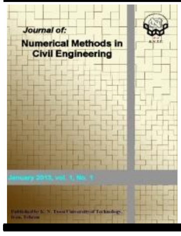

Numerical Methods in Civil Engineering

Numerical Investigation of Cavitation on Spillways. A Case Study: Aydoghmush dam

Mohammad Barzegari[*], Ramtin Sobhkhiz Foumani[**], Mohsen Isari[***], Reza Tarinejad[****], Seyed Amir Alavi[*****]

ARTICLE INFO
Article history:
Received:
May 2019.
Revised:
July 2019.
Accepted:
August 2019.

Keywords:
Cavitation;
Cavitation Index;
FLUENT;
VOF;
Standard k-ε;
Aydoghmush Dam Spillway

Abstract:

Cavitation is among the most complicated and common damages of spillway structures. This phenomenon is controlled by different parameters including the pressure, flow velocity, spillway surface material, operation time, and air flow content. The cavitation index is calculated along the spillway and compared with its critical value using the measured values of the flow's hydraulic parameters. The high cost of experimental models for determining hydraulic parameters, the time required for developing experimental models, and the ever-increasing capabilities of computational fluid mechanics (CFD) models have led to the use of numerical simulation in the flow analyses. The present study employs ANSYS FLUENT to simulate the flow on the spillway of Aydoghmush Dam (Iran), calculate flow parameters, and determine the cavitation index at the flow rates of 35, 800, 1500, and 1850 m^3/s. The standard k-ε equations were applied to model the turbulent flow, while the volume of fluid (VOF) method was employed to determine the flow's free surface profile. The results showed acceptable consistency between the FLUENT and physical model results. It was also found that cavitation did not occur at any of the flow rates.

1. Introduction

Cavitation-induced damages are well-known events in hydraulic structures. They introduced problems complications on the functioning of an American dam drainer in early 1915. However, the first important cavitation-induced spillway failure occurred in 1941, when cavitation was described only as one of the causes of the failure. Today, it is known that cavitation is the main cause of dam failure [1]. At present, investigating cavitation-caused damages and estimating the cost of repairing such damages are essential topics in engineering projects.

**M.Sc. Student of Civil Engineering Department, Ahar Branch, Islamic Azad University, Ahar, Iran.*
***PhD Student of Civil Engineering Department, University of Qom, Qom. Iran.*
**** Corresponding Author: Ph.D. Faculty of Civil Engineering, University of Tabriz, Tabriz, Iran. E-mail: isari.mohsen@tabrizu.ac.ir*
*****Associate Professor, Faculty of Civil Engineering, University of Tabriz, Tabriz, Iran .*
******PhD Student of Civil Engineering Department, Boshehr Branch, Islamic Azad University, Boshehr, Iran.*

The results of some such studies have been implemented in real-life cases. In recent years, due to the invention of advanced and precise computer solutions, numerical methods have been employed to design complicated hydraulic structures.

Several studies have investigated the massive hazards and damages of cavitation to be cognizant of the cause and extensiveness of cavitation. Despite extensive knowledge, even currently designed structures are exposed to frequent cavitation problems [2, 3].

Cavitation-induced damages have been discovered for large spillways across the world, including Russia, Pakistan, Venezuela, and Iran. Cavitation caused damage to a large portion of the spillway of Karun 1 Dam in Iran. Hence, many studies have been undertaken on cavitation for both models of different sizes and real-life cases [4].

The water height at different locations, the pressure applied to the bed, and flow velocity, are among the most important design factors in spillway and chute cavitation analyses. These factors are determined after analyzing the spillway flow. Numerical models provide a suitable opportunity to simulate physical phenomena, which can be extensively applied to ensure the performance of hydraulic structures [5,6]. Iran's water industry requires to consider the consequences of potential failure of important structures, including spillways, based on more accurate numerical models. Accordingly, in the case of Aydoghmush Dam, numerical models are inevitable in evaluating the possibility of cavitation and making a proper estimation of the structure's safety against cavitation.

The present study primarily aims to numerically simulate Aydoghmush Dam's spillway flow by ANSYS FLUENT, determine the pressure on the bed, estimate the spillway flow velocity, and calculate the cavitation index along the spillway. FLENT was employed to calculate the cavitation number (Ca) based on depth, velocity, and pressure.

2. Literature review

Jalal and Mehri [7] studied cavitation in Balaroud Dam's spillway in Iran. They built the spillway's model at a scale of 1:110. Tests were carried out at 14 flow rates for real-life conditions by measuring important flow parameters, including the depth, velocity, and pressure in the middle and lateral axes. The results indicated that an increase in the velocity of the chute reduced the cavitation index. The least cavitation occurred at the beginning of the bucket-type thrower in the middle axis [7].

Cassidy [8] applied a numerical model to determine the two-dimensional pressure on a spillway crest based on the potential flow. The numerical results were almost the same as the experimental results, suggesting the insignificant effect of viscosity on the free surface.

Ikegawa and Washizu [9] and Bett [10] employed the linear finite element method (FEM) to solve the equations governing the flow field. They confirmed the results of Cassidy [8]. However, the convergence rate considerably increased in their numerical analysis.

Li et al. [11] obtained more accurate results by separating the flow field using the FEM and solving the flow field as a potential field in two dimensions. Olsen & Kjellesvig [12] analyzed the spillway flow in two and three dimensions using Reynolds-averaged Navier–Stokes (RANS) equations and standard *k-ε* equations by the finite volume method (FVM). They reported the effect of the diffusion model on the spillway flow rate to be insignificant.

