How to cite: VARÇİN H., ÜNEŞ F., GEMİCİ E., KAYA Y. Z. (2024) Examining the Change in Pressure Parameters in the Chute Spillway with a Numerical Model. *2024 "Air and Water – Components of the Environment" Conference Proceedings*, Cluj-Napoca, Romania, p. 137-145, DOI: 10.24193/AWC2024_13.

Examining the Change in Pressure Parameters in the Chute Spillway with a Numerical Model

Hakan VARÇİN[1] , Fatih ÜNEŞ[1] , Ercan GEMİCİ[2] , Yunus Ziya KAYA[3]

DOI: 10.24193/AWC2024_13

ABSTRACT. Examining the Change in Pressure Parameters in the Chute Spillway with a Numerical Model. Knowing hydraulic parameters is very important in the design of spillways, which are the safety structures of dams. While obtaining these parameters, traditionally theoretical and empirical approaches are used and scale model experiments are performed. Today, with the increase in computer processing capacity, numerical modeling techniques are used as an alternative to long and costly experimental studies. In this study, the pressure parameter changing along the longitudinal section in the chute spillway was examined using computational fluid dynamics (CFD). In the 3D CFD model of the spillway, the VOF method and the standard k-e turbulence model, which can solve two-phase flows, were used for the design discharge. The obtained pressure parameter results were examined and interpreted along the spillway section. According to the pressure results obtained as a result of numerical analysis, points along the spillway where cavitation may occur have been determined.

Keywords: Spillway, CFD Model, VOF Method , Pressure

1. INTRODUCTION

Spillways are structures that pass water by controlling the flow of water from upstream to downstream, have important functions related to the operation and stability of the dam and have an important place in the cost of the dam. For a dam to be safe, the spillway must be able to pass large floods without putting the dam at risk. Adequate planning, design, construction and reliable operation of the spillway are of great importance for the reliability of the dam and its downstream inhabitants. Due to their importance, spillways are carefully studied in their project design and their calculations are supported by modelling when necessary. Although physical model experiments are the commonly used method in this regard, numerical models using advanced techniques have also been widely used today (Aydin et al., 2020; Kumcu, 2017). Since numerical models are more advantageous than experimental models in terms of time and cost, only numerical models can be used in some cases. Hydraulic analyses of spillways of various types and capacities have been carried

[1] Iskenderun Technical University, Civil Engineering Department , e-mail: hakan.varcin@iste.edu.tr fatih.unes@iste.edu.tr
[2] Bartin University, Civil Engineering Department , e-mail: egemici@bartin.edu.tr
[3] Osmaniye Korkut Ata University, Engineering Faculty, Civil Eng. Department, yunuszkaya@osmaniye.edu.tr

out by many researchers using experimental and numerical methods. Some of the recent studies on the subject can be summarized as follows:

Daneshfaraz and Ghaderi (Daneshfaraz et al., 2016) investigated the effect of the curvature diameter of the inverted curve 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. 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.

In this study, a three-dimensional numerical model of a prototype dam spillway was created. The VOF method, which can solve two-phase flows, and the standard k-e turbulence model were used in the numerical model. In the numerical analysis, the pressure parameter of the flow occurring in the spillway design discharge was examined in three different length sections along the spillway length and discussed by making comparisons.

2. MATERIAL AND METHODS

2.1. Case Study

In this study, a numerical model of a prototype spillway structure was created. For this purpose, a chute spillway structure with two slopes in the discharge channel was chosen. The spillway structure is a radial gate spillway belonging to a dam that is still in operation and was built to generate electrical energy (DSI, 1985). The spillway, designed with a capacity of 10055 m3 s-1, is 82.5 m wide and 588 m long. Discharge channel slopes are 0.03 and 0.17 and are connected to a vertical curve (Figure 1).

