How to cite: Varçin, H., Üneş, F., Gemici, E., Taşar, B., Turhan, E. (2023) Numerical Analysis of the Flow Over the Dam Spillway. *2023 "Air And Water – Components of the Environment" Conference Proceedings*, Cluj-Napoca, Romania, p. 121-129, DOI: 10.24193/AWC2023_12.

NUMERICAL ANALYSIS OF THE FLOW OVER THE DAM SPILLWAY

Hakan VARÇİN[1] , Fatih ÜNEŞ[1] , Ercan GEMİCİ[2], Bestami TAŞAR[1], Evren TURHAN[3]

DOI: 10.24193/AWC2023_12

ABSTRACT. The determination of hydraulic parameters is very important in the design of the dam spillway structure. Hydraulic parameters are obtained by theoretical and empirical approaches according to the design flow discharge. In general, before the application project, tests are made on the hydraulic model and the design is given its final shape. Advanced numerical modeling techniques can be used in conjunction with or as an alternative to experimental studies. This study investigated hydraulic parameters in three dimensions under design flow in a dam spillway using computational fluid dynamics (CFD). In the numerical model, the VOF method, which can solve two-phase flows, and the standard k-e turbulence model are used. Obtained results were compared with experimental results. It was determined that the experimental and numerical model results were quite compatible with each other.

Keywords: Dam, Spillway, Experimental, CFD Model, VOF Method

1. INTRODUCTION

Spillways are safety and control structures used for the discharge of excess water in the reservoir and reservoir operations during flooding. These structures, which are built with and without control, are one of the most important elements of dams in terms of hydraulics(Design of small dams, 1977). It is very important to detect and solve problems by analyzing the flows in the spillway chutes in advance, in terms of the safety and cost of such structures. Hydraulic calculations are made according to the design flow discharge determined by considering the dam type and downstream conditions. Since the hydraulic calculations are made using theoretical and empirical approaches at the provincial stage of the design, the model study is required to confirm the accuracy of the design (Willey et al., 2012). Although the method commonly used in this regard is physical model experiments, numerical models using advanced techniques have also

[1] Iskenderun Technical University, Civil Engineering Department , e-mail: hakan.varcin@iste.edu.tr fatih.unes@iste.edu.tr bestami.tasar@iste.edu.tr
[2] Bartin University, Bartin – TURKEY, e-mail: egemici@bartin.edu.tr
[3] Adana Alparslan Türkeş Science and Technology University, Civil Engineering Department, e-mail: eturhan@atu.edu.tr

started to be widely used today. The key is to use both methods together to take advantage of them. Since numerical models are more advantageous in terms of time and cost than experimental models, only numerical models can be used in some cases. From the past to present, hydraulic analyzes of spillways of various types and capacities have been made by using experimental and numerical methods by many researchers. Some of the recent studies on the subject can be summarized as follows: Rodi (1980) tested the flow over a conventional spillway using the standard k - ε model, which is the most commercially commercial turbulence model, and proved that this turbulence model is quite accurate. Olsen and Kjellesvig (1998) numerically modeled the water flow over a two- and three-dimensional spillway by choosing different geometries to estimate the spillway capacity. The k - ε turbulence model is chosen and the equations of motion are solved accordingly. Numerical results and experimental studies were compared and close values were obtained. Savage and Johnson (2001) completed their work using Flow 3D to compare the flow parameters on a standard ogee-crested spillway between physical model test results, current spillway literature design guidelines created by USACE and USBR, and numerical simulation results. Numerical simulations were solved with Reynolds Mean Navier-Stokes equations using the finite volume method. This study showed that the numerical tools for calculating the discharge and pressure on the spillway are sufficiently advanced. Today, physical model studies are still considered the basis against which other methods are compared, but numerical simulations have improved accuracy to derive discharge capacities and pressures. Kumcu [2016] conducted a numerical study to examine the flow characteristics of the Junction Dam and spillway design. In the article, the experimental data of the Kavşak Dam spillway and the results obtained by CFD analyzes are compared. Daneshfaraz and Ghaderi [2017] investigated the effect of the curvature diameter of the inverted curve the downstream of an ogee spillway on the base pressure with a 2D numerical model using the FLUENT software. Demeke et al. (2019) used a 3D hydrodynamic model to unveil safety issues in an Ethiopian dam spillway. Yang et al. (2019), after a review of Sweden dams, highlights the potentiality of CFD to be used in conjunction with physical modeling. Gadhe et al. (2021) compared experimental and numerical simulations to an upgraded spillway of an Indian dam, demonstrating how the numerical model can capture the water surface profile along the spillway very accurately and can be a complementary tool for assessing the hydraulic performance of structures. Based on these studies in the literature, the VOF model and the k-e turbulence model were used in the CFD part of this study.

