International Journal of Science and Research Archive

eISSN: 2582-8185

Cross Ref DOI: 10.30574/ijsra

Journal homepage: https://ijsra.net/

(Review Article)

A review of numerical simulations on stepped spillways using the k-ε turbulence model

Nastaran Sabahi *

Department of Hydraulics, Faculty of Civil Engineering, Middle East Technical University, Ankara, Turkey.

International Journal of Science and Research Archive, 2024, 12(01), 978–989

Publication history: Received on 16 April 2024 revised on 23 May 2024; accepted on 25 May 2024

Article DOI: https://doi.org/10.30574/ijsra.2024.12.1.0940

Abstract

For centuries, stepped spillways and cascades have been employed to effectively dissipate the kinetic energy of flowing water and promote aeration. The origins of stepped chute construction can be traced back to ancient Greece and Crete. In antiquity, this design served three main purposes: as stepped spillways, for stormwater and irrigation channels, and in municipal water supply systems. Initially crafted from cut-stone masonry or timber, the 19th century witnessed the introduction of various construction materials, such as composite materials and non-reinforced concrete. Over the last forty years, renewed interest in the staircase chute design has emerged, thanks to advancements in construction materials and techniques. The stepped invert in these designs significantly enhances energy dissipation above the steep chute, reducing the need for large downstream stilling structures. Stepped chutes are now applied not only in traditional uses but also in in-stream re-aeration and water treatment plant cascades, improving the transfer of atmospheric gases and volatile organic components between air and water. However, the engineering of stepped spillways is complex due to hydrodynamic challenges, involving different flow regimes, intricate two-phase air-water fluid dynamics, and a substantial rate of energy dissipation above the stepped chute. This study aims to compile recent research on stepped spillways, employing numerical approaches to address these engineering challenges. FLUENT and Flow-3D have been the most useful software in simulating stepped spillways among the researchers and compared with the experimental results they have proved to be quite beneficial. Here we provide a review of numerical simulations on stepped spillways using FLUENT and k-ε turbulence model, the most utilized methods in this software. The study illustrates that the RNG k-ε turbulence model has been the most popular model among researchers to simulate the flow over stepped spillways and is suggested as one of the models that shows a great correlation with the experimental results.

Keywords: Stepped Spillways; Energy Dissipation; Fluid Dynamics; Numerical Simulations; FLUENT.

1. Introduction

Presently, flood issues frequently arise from both human activities and natural factors. Weirs and spillways are constructions employed for managing, controlling, and redirecting the flow of water, typically across its path and often at right angles to the direction of the flow [27]. Constructing dams is one approach employed to address this problem, with dam spillways specifically designed to release excess floodwater when the reservoir capacity is exceeded. Additionally, dams can store significant amounts of water for utilization during dry seasons. However, the construction of dams requires careful consideration of factors such as geographical area, environmental impact, and population density to assess the feasibility and benefits of the construction. However, topology is one problem that can limit the frequent use of dams, as much as other energy generators or dissipator structures [4-6].

Stepped spillways are exceptional hydraulic structures known for their ability to dissipate energy and control water flow effectively. Their stepped configuration minimizes downstream erosion by creating turbulence and aeration,

* Corresponding author: Nastaran Sabahi

reducing water velocity, and preventing hydraulic jumps. Beyond functionality, the aesthetic design of stepped spillways makes them suitable for both urban and natural landscapes. They offer versatile applications in dams, reservoirs, irrigation channels, and stormwater management, accommodating various flow conditions. Stepped spillways contribute to safety by mitigating the risk of downstream flooding and erosion. Additionally, their environmental impact is reduced, positively affecting aquatic habitats and ecosystems. The gradual energy dissipation and controlled flow make stepped spillways a cost-effective choice in many water-management projects. It should be pondered that the problem of energy is of great concern to the engineers [7]. Overall, the exceptional features of stepped spillways lie in their combination of functional efficiency, aesthetic appeal, safety improvements, and environmental considerations [10-12].

The type of stepped flow regime is a function of the discharge and step geometry [25-27]. At low flow rates and for relatively large step heights, the water flows from one step onto another as a succession of free-falling nappes. At larger flow rates, the flow skims over the step edges with the formation of recirculating vortices between the mainstream and the step corners. The hydraulic characteristics differ significantly between each regime [8]. The transition between nappe and skimming flow is related to the flow rate, chute slope, step geometry, and local flow properties at each step [9].

1.1. CFD

CFD software works by implementing algorithms and numerical methods to solve the governing equations of fluid flow within a computational domain. Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation is:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\overrightarrow{V}) = 0 \qquad (1)$$

and the conservation of momentum equation is:

$$\rho\frac{\partial \vec{V}}{\partial t} + \rho\big(\vec{V}.\nabla\big)\vec{V} = -\nabla p + \rho\vec{g} + \nabla.\tau_{ij} \qquad (2)$$

These equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD) [4].

A fundamental consideration for CFD code developers is the choice of suitable techniques to discretize the modeled fluid continuum. Of the many existing techniques, the most important include finite difference, finite elements, and finite volumes. Although all these produce the same solution at high grid resolutions, the range of suitable problems is different for each. This means that the employed numerical technique is determined by the conceived range of code applications.