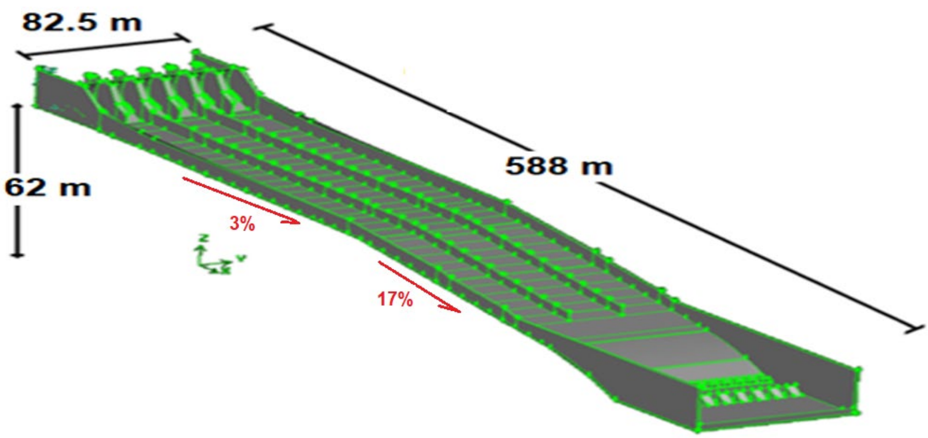

Fig.1. 3D prototype spillway geometry

2.2. CFD Study

The flow over the spillway is open channel flow. In these flows, the flowing fluid has a free surface at atmospheric pressure and the fluid moves with the effect of gravity. This type of free surface flow is simulated by the volume of fluid (VOF) method as a water-air 2-phase flow problem. Flows over spillways are turbulent and at high velocities. According to the literature, the standard k-ε turbulence model can be used for three-dimensional numerical simulation of the flow(Hirt & Nichols, 1981).

The three-dimensional geometry of the spillway structure was designed with the GAMBIT drawing program, approximately two million triangular mesh were created in this geometry and the boundary conditions of the flow were defined. Accordingly, the entrance section of the structure, along with the velocity inlet, outlet and surface pressure outlet, was determined as the wall boundary condition for all other sections (Figure 2). In case of a flood flow rate of 10 055 m^3/s in the prototype spillway, the gates are fully open and the flood height in the approach channel is calculated as 25 m. The energy breaker pool at the end of the spillway structure was not included in the numerical model(Özdemir et al., 2022; Taşar et al., 2023, 2021; Varçin, Üneş, Gemici, & Zelenakova, 2023; Varçin, Üneş, Gemici, TAŞAR, et al., 2023).

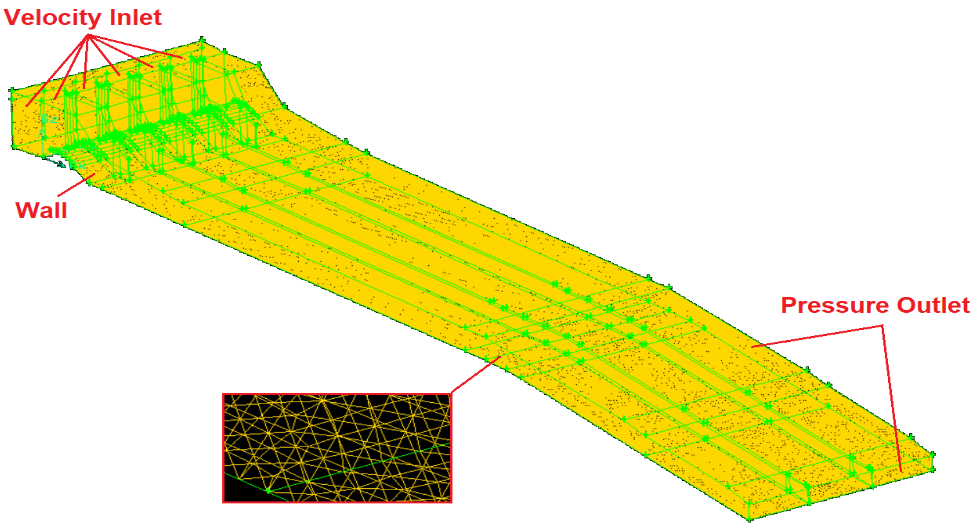

Fig. 2. Boundary conditions defined for the flow in the three-dimensional spillway model

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 addition, in accordance with the prototype spillway project, the approach channel length is 15 m and the length between the approach channel and the spillway outlet is 402 m.

