ISSN 2303-4521
Periodicals of Engineering and Natural Sciences *Original Research*
Vol. 11, No. 2, April 2023, pp.307-314

Simulation of flow over stepped and traditional spillways

Abdulnaser A.Ahmed[1], Adnan A. Ismael[1], Saleh J. Shareef[3]

[1] Technical Institute, Northern Technical University, Mosul, Iraq
[2] Engineering Technical College, Northern Technical University, Mosul, Iraq

ABSTRACT

There are several challenges that are faced by these structures, such as cavitation and high speed of flow during the dissipation of flow energy. The spillway is one of the key basic elements of dams that are utilized to pass huge volumes of discharge. The current work used computational fluid dynamics (CFD) to numerically estimate the water surface flow and pressure distribution over stepped and traditional spillways. The numerical findings were compared with experimental data using the Renormalization group k- model. The numerical and experimental data have been compared, and it has been determined that there is an acceptable agreement between the two sets of findings.

Keywords: Stepped and traditional spillways, computational fluid dynamics, $k-\varepsilon$ model, pressure distribution, CFD.

Corresponding Author:

Abdulnaser A.Ahmed
Technical Institute
Northern Technical University
Mosul, Iraq
Email:-naser102525@yahoo.com

1. Introduction

The spillway is one of the main constituent parts of dams that are used to pass large amounts of discharge, and there are many difficulties facing this structures like cavitation and high speed of flow during the dissipation of flow energy. It contains a full integration of problem specification, analysis, and results display, in addition to other characteristics such as a sophisticated coupled solver that is reliable and robust. Finally, it includes a setup procedure that is user-friendly and interactive, consisting of menus and sophisticated graphics. Many researchers have used it to simulate the flow across spillways by employing a variety of techniques and software applications, such as ANSYS cfd as well as opening foam [1-5].

Ismael et al. [6].The authors used two-phase flow to simulate the flow field over step pool spillway by k-epsilon model and compared numerical result with experimental data. They found that the mixture model with the Reynolds stress turbulence model are more agreement with experimental results. The VOF method was utilized by Zang et al. [3] in order to simulate the turbulent flow in the 3D flow field of the spillway. The experimental data and the numerical results for the surface elevation, flow velocity, and pressure along the spillway match up quite well. The installation of a breaker over a stepped spillway causes an increase in the amount of energy that is lost. Zhan et. [1]In order to model the skimming flow, we made use of three distinct numerical approaches: the volume of fluid (VOF), the mixing, and the Eulerian methods. The experimental value is most closely matched by the air volume fraction that was anticipated using the VOF+LES approach. The aim of this study is to simulate the free water surface flow, pressure distribution and velocity distribution on traditional and stepped spillway. The experimental results of traditional and stepped spillway were compared to the numerical results.

2. Methods

This study demonstrates the simulation of a free water surface flow, pressure distribution and velocity distribution on traditional and stepped spillway. The geometry consists of a 3D channel in which the bottom of the channel is interrupted by a traditional and stepped spillway with semicircular crest of radius 6cm. The normal inlet speed is range from (0.008 to 0.1898 m/s) for traditional spillway and from (0.017 to 0.178 m/sec) for stepped spillway; the incoming water has a turbulence intensity of 5% as shown in figures 1 and 2. The flow upstream of the customary and stepped spillway is subcritical.
Physical Model used in the Present Study

Two types of spillways are detailed as follows;

i) The smooth spillway) with downstream slope (α=53.13°=V:H=1:0.75), flow discharges of 1.28, 4.03, 8.67,17.53,25.62 and 38.82 L/sec were used.
ii) Stepped spillway with downstream slope (α=53.13°=V: H=1:0.75), number of step N_s=3, h = 0.06 m and l = 0.0448 m and flow discharges are 2.79, 5.82, 14.85, 18.58, 22.22 and 34.82 L/sec, respectively.

