of outlet systems and predicting their performance under flood and high sediment load conditions. Flow3D demonstrates high accuracy in simulating hydrodynamic flows in bottom outlet systems. The impact of suspended sediments on reducing both outflow rate and discharge coefficient is especially significant at high sediment concentrations, a factor that should be considered in the design and operation of outlet systems.

Using more advanced numerical models, such as LES, allows for a more detailed examination of complex flows and the impact of suspended sediments, as these models offer higher accuracy in turbulent and multiphase flow conditions. Broader studies with various sediment concentrations and particle sizes are beneficial for assessing the effects of different sediment types on flow rate and discharge coefficient. Simulating bottom outlet systems in other dams helps generalize the results. Conducting large-scale physical experiments and comparing them with simulation results enhances accuracy and validates the models. Examining environmental factors like temperature, weather conditions, and sediment types is essential to understand their effects on outlet system performance. Finally, optimizing the design of outlet systems improves performance under high sediment load conditions. Future studies could further investigate the impact of varying sediment shapes and sizes on system efficiency.

Funding

This research was self-funded.

Conflicts of Interest

The author declares no conflict of interest.

References

Aminian, P., Ahmadi, A., & Emamgholizadeh, S. (2019a). Experimental investigation on the deviated sediment and flow to sediment bypass tunnels (SBTs) using submerged plates. *Journal of Hydraulic Structures*, *5*(2), 18-31.

Aminian, P., Ahmadi, A., & Emamgholizadeh, S. (2019b). Experimental Study of the Effects of Hydraulic and Geometric Parameters of the Sediment Transport Tunnel on the Deviation Flow and Transmitted Sediment. *Iranian Journal of Soil and Water Research*, *50*(2), 505-514.

Aminian, P., Jorabloo, M., & Ahmadi, A. (2023). Numerical Study of the Effects of Inlet Geometry on the Performance of the Sediment Bypass Tunnel using SSIIM Numerical Model. *Journal of Hydraulic and Water Engineering*, *1*(1), 65-75.

Arman, A., Fathi-Moghadam, M., Samadi, H., & Emamgholizadeh, S. (2009). Fall velocity of cohesive sediments in Dez Dam reservoir.

Barnea, D., Shoham, O., Taitel, Y., & Dukler, A. (1985). Gas-liquid flow in inclined tubes: flow pattern transitions for upward flow. *Chemical Engineering Science*, *40*(1), 131-136.

Borodina, L. K. (1969). Aeration behind low-level gates. *Power Technology and Engineering*, *3*(9), 826-834.

Bureau, O. R. (1977). *Design of small dams*. Govt. Print. Off.

Dargahi, B. (2010). Flow characteristics of bottom outlets with moving gates. *Journal of Hydraulic Research*, *48*(4), 476-482.

Emamgholizadeh, S., Bateni, S. M., & Jeng, D.-S. (2013). Artificial intelligence-based estimation of flushing half-cone geometry. *Engineering Applications of Artificial Intelligence*, *26*(10), 2551-2558.

Emamgholizadeh, S., Bateni, S. M., & Nielson, J. R. (2018). Evaluation of different strategies for management of reservoir sedimentation in semi-arid regions: a case study (Dez Reservoir). *Lake and Reservoir Management*, *34*(3), 270-282.

Emamgholizadeh, S., Bina, M., Fathi-Moghadam, M., & Ghomeyshi, M. (2006). Investigation and evaluation of the pressure flushing through storage reservoir. *ARPN Journal of Engineering and Applied Sciences*, *1*(4), 7-16.

Emamgholizadeh, S., & Fathi-Moghdam, M. (2014). Pressure flushing of cohesive sediment in large dam reservoirs. *Journal of Hydrologic Engineering*, *19*(4), 674-681.

Emamgholizadeh, S., Khademi, N., & Hosini, H. (2020). Prediction of input sediment to the Shirin-Darreh dam reservoir using HEC-RAS numerical model. *Journal of watershed management research*, *11*(21), 208-222.

Emamgholizadeh, S., & Samadi, H. (2008). Desilting of deposited sediment at the upstream of the Dez reservoir in Iran. *Journal of Applied Sciences in Environmental Sanitation*, *3*(1), 25-35.

Fathi-Moghadam, M., Arman, A., Emamgholizadeh, S., & Alikhani, A. (2011). Settling properties of cohesive sediments in lakes and reservoirs. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, *137*(4), 204-209.

Heller, V., Hager, W. H., & Minor, H. E. (2005). Ski jump hydraulics. *Journal of Hydraulic Engineering*, *131*(5), 347-355.

Jabary, A., Hosseini, A., Haghiabi, A. H., Emamgholizadeh, S., & Behnia, A. (2014). Prediction of the sediment load in the river by HEC-RAS. *Irrigation and Water Engineering*, *4*(4), 12-23.

Khatsuria, R. M. (2004). *Hydraulics of spillways and energy dissipators*. CRC press.

Liu, T. (2014). *Modelling air—water flows in bottom outlets of dams* KTH Royal Institute of Technology].

Manual, F. D. U. (2011). Flow3D user manual, v9. 4.2, flow science. *Inc., Santa Fe, NM*.

Rajaratnam, N., & Beltaos, S. (1977). Erosion by impinging circular turbulent jets. *Journal of the Hydraulics Division*, *103*(10), 1191-1205.

Razavi, A. R., & Ahmadi, H. (2017). Numerical modelling of flow in morning glory spillways using FLOW-3D. *Civil Engineering Journal*, *3*(10), 956-964.

Sharma, K. P., Sharma, S., Sharma, S., Sharma, P. K., Swami, R. C., Singh, P. K., & Rathore, G. S. (2007). *Mansagar Lake: Past, present and future* Proceedings of Taal,

Speerli, J., & Hager, W. H. (2000). Air-water flow in bottom outlets. *Canadian Journal of Civil Engineering*, *27*(3), 454-462.

Taghavi, M., & Ghodousi, H. (2015). Simulation of flow suspended load in weirs by using Flow3D model. *Civil Engineering Journal*, *1*(1), 37-49.

Te Chow, V. (1959). *Open channel hydraulics*. McGraw-Hill.

Wilcox, D. C. (1998). *Turbulence modeling for CFD*. DCW Industries.

Yakhot, V., & Orszag, S. A. (1986). Renormalization group analysis of turbulence. I. Basic theory. *Journal of Scientific Computing*, *1*(1), 3-51.