The mesh qualities are controlled with equiangular skewness, equalize skewness and aspect ratio in the GAMBIT program. The skewness error increases in the model due to the shape of the spillway. Those errors are defined and corrected by increasing the amount of mesh with GAMBIT. However, when the mesh gets smaller, the number of mesh is increased. Obviously, this requires more computer capacity and time(Taşar et al., 2021; Unes, 2008; Üneş, 2008; Üneş et al., 2016; Üneş & Ağıralioğlu, 2017; Unes & Varcin, 2017; Üneş & Varçin, 2015).

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. 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. Numerical analyzes were made with ANSYS-Fluent Relesa19. A workstation with an four- core Xeon 4.0 GHz processor and 32 GB RAM was used in the numerical analysis. The analysis took approximately 12 hours. Discretization methods and solver settings are presented in Table 1. From Table 1, convergence criteria, discretization methods, etc. is seen. (ANSYS, 2015).

Table 1. Numerical model details.

Solver set	**Solver Space-Time**	**Pressure based 3D, unsteady**
Model	Multiphase Model Viscous Model	VOF k-ε
Phase	Primary Phase Secondary Phase	Air water
Discretization	Pressure Momentum	Presto Second order upwind
Pressure-Velocity Coupling	Method	Coupled
Convergence Criterion	Residuals	0.001 (Continuity) 0.001 (Momentum)

3. RESULTS AND DISCUSSIONS

In the numerical model, flow pressure measurements were made on six longitudinal sections (Longitudinal sections 1, 2, 3, 4, 5, 6) along the spillway. The spillway discharge channel consists of three parts: right middle and left chute channel. The right shot channel is represented by longitudinal section 1, the middle shot channel is represented by longitudinal section 3, and the left shot channel is represented by longitudinal section 5, and the examinations were made on these longitudinal sections. In Figure 3, longitudinal sections are shown on the spillway geometry.

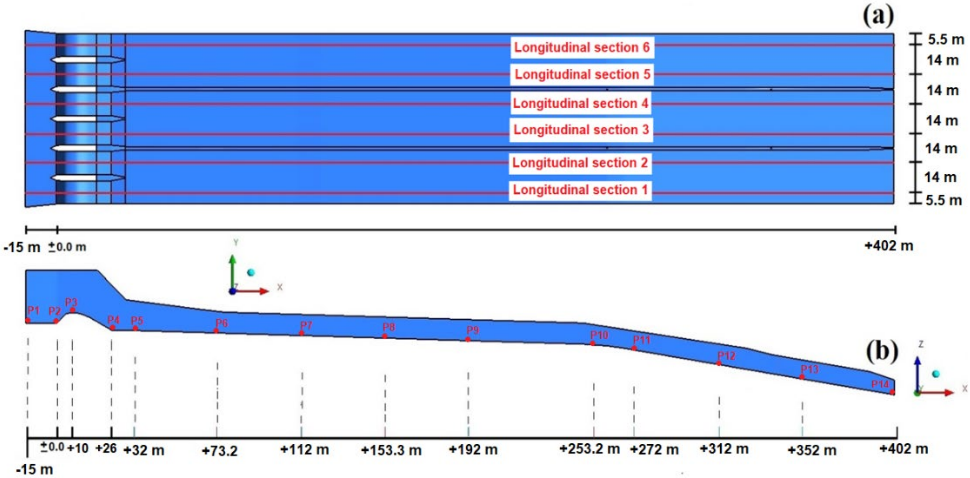

Fig. 3. Longitudinal section points measured on the spillway model a) Spillway plan section b) Spillway longitudinal section

As a result of the numerical analysis performed for the prototype spillway, the general view of the flow formed on the spillway model is shown in Figure 4.

