Polytechnic Journal. 2022. 12(2): 47-52
ISSN: 2313-5727
http://journals.epu.edu.iq/index.php/polytechnic

RESEARCH ARTICLE

Performance of the Gomaspan Dam's Stepped Spillway Against Cavitation using CFDcity

Abo A. A.[1]

[1] *Department of Water Resources Engineering, College of Engineering, Salahaddin University - Erbil, Kurdistan Region, Iraq*

***Corresponding author:**
Abdulla Abdulwahid Abo
Department of Water Resources Engineering, *College of Engineering*, Salahaddin University- Erbil, Iraq Kurdistan Region, Iraq

E-mail:
Abdulla.abo@su.edu.krd

Received: 20 Jul. 2022
Accepted: 11 Aug. 2022
Published: 1 February. 2023

DOI
10.25156/ptj.v12n2y2022.pp47-52

ABSTRACT

This study aims to determine how well the stepped spillway at Gomaspan dam performs in terms of cavity index. For this purpose, a computational fluid dynamics (CFD) code called FLUENT is used. To begin, the code was validated by applying it to the experiments work of (Li *et* al. 2021), and the findings demonstrate great agreement between CFD and the aforementioned experimental data. Second, the algorithm is applied to the designed stepped spillway of the Gomaspan dam, and the cavity index is obtained at each step. The results show that the design is unsatisfactory, and the steps are particularly vulnerable to cavitation because the inception point located at step 19 and the velocity of water after inception point ranges between (22m/s-34.6m/s), it may also cause the cavitation after a long period of time.

Keywords: CFD, Renormalized $k-\varepsilon$ turbulent model, stepped spillway, Cavity index, Fluent.

INTRODUCTION

Vapour cavities are formed in a liquid when the pressure drops below the saturated water pressure and the temperature remains constant. Cavitation is the process of producing vapour cavities. Once they've been carried to greater pressures, the vapour soon condenses, the cavities collapse, and they fill with water. In addition to being disruptive to the flow of water, the implosion may cause significant damage if it occurs near the spillway's surface (Novak, *et al.* 2007). The occurrence of cavitation is associated with inadequate design, misalignment, and surface roughness. For example, if the curvature change of the spillway crest is too abrupt, flow separation arises, which might lead to low pressure and cavitation. Based on both factors(flow velocity and cavitation index), the largest damage occurs at end sections of the chute; nevertheless, the cavitation index factor produces superior forecasts of cavitation damage levels and places compared with the observed damage(Kermani, Barani and Ghaeini 2013). Stepped dam spillway releases are efficient in managing the cavitation phenomena, such that by lowering flow rate and raising cavitation index, the avoidance of this phenomenon may be accomplished using this method(Payman, Khosrojerdi, and Shafai 2014). Owing to the rising of the flow depth along the walls, the pressure rises along the chute walls proportionally, the cavitation index rises, and as a result, the least-convergent model has the greatest cavitation risk (Reisi, Salah, and Kavianpour 2015). On stepped spillways, cavitation may occur in nonaerated flows due to the friction coefficient of these uniformly distributed macro-roughness components, and the incipient cavitation index is related to this (K.W. Frizell1 and F.M. Renna, 2011).

the sex-based difference in inflammatory, coagulation, and cardiac biomarkers in COVID-19-positive patients in Erbil city.

COMPUTATIONAL FLUID DYNAMICS

Fluid mechanics has a subfield known as computational fluid dynamics (CFD), where numerical analysis is used with a computer to address problems involving fluid flow, heat and mass transport, and chemical reactions. Engineers have utilized it widely in a wide range of applications since it was first created in the early 1990s (Abo A. A. 2013). Computational fluid dynamics may be used to examine the behavior of laminar, turbulent, reacting, and non-reacting fluids. You can rapidly and economically build, develop, and evaluate results using this tool. This is a crucial advantage to bear in mind. Finite difference and finite volume element are two of the most used CFD methods. Nuclear reactors may also be simulated using CFD. In fluid flow calculations, it's often used since it's based on a particular finite difference. A fluid model's volume is often utilized to establish the position of the free surface in free surface cases (Saleh 2019).