Burgisser and Ruschmann [13] analyzed the normal two-dimensional flow on a spillway crest by the FEM assuming an incompressible turbulent flow. The governing equations included the RANS equations. The numerical results of the spillway flow rate and pressure distribution on the spillway were similar to the experimental results. Song and Zhou [14] proposed a three-dimensional model of a tunnel-shaped spillway's flow. They applied the large eddy simulation (LES) to determine the effects of turbulence and the free flow surface was determined based on the Eulerian-Lagrangian approach. The steady spillway flow was solved by the Bernoulli formula. Subsequently, the problem was analyzed in three dimensions based on the governing equations and LES equations, considering the fixed free flow surface. Finally, the entire field was calculated based on a variable free flow surface. In comparison to the experimental results, the numerical results showed high accuracy.

Unami et al. [15] developed a two-dimensional model of spillway free surface with a triangular irregular grid by combining the FVM and FEM. The temporal algorithm was based on the fourth-order Runge-Kutta method. The numerical results did not show high accuracy in comparison to the experimental results because the flow was treated as sheet flow.

Tufi [16] analyzed the two-dimensional spillway crest flow by assuming potential flow and applying the Neumann condition to the flow field's boundaries using the finite difference method (FDM). In comparison to the experimental results, the numerical results suggested an insignificant effect of viscosity on the flow field. The numerical free surface was also compared to the experimental results, showing reliable accuracy for the viscosity method.

Bruce and Michael. [17] physically and numerically modeled the standard ogee spillway flow. The two-dimensional turbulent flow results obtained from the standard *k-ε* equations by the FVM in FLOW 3D were very close to the experimental results.

Barani and Bahrami [18] simulated the flow on the chute and determined air concentration on the chute bottom to investigate the possibility of cavitation on the chutes and spillways in FLOW 3D. Comparing the numerical results with the experimental results revealed that the initial concentration, the Froude number, and the chute slope had large effects on the underlying air's concentration in the range of $C_{b,min} \leq C_{b,det} \leq C_{bo}$ and declined it exponentially.

Mirbagheri and Mansouri [19] developed a mathematical model to study the flow field and cavitation on a dam's spillway. They obtained the Navier-Stokes equations to determine the pressure field, flow velocity, and the free surface water profile based on viscosity. The convergence factor was applied to the all locations in the three-dimensional system. Cavitation was addressed after calculating the pressure and velocity.

Hoseynzadeh et al. [20] evaluated cavitation for the chute of Garmichay Dam (Iran) using the HEC-RAS model. They determined flow parameters including the depth, velocity, and pressure along the chute path at different flow rates. After obtaining the velocity and depth in different sections of the chute, the cavitation index was calculated for the sections, followed by discussions regarding the possibility of cavitation along the chute at the flow rates. It was concluded that the minimum cavitation index for the three flow rates was 0.55. In other words, it was concluded that cavitation would be non-problematic along the spillway, and the channel could be protected from cavitation by surface finishing.

Khorshidi et al. [21] studied cavitation in the underpass drainer channel of Sefidrood Dam, Iran, in FLUENT. They modeled the two-phase flow of water and air using the fluid volume model. The pressure and velocity were determined along the channel for estimating the cavitation size. They concluded that the finite volume model of FLUENT could accurately model the flow in underpass drainer channels.

Kavianpour et al. [22] numerically analyzed aeration flow in the downstream of the aerators. They employed the k-ε model to simulate flow turbulence. Comparing the experimental and numerical results showed acceptable consistency.

Shamsaei and Mohammad [23] numerically studied the effect of the step geometry on the pressure loss for stepped spillways in FLUENT to predict the major flow properties, including the free surface, velocity, step pressure, and energy loss. They observed the numerical results to follow the spillway pressure variation. The results demonstrated that the minimum pressure occurred on the vertical surfaces and near the step's upstream edge.

Dargahi [24] experimentally and numerically investigated the three-dimensional spillway flow for evaluating the flow field on the spillway. They properly simulated the upstream water surface with the numerical model and obtained acceptable velocity and pressure results. The maximum relative error occurred at $X_n = 2.2$ at all three heads.

Shafaei-Bajestani and Arianfar [25] numerically studied the flow pattern around the piers of a cylindrical bridge in FLUENT. The numerical FLUENT model was validated by comparing the experimental results and other works studies. The results indicated that the k-ε (RNG) and Reynolds stress (RSM) models produced closer results than other models to the experimental results. Also, the k-ε (RNG) model was more accurate than the RSM model in calculating the flow parameters, and reached convergence in a shorter time.

Cheng et al. [26] simulated the two-phase water-air flow on stepped spillways in FLUENT. For a mixed flow in the stepped spillway, they obtained successful numerical results including reactions between the air bubbles, the re-flowing of the bubbles in the flow regime, and the velocity distribution and pressure profile on the surfaces of the steps.

3. Materials and methods

Initial information on the subject is required for any study. The present study investigates the flood drain system of Aydoghmush Dam. The solving strategy adopted for the numerical model was using the FLUENT to obtain the hydraulic parameters of the spillway.

3.1. The basic information of Aydoghmush Dam

Aydoghmush Storage Dam is located on the main branch of Aydoghmush River, southwest of Mianeh County, North Azarbaijan Province, Iran. It was constructed to supply water for more than 14,000 hectares of farmlands. Aydoghmush River covers an area of over 1800 km^2. It originates from Belgheys, Gharedash, and Ghareaghach Mountains. It passes through 120 km of the path, joins Ghezel Uzun River along with Gharangho and Shahchay Rivers, and flows toward the Caspian Sea [27]. Aydoghmush Basin has a semi-arid climate with annual precipitation of 410 mm. Its runoff is approximately 210 million cubic meters. Due to the non-coordination between the river and farm requirements, only about 1,000 hectares of the region's farmlands (12% of the entire cultivable lands) are equipped with irrigation farm facilities. Studies on the agricultural situation of the region showed the necessity of dam construction for water storage. In 1992, Aydoghmush Dam was planned to be constructed in the west of Mianeh County to store water for irrigating 11,100 hectares of farmlands. Tables 1 and 2 provide the technical specifications of Aydoghmush Dam [27].