Finite difference techniques are of limited use in many engineering flows due to difficulties in their handling of complex geometries. This has led to increased use of finite elements and finite volumes, which employ suitable meshing structures to deal appropriately with arbitrary geometry. Finite elements can be shown to have optimality properties for some types of equations [14]. However, only a limited number of commercial finite element packages exist, which is undoubtedly a reflection of the difficulties involved in the programming and understanding of this technique [20].

Fortunately, such difficulties are obviated through the implementation of finite volumes methods. When the governing equations are expressed through finite volumes, they form a physically intuitive method of achieving a systematic account of the changes in mass, momentum, and energy as fluid crosses the boundaries of discrete spatial volumes within the computational domain [30]. The ease in the understanding, programming, and versatility of finite volumes has meant that they are now the most commonly used techniques by CFD code developers [22].

1.2. Turbulence Models

In computational fluid dynamics (CFD), turbulence models like Reynolds-Averaged Navier-Stokes (RANS) and Direct Numerical Simulation (DNS) are used to simulate turbulent flows.

1.2.1. RANS

RANS is a widely used turbulence modeling approach where the Navier-Stokes equations are averaged over time to obtain Reynolds-averaged equations. This averaging process separates the flow into mean and fluctuating components. In RANS, the problem of turbulence closure arises because the averaged equations still contain terms representing the effects of turbulence that need to be modeled. Various turbulence models are used to close the equations, such as the k-ε, k-ω, and Spalart-Allmaras models. These models provide relationships between the turbulent quantities and flow parameters based on empirical or physical principles. RANS models are computationally less expensive compared to DNS and are suitable for a wide range of industrial applications where accurate predictions of mean flow properties are needed [31].

Different classes of RANS turbulence models exit:

I. Zero-equation models: In these models, the only solved equation is a system of partial differential equations (PDEs) for the mean field, and no other PDEs are used. These models are also called mean-velocity field (MVF) closures [22].
II. One-equation models: This course includes an extra transport equation used to compute the turbulence velocity scale, typically expressed in relation to the average turbulent kinetic energy (K) [19].
III. Two-equation models: This course includes an extra transport equation for determining the turbulence length scale, often expressed in relation to the scalar dissipation rate of turbulent kinetic energy (ε). Classes (II) and (III) are called as mean turbulent energy (MTE) closures-representing a subset of the wider category of mean turbulent field (MTF) closures [21].
IV. Stress-equation models: This course includes several extra transport equations for the components of the Reynolds-stress tensor (τ_{ij}) and one for the scalar dissipation rate (ε). For this reason, these models are also called mean Reynolds-stress (MRS) closures [1-4].

The k-ε models are a type of two-equation RANS model that relies on solving transport equations for k and epsilon. These equations play a crucial role in determining the eddy viscosity and influencing the mean flow behavior.

The Reynolds Decomposition and Averaging of the zero-equation turbulence model for the flow of viscous incompressible fluid with constant properties which is governed by Navier-Stokes equations is as below:

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \vartheta \frac{\partial^2 u_i}{\partial x_j \partial x_j} \qquad (3)$$

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad (4)$$

Where u_i is the fluid velocity, p is the pressure which is divided by the density ρ and υ is the fluid kinematic viscosity, and body forces do not appear explicitly [1].

The system of partial Differential equations to be solved in the zero-equation class of models for the calculation of the zero-equation turbulence model is as below:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_j} + \vartheta \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \qquad (5)$$

$$\tau_{ij} = \frac{2}{3} K \delta_{ij} - \vartheta_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) \qquad (6)$$

In the second equation υ_T is the turbulent or eddy viscosity, and K is the average kinetic energy of the velocity fluctuations. In a closed system of PDEs, due to the flow condition and considering incompressible flows, K does not need to be calculated explicitly [1-3].

K-Epsilon (k-ε) Turbulence Models:

The k-ε model is a widely used turbulence model in Computational Fluid Dynamics (CFD) for simulating **mean flow characteristics** in turbulent flows. It belongs to a class of models called **eddy viscosity models**, which aim to capture the effects of turbulence by introducing an additional viscosity term.

Here's a breakdown of the k-ε model:

- **Two-Equation Model:** It solves two transport equations in addition to the standard conservation equations. These equations represent:
  - **Turbulent Kinetic Energy (k):** This signifies the energy present in the turbulence.
  - **Turbulent Dissipation Rate (ε):** This determines the rate at which turbulent kinetic energy is dissipated.
- **Eddy Viscosity:** By combining k and ε, the model calculates an **eddy viscosity (μ_T)**. This eddy viscosity acts like a virtual viscosity added to the molecular viscosity of the fluid, accounting for the momentum transfer due to turbulence.