In this study, hydraulic parameters under design flow in a dam spillway are investigated in three dimensions using both experimental and computational fluid dynamics (CFD). In the experimental studies, velocity and water depth measurements were made on a section determined on the 1/200 scale spillway model. In the other part of the study, a three-dimensional numerical model of the spillway model was created and similar parameters were examined. In the numerical model, the VOF method, which can solve two-phase flows, and the standard k-e turbulence model are used. The obtained results were compared with the experimental results and a very good agreement was obtained. Thus, the advantages of numerical models as well as hydraulic model studies of such projects are discussed.

2. MATERIAL AND METHODS

The spillway structure of the dam used in this study is a radial-covered spillway. The spillway, designed with a capacity of 10055 $m^3 s^{-1}$, has 6 chambers, each of which is 11 m wide and 15.60 m high, equipped with a radial cover. The discharge channel, which extends downstream of the ogee crested sill, is 81 m wide (Fig. 1).

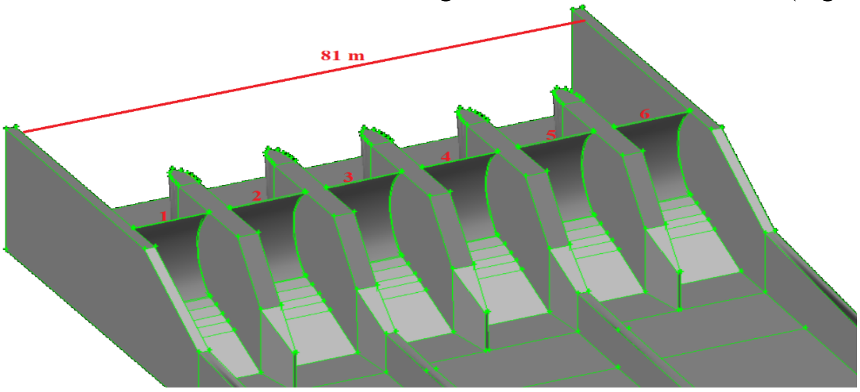
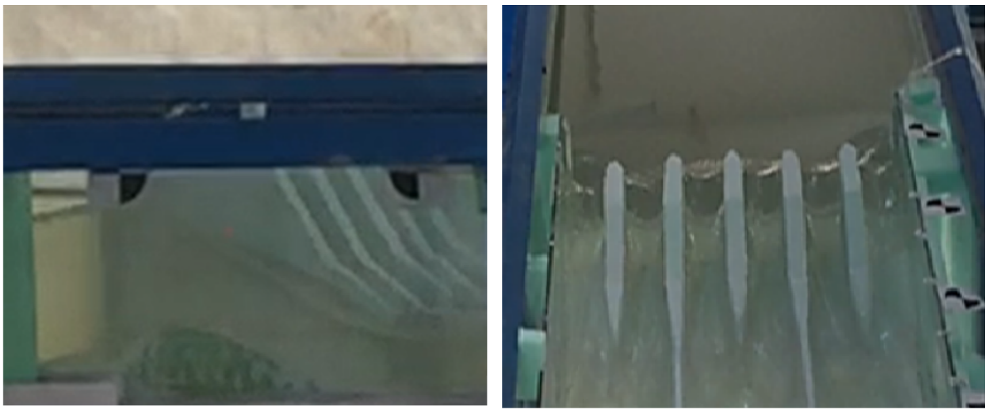