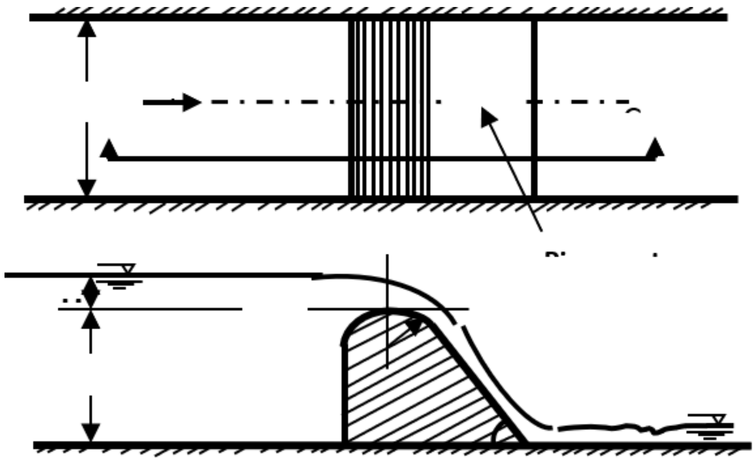


Figure 1. Traditional spillwayand piezometer location with simicircular crest

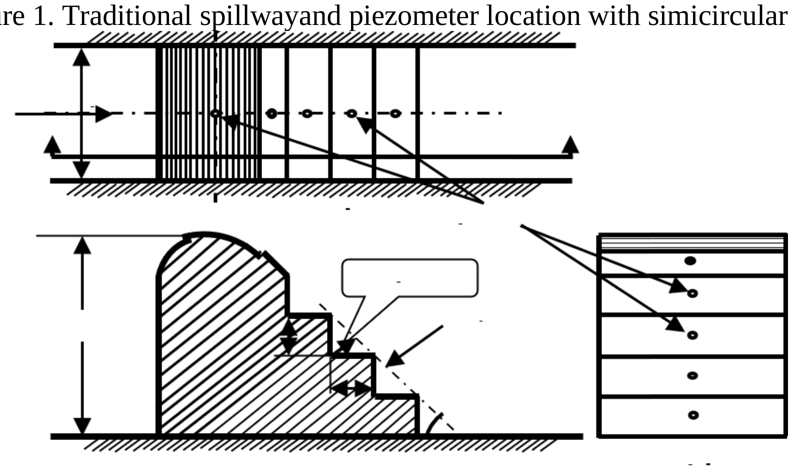


Figure 2. Stepped spillwayand piezometer location

2.1. Mesh refinement

In contrast, structured quadrilateral meshes will, in most cases, cause cells to be put in areas of the flow domain where they are not required, whereas triangular meshes make it possible for cells to congregate in certain portions of the flow domain. (Pulliam, 1994). As seen in figures 3 and 4, quadrilateral elements have the ability to accommodate a significantly higher aspect ratio than triangle cells, which is one of the characteristics that could potentially make quadrilateral elements more cost-effective in certain contexts. For a typical spillway, the tetrahedral mesh should have 88890 nodes and 426362 elements, respectively as can be seen in Table 2 below:

Table 2. Quality of the simulation's mesh according to the analysis

Weir diameter	Obliqueness angle	Mesh Skewness	Cell class
0.114	90	0.067	Excellent
0.114	135	027	Good
0.114	150	0.26	Good
0.09	90	0.059	Excellent
0.09	135	0.26	Good
0.09	150	0.26	Good
0.0635	90	0.078	Excellent
0.0635	135	0.26	Good
0.0635	150	0.25	Excellent