There are different variations of the k-ε model, each addressing some limitations of the original formulation. Here are two main classifications:

1. **Standard k-ε Model:** This is the original and most basic version. It's computationally cheap but has limitations in accuracy, particularly for flows with large adverse pressure gradients.
2. **Realizable k-ε Model:** This is a more advanced version that addresses some shortcomings of the standard model. It incorporates mathematical constraints to ensure the model adheres to the physical principles of turbulent flows, leading to improved accuracy in certain scenarios [13].

While other k-ε model variations exist, these two are the most commonly used in CFD simulations.

The equations for both the k-ε models are summarized below [23]:

$$\rho \frac{Dk}{Dt} = P - \rho\varepsilon + \frac{\partial}{\partial x_i}(\mu\alpha_k \frac{\partial k}{\partial x_i}) \qquad (7)$$

$$\rho \frac{D\varepsilon}{Dt} = \frac{\varepsilon}{k}\left(C_{\varepsilon_1}P - C_{\varepsilon_2}\rho\varepsilon\right) - \rho R + C\varepsilon_3\rho\varepsilon\nabla.U + \frac{\partial}{\partial x_i}(\mu\alpha_\varepsilon \frac{\partial \varepsilon}{\partial x_i}) \qquad (8)$$

$$P = 2C_\mu\rho\frac{k^2}{\varepsilon}\left[S_{ij}S_{ij} - \frac{1}{3}(\nabla.U)^2\right] - \frac{2}{3}\rho k\nabla.U \qquad (9)$$

$$R_{STAND} = 0 \quad R_{RNG} = \frac{C_\mu\eta^3(1-\frac{\eta}{\eta_0})}{1+\beta\eta^3}\frac{\varepsilon^2}{k} \qquad (10)$$

$$C_{\varepsilon_1 STAND} = 1.44 \quad C_{\varepsilon_1 RNG} = 1.42$$

$$C_{\varepsilon_2 STAND} = 1.92 \quad C_{\varepsilon_2 RNG} = 1.68$$

$$C_{\varepsilon_3 STAND} = -1$$

$$C_{\varepsilon_3 RNG} = \frac{-1+2C_{\varepsilon_1}-3m_1(n_1-1)+(-1)^\delta\sqrt{6}C_\mu C_\eta}{3} \qquad (11)$$

$$C_\eta = \frac{\eta(1-\frac{\eta}{\eta_0})}{1+\beta\eta^3} \qquad (12)$$

$$\beta = 0.012$$

$$\eta_0 = 4.38$$

$$\eta = \frac{Sk}{\varepsilon} \qquad S = (2S_{ij}S_{ij})^{1/2} \qquad (13)$$

$$\delta = 1 \qquad \nabla.U < 0$$

$$\delta = 0 \qquad \nabla.U > 0$$

$$C_{\mu\, STAND} = 0.09 \quad C_{\mu\, RNG} = 0.0837$$

$$\alpha_{k\,STAND} = 1.0 \quad \alpha_{\varepsilon\,STAND} = 0.769$$

$$\alpha_{k\,RNG} = \alpha_{\varepsilon\,RNG} = 1.39$$

1.2.2. DNS

DNS is a method where the Navier-Stokes equations are solved directly without any turbulence modeling. It resolves all spatial and temporal scales of turbulence in the flow domain. Since DNS directly solves the governing equations, there is no need for turbulence modeling or closure assumptions [18]. DNS is computationally very demanding because it resolves all turbulent scales. It requires high spatial and temporal resolutions, making it feasible only for relatively simple flows or academic research purposes. DNS provides highly accurate results, capturing all turbulent structures and their interactions. It serves as a benchmark for validating other turbulence models. Due to its computational cost, DNS is typically limited to low Reynolds number flows or smaller computational domains.

Fluent

ANSYS Fluent stands as a pinnacle in computational fluid dynamics (CFD) software, meticulously designed by Fluent Inc. to address the complexities of fluid flow and heat transfer simulations across diverse industries. Renowned for its versatility and robust capabilities, ANSYS Fluent provides engineers and researchers with a sophisticated platform for modeling complex physical phenomena, encompassing multiphase flows, heat transfer mechanisms, turbulence, and chemical reactions. The software's intuitive interface, coupled with advanced features like parallel processing and intricate meshing tools, empowers users to simulate and analyze fluid dynamics scenarios with precision. Widely employed in aerospace, automotive, energy, and manufacturing sectors, ANSYS Fluent remains an indispensable tool for gaining insights into fluid behavior, optimizing designs, and accelerating innovation in engineering applications. Utilizing ANSYS Fluent as a simulation tool is proved to be beneficial in time and costs in fields which demand much of time and money to be physically modeled, such as electrical device cooling system designs [29].

ANSYS Fluent provides high-fidelity fluid dynamics modeling, offering advanced turbulence models and multiphase flow capabilities crucial for accurate representation. Robust meshing tools ensure precise geometry representation and parallel processing efficiency accelerates complex simulations. The user-friendly interface facilitates setup, execution, and post-processing, while comprehensive tools aid in visualization and analysis. Integration with the ANSYS ecosystem allows for a multiphysics simulation environment, and the software supports validation through comparison with experimental data [16].