3.2. Physical spillway model

Since spillway of Aydoghmush Dam perfectly matches Sahand Dam Spillway, the present study was conducted based on the data of Sahand Dam Spillway obtained from Plexiglas at a scale of 1:40 in the Iran Water Research Center to validate the FLUENT results. The static pressure was measured by 11 piezometers installed in the spillway's crest. The concentration of piezometers was larger in the curvatures of the spillway and locations with higher slope gradients. The velocity was measured as the mean velocity in half depth of the flow [27].

Table 1. The general specifications of Aydoghmush Dam's Spillway [27]

Spillway Type	**Free Spillway**
Threshold	Concrete, Ogee-shaped
Crest Length	65 m
Weir crest elevation	1341.5 m above the sea level
Chute Length	142.8 m
Designed Capacity	2730 m^3/s
Energy Damping System	Throwing Bucket

Table 2. The specifications of the reservoir and body [27]

Dam Type	**Earth Dam with a Middle Aquiclude Clay Core**
Crest Elevation	1350 m above the sea level
River Bed Elevation	1283 m above the sea level
Foundation Elevation	1268 m above the sea level
Dam elevation from the river bottom	67 m
Maximum dam elevation from the bedrock	82.1 m

4. Analyzing the results

This section investigates the cavitation potential of Aydoghmush Dam's spillway by calculating the flow height, velocity, hydraulic pressure, and cavitation index. The numerical results of the flow rates of 800, 1500, and 1850 m^3/s are compared with the results obtained from the physical model at a scale of 1:40 to validate the FLUENT results. Finally, it is discussed whether cavitation occurs based on the calculated cavitation index. Since the surplus water of the dam overflew last year and the dam experienced a maximum overflow rate of 35 m^3/s, the present study simulated the flow rate of 35 m^3/s and validated the results according to the observation data.

4.1. Geometrical model and producing a suitable grid

The dam's spillway was modeled in the real dimensions in GAMBIT. The coordinates of the ogee spillway were drawn in two dimensions. Joining the points produced lines and joining the lines formed surfaces. Once the geometric design was completed, a suitable grid was applied to the geometric model. The grid-type was defined based on the problem and flow analysis. In CFD problems, the type of the grid and the number and shapes of elements in different locations play key roles in dam parts. Moreover, they provide high accuracy to minimize errors and save memory to achieve convergence in a shorter time.

Here, a regular grid and pave tetragonal elements were chosen due to the complicated geometry of the problem. Moreover, irregular and triangular elements were discarded to avoid large number of meshes and thus, long computation times. Two separates were considered for the beginning of the spillway; the lower part functioned as the water inlet, while the upper part served as the air inlet.

Considering the nature of the problem, the boundary conditions were chosen as follows (Fig. 1).

The water velocity in the spillway's inlet was defined as the velocity input as it was known. For the depth of the inlet water, a suitable height was separated based on the water depth in Aydoghmush Dam's design and the remaining height was selected for the air.

The velocity of the in-water phase and in-air phase were considered.

The rigid boundary of the spillway was determined to be a wall.

Since the results will not be altered if the wall boundary is defined, the upper boundary was considered as the pressure outlet and the outlet boundary was treated as the atmospheric air outlet pressure.

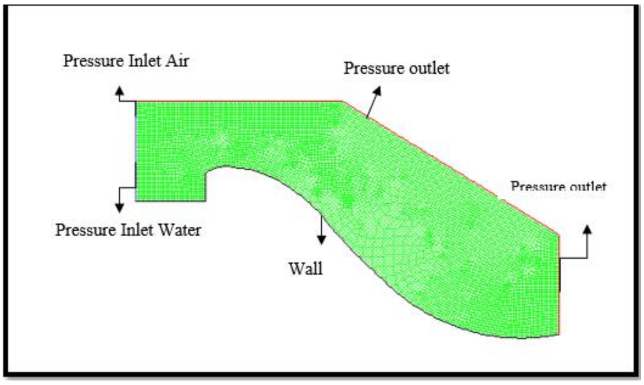

Fig. 1: The boundary conditions

A) Numerical accuracy

Improved upwind indicates lower software error in rounding the results of solving equations. The problem was solved with the following two accuracy degrees:

1. First-order upwind: It is used only to achieve convergence. The software results are not provided under first-order upwind due to their low value.
2. Second-order upwind: This separation method is more accurate than the previous one and was used to achieve final results with high accuracy. The entire results provided in this work were analyzed by the second-order upwind.

B) Geometric accuracy

The geometric accuracy is improved by making meshes finer. Three accuracy degrees were employed as follows:

1. Initial accuracy: It is the initial meshing in GAMBIT. The total number of meshes was 6,395 in the initial accuracy.
2. Secondary accuracy: To achieve higher accuracy, after reaching convergence in solutions at the initial accuracy, each mesh was divided into four smaller meshes. The total number of meshes came up to 26,039.
3. Final accuracy: To achieve the highest accuracy after reaching convergence in the solutions at the secondary accuracy, the meshes became smaller to reach a total number of 31,892.