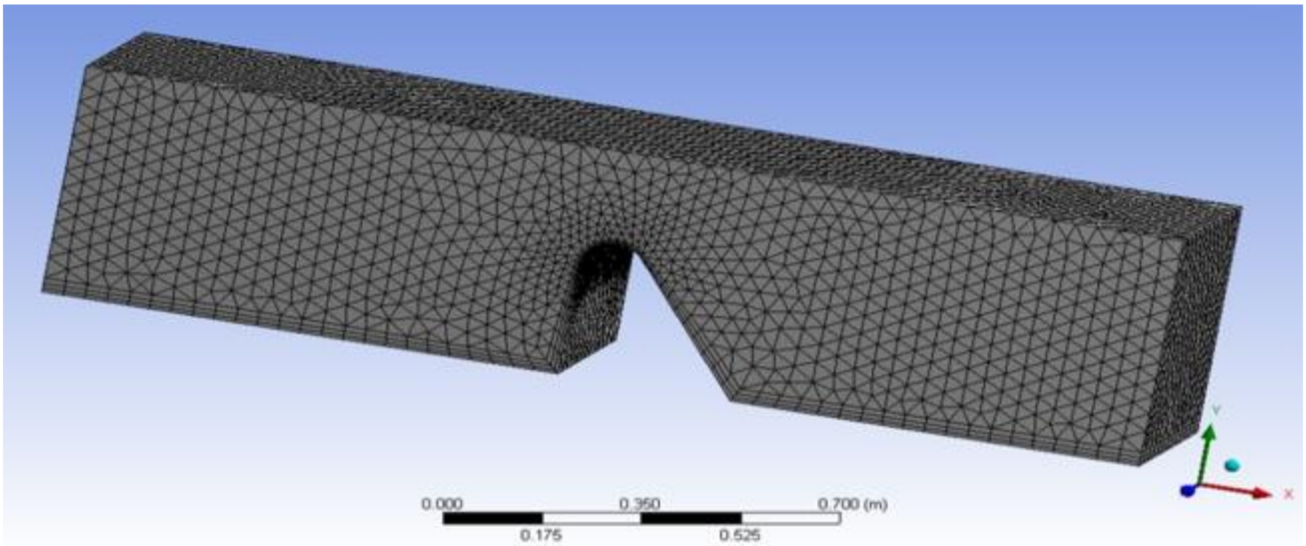


Figure 3. Mesh generation of traditional model in (ANSYS) ICEM CFD

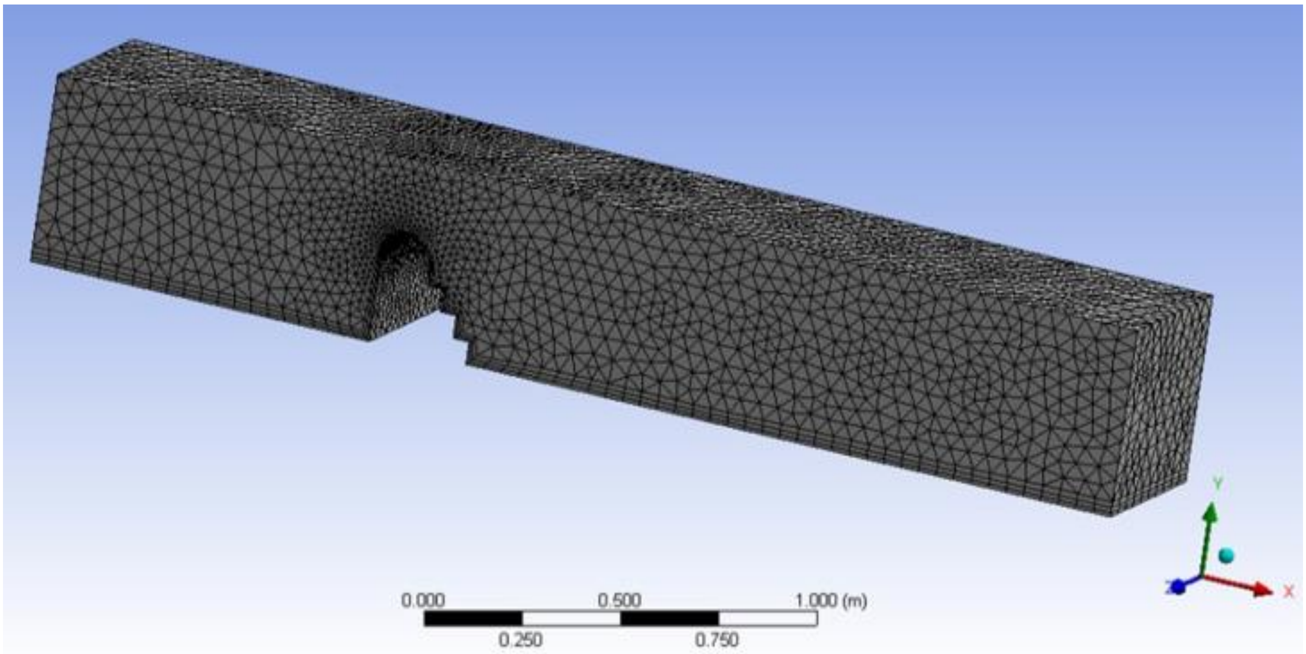


Figure 4. Mesh Generation of Stepped Model in (ANSYS) ICEM CFD).

The equations for k and ε in the RNG k- ε turbulence can be determined as shown below:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i)$$
$$= \frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \text{----------------------------------(4.30)}$$
$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i)$$
$$= \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \text{---------------(4.31)}$$

Where: α_k,α_ε = inverse effective Prandtl amounts for k andε,μ_{eff}= effective viscosity= $\mu + \mu_t$,$C_{1\varepsilon} = 1.42$,$C_{2\varepsilon} = 1.68$,$C_{3\varepsilon} = 1.0$. The term $R_\varepsilon = C_\mu \rho \eta^3 (1 - \eta/\eta_0)\varepsilon^2/k(1 + \beta\eta^3)$where:$C_\mu = 0.0845$, $\eta_0 = 4.38$, $\beta = 0.012$.