Potential challenges in simulating stepped spillways with Fluent include a steep learning curve for new users, substantial computational resource requirements for large and complex simulations, and associated licensing costs. Results may be sensitive to input parameters, requiring careful calibration, and the accuracy of simulations depends on chosen models and assumptions. Post-processing tools, while comprehensive, may require time to master. Users must weigh these considerations against the benefits of obtaining detailed insights into the hydraulic behavior of stepped spillways [21].

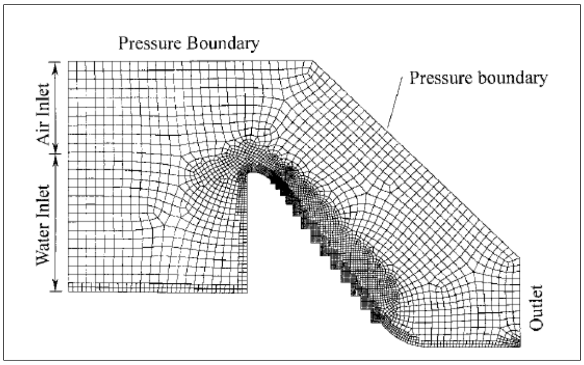

Figure 1 Example of mesh and boundaries in stepped spillways

Simulating stepped spillways using ANSYS Fluent involves several steps, including geometry creation, meshing, defining boundary conditions, setting up solver settings, running the simulation, and post-processing the results. Should the first step of simulating be creating a 2D or 3D model of the stepped spillway using CAD software (such as ANSYS DesignModeler or other compatible tools). Then the proper and high-quality mesh can be generated. The most important point here is to consider mesh refinement near critical areas, such as steps and regions with expected flow separation.

Defining the boundary condition for the simulation and specifying the inlet and outlet condition, as well as the material properties of the fluid is the third step. Next, the appropriate turbulence model should be selected based on the expected flow condition. ANSYS Fluent offers various turbulence models like K-epsilon, SST, etc. If the flow involves multiphase behavior, the relevant multiphase should be applied. The simulation can be run after configuring the solver settings, such as the discretization schemes, solution methods, and convergence criteria. In the case of free-surface flow, we can consider using the Volume of Fluid (VOF) model. Then the key parameters such as flow velocity, pressure distributions, and water level along the stepped spillways can be analyzed and compared with the experimental data. On the other step, all the obtained results can be verified by other simulating software, such as MATLAB/Simulink [4].

The k-ε Model on Stepped Spillways

The flow across a stepped spillway undergoes three distinct regimes based on the flow rate: skimming, transition, and nappe flow regimes, each corresponding to decreasing flow rates. The skimming flow regime is prevalent at higher flow rates, characterized by coherent turbulent flow skimming over the pseudo-bottom formed by the step edges. Under the pseudo-bottom, recirculation and vertical structures exist, filling the cavities. Transition flows occur at intermediate flow rates, displaying strong fluctuations and splashing near the free surface. At lower discharge conditions, nappe flows take place, where free-falling nappes are observed at each step edge, followed by their impacts on the subsequent step [25]. Figure 2 illustrates these three flow regimes over the stepped spillways.

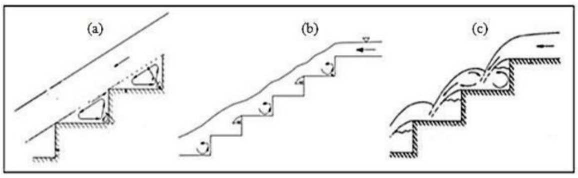

Figure 2 Schematic view of different flow regimes ((a): skimming flow regime, (b): transition flow regime and (c): nappe flow regime) in stepped spillways

This article reviews the most useful turbulence models and free-surface flow models to optimize the upcoming studies and in some cases to assist those searching for the best models to use in ANSYS Fluent software. In 2014 Daneshfaraz et al. [11] analyzed the flow over stepped spillways using the Fluent software considering three turbulence models: (a) standard k-epsilon, (b) Renormalized Group Theory k-epsilon (RNG) model, and (c) k-omega model.

The model underwent verification and was assessed by comparing its results with experimental data in terms of water surface profiles. The evaluation employed the Root Mean Square Error (RMSE) criterion, calculated using the following formula:

$$RMSE = \sqrt{\frac{\sum_{i=1}^{N}(y_{mi}-y_{pi})^2}{N}} \qquad (14)$$

They also examined the energy dissipation rate for the stepped spillways. To conduct the simulation, a finite element mesh comprising rectangular four-node elements was utilized for all the stepped spillways. In Fig. 3, the mesh for the first type of spillway is depicted, with the mesh resolution highlighted at three different locations labeled 1, 2, and 3. A high-resolution mesh (small-sized elements) was employed for the steps to accurately capture the formed vortices. Similar meshes were used for the other three types.

For simulating the flow using the FLUENT software, the initial conditions of the water surface need to be specified first (Fig. 4). Subsequently, the equations are solved iteratively with a time step of 0.001. This process is repeated until convergence is achieved, where the convergence criterion is that the relative error between two consecutive iterations is less than or equal to 0.0001. The flow field is considered to consist of separate control volumes. The governing flow equations are integrated in each control volume, and the algebraic equations are discretized using different discontinuity schemes. The GAMBIT software was utilized to generate the geometry of the flow field and mesh.