Fig. 1. ***3D prototype spillway geometry***

2.1. Experimental Study

Experimental studies on the spillway model created according to 1/200 scale Froude similarity were carried out in the hydraulic laboratory. Velocities and depths in the sections determined in the experiments were measured. Velocity measurements were determined with the Acoustic Doppler Velocimeter (ADV) device (Fig. 2).



a b

Fig. 2. ***Approach and entrance of the spillway structure shown during the experiment a) Side view b) Top view***

2.2. CFD Study

The flow over the spillway is an open channel flow. In open channel flow, the flowing fluid has a free surface at atmospheric pressure and the driving force is gravity(Chaudhry, 2008). When the literature is examined, it is seen that such free surface flow is simulated by the fluid volume (VOF) method as water-air 2-phase flow problems(Morovati et al., 2016). Flows over the spillway are high velocities and turbulent. According to the literature, the standard k-ε turbulence model can be used in the three-dimensional numerical simulation of the flow(Hirt & Nichols, 1981).

The investigated open channel flow is a 3D, turbulent, steady free surface flow. Equations used in the numerical model; continuity, momentum, and turbulence equations. The standard k-e turbulence equation was used as the turbulence equation.

In this study, the volume of fluid (VOF) method was used to calculate the water-air interface. The VOF method essentially determines whether the element volumes in the computational mesh are empty, partially filled, or completely filled with water. (Hirt & Nichols, 1981)

1/200 scale geometry of the spillway model was made with the GAMBIT drawing program, and approximately 800 thousand triangular mesh was created within this geometry (DSI, 1985). The three-dimensional geometry has been created, and the boundary conditions of the flow formed in the spillway have been defined. Accordingly, the entrance part of the structure was determined as velocity inlet, outlet, and surface pressure outlet as a wall boundary condition in all other parts (Fig. 3).

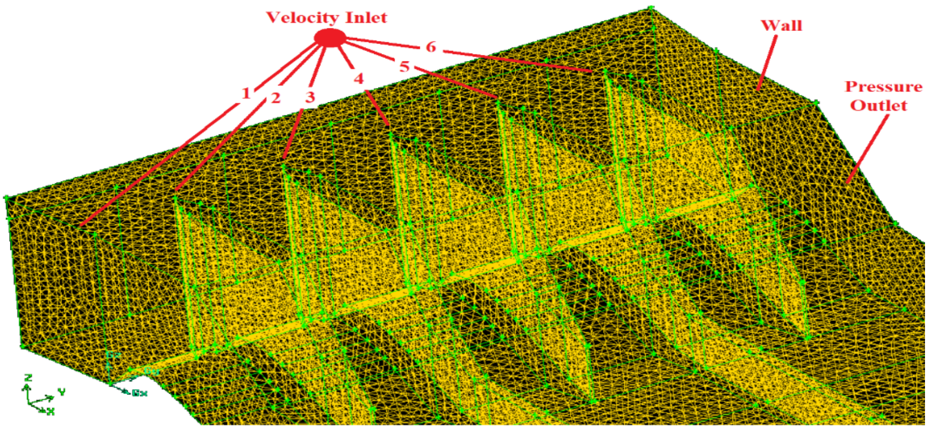


Fig. 3. ***Boundary conditions defined for the flow in the three-dimensional spillway model***

The flow on the spillway passes around five sluice pillars on the sill structure, resulting in six different entry velocities. Then, it flows from three separate branches with two separating walls on the discharge channel. In spillway projects, when the gates are fully open at the time of flooding, the initial velocities in the approach channel were adjusted according to the Froude similarity for the 1/200 scale model, and these values were used in CFD analyses. Initial velocity values are given in Table 1 (DSI, 1985).