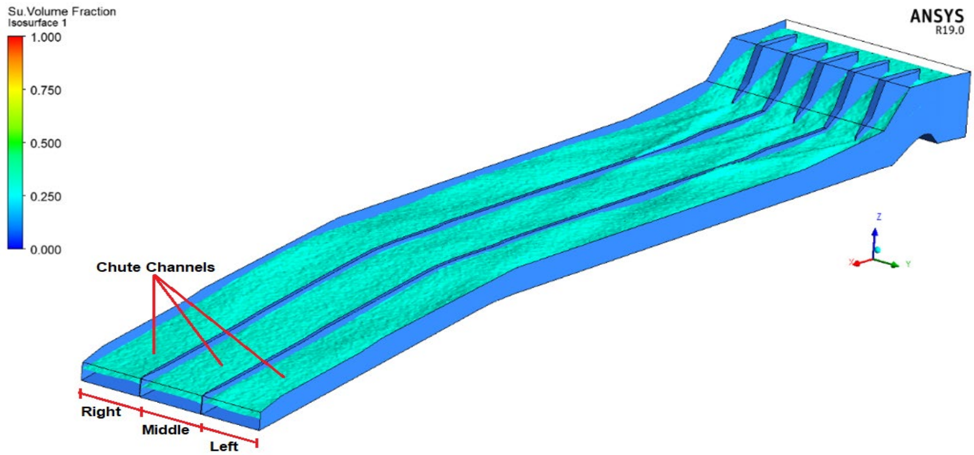

Fig. 4. CFD analysis water surface overview

As seen in Figure 4, as a result of the numerical analysis, it is seen that the prototype spillway can transfer the given flow rate from upstream to downstream. Water overflows were observed at some points on the separation walls at the middle points of the discharge channel.

3.1. Comparison of Pressure

The pressure values of the flow along the right, middle and left flume channel of the spillway are shown in Table 2 and Figure 5. The right chute channel represents longitudinal section 1, the middle chute channel represents longitudinal section 3, and the left chute channel represents longitudinal section 5.

Table 2. Pressure fluctuations in longitudinal sections obtained as a result of CFD analysis

Point No	Location at x direction (m)	*Longitudinal section 1 Pressure* (kPa)	*Longitudinal section 3 Pressure* (kPa)	*Longitudinal section 5 Pressure* (kPa)
1	-015.0	210.650	203.703	201.194
2	+/-00.0	227.231	215.446	212.038
3	+010.0	022.341	025.837	025.839
4	+026.3	215.114	210.397	206.654
5	+032.0	143.785	131.103	130.473
6	+073.2	063.645	062.045	069.041
7	+112.0	066.261	060.133	063.350
8	+153.3	068.721	059.767	057.639
9	+192.0	062.989	057.467	060.676
10	+253.2	035.841	031.673	035.308
11	+272.0	043.961	039.664	040.210
12	+312.0	054.632	046.867	045.275
13	+352.0	047.596	043.079	042.744
14	+402.0	016.690	015.113	017.093

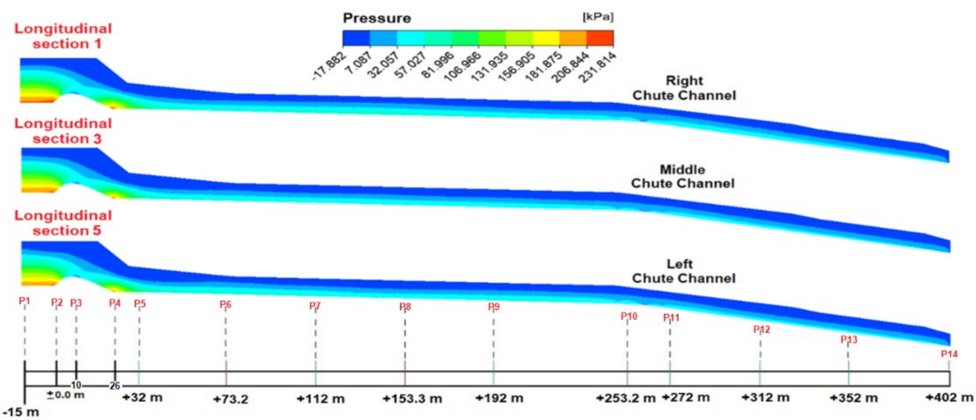

Fig. 5. Flow pressure contours in longitudinal sections as a result of CFD analysis