GOVERNING EQUATION

Using a computer, one may solve the Navier-Stokes equation, which is based on the conservation of mass and energy and momentum. Stepped spillways are modeled using computer-aided design (CFD) tools. Fluent software uses the finite volume method to discretize the Navier-Stokes equation (FVM). In CFD, the FVM technique is most often used to discretize dynamic equations. Solving the governing formula using algebraic equations rather than numerical answers is the goal of this strategy. Essentially, the integration equations of the control volume are solved (Raza and Wan 2021).

$$\frac{\partial v}{\partial \chi}+\frac{\partial v}{\partial y}=0 \quad (1)$$

Where:

u= Fluid velocity in x direction in (m/s), v= Fluid velocity in y direction in (m/s)

X-Momentum equation:

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\partial uv)}{\partial y}=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (2)$$

Y-Momentum equation:

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\partial vu)}{\partial x}+\frac{\partial(\partial v^2)}{\partial y}=\rho g_y-\frac{\partial \mathcal{P}}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \quad (3)$$

Where:

ρ=fluid density (kg/m3), μ=fluid viscosity (Pa.s), gx=gravity acceleration in the x-direction (m/s2), gy=gravity acceleration in the y-direction (m/s2),

$\frac{\partial p}{\partial x}$ =pressure gradient in x-direction (Pa/m),

$\frac{\partial p}{\partial y}$ =pressure gradient in y-direction (Pa/m)

The code can capture free surface flow by using the technique of volume of fluid VOF multiphase model was developed by (Hirt and Nichols 1981). When there are several steps in a process, it is most often used. Using this technique, each step is treated as a singular group. This study's research aims to track air-water interaction in free-surface flow. As a result, VOF may be used to pinpoint the exact position of the transition point between phases. In contrast to other multiphase models that just record bubbles, VOF captures the interface as a mixed cell. Other multiphase models' results are insufficient when the interface is taken into consideration when using the VOF approach. In this study, the VOF model is used to accurately forecast the air-water interface. VOF may be defined as 0 for cells that do not contain water, and 1 for cells that are totally filled with water. If the value is between 0 and 1, the cell is a mixture of water and air with a free surface in the middle. The value of each cell's phase value is represented by an. The volume portions of air and water are referred to as a and w, respectively (Raza and Wan 2021).

$$\alpha_a+\alpha_w=1 \quad (4)$$

Where:

α_a: is volume portions of air andα_w: is volume portions of water

A cell's characteristics might indicate either one phase or the other. Any and all variables inside a single cellular unit are shared due to the concept of volume fraction. The cell's total volume fraction is 1 if all of the cell's phases have the same volume percentage. Eq -4 is used to compute the cell density ρ.

$$\rho=\alpha_w\rho_w+\alpha_a\rho_a=\alpha_w\rho_w+(1-\alpha_w)\rho_a \quad (5)$$

Where:

ρ_a: is density of air andρ_w: is density of water

In order to calculate the volume fraction of each phase, Equations (6) & (8) are tracked using the VOF technique.

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho \mu i}{\partial xi}=0 \quad (6)$$

$$\frac{\partial \rho \mu i}{\partial t}+\frac{\partial \rho \mu i \mu j}{\partial xj}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}(\mu+\mu t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

7

$$\frac{\partial \alpha w}{\partial t} + \mu i \frac{\partial \alpha w}{\partial x_i} = 0 \qquad 8$$

Renormalized *k-ε* Turbulent Model

The roughness of a spillway is greatly influenced by the steps. A stepped spillway has a turbulent flow because of its roughness. Increasing the turbulence by bouncing two-phase water over an angled spillway is a possibility. A turbulent model therefore predicts the presence of turbulence. Realistically constrained k- model has been presented for turbulent flow with high Reynolds number. Because of the high turbulent Reynolds number, a new equation for the dissipation rate (ε) has been proposed that provides better results than previous models. Improved k- ε model performance and excellent results for recirculation flow have been achieved. Standard k-ε and shear stress transport k-ε models, as well as $RNG\ k-$ ε and $standard\ k-$ ε RANS models, work well for simulating many types of flows, such as pipe or laminar flow. $k-$ ε is used in this study since existing RANS models for recirculating flow were insufficient and the flow in the step spillway is recirculating. Turbulence dissipation rate (ε) and turbulence kinetic energy (k) transmission are given by Equations (9) and (10), respectively (Raza and Wan 2021).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial xi}(\rho kuj) = \frac{\partial}{\partial xi}\left[\left(\mu + \frac{\mu t}{\sigma k}\right)\frac{\partial k}{\partial xj}\right] + G_k + G_b - \rho\epsilon - \gamma_M + S_k \qquad 9$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial xj}(\rho\varepsilon\mu j) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{\mu t}{\sigma\epsilon}\right)\frac{\partial \varepsilon}{\partial xj}\right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\varepsilon^2}{k+\sqrt{v\epsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S_\varepsilon \qquad 10$$