3. Results and discussion

ANSYS (CFX) program was used to develop numerical models for the two physical models in order to study the water surface profile and pressures along the center line of traditional and stepped spillway. The simulation was verified with the physical model test data. This verification consisted of: (i) flow water surface profiles along traditional and stepped spillways and (ii) pressure head distribution over smooth and stepped spillway. Water surface profile Water surface profile simulated by Fluid flow CFX. The predicted data were compared with measured results (Tadayon and Ramamurthy 2009). The outcomes of the simulations carried out using the numerical model are presented in Figures 5 and 6. Figures 7 and 8 illustrate the water velocity vectors for a standard spillway and a stepped spillway, respectively. According to these numbers, there is a good agreement between the results that were predicted and those that were measured, with a maximum error rate of 3.2%.

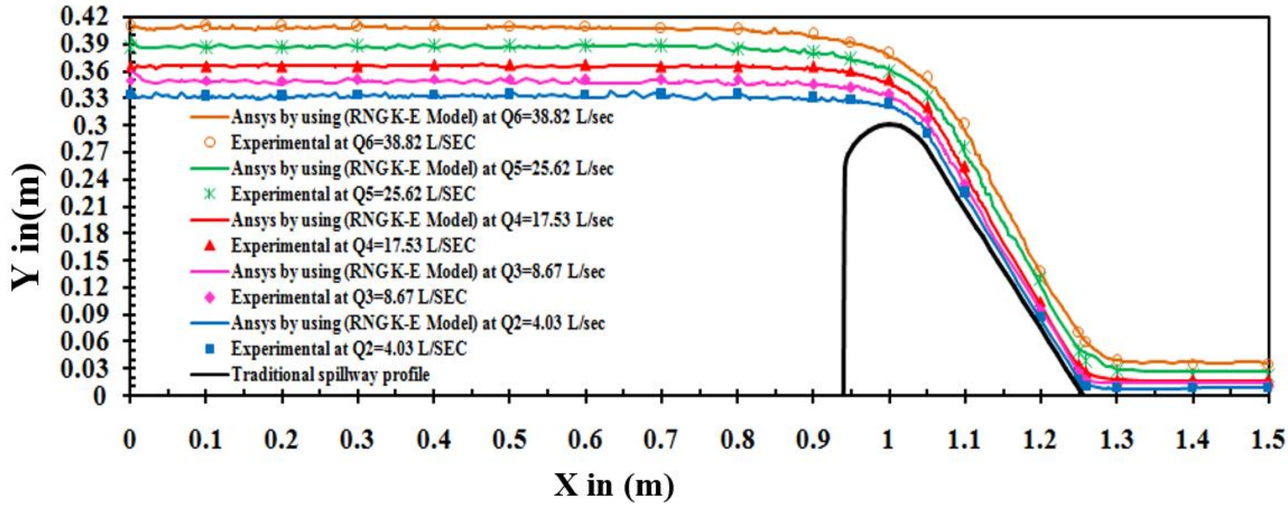


Figure 5. Comparison between calculated and experimental data by ANSYS CFX for Water surface profile over traditional model.