When Table 2 and Figure 5 is examined, it is determined that the pressure values calculated at fourteen different points along the spillway in all three length sections are close to each other. At points P1 and P2, where the approach channel is located, the pressure values were calculated high due to the high water height. The highest pressure value at P2, which is the beginning of the weir threshold, was obtained as 227.231 kPa in the 1st section representing the right chute channel, 215.444 kPa in the 3rd section representing the middle chute channel, and 212.038 kPa in the 5th section representing the left chute channel. It means that the flow height in the right chute channel is high and the flow accumulates towards the right side. At the P3 point, located at the top of the weir threshold, the water velocity increased suddenly due to the effect of gravity, causing sudden decreases in pressure values. The pressure values at point P3 were calculated as 22.341 kPa, 25.837 kPa and 25.839 kPa, respectively. Pressures are low at this point but no negative pressure has occurred. Damages may occur due to the vacuum effect that will occur as a result of negative pressures, especially at the tops of the weirs. At the P4 point, which is the end of the weir threshold and the beginning of the discharge channel, the pressure values of the flow rapidly exceeding the threshold increased rapidly. The pressure values at point P4 were calculated as 215.114 kPa, 210.397 kPa and 206.654 kPa, respectively. It has been observed that the pressure values gradually decrease at the points P5, P6, P7, P8, P9, and P10, which are located on the 3 percent slope of the discharge channel in each of the three sections. While the pressure values at P5 point were 143.785 kPa, 131.103 kPa and 130.473 kPa, respectively, they decreased to 35.841 kPa, 31.673 kPa and 35.308 kPa, respectively, at P10 point. A pressure drop due to sudden velocity occurred at the P10 point, where the slope of the discharge channel changed from 3 percent to 17 percent. Therefore, damage due to cavitation may occur at this point. Similarly, it was observed that the pressure values gradually decreased at points P11, P12, P13 and P14, which are located at a 17 percent slope of the discharge channel. The lowest pressure values were obtained at the P14 point, located at the exit of the spillway, where the velocity values are maximum. The pressure values at point P14 were calculated as 16.690 kPa, 15.113 kPa and 17.093 kPa, respectively.

As can be seen from the table and graph, damage due to cavitation is expected to occur at points along the spillway where velocity values are high and pressure values are low. Damage from cavitation may occur at point P3 above the weir threshold, at point P10 where the discharge channel slope changes, and at point P14 at the discharge channel outlet. Therefore, precautions should be taken at these points. At these points, the strength of concrete surfaces can be increased by adding various additives to the concrete during the manufacturing phase. It can also be used in anti-cavitation aerators. However, these measures are matters that affect the cost.

4. CONCLUSIONS

In this study, the flow passing over a two-slope prototype chute spillway structure was CFD simulated with the ANSYS Fluent program, and the pressure values in the sections determined along the spillway were calculated. The results obtained as a result of the studies are listed as follows;

- As a result of the studies, it was observed that the differences between the numerical analysis pressure values for three length sections were not very high.
- When the pressure values for three length sections were compared, it was seen that the pressure values were higher in the 1st longitudinal section, which represents the right chute channel.
- Damage caused by cavitation may occur at the P3 point above the weir threshold where the velocity values are high and the pressure values are low, at the P10 point where the discharge channel slope changes, and at the P14 point at the discharge channel exit.
- Depending on the pressure values obtained from the numerical model, necessary precautions should be taken at risky points to prevent damages that may occur due to cavitation on the spillway.