CODE VALIDATION

In order to cover the features of warping dams in the Loess Plateau area of China, several dam heights of 15 m, 30 m, and 45 m were chosen for the study. This was done to ensure that the results were comprehensive. The dams were examined under a single flow width of (2.5, 4.5, and 5.5) $m^2 . s^{-1}$ for the varied heights (Li *et* al. 2021).

To validate the code one of the models used to check the results. Height 15m and flow $4.5\mathrm{m}^2 . \mathrm{s}^{-1}$ is used.

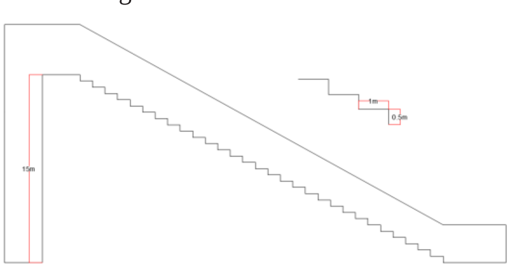


Figure 1: prototype used for code validation, Height 15m and flow $4.5m^2 . s^{-1}$.

According to experimental results the energy dissipation rate for the prototype (H=15m and flow $4.5m^2 . s^{-1}$) is about 46.38%, and the percentage of head loss computed by code is 45.36% see figures 2.

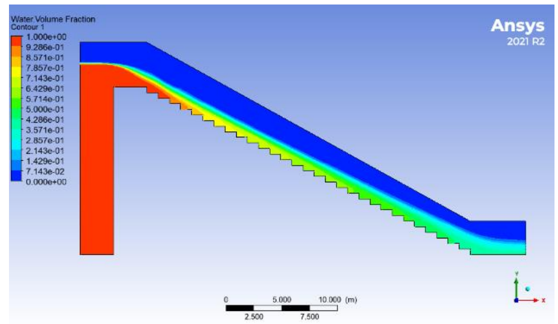


Figure 2: contour of water volume fraction of numerical solution.

The variation between the experimental and code results is 1.02% so there is a good agreement between the results of experimental and the numerical solution.

RESULTS & DISCUSSION

for flow domain creation the ANSYS Workbench is used, which can then be exported as ANSYS-FLUENT code. It was utilized to create the geometry for this study, the spillway has 73 step risers 0.9m and 71 step threads 0.72m with $q=27.044m^2/s$ see figure 3.

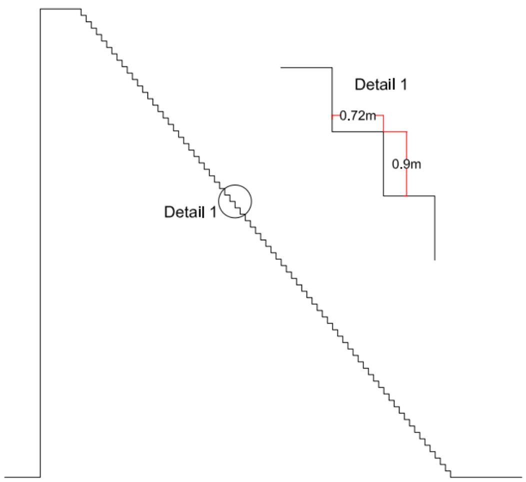


Figure 3: Typical section of Gomaspan dam spillway

The mesh element size 0.1m was used (148829 nodes , 146931 elements), Skewness=0.0055

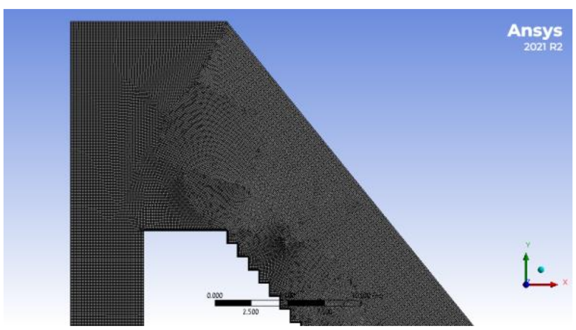


Table 1: Cavitation Damage level (Fadaei, Barani and Ghaeini 2013)