Furthermore, the PRESTO scheme was employed for interpolating the pressure, the Quick scheme was implemented for discretizing the terms of the momentum equation, the first-order upwind scheme was used for discretizing the turbulence equations, and the PISO algorithm was employed for coupling the pressure and velocity. To avoid solution divergence, under-relaxation factors less than one were used for the pressure, momentum, and Reynold's stress.

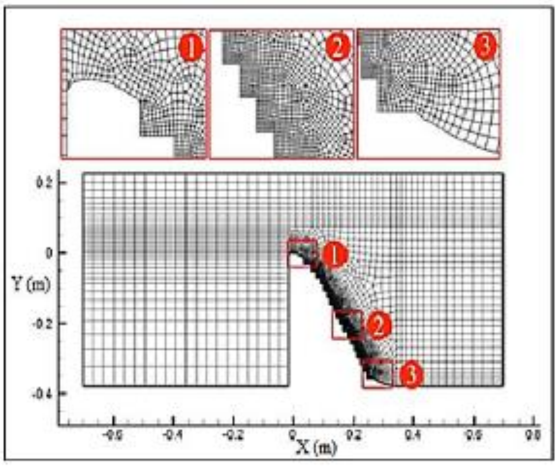

Figure 3 The finite element mesh for the first type of the stepped spillway

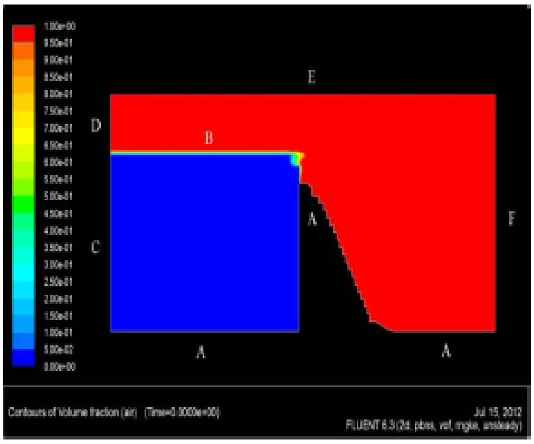

Figure 4 Boundary conditions and initial conditions for the first type of stepped spillway

The comparison between the modeled water depth and experimentally measured results revealed that the RNG k-epsilon model was more accurate in predicting the water surface profile. Conversely, when calculating the rate of energy dissipation using the energy equations, it was observed that the first type of stepped spillways, characterized by larger steps, resulted in the highest rate of energy dissipation. Table 1 illustrates the RMSE values for three turbulence models.

Table 1 RMSE values of the three turbulence models for all four types of the spillways

Type	Standard k-omega model	RNG k-epsilon model	Standard k-epsilon model
1	17.15	16.56	16.80
2	18.48	19.21	25.77
3	18.93	15.10	16.55
4	17.68	11.03	14.26

In another study, Kositgittiwong et al. (2013) [17] numerically simulated the flow velocity profiles along a stepped spillway with 25 steps which are in 0.61 m height and 1.22 m in length. The study was carried out by using 5 different turbulence models including the Standard k-epsilon, the Realizable k-epsilon, the Renormalization group k-epsilon, the Standard k-omega, and the Shear stress transport k-omega. Then the CFD simulation results were compared with the laboratory measurements from a large-scale physical model with flow velocities up to 15 m/s. The study considers some limitations for the flow along the stepped spillway including the Reynolds number of $1.68 \times 10^6 \leq Re \leq 7.21 \times 10^6$. The coefficient of Manning was considered as 5.09. This study sought to determine the most suitable turbulence model for simulating skimming flow over stepped spillways and to predict flow velocity with the least deviation, as measured by the root mean square error (RMSE). The VOF (Volume of Fluid) method was employed as a multiphase model. Velocity measurements were taken from the pseudo-bottom to the free surface, specifically at an air concentration of 90%, denoted as V_{90}.

