ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

Cavitation Investigation in the Sluice Section of the Sefid Rood Dam Using Flow 3D Software for Environmental Purposes

[1]Sina Tarighi; [2]Parisa Ghasemzadeh; [3]Behnam Jabbari kalkhoran
[1]Department of Civil, Chemical and Environmental Engineering, University of Genova, Italy
[2]Department of Civil, Chemical and Environmental Engineering, University of Genova, Italy
[3]C3A - Center Agriculture Food Environment, University of Trento, Italy

Abstract:- It is evident that water resources are essential for the existence of living organisms, particularly human life. Outlets are a series of structures employed to transfer water from the dam reservoir to the discharge point downstream. Due to the significance of this section of the dam, the performance analysis of the outlet, including the channel, gates, and their outlet, is sensitive. The presence of pressurized flow in the upstream of the outlet gate, energy dissipation due to various factors, and the very low values concerning the gate opening compared to the water head over the outlet gate cause significant errors in determining various parameters related to the outlets. This includes pressure drops across the gates and their discharge capacities when using theoretical methods.

This research aims to investigate pressure distribution at various points along the outlet channel, determine the gate discharge capacity, and calculate its discharge coefficient. It explores the possibility of cavitation occurrence, compares the presented scenarios for post-service and emergency gate operations in the simultaneous operation of two gates, and determines the main loss coefficients in the channel, including frictional losses, conversion losses, and gate losses. This investigation utilizes data obtained from the physical model of the spillway outlet constructed at the Soil and Watershed Conservation Research Center laboratory. The physical model includes the channel and gates (service and emergency), and necessary experiments were conducted. The pressure values at different points, gate discharge rates at three opening levels (60%, 80%, and 100%), were measured in the reservoir, and the results are presented in corresponding tables and graphs.

Additionally, the Flow 3D software was employed to numerically model the outlet discharge under three gate openings (60%, 80%, and 100%) for comparison between experimental and numerical results and with previous findings in this research. Subsequently, it will be demonstrated that, under single-gate operation and simultaneous operation, the cavitation index in critical areas, such as gate slots and between gates, in the single-gate mode falls within an acceptable range, practically eliminating the risk of cavitation. However, in simultaneous operation mode, negative pressures occur in some gate openings, posing the possibility of cavitation occurrence.

Keywords:- *Cavitation, Gate Slot, Hydraulic Structures, Physical Model.*

I. INTRODUCTION

For years, human water consumption has commenced in various sectors, including agriculture, drinking, industry, and power generation. Water is, indeed, a crucial factor for the progress of different societies worldwide, and its proper transfer is pivotal both technically and economically[1]. The world is seeing more and more areas struggling with water shortages. Because of this, traditional methods of managing water supplies just aren't enough anymore. Water reallocation is a new approach that allows for more flexibility in how water is distributed. This could be helpful in dealing with water scarcity as situations change due to things like climate, economics, and the environment [2].

In cases where the reservoir depth of a dam is substantial, submersible outlets are used for downstream water usage, emergency dam discharge, and, in some instances, sediment discharge accumulated in the dam reservoir[3]. Common components of outlets include the water intake channel or intake structure, water conveyance channel, spillway or tunnel, downstream gate chamber, chute or spillway, energy dissipation, and outlet channel. An outlet may comprise all or some of these components. [1].

Outlets are often employed for diverting water during construction, and if highly reliable, they may also assist in flood discharge schemes. In arch or relatively small gravity dams, outlets are smaller, but in large earth dams, a large outlet is usually divided into two pressured sections (controlled by one gate with high head) and an outlet tunnel (transferring subcritical flow to the atmosphere)[13].

In submersible outlets, two-phase flow of water and air occurs at high velocities. The sudden separation and transformation of the flow from the pressurized state to the free state result in a significant drop in downstream pressures.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

Minimizing negative pressures generated downstream of the gate is crucial. These negative pressures can lead to damage to the downstream structure and the gate itself. Cavitation, one of the most undesirable hydrodynamic phenomena, occurs due to the negative pressures created downstream of the gate. Usually, air injection through an air duct is used to control this phenomenon [2].

Fundamentally, One significant challenge in hydraulic and hydropower projects is cavitation. This phenomenon arises when the pressure within a liquid falls below its vapor pressure, leading to the formation of vapor bubbles [14]. These pressure fluctuations are common in internal flows within pumps, venturis, valves, and hydraulic turbines. Cavitation is characterized by the repeated creation and forceful collapse of these vapor bubbles, which can ultimately damage the components [15][16].