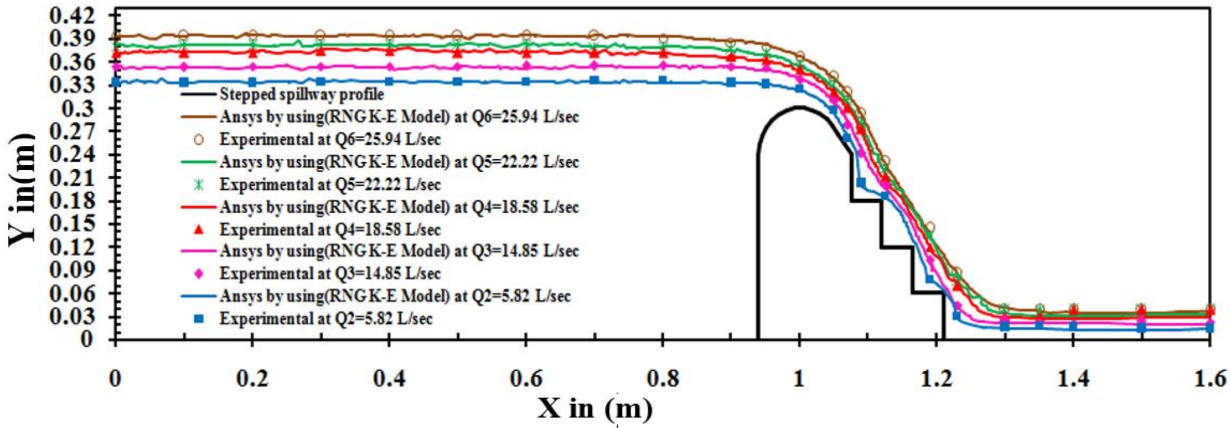


Figure 6. Comparison between calculated and experimental data by ANSYS CFX for Water surface profile over stepped model

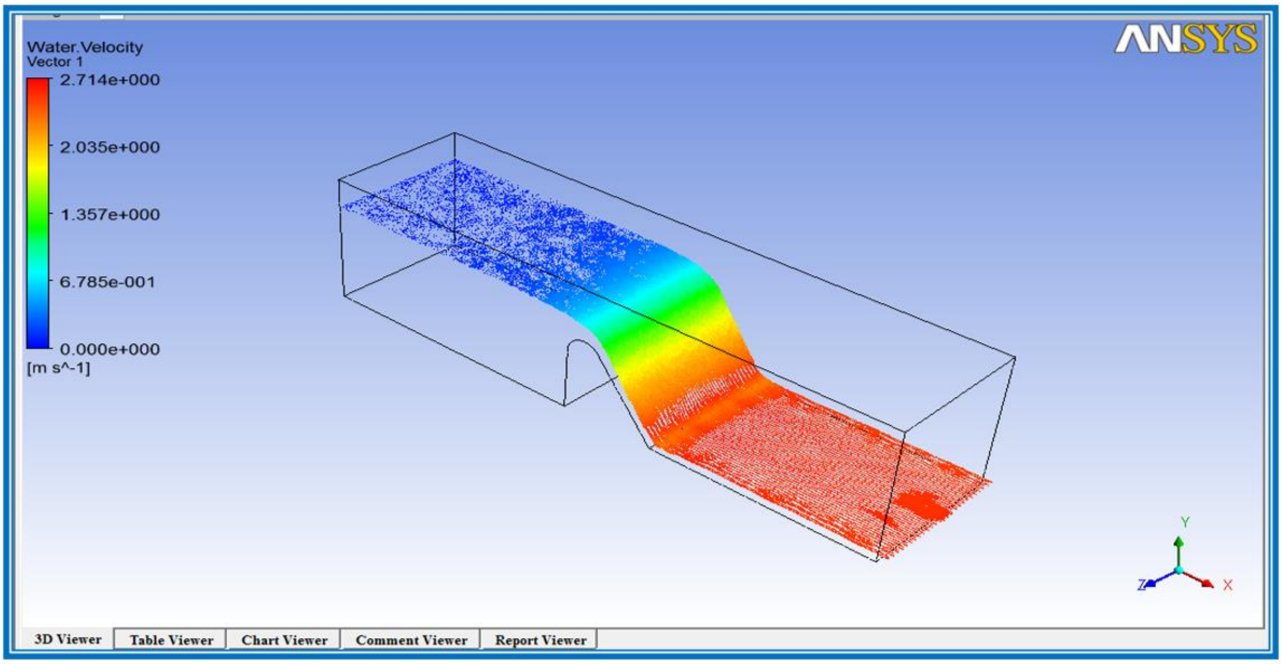


Figure 7. Vector of water velocity over traditional model at Q5=25.62 l/sec by using RNG K-Epsilon model