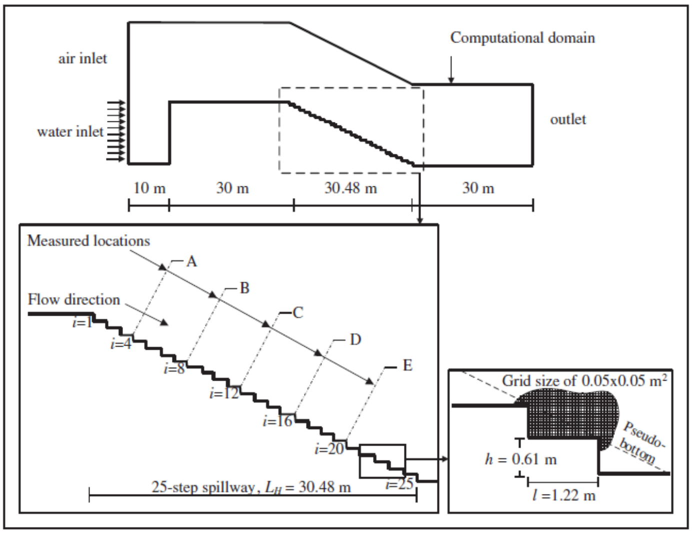

Figure 5 Schematic diagram of the stepped spillway

Five distinct turbulence models underwent testing against large-scale physical model experiments conducted at Colorado State University, featuring a unit discharge as high as 2.69 m^2/s. The discrepancies in root mean square error (RMSE) between the most and least effective models ranged from 0.96 to 1.07, and there were no significant differences observed in the profiles. The k-omega models exhibited slightly better suitability for the near-wall zone in the lower region of the velocity profile, while the RI k-epsilon model yielded slightly superior results in the upper part of the velocity profile. A reasonably accurate initial approximation can be achieved using a power law for the velocity profile with n=5.09. This analysis of velocity profiles on large stepped spillways serves as a valuable complement to previous studies conducted on a much smaller scale.

The next study discussed in this article is conducted by Qian et al. (2009) [24]. The numerical simulation of water flow over the stepped spillway employs a mixture multiphase flow model, incorporating different turbulence models to formulate the governing equations. The investigated turbulence models include the realizable k-epsilon model, SST k-omega model, v2-f model, and LES model. The computational results obtained from these four turbulence models are compared with experimental data across various aspects, namely mean velocity, spanwise vorticity, and the growth of turbulent boundary layer thickness in the streamwise direction.

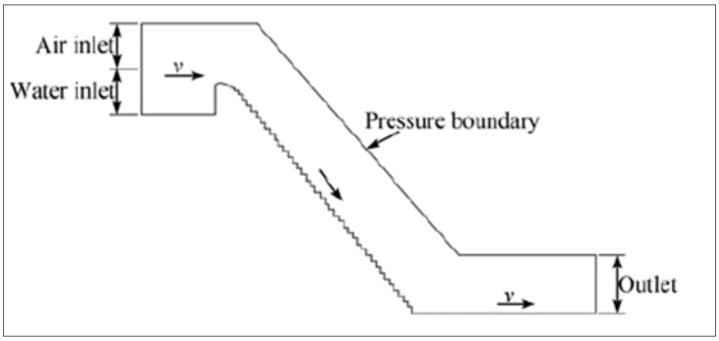

Figure 6 Computational domain and boundary conditions

The experimental model features dimensions of 0.50 m in width and 2.0 m in height (from crest to toe). The bottom inclination corresponds to a slope of 51.3° or 1:0.8. The spillway face is equipped with a total of 40 steps, with the first

37 steps maintaining a uniform height of 0.05 m, while the last 3 steps are transitional, with variable heights, numbered from toe to crest.

The comparison reveals that the realizable k-epsilon model, which incorporates the rotation tensor, stands out as the most effective turbulence model for accurately simulating water flow characteristics over stepped spillways. The v^2-f model struggles to predict flow in the recirculation region near the wall, while the SST k-omega model tends to produce excessively large turbulence levels in areas with significant normal strain, such as stagnation regions and regions with strong acceleration. This leads to a turbulence boundary layer thickness much smaller than the actual value. Additionally, the mean velocity profile normal to the pseudo-bottom and the pressure field on the steps were examined based on computational results using the Realizable k-epsilon model. The velocity distribution in the boundary layer can be reasonably approximated by a power law at the step edges, expressed as $\frac{U}{U_0} = (\frac{y}{\delta})^{1/3}$. The lowest pressures for each step are situated on the vertical surface near the outer edge of the steps.

Salmasi and Samadi (2018) [28] explore the dynamics of flow on a stepped spillway and quantify the associated energy loss. Additionally, the research compares the fluctuation of velocity vectors, shear stress, and pressure during the flow on each step. To achieve this, a physical model of the stepped spillway was constructed with a slope at a ratio of 2:1 (horizontal to vertical), and experiments were conducted with ten distinct flow rates. Numerical simulations were also carried out under identical conditions using FLUENT software, employing the RNG k–ε turbulence model.

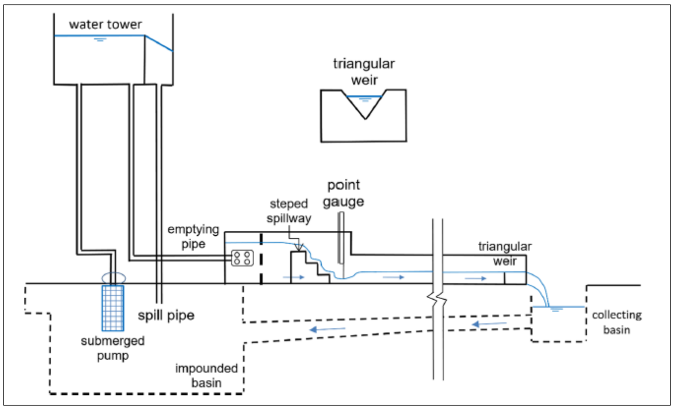

Figure 7 Schematic representation of the experimental equipment

In this study, the channel and spillway walls were specified using the wall boundary condition. The water depth at the entrance and the freeboard of the channel was defined by the velocity inlet condition, while the output stream face was characterized by the pressure outlet condition. To assess the impact of walls on the flow, the standard wall function was employed. Given the rotational nature of the flow on the spillway, the RNG k–epsilon turbulence model was chosen for simulation. It is noteworthy that the RNG model is recommended for simulating rotating and curved flows [14]. The results of the study approve the beneficial use of the RNG k-epsilon model compared to the experimental results. The authors also highlight the point that according to the numerical simulations, it is determined that when the flow discharge is increased, the shear stress and pressure will be decreased as a result.