4.3. Input velocity

The spillway was investigated for the flow rates of 800, 1500, and 1850 m^3/s, providing the cavitation index for the mentioned flow rates. The input velocity was selected to be 2.30, 3.44, and 4.59 m/s for the flow rates of 800, 1500, and 1850 m^3/s, respectively. The turbulence parameter (the turbulence parameter is the ratio of the desired point velocity to the average velocity in terms of Reynolds number, which does not have a unit) was calculated to be 2.02, 1.93, and 1.89 for the flow rates of 800, 1500, and 1850 m^3/s. The turbulence of the flow was small at a turbulence intensity of below 1%, while it was very high at a turbulence intensity of above 10%.

The volume of fluid (VOF) varied between 0 and 1. In the flow inlet, VOF was considered to be 1, indicating that the entire cell volume is filled up with the fluid. A VOF of zero implies that the entire cell volume is empty. The air inlet was specified with a wall boundary condition, and its parameters were set as default, except for VOF. VOF was treated to be 0 in the air inlet. Also, the upper surface of the walls was considered to be air, and the input pressure was set to be its boundary condition, where VOF was set to 0. In the outlet, the pressure outlet condition was applied with the default settings.

4.4. Measuring the water depth in the numerical model

4.1.1. Comparing the spillway water surface level profile to the observed data at a flow rate of 35 m^3/s

The location with a VOF of 0.5 was the free flow surface. Interpolation between points was used for heights with a VOF of below 0.5 to find the height at which the VOF was 0.5. The heights of lines are represented as graphs in FLUENT. Fig. 2 demonstrates diagrams to compare the physical and numerical results. Acceptable consistency can be seen between the numerical and observation results.

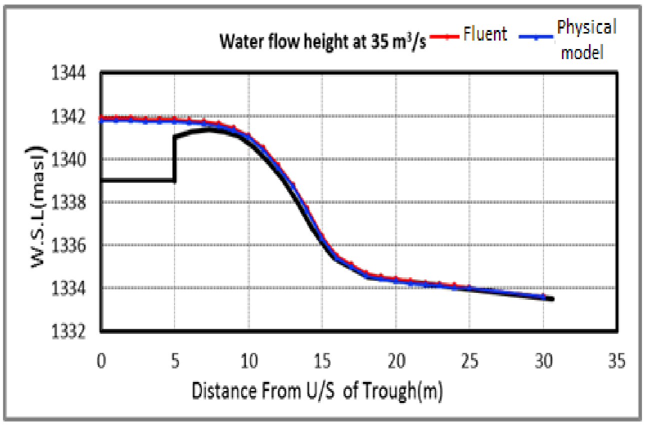

Fig. 2: A comparison of the numerical spillway water surface level profile and observation data at a flow rate of 35 m^3/s

4.4.2. Comparing the spillway water surface level profile to the observed data at a flow rate of 800 m^3/s

Fig. 3 shows the water surface level (WSL) values at a flow rate of 800 m^3/s. As can be seen, there was acceptable consistency between the numerical results and observation data, particularly at the end spillway stations. Moreover, the software was able to predict the water surface with satisfying accuracy.

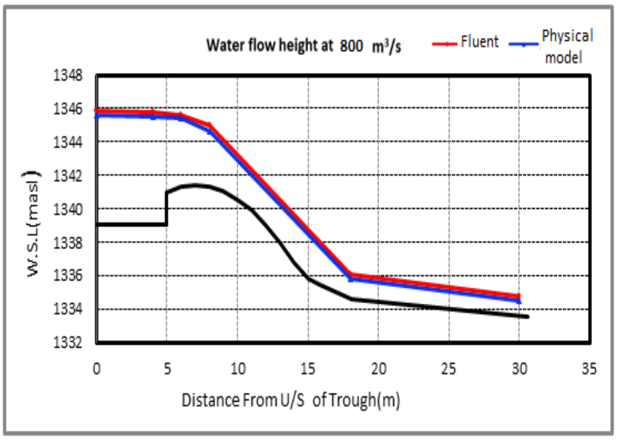

Fig. 3: A comparison of the numerical spillway water surface level profile and observation data at a flow rate of 800 m^3/s

4.4.3 Comparing the spillway water surface level profile to the observed data at the flow rates of 1500 and 1850 m^3/s

Figs. 4 and 5 depict WSL values at the flow rates of 1500 and 1850 m^3/s, respectively. Thereby, good agreement exists between the numerical results and observation data, particularly at end stations, and the software was able to predict the WSL with reliable accuracy.

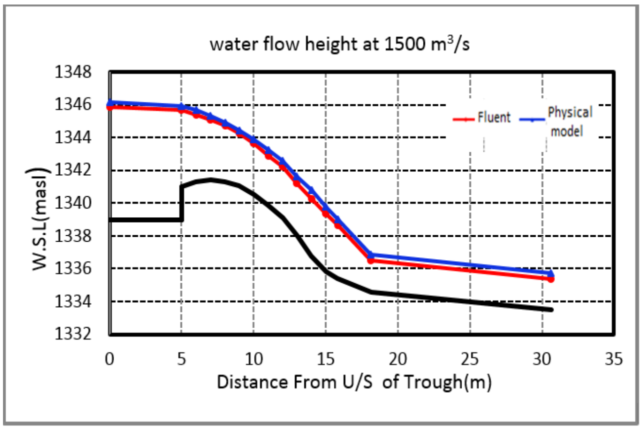

Fig. 4: A comparison of the numerical spillway water surface level profile and observation data at a flow rate of 1500 m^3/s