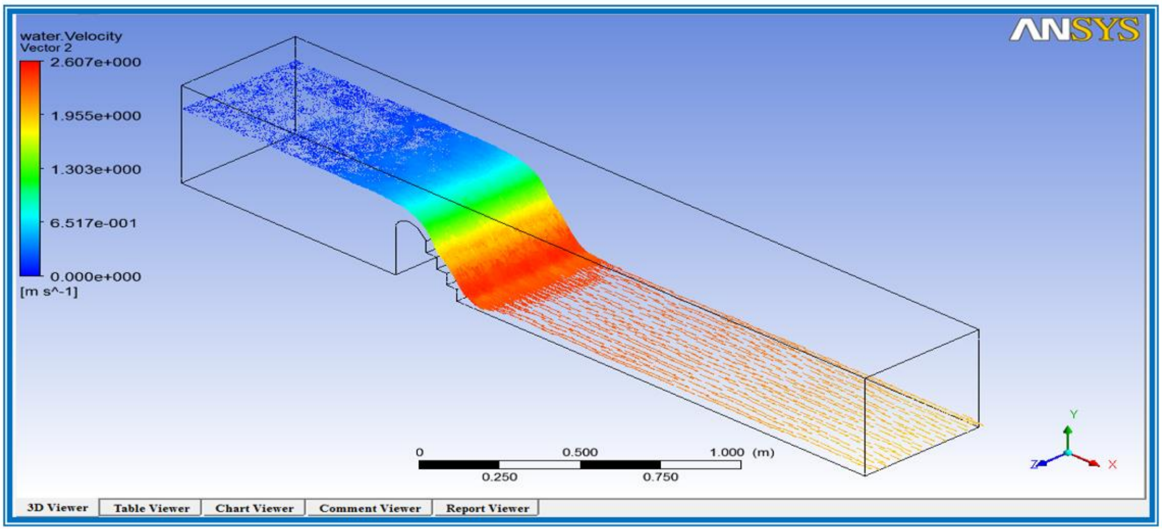


Figure 8. Vector of water velocity over Stepped model at Q5=22.22 l/sec by using RNG K-Epsilon model

To make sure that the pressure distributions computed by the program Fluid Flow (CFX) are scientifically acceptable and logically justified, the computed results are compared with those measured experimentally along the surface of traditional and stepped spillway model starting from the crest as a zero- distance location. Experimental and numerical results obtained from the Fluid Flow (CFX) program related to average pressure distributions over the traditional and stepped spillway with semicircular crest were utilized to validate the predicted results. Both measured and predicted average pressure head distributions over both types of spillway are shown in figures (9 and 10). For completely discharge rates. the average pressure head decreases with increasing the discharge at the crest of traditional and stepped spillway, while; at the end of sloping straight line. The minimum pressure occurs in a region at the crest, after the crest and at piezometer No.4; the comparison showed that there are some differences between the computed and measured results. These differences at the region of crest are attributed to waves on the upstream water surface, whereas the differences at the toe region are attributed to head fluctuations, high velocity, and high turbulence of flow. Additionally, in order to differentiate the model performance, the Root Mean Square Error, or RMSE value, is calculated to compare the average pressure head along the center line of the physical model to that of the predictive model. As a result, the findings of the comparison revealed a high level of congruence between them. Because, the value of the

RMSE is low and equal to 0.0233 at the discharge Q_6=38.82 L/sec and 0.0266 at the discharge Q_6=25.94 L/sec for both traditional and stepped spillway which is reflect to a high accuracy in the numerical prediction. This work demonstrates that a numerical model may successfully forecast flow variables over a complex stepped spillway by making use of the variable order flow (VOF) technique.