Hamedi et al. (2016) [15] conducted a 2D numerical simulation over the stepped spillways. The study focuses on juxtaposing 2-D numerical simulations with experimental observations in stepped spillways that feature inclined steps and an end sill. It introduces an efficient, dependable, cost-effective, and non-experimental method for designing these steps. The complexity of the novel and intricate geometry poses challenges in simulation, particularly around the end sills, requiring heightened precision compared to horizontal steps. The VOF Method and the standard k-ε turbulence model are recommended for simulating flow patterns and assessing energy loss across the stepped spillway. The study considers different slopes, end sill heights and thicknesses for the steps and solves the numerical simulations in various attempts. Flow depth and velocity are calculated numerically in critical sections, and energy loss is computed at each section.

Table 2 Energy loss -Various end sills -Calibration

	Energy Loss %		
Run	Experimental	Numerical	Error%
1	59.56	55.93	6.09
2	60.09	56.51	5.95
4	60.60	56.61	6.58
5	60.86	56.87	6.56
7	58.47	55.79	4.58
8	59.16	56.15	5.08
10	59.31	56.41	4.89
12	59.48	56.37	5.23

Table 3 Energy loss -Various end sills -Validation

	Energy Loss%		
Run	Experimental	Numerical	Error%
3	60.54	56.47	6.72
6	61.12	57.74	5.53
9	59.21	56.20	5.09
11	59.36	56.63	4.59

The error between the experimental and numerical simulations is acceptable, and the results shows that k-ε turbulence model is capable of simulating the flow over the stepped spillways.

2. Conclusion

The aim of this study is to review the k-ε turbulence models and their functionality in simulating the stepped spillways and finally to compare the two types of k-ε model. The k-epsilon model is a popular choice for simulating turbulent flow over stepped spillways, but there are different variations within this model family. Considering the point that almost all of the models are simulated using the VOF model, the results indicates that the RNG k-ε model is considered a good balance between accuracy and computational efficiency for stepped spillways. It incorporates the effects of mean flow rotation and improves performance in boundary layers and recirculating regions. However, the k-ω model might even be slightly better model for the near wall eddies. This Standard k-ε model can struggle with flows with strong streamline curvature and adverse pressure gradients, which are common on stepped spillways. Also, Shear Stress Transport (SST) k-omega model can be a good alternative, particularly for complex flows with separation and reattachment. On the other hand, it can be concluded that the linear RNG k-ε model is more reasonable in predicting the recirculatory flows. In the case of nonlinear model, RNG k-ε model superiority over standard k-ε model is uncertain. Also, the nonlinear RNG k-ε model does not seem to be very appealing alternative in comparison with the linear model which can cost much less.

Compliance with ethical standards

Disclosure of conflict of interest

The author has no conflict of interest in this study.

References

[1] Alfonsi G. Reynolds-averaged Navier–Stokes equations for turbulence modeling. 2009.

[2] Almasi, P., Premadasa, R., Rouhbakhsh, S., Xiao, Y., Wan, Z., & Zhang, Q. A review of developments and challenges of preflight preparation for data collection of UAV-based infrastructure inspection. 2024.

[3] Babazadeh Dizaj R. DEVELOPMENT OF LSF-BASED DUAL-PHASE CATHODES FOR INTERMEDIATE TEMPERATURE SOLID OXIDE FUEL CELLS: Middle East Technical University; 2022.

[4] Bhaskaran R, Collins L. Introduction to CFD basics. Cornell University-Sibley School of Mechanical and Aerospace Engineering. 2002:1-21.

[5] Bhuvela P, Taghavi H, Nasiri A, editors. Design Methodology for a Medium Voltage Single Stage LLC Resonant Solar PV Inverter. 2023 12th International Conference on Renewable Energy Research and Applications (ICRERA); 2023: IEEE.

[6] Bhuvela P, Taghavi H, Nasiri A, editors. 13.8 kV, 1MW Resonant Direct AC Medium Voltage Single Stage Solar PV Inverter. 2024 IEEE Applied Power Electronics Conference and Exposition (APEC); 2024: IEEE.

[7] Chalaki HR, Babaei A, Ataie A, Seyed-Vakili S-V, editors. The Effect of Impregnation of Ceramic Nano-particles on the Performance of LSCM/YSZ Anode Electrode of Solid Oxide Fuel Cell. 5th International Conference on Materials Engineering and Metallurgy; 2016.

[8] Chanson H. Hydraulic Design of Stepped Cascades. Channels. 1994.