In the time-dependent solution process, the initial condition is F=1 at the entrance boundary of the solution region, and F=0 at the exit boundary of the other regions and the solution region.

Table 1. Froude similarity values for prototype and model

Parameters		Dimension	Froude Scale Ratio λ =1/200	Prototype Value	Similarity Account	Model Value
Discharge (Q)		L^3T^{-1}	$\lambda^{5/2}$	10055 $m^3\ s^{-1}$	$Q_p{*}\lambda^{5/2}$	17,7 lt s^{-1}
Velocity (V)	Inlet 1	LT^{-1}	$\lambda^{1/2}$	6.5 m s^{-1}	$V_p{*}\ \lambda^{1/2}$	0.46 m s^{-1}
	Inlet 2			6 m s^{-1}		0.42 m s^{-1}
	Inlet 3			5,52 m s^{-1}		0.39 m s^{-1}
	Inlet 4			5,42 m s^{-1}		0.38 m s^{-1}
	Inlet 5			5,4 m s^{-1}		0.36 m s^{-1}
	Inlet 6			5,9 m s^{-1}		0.42 m s^{-1}

The time step for the turbulence model used in the numerical modeling is Δt=0. It was chosen as 0.1 second and a solution was made for 120 seconds, during which the numerical solution became stable. The numerical solution of the fundamental equations according to the boundary conditions was made using the ANSYS-Fluent package program based on the finite volume method (ANSYS, 2015).

3. RESULTS AND DISCUSSIONS

The general view of the flow formed on the spillway model as a result of the CFD analysis and section A are shown together (Fig. 4).

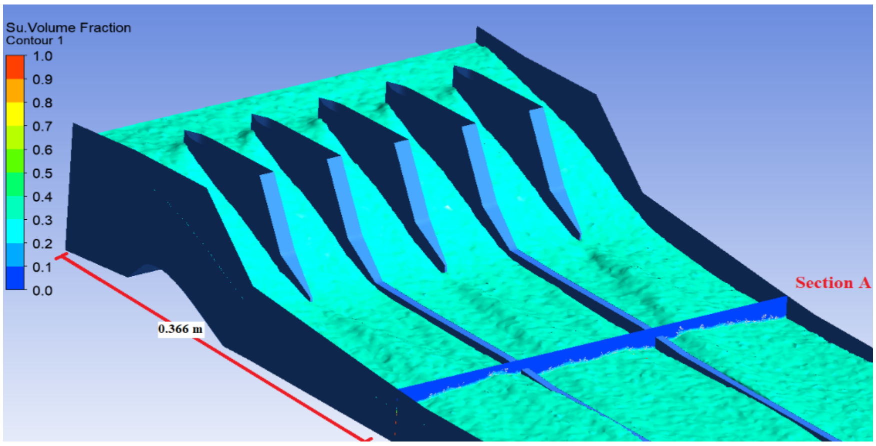


Fig. 4. ***CFD analysis water surface overview***

When Figure 4 is examined, it is seen that the spillway model is sufficient for the flood flow. Overflows are observed at some points of the discharge channel separation walls, which are lower than the spillway side walls. The reason for this is the turbulent flow in the flood regime. Velocity and depth measurements were made at three points in section A, which is approximately 0.36 m from the spillway starting point.

3.1. Comparison of Velocity

The experimental and numerical model velocity values obtained in Section A is shown in Table 2. The error rates by comparing the experimental and CFD analysis results.

Table 2. CFD and experimental results for velocity fluctuations for section A

	Point No	Location at y direction (m)	Experimental Results (m s^{-1})	CFD Results (m s^{-1})	Error (%)
Section A	**P1**	0.0700	1.149	1.373	19.5
	P2	0.2025	1.378	1.266	8.1
	P3	0.3337	1.273	1.361	6.9

When Table 2 is examined, the error rates in velocity values at P1, P2 and P3 points in section A of the discharge channel were calculated as 19.5,8.1 and 6.9, respectively. The P1 error percentage was higher than the others. Since the P1 point is close to the right side wall of the chute channel, we think that the turbulence occurring at this point may have caused errors in the measurements. The numerical model velocity contours obtained in Section A are shown in Fig. 5.