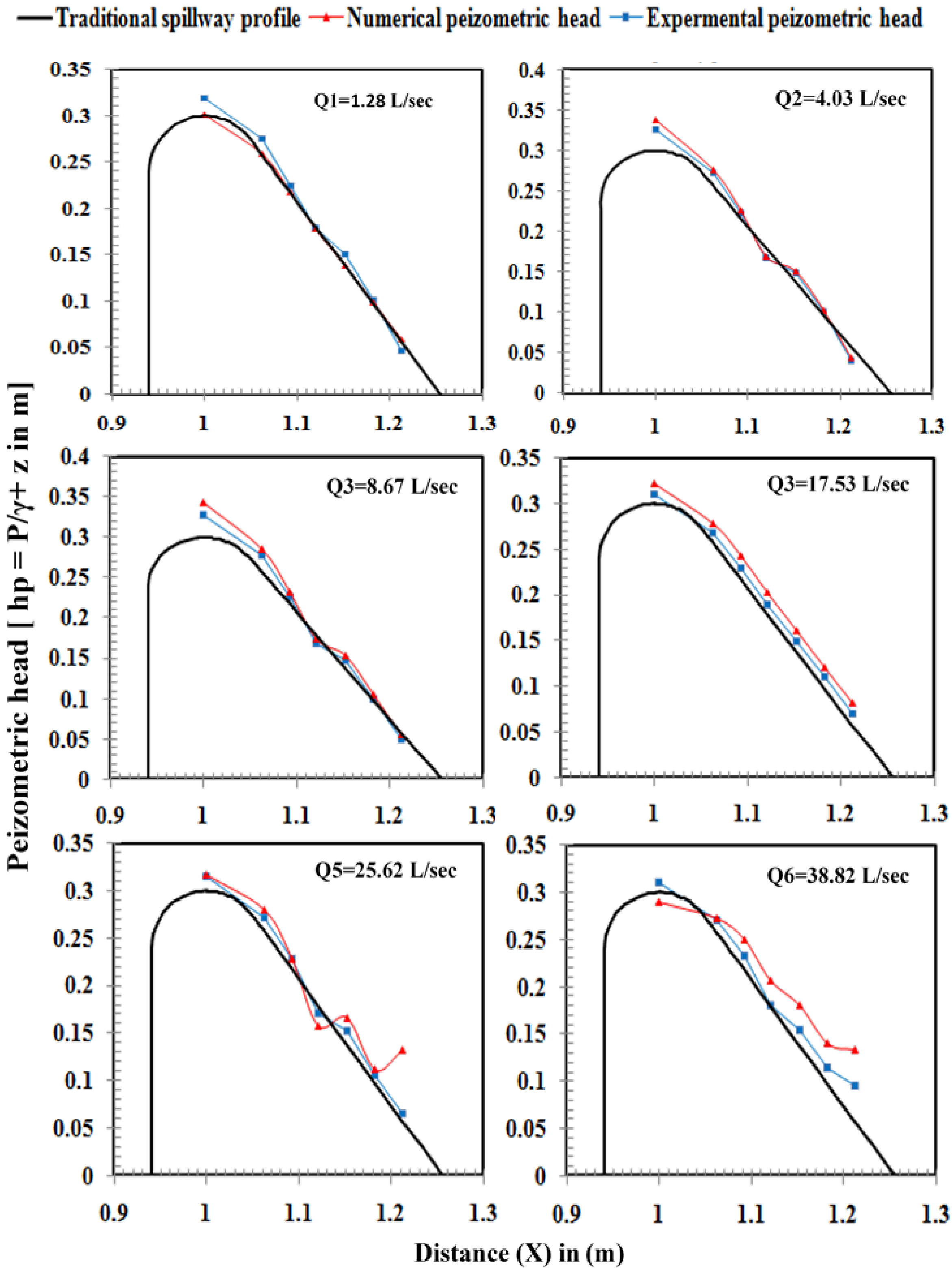
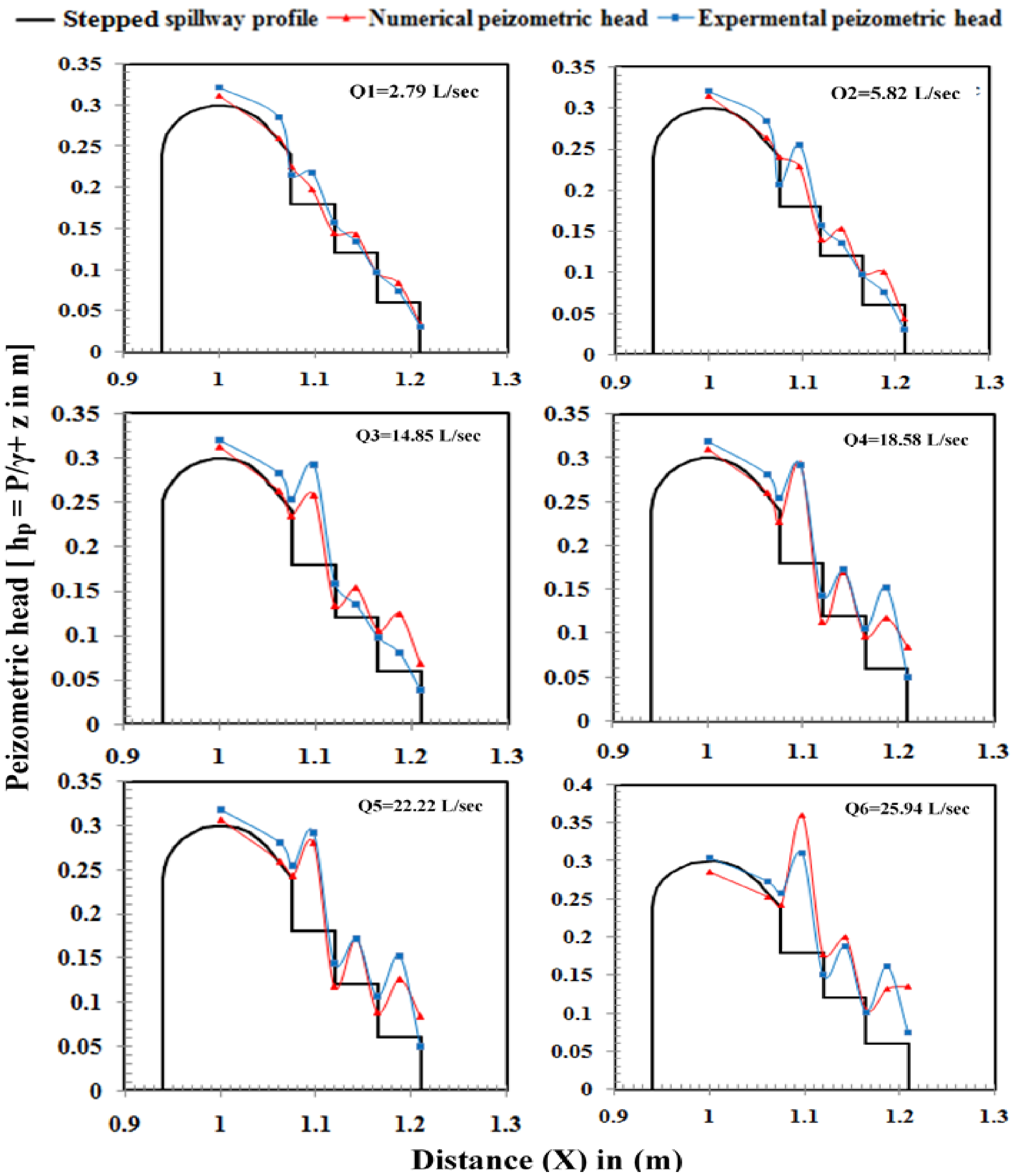