[9] Chanson H, editor A review of accidents and failures of stepped spillways and weirs. Proceedings of the Institution of Civil Engineers-Water and Maritime Engineering; 2000: Thomas Telford Ltd.

[10] Chen Q, Dai G, Liu H. Volume of fluid model for turbulence numerical simulation of stepped spillway overflow. Journal of Hydraulic Engineering. 2002;128(7):683-8.

[11] Daneshfaraz R, Sadeghfam S, Kashani M. Numerical simulation of flow over stepped spillways. Research in civil engineering and environmental engineering. 2014;2(4):190-8.

[12] Dizaj RB, Sabahi N. Laboratory preparation of LSM and LSF sputtering targets using PTFE rings for deposition of SOFC thin film electrodes. World Journal of Advanced Engineering Technology and Sciences. 2023;10(2):203-12.

[13] Dizaj RB, Sabahi N. Optimizing LSM-LSF composite cathodes for enhanced solid oxide fuel cell performance: Material engineering and electrochemical insights. World Journal of Advanced Research and Reviews. 2023;20(1):1284-91.

[14] Ferziger JH, Perić M. Computational methods for fluid dynamics: Springer; 2002.

[15] Hamedi A, Hajigholizadeh M, Mansoori A. Flow simulation and energy loss estimation in the nappe flow regime of stepped spillways with inclined steps and end sill: a numerical approach. Civil Engineering Journal. 2016;2(9):426-37.

[16] Kaveh A, Almasi P, Khodagholi A. Optimum design of castellated beams using four recently developed meta-heuristic algorithms. Iranian Journal of Science and Technology, Transactions of Civil Engineering. 2023;47(2):713-25.

[17] Kositgittiwong D, Chinnarasri C, Julien PY. Numerical simulation of flow velocity profiles along a stepped spillway. Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering. 2013;227(4):327-35.

[18] Li H, Wang L, Wu W, Bian W, Ding D, Chen F, editors. C2H6 Dehydrogenation and Electrical Power Production in a Protonic Conducting Fuel Cell with in-Situ Exsolved Metal Nanoparticle Catalyst. Electrochemical Society Meeting Abstracts 239; 2021: The Electrochemical Society, Inc.

[19] Li H, Wang W, Park K-Y, Lee T, Chen F, editors. A-Site Doping Effect on the Performance of Sr2Fe1. 4 Ni0. 1Mo0. 5O6-Δ Anodes for SOFCs. Electrochemical Society Meeting Abstracts 242; 2022: The Electrochemical Society, Inc.

[20] Li H, Wang W, Zhao K, Park K-Y, Lee T, Dizaj RB, et al. A Redox-Reversible A/B-site Co-doped BaFeO3 Electrode for Direct Hydrocarbon Solid Oxide Fuel Cells. Journal of Materials Chemistry A. 2024.

[21] Mellor GL, Herring HJ. A survey of the mean turbulent field closure models. AIAA journal. 1973;11(5):590-9.

[22] Norton T, Sun D-W. Computational fluid dynamics (CFD)–an effective and efficient design and analysis tool for the food industry: a review. Trends in Food Science & Technology. 2006;17(11):600-20.

[23] Papageorgakis G, Assanis DN. Comparison of linear and nonlinear RNG-based k-epsilon models for incompressible turbulent flows. Numerical Heat Transfer: Part B: Fundamentals. 1999;35(1):1-22.

[24] Qian Z, Hu X, Huai W, Amador A. Numerical simulation and analysis of water flow over stepped spillways. Science in China Series E: Technological Sciences. 2009;52(7):1958-65.

[25] Rajaratnam N. Skimming flow in stepped spillways. Journal of Hydraulic Engineering. 1990;116(4):587-91.

[26] Rostaghi Chalaki H, Babaei A, Ataie A, Seyed-Vakili SV. LaFe 0.6 Co 0.4 O 3 promoted LSCM/YSZ anode for direct utilization of methanol in solid oxide fuel cells. Ionics. 2020;26:1011-8.

[27] Salmasi F, Sabahi N, Abraham J. Discharge coefficients for rectangular broad-crested gabion weirs: experimental study. Journal of Irrigation and Drainage Engineering. 2021;147(3):04021001.

[28] Salmasi F, Samadi A. Experimental and numerical simulation of flow over stepped spillways. Applied Water Science. 2018;8(8):229.

[29] Taghavi H, El Shafei A, Nasiri A, editors. Liquid Cooling System for a High Power, Medium Frequency, and Medium Voltage Isolated Power Converter. 2023 12th International Conference on Renewable Energy Research and Applications (ICRERA); 2023: IEEE.

[30] Versteeg H, Malalasekera W. Computational fluid dynamics. The finite volume method. 1995:1-26.

[31] Wang W, Li H, Park KY, Lee T, Ding D, Chen F. Enhancing Direct Electrochemical CO2 Electrolysis by Introducing A-Site Deficiency for the Dual-Phase Pr (Ca) Fe (Ni) O 3− δ Cathode. Energy & Environmental Materials. 2024:e12715.