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

ANSYS. (2015). *FLUENT Theory Guide*. ANSYS Inc.

Aydin, M. C., Isik, E., & Ulu, A. E. (2020). Numerical modeling of spillway aerators in high-head dams. *Applied Water Science*, *10*(1), 42. https://doi.org/10.1007/s13201-019-1126-2

Daneshfaraz, R., Joudi, A. R., Ghahramanzadeh, A., & Ghaderi, A. (2016). Investıgatıon of flow pressure dıstrıbutıon over a stepped spıllway. *Advances and Applications in Fluid Mechanics*, *19*(4), 811–828. https://doi.org/10.17654/FM019040811

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

DSI. (1985). Çatalan dam spillway model experiments report. In *General Directorate of State Hydraulic Works-Hydraulic Model Laboratory*.

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

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

Kumcu, S. Y. (2017). Investigation of flow over spillway modeling and comparison between experimental data and CFD analysis. *KSCE Journal of Civil Engineering*, *21*(3), 994–1003. https://doi.org/10.1007/s12205-016-1257-z

Özdemir, Y. C., Üneş, F., Taşar, B., Varçin, H., & Gemici, E. (2022). 3D Numerical Simulation of Submerged Vane. *International Journal of Environment, Agriculture and Biotechnology*, *7*(6). https://doi.org/10.22161/ijeab.76.21

Taşar, B., Üneş, F., & Gemici, E. (2023). Laboratory and numerical investigation of the 2-array submerged vanes in meandering open channel. *Mathematical Biosciences and Engineering*, *20*(2). https://doi.org/10.3934/mbe.2023153

Taşar, B., Üneş, F., Gemici, E., & Varçin, H. (2021). *Numerical Simulation of Channel Flow Using Submerged Vane in River Arrangements*. 119–130. https://doi.org/10.24193/AWC2021_11

Unes, F. (2008). Analysis of plunging phenomenon in dam reservoirs using three-dimensional density flow simulations. *Canadian Journal of Civil Engineering*, *35*(10), 1138–1151. https://doi.org/10.1139/L08-061

Üneş, F. (2008). Investigation of density flow in dam reservoirs using a three-dimensional mathematical model including Coriolis effect. *Computers & Fluids*, *37*(9), 1170–1192. https://doi.org/10.1016/j.compfluid.2007.11.004

Üneş, F., & Ağıralioğlu, N. (2017). Numerical Investigation of Temporal Variation of Density Flow and Parameters. *Journal of Applied Fluid Mechanics*, *10*(1), 81–94. https://doi.org/10.18869/acadpub.jafm.73.238.25947

Üneş, F., Demirci, M., & Varçin, H. (2016). 3-D Numerical Simulation of a Real Dam Reservoir: Thermal Stratified Flow. *Springer Water*, 377–394. https://doi.org/10.1007/978-981-287-615-7_26/COVER

Unes, F., & Varcin, H. (2017). 3-D real dam reservoır model for seasonal thermal densıty flow. *Environmental Engineering and Management Journal*, *16*(9), 2009–2024. https://doi.org/10.30638/eemj.2017.209

Üneş, F., & Varçin, H. (2015). Investigation of seasonal thermal flow in a real dam reservoir using 3-D numerical modeling. *Journal of Hydrology and Hydromechanics*, *63*(1), 38–46. https://doi.org/10.1515/johh-2015-0007

Varçin, H., Üneş, F., Gemici, E., TAŞAR, B., & Turhan, E. (2023). NUMERICAL ANALYSIS OF THE FLOW OVER THE DAM SPILLWAY. *Air And Water – Components of the Environment*, 121–129. https://doi.org/10.24193/AWC202312

Varçin, H., Üneş, F., Gemici, E., & Zelenakova, M. (2023). Development of a Three-Dimensional CFD Model and OpenCV Code by Comparing with Experimental Data for Spillway Model Studies. *Water*, *15*(4), 756. https://doi.org/10.3390/w15040756