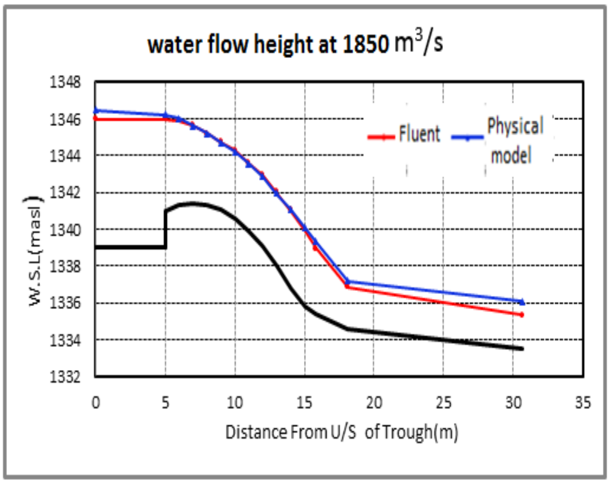

Fig. 5: A comparison of the numerical spillway water surface level profile and observation data at a flow rate of 1850 m^3/s

4.5. Measuring the numerical flow velocity at the four flow rates

4.5.1. Measuring the numerical flow velocity at 35 m^3/s

The water flow velocity can be calculated in different locations of the drawn lines. Since the velocity was measured at the half flow height in the report, the corresponding velocity was measured at the half flow depth as the flow velocity in the numerical model after the spillway flow was obtained. The velocity was calculated on the spillway in the observation report. However, since FLUENT was able to calculate the velocity in different locations, it was acquired separately in different distances. Fig. 6 illustrates the diagram of numerical velocity on the spillway at 35 m^3/s.

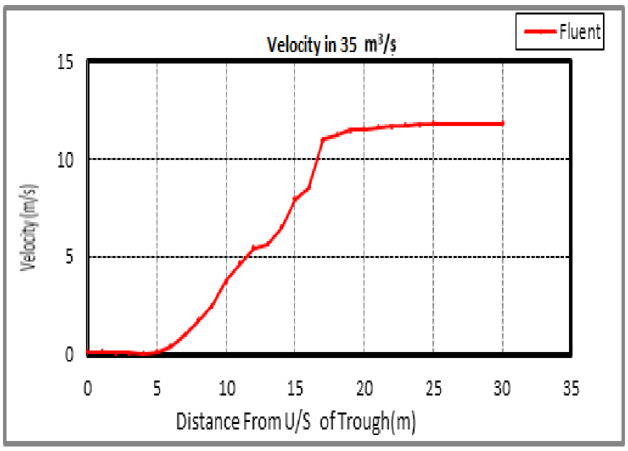

Fig. 6: The numerical diagram of the velocity on the spillway at *35 m^3/s*

4.5.2. Comparing the numerical velocity results to the observation data at 800 m^3/s

Six stations were embedded in the model to record the flow velocity. A comparison of the numerical and physical results showed that there was no significant consistency between the results at stations near the spillway crest. This could be attributed to water velocity measurement errors or inaccurate observation results. Moreover, the observation results provided the velocity only at the half depth, while the mean velocity had to be reported at a depth of 0.6, for FLUENT to be able to calculate the velocity in every location.

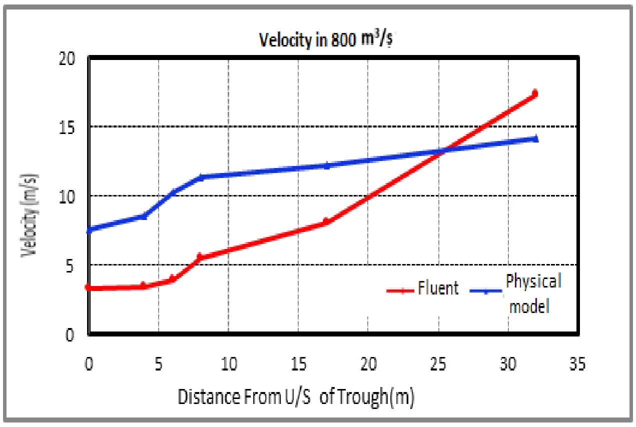

Fig. 7: Comparing the numerical velocity results to the physical data

4.5.3. Comparing the numerical velocity results to the observation data at 1500 and 1850 m^3/s

Figs. 8 and 9 compare the numerical velocity results to the observation data at 1500 and 1850 m^3/s, respectively. As a result, there was acceptable agreement between the numerical and observation results in the selected locations.

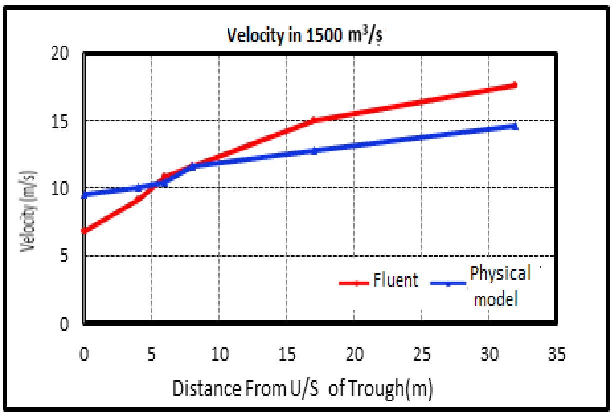

Fig. 8: A comparison of the numerical velocity results to the physical model data at 1500 m^3/s

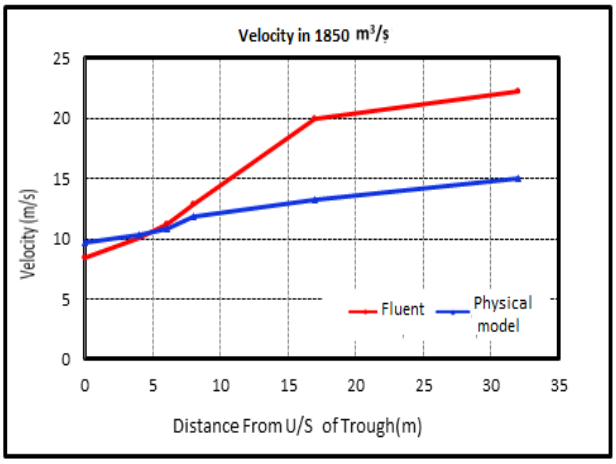

Fig. 9: A comparison of the numerical velocity results to the physical model data at 1850 m^3/s