Figure 9. Comparison between experimental and calculated Piezometric head by ANSYS CFX over traditional model (No.1)



Figures 10. Comparison between experimental and calculated Peizometric head by ANSYS CFX over stepped model (No.1).

4. Conclusion

A numerical simulation research using commercially available CFD software (CFX) was carried out in order to examine flow characteristics for traditional and stepped spillway, which is a structure that is frequently used to dispose of excess water and to quantify discharge. Simulations were carried out to determine the free water surface flow, pressure distribution, and velocity distribution on both a standard and stepped spillway. According to the findings of the study, a numerical model may successfully estimate flow variables through a complex stepped spillway by employing the VOF technique.

A comparison was made between the experimental findings of a traditional spillway and a stepped spillway and the numerical findings. The experimental and anticipated findings were able to reach a satisfactory level of congruence. In conclusion, it is possible to assert that CFX is an effective tool for the resolution of hydraulic issues that are associated with flow measuring structures.

Conflict of interest

All the authors have confirmed that they do not have any competing interests, and they have come to an agreement that this research can be published in accordance with academic ethics.

Author contributions

Each of the authors contributed to the manuscript on an equal level.

Funding

The completion of the work was solely dependent on the efforts that we put in; it did not receive funding from any official Institute or firm.

References

[1] J. Zhan, J. Zhang, and Y. Gong, "Numerical investigation of air-entrainment in skimming flow over stepped spillways," *Theoretical and Applied Mechanics Letters,* vol. 6, no. 3, pp. 139-142, 2016.
[2] S. Haun, N. R. B. Olsen, and R. Feurich, "Numerical modeling of flow over trapezoidal broad-crested weir," *Engineering Applications of Computational Fluid Mechanics,* vol. 5, no. 3, pp. 397-405, 2011.
[3] Q. Zhang, Y. Diao, X. Zhai, and S. Li, "Experimental study on improvement effect of guide wall to water flow in bend of spillway chute," *Water Science and Technology,* vol. 73, no. 3, pp. 669-678, 2016.
[4] M. I. Yuce, A. A. Al-Babely, and M. A. Al-Dabbagh, "Flow simulation over oblique cylindrical weirs," *Canadian Journal of Civil Engineering,* vol. 42, no. 6, pp. 389-407, 2015.
[5] A. Parsaie and A. H. Haghiabi, "Evaluation of energy dissipation on stepped spillway using evolutionary computing," *Applied Water Science,* vol. 9, pp. 1-7, 2019.
[6] A. Ismael, H. Hussein, M. Tareq, and M. Gunal, "Simulation of Three-Dimensional Flow Field around Unconventional Bridge Piers," *Eurasian Journal of Science and Engineering,* vol. 3, no. 2, pp. 90-99, 2017.