Level	Cavitation damage risk	Interval of velocity m/s	Interval of cavitation index
1	No cavitation damage	V≤5	σ>1
2	Possible cavitation damage	5<V≤16	0.45<σ≤1
3	Cavitation damage	16<V≤25	0.25<σ≤0.45
4	Serious damage	25<V≤40	0.17<σ≤0.25
5	Major damage	V>40	σ≤0.17

Figure 4: Typical mesh with inflation technique

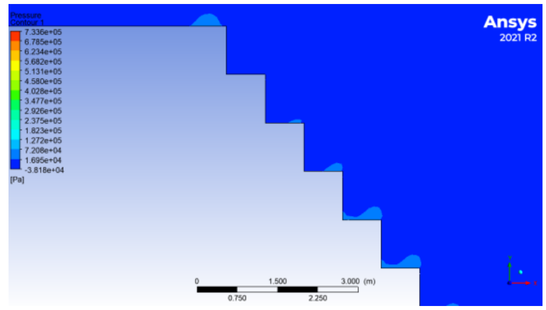


Figure 5: Pressure contour for first steps

Due to the negative pressure on the first step the cavity index has a small value but not produce cavitation because of the low velocity which is lead to low dynamic forces, then the pressure will be positive on the other steps nonlinearly increase and decrease. The velocity increase linearly from the first to last step, it makes serious problem to produce cavitation specifically starts at step 12 to step 19. From step 19 to the last step the water is aerated but still not safe because of the high value of dynamic forces due to high velocity ranges from (22m/s-34.6m/s), it may also cause the cavitation after a long period of time. The inception point was found at the step 19 (21.899m) far from the crest. Table 1 represented the risk cavity damage which is invented by (Fadaei, Barani and Ghaeini 2013)

The cavity index on each step is determined using equation 11. Figure 6 represents the value of cavity index on each step.

$$\sigma = \frac{(P-P_v)}{0.5\rho V^2} \quad (11)$$

Where:-

P: is the pressure of flowing water over the steppes at the particular temperature,

P_v: is the vapor pressure at the specific temperature, ρ: is the density of the water and the average velocity

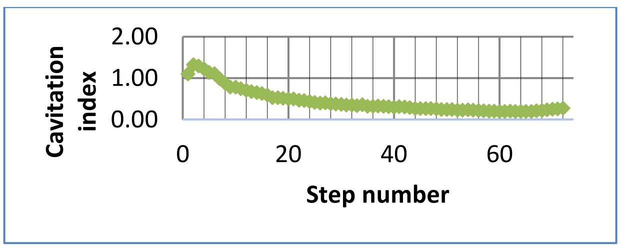


Figure 6: Cavitation index with respect to steps.

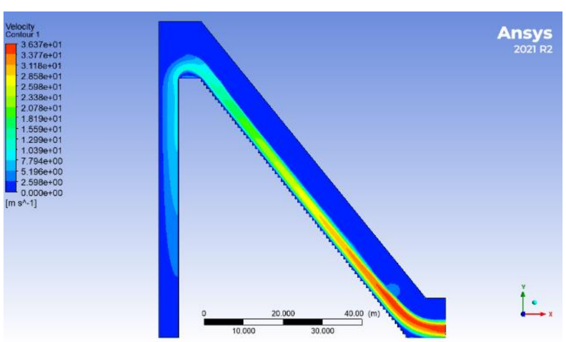


Figure 7: contour of velocity profile.

The maximum negative pressures expected just downstream first step, this is high and hence aeration arrangements are necessary. To control negative pressures and consequently cavitation damage aeration pipe 25 mm diameter at 3 m c/c along the spillway face below gate lip will be provided (R. S. Varshney *et al.* 1979). These pipes will be connected to a bigger size header. While from step 19 to the end cavitation will be occur because for high velocity recommended by (Fadaei, Barani and Ghaeini 2013) see figure 7 as well.