4.6. Measuring the numerical static pressure at the four flow rates

It is essential to have sufficient information on the spillway pressure to investigate the possibility of cavitation. FLUENT calculates the static pressure as a pressure curve in different locations of the drawn lines. Here, the static pressure near the flow bed is of high significance. FLUENT provides the pressure in kPa. To compare the numerical and physical static pressure results, we should compare the unit of pressure results from Pa into m of water height. Figs. 10-13 compare the numerical and physical static pressure results at the flow rates of 35, 800, 1500, and 1850 m^3/s, respectively.

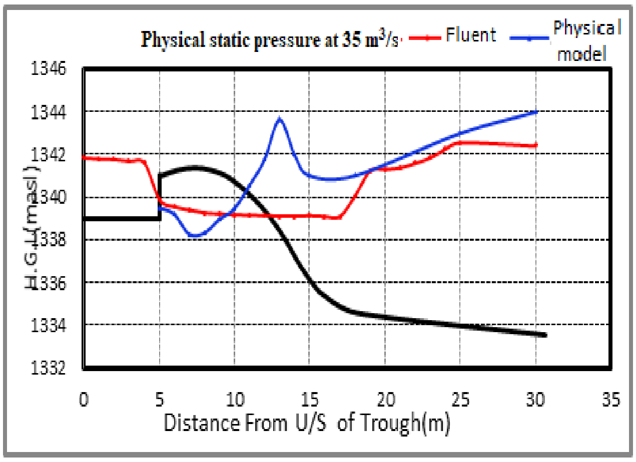

Fig. 10: A comparison of the numerical and physical static pressure results at 35 m^3/s

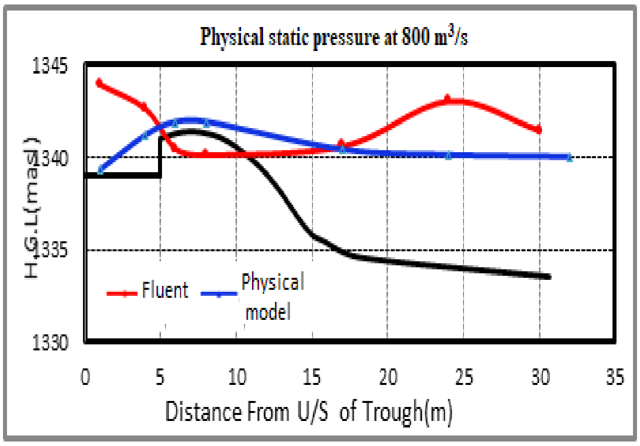

Fig. 11: A comparison of the numerical and physical static pressure results at 800 m^3/s

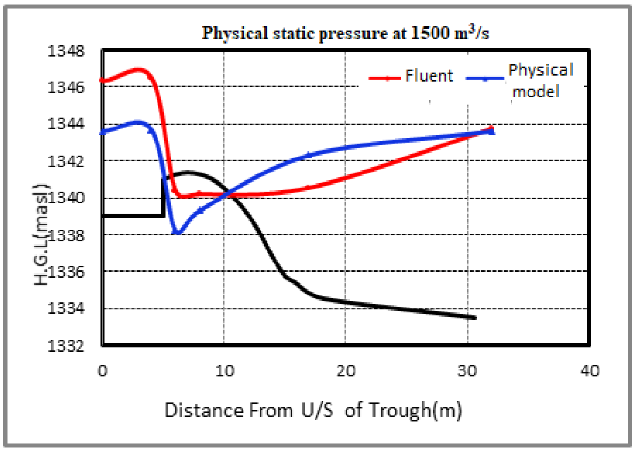

Fig. 12: A comparison of the numerical and physical static pressure results at 1500 m^3/s

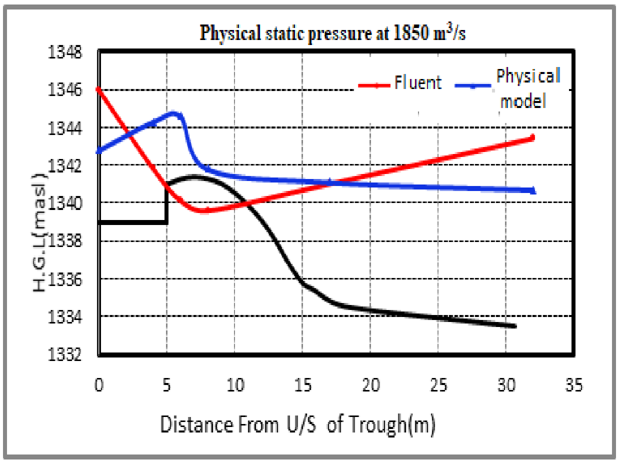

Fig. 13: A comparison of the numerical and physical static pressure results at 1850 m^3/s

4.7. Calculating the cavitation index

4.7.1. Calculating the cavitation index at 35 m^3/s

In the present study, the cavitation index was calculated as

$$\sigma = \frac{\left(\frac{p}{\gamma} + \frac{p_{atm}}{\gamma} - \frac{P_v}{\gamma}\right)}{\left(\frac{V_0^{\,2}}{2g}\right)} \qquad (1)$$

where the vapor pressure of water was considered as 0.236 m of water height at 20°C, while the atmospheric pressure was treated as 10.3 m of water height. Fig. 14 depicts the cavitation index values at 35 m^3/s. Accordingly, the numerical cavitation index of Aydoghmush Dam's spillway was predicted to be above 0.2, considering the very small flow velocity.