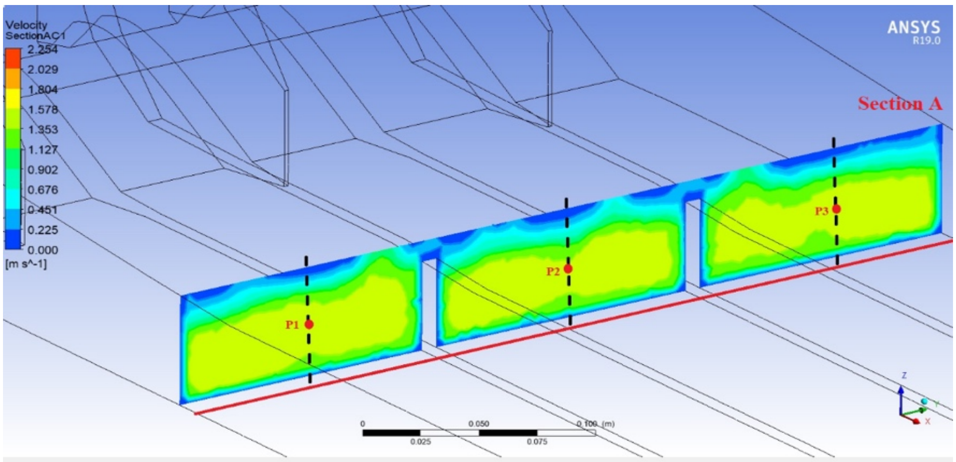


Fig. 5. ***Flow velocity contours for section A as a result of CFD analysis***

In Figure 5, it is seen that the velocity values increase from the wall edges to the center. If the velocity values in the cross-sections are examined, it can be observed that the velocity values of the air phase are lower after the water–air separation line.

3.2. Comparison of Water Depth

The experimental and numerical model water depth obtained in Section A is shown in Table 3. Table 3 show the error rates by comparing the experimental and CFD analysis results.

Table 3. CFD and experimental results for water depth fluctuations for section A

	Depth No	Location at y direction (m)	Experimental Results (m s^{-1})	CFD Results (m s^{-1})	Error (%)
Section A	D1	6.4375	4.5	4.315	4.1
	D2	20.2500	4.5	4.015	10.8
	D3	34.0625	4.5	3.915	13.0

When Table 3 is examined, the error rates in water depth at D1, D2 and D3 points in section A of the discharge channel were calculated as 4.1,10.8 and 13.0 respectively. The error rates in the measurement values are considered to be at an acceptable level. Turbulent fluctuations created by the incoming flood flow are thought to affect the measurement results as the cause of the error. The numerical model water depth contours obtained in Section A are shown in Figure 6.

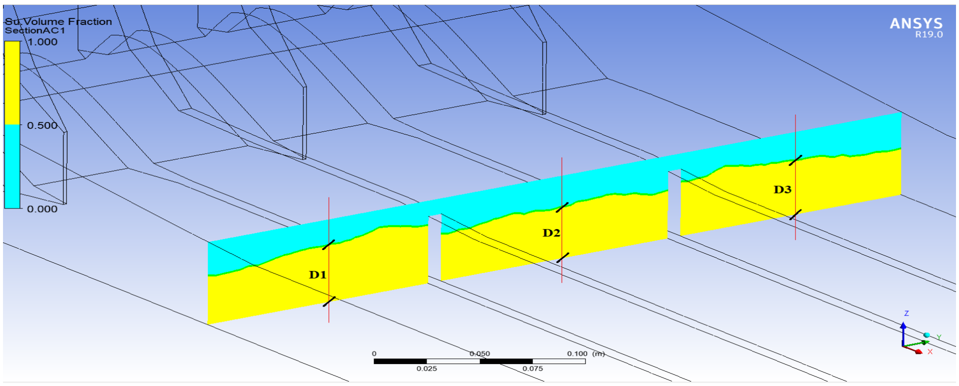


Fig. 6. ***Flow water depth for section A as a result of CFD analysis***

When Figure 6 is examined, it is seen that the water depths in the right and left chute channels near the separation walls are higher than in the middle chute channel. At some points, it has risen by exceeding the separation walls in the middle of the chute channels, but the situation is not troublesome as long as the overflow remains in the gutter.