The reduction in local pressure in fluid flow can be due to factors such as a decrease in total energy, an increase in local velocity, vortex formation, or excessive flow separation. The water flowing through hydraulic structures contains air bubbles of various sizes and different impurities. These conditions are necessary for initiating cavitation, determining potential damage, and generating noise.

With the continuous decrease in pressure in fluid flow and the increase in velocity, critical conditions are reached, initiating cavitation. This threshold of cavitation is referred to as the cavitation threshold. According to the above definition, the following can be highlighted regarding cavitation:

- Cavitation is a fluid process and does not occur in the solid or gas phases under normal conditions.
- Cavitation results from a reduction in fluid pressure, and therefore, controlling the amount of this reduction or, in other words, minimizing the absolute pressure is crucial.
- Cavitation is a dynamic process that examines the growth and explosion of cavitation bubbles.
- There is no indicator showing that the fluid is static or in motion. Therefore, cavitation can occur in both states [3].

Velocity is an essential parameter that plays a fundamental role in analyzing cavitation occurrence and vibration, in addition to determining the downstream flow pattern. Due to the high flow velocity in the channel and under the gate, the Reynolds number of the flow is usually greater than 10, indicating a fully turbulent flow. According to studies, when the flow velocity in the outlet tunnel exceeds 10 meters per second, air injection is necessary to prevent cavitation [4].

The role of air conduits in these flows is to prevent cavitation inside the channel. Air conduits are usually embedded in the upstream area, where the cavitation index is below the critical value. These conduits reduce the flow velocity and, consequently, the cavitation hole explosion process. Moreover, the presence of air conduits in the channel's roof transforms the flow from pressured to free flow, reducing negative pressures generated behind the gate due to contact with the atmosphere [5].

Flow in submersible channels is highly sensitive to geometric parameters, and even slight changes can alter the results [6]. Areas most at risk of cavitation are located in the lower part of the service gate, between the two service and emergency gates (in the case of simultaneous operation of two gates), and channels with gate slots creating an uneven surface against the flow. The flow passing under the gate generates rotational flow downstream, characterized by a severe pressure drop. The created drop is a function of gate opening, head above the gate, and channel geometry. On the other hand, intense pressure fluctuations lead to a decrease in local pressure in that area, and given the high flow velocity, the potential for cavitation increases [6].

Key variables in pressured channels include geometric parameters (length, width, height, slope) and hydraulic parameters (velocity, passing flow rate, head above, and pressure). In each gate placement, due to the moment and high flow velocity, pressure decreases along the lower surface of the gate, and behind it, while pressure changes minimally upstream of the gate.

In analyzing cavitation issues, it is essential to define an indicator that describes flow conditions regarding cavitation existence, initiation, or various stages of its expansion [7].

To define a cavitation indicator, we consider the fluid flow in a Venturi tube according to Figure (1). The pressure difference between two points, one on the body and the other in the untouched fluid upstream at a distance proportional to the square of the relative velocity, can be written as a negative coefficient of upstream pressure.

$$C_p = \frac{(P_{vp} - P_t)}{0.5\rho\, V_{vp}^2} \qquad (1)$$

In this relation, fluid density, fluid velocity relative to the wall, upstream fluid pressure, and pressure in the Venturi throat are represented.

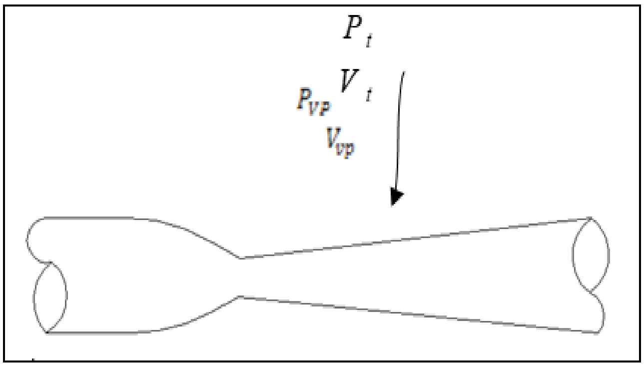

Fig 1 Flow in a Venturi Tube and Definition of Cavitation Index

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

One can consider conditions where the magnitude decreases sufficiently for cavitation to occur. This can be achieved by either increasing the relative velocity while keeping the speed constant or continuously reducing the speed while maintaining a constant relative velocity. Both methods lead to a reduction in the absolute values of all localized pressures on the body's surface. Disregarding surface tension, the pressure will be equal to the pressure of cavitation-containing voids. If we designate this expression as the bubble pressure and assume that cavitation occurs when the normal stress at a point in the fluid decreases to zero