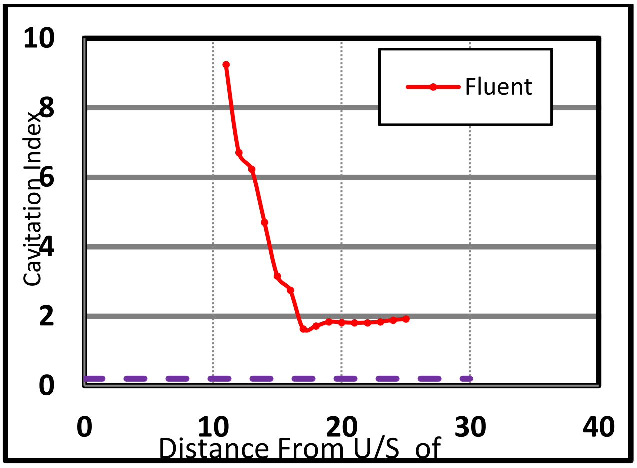

Fig. 14: A comparison of the cavitation index results at 35 m^3/s

4.7.2. Calculating the cavitation index at 800, 1500, 1850 m^3/s

The cavitation index was calculated using the height, velocity, and static pressure results. Figs. 15-17 compare the numerical and observation cavitation index results at the flow rates of 800, 1500, and 1850 m^3/s, respectively.

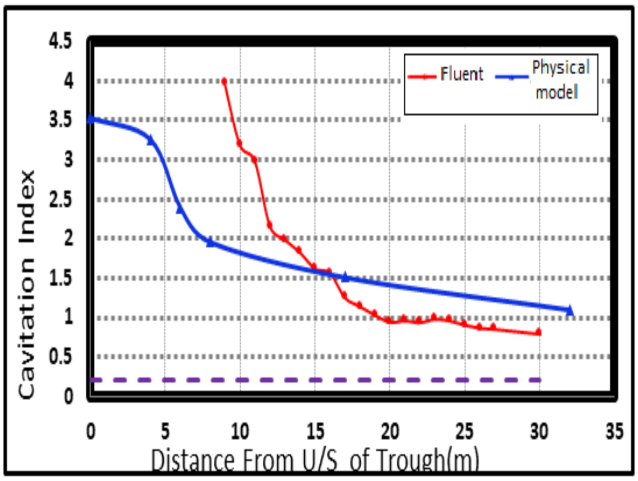

Fig. 15: A comparison of the numerical and observation cavitation index results at 800 m^3/s

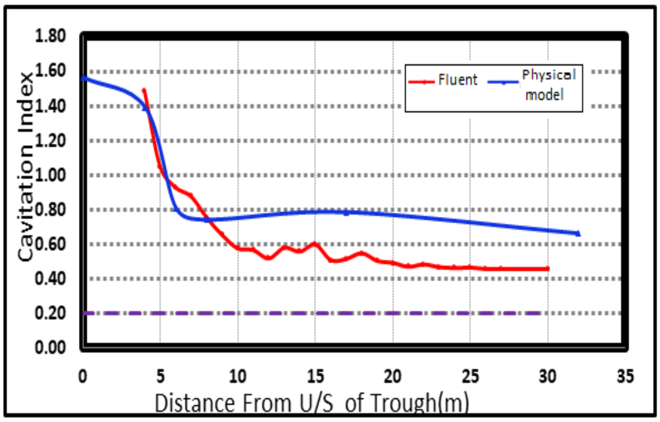

Fig. 16: A comparison of the numerical and observation cavitation index results at 1500 m^3/s

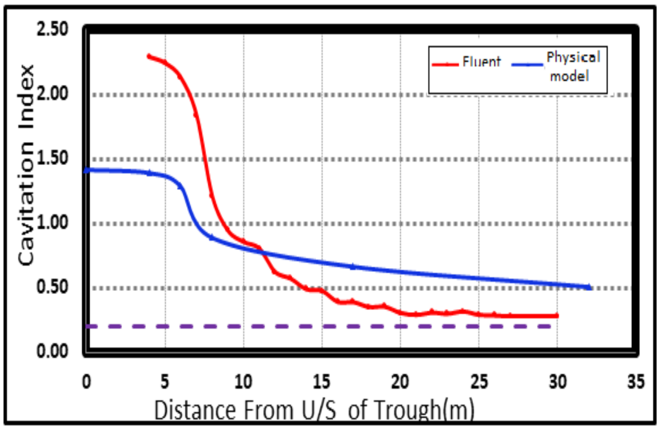

Fig. 17: A comparison of the numerical and observation cavitation index results at 1850 m^3/s

5. Conclusion

The comparison of the numerical and physical models indicated that the flow depth and static pressure errors were small and negligible. The investigation of the cavitation index diagrams demonstrated that the lowest cavitation was estimated to be 0.28 at a flow rate of 1850 m^3/s. It can be stated that cavitation is less likely in Aydoghmush Dam's spillway. Unlike experimental methods, numerical methods can model complicated geometries with different boundary conditions at a lower cost. Although reducing the size of meshes produces more accurate results, it considerably prolongs the analysis time. Thus, it is required to select a suitable element size in numerical analyses by accepting a specific approximation. To achieve a more realistic result, the numerical and physical results are compared to modify assumptions to match the theoretical and practical results if needed. The use of numerical models instead of experimental models saves cost and time. The FVM FLUENT can serve as a suitable means of modeling spillway flow. This is confirmed by the results of the present work. As predicted, despite differences, the numerical and experimental results were consistent. Furthermore, the data of the proposed model can be employed along with experimental results to control cavitation damage, as well as analyzing and designing the spillway flow.