CONCLUSION

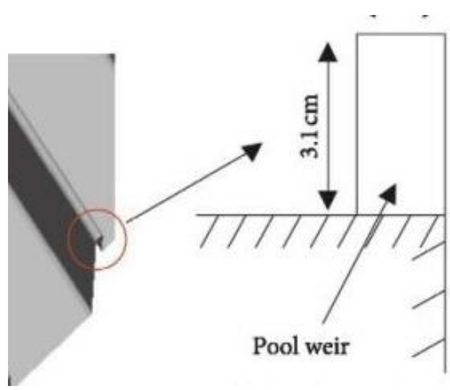
ANSYS-FLUENT code may be used to simulate flow on stepped spillways and examine hydraulic flow, according to the results of the numerical model. The hydraulic characteristics of the Gomaspan spillway are explored deeply in this analysis, including flow velocity, depth, cavitation index, pressure, and distribution pattern. The flow of the spillway was numerically simulated in two dimensions using these initial and boundary conditions. The findings of this study show that depending on the influence of both components (flow velocity and negative pressure) on the cavitation index, there is a significant risk of major cavitation damage in many points of the spillway for a given discharge. Furthermore, when compared to the damage numbers for the Gomaspan dam spillway, the cavitation index factor-based approach delivers improved estimates of cavitation damage levels and cavitation damage levels and locations.

Recommendation

i. Re-design the proposed spillway or add PVC air pipe on the crest of the spillway to be safe against cavitation damage

ii. Step heights should be increased to increase the roughness of the spillway which is lead to increase velocity.

iii. Negative pressures acting on the vertical step surfaces are reduced with the help of the pool weirs, which is beneficial to the reduction in cavitation risk.



ACKNOWLEDGMENTS

As this project has been performed as a sub study in an on-going project at Consultant Company with office in Gomaspan, Erbil, I would like to send my gratitude to consultancy employees Mamosta Shahin Sabir Ahmed, and Mr Abdulla Gardy the contractor of the aforementioned project who have helped and supported me in my work. I would like to thank Pshtiwan Othman MSc student, for helping during study preparation.

REFERENCES

-Abo, A., 2013. *A three-dimensional flow model for different cross-section high-velocity channels* (Doctoral dissertation, University of Plymouth).

-Frizell, K.W. and Renna, F.M., 2011. Laboratory studies on the cavitation potential of stepped spillways. *Apelt, CJ (Editor); Ball, J (Editor)*, pp.2420-2427.

-Frizell, K. H. & Mefford, B. W. (1991) 'Designing spillways to prevent cavitation damage'. Concrete International, 13 (5). pp 58-64.'

-Hirt, C.W. and Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of computational physics*, *39*(1), pp.201-225.

-Kermani, E.F., Barani, G.A. and Ghaeini-Hessaroeyeh, M., 2013. Investigation of cavitation damage levels on spillways. *World Applied Sciences Journal*, *21*(1), pp.73-78.

-Li, G., Zhang, H., Li, X., Guo, L., Gao, Y. and Cai, D., 2021. Numerical Simulation of Stepped Spillways with Front Step Deformation. *Mathematical Problems in Engineering*, *2021*.

-Novak, P., 2007. Moffat. AIB, Nalluri, C. and Narayanan, R., Hydraulic Structures.

-Payman Karami, S., Khosrojerdi, A. and Shafai Bajestan, M., 2014. Numerical Modelling of Hydraulic Flow in Dam Stepped Spillway and Study of Cavitation Phenomenon. *European Online Journal of Natural and Social Sciences: Proceedings*, *3*(3 (s)), pp.pp-283.

-Raza, A., Wan, W. and Mehmood, K., 2021. Stepped Spillway Slope Effect on Air Entrainment and Inception Point Location. *Water*, *13*(10), p.1428.

-Reisi, A., Salah, P. and Kavianpour, M.R., 2015. Impact of chute walls convergence angle on flow characteristics of spillways using numerical modeling. *Int. J. Che, Env & Bio. Sci*, *3*(3), pp.245-251.

-Varshney, R. S., Gupta, S. C., & Gupta, R. L. (1979). Theory and design of irrigation structures. Vol. 2, Canal and storage works. Roorkee: NEM Chand & Bros.

-Saleh, S.M. and Husain, S.M., 2020. Computational Study to Predict the Free-Surface Flow over Non-uniform Stepped Spillway Using ANSYS-CFX. *Polytechnic Journal*, *10*(1), pp.43-50.