5. CONCLUSIONS

In this study, a 1/200 scale model of a spillway structure was created and experimental studies were carried out in the hydraulic channel for the flood situation After the experimental studies, a CFD simulation of the model was made with the ANSYS Fluent program, and velocity values and water depths were measured for a determining section. The results obtained as a result of the studies are listed as follows;

- As a result of the studies, it was observed that the error rate was not very high between the experimental and numerical analysis velocity and depth results.
- It was observed that the error rate of the edge flow in contact with the wall on the spillway model was higher than the flow in the middle region. It is thought that this situation is caused by the wall roughness.
- Turbulence effects in the flow in flood regime caused inaccuracies in the measurements.
- The good compatibility of the results shows that the numerical model is successful and can be used. It is thought that it will save both time and cost for future spillway model experiments.

ACKNOWLEDGMENTS

The authors thank the General Directorate of State Hydraulic Works, Technical Research and Quality Control Department (DSI TAKK) for their support to this study.

REFERENCES

1. ANSYS. (2015). *FLUENT Theory Guide*. ANSYS Inc.
2. Chaudhry, M. H. (2008). Open-channel flow: Second Edition. In *Open-Channel Flow: Second Edition*. Springer US. https://doi.org/10.1007/978-0-387-68648-6
3. Demeke, G. K., Asfaw, D. H., & Shiferaw, Y. S. (2019). 3D Hydrodynamic Modelling Enhances the Design of Tendaho Dam Spillway, Ethiopia. *Water*, *11*(1), 82. https://doi.org/10.3390/w11010082
4. Design of small dams. (1977). *Design of small dams* (B. of Reclamation (ed.)). U.S. Government Printing Office.
5. DSI. (1985). Çatalan dam spillway model experiments report. In *General Directorate of State Hydraulic Works-Hydraulic Model Laboratory*.
6. Gadhe, V., Patil, R. G., & Bhosekar, V. V. (2021). Performance assessment of upgraded spillway – case study. *ISH Journal of Hydraulic Engineering*, *27*(3), 327–335. https://doi.org/10.1080/09715010.2018.1550730
7. Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, *39*(1), 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

8. Morovati, K., Eghbalzadeh, A., & Soori, S. (2016). Numerical Study of Energy Dissipation of Pooled Stepped Spillways. *Civil Engineering Journal*, *2*(5), 208–220. https://doi.org/10.28991/CEJ-2016-00000027
9. Olsen, N. R. B., & Kjellesvig, H. M. (1998). Three-dimensional numerical flow modelling for estimation of spillway capacity. *Journal of Hydraulic Research*, *36*(5), 775–784. https://doi.org/10.1080/00221689809498602
10. Rodi, W. (1980). Turbulence models and their application in hydraulics - A state of the art review. *International Association for Hydraulic Research (IAHR)*.
11. Savage, B. M., & Johnson, M. C. (2001). Flow over Ogee Spillway: Physical and Numerical Model Case Study. *Journal of Hydraulic Engineering*, *127*(8), 640–649. https://doi.org/10.1061/(ASCE)0733-9429(2001)127:8(640)
12. Willey, J., Ewing, T., Wark, B., & Lesleighter, E. (2012). Complementary use of physical and numerical modelling techniques in spillway design refinement. *Commission Internationale Des Grands Barrages*.
13. Yang, J., Andreasson, P., Teng, P., & Xie, Q. (2019). The Past and Present of Discharge Capacity Modeling for Spillways—A Swedish Perspective. *Fluids*, *4*(1), 10. https://doi.org/10.3390/fluids4010010