References

[1] Colgate, Donald.,(1977) .Cavitation Damage in Hydraulic Structures. Wear of Materials, American Society of Mechanical Engineers, New York.

[2] Parsaie, A., Dehdar-Behbahani, S. & Haghiabi, A.H. (2016). Numerical modeling of cavitation on spillway's flip bucket. *Front. Struct. Civ. Eng.* 10, 438–444. https://doi.org/10.1007/s11709-016-0337-y

[3] Dehdar-Behbahani, S., & Parsaie, A. (2016). Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran. *Alexandria Engineering Journal*, *55*(1), 467–473. https://doi.org/10.1016/j.aej.2016.01.006.

[4] Zandi, Y. , (2005). Cavitation in overflows, Tabriz Islamic Azad University Press.

[5] Parsaie, A., Haghiabi, A.H. & Moradinejad, .A (2015). CFD modeling of flow pattern in spillway's approach channel. *Sustain. Water Resour. Manag.* 1, 245–251. https://doi.org/10.1007/s40899-015-0020-9.

[6] Parsaie, A., & Haghiabi, A. (2015). The Effect of Predicting Discharge Coefficient by Neural Network on Increasing the Numerical Modeling Accuracy of Flow Over Side Weir. *Water resources management, 29*, 973-985. doi: 10.1007/s11269-014-0827-4

[7] Ibn Jalal, R. And Mehri, M., (2008). Evaluation of vacuum over Ballaroud Dam overflow using physical model",4th National Congress of Civil Engineering, University of Tehran,.

[8] Cassidy J J (1965). Irrotational flow over spillways of finite height", J. Eng. Mech. Div., ASCE91, , (6):155–173.

[9] Ikegawa M, Washizu K.(1973). Finite element method applied to analysis of flow over a spillway crest",J. Numer. Methods Eng.,6: 179–189.

[10] Betts, P. (1979).A variational principle in terms of stream function for free-surface flows and its application to the finite element method. Computers & Fluids - COMPUT FLUIDS, 7. 145-153. 10.1016/0045-7930(79)90030-6.

[11] Li W, Xie Q, Chen C J (1989). Finite analytical solution of flow over spillway.J. Eng. Mech. ASCE115: 2635–2647.

[12] Olsen, N & Kjellesvig, H,(1998). Three-Dimensional Numerical Flow Modelling for Estimation of Spillway Capacity. Journal of Hydraulic Research. 36. 775-784. 10.1080/00221689809498602.

[13] Burgisser M F, Rutschmann P (1999). Numerical solution of viscous 2-D vertical free surface flows:Flow over spillway crests.Proc. 28th IAHR Congress, Technical University, Graz, Austria.

[14] Song, C & Zhou, F (1999).Simulation of Free Surface Flow over Spillway. Journal of Hydraulic Engineering-asce - J HYDRAUL ENG-ASCE. 125. 10.1061/(ASCE)0733-9429(1999)125:9(959).

[15] Unami K, Kawachi T, Munir Baber M, Itagaki H (1999).Two dimensional numerical model of spillwayflow.J. Hydrol. Eng. ASCE125: 369–375.

[16] Tufi M A (2001).Solution for spillway flow by finite difference method. J.Hydr.Res.vol.39 (3),pp:241-247.

[17] Bruce M S, Michael C J (2001). Flow over Ogee spillway; physical and numerical model case study.J. Hydrol. Eng. ASCE127: 640–649.

[18] Barani,GH. And Bahrami, A. (2008). Investigation of effective factors in aeration and role of aeration in preventing cavitation in dam overflow, 3rd Iranian Water Resources Management Conference, Tabriz University.

[19] Mirbagheri, S.A., Mansouri, M. N. (2005). *Solution of Flow Field Equation & Verification of Cavitation Problem On Spillway Of The Dam*, International Journal of Engineering.

[20] Hosseinzadeh, A., Azar partovi, Sh. And Farsadizadeh, d. , (2008). Flow analysis and cavitation analysis in the overflow firing of the Grammys dam using the HEC-RAS model, 3rd Iranian Water Resources Management Conference, Tabriz University.

[21] Khorshidi, h. Taleb bidokht,N. Nikseresht, A. H. (2008). Investigation of cavitation phenomena in inferior dischargers of Sefidrood dam with Fluent, 4th National Congress of Civil Engineering, University of Tehran.

[22] Kavianpour, M.E. Hajikandi,H. , Pirouz,B. (2008). Numerical Solution of Aeration Flow Downstream of Aerators in Lower Sefidroud Dam Dischargers, 3rd Iranian Water Resources Management Conference, Tabriz University.

[23] Shamsaei, A. And Mohammadi, AS. (2008). Study of the effect of step geometry on energy loss in stepped weirs by numerical simulation, 3rd Iranian Water Resources Management Conference, Tabriz University.

[24] Dargahi, B. (2006).*Experimental Stady & 3D Numerical Simulation For A Free-Overflowe Spillway*, J. Hyd. Eng., ASCE.

[25] Shafaei Bejestan, M.S. And Arianfar, AS. (2008). Numerical study of flow pattern around cylindrical bridge foundations by Fluent, 4th National Congress of Civil Engineering, University of Tehran.

[26] Cheng, Xiangju & Chen, Yongcan & Luo, Lin. (2006). Numerical simulation of air-water two-phase flow over stepped spillways. Science in China Series E: Technological Sciences. 49. 674-684. 10.1007/s10288-006-2029-2.

[27] Bandab Consulting Engineers (1996). Landscape of Aidoghmush Dam Project, East Azerbaijan Regional Water Organization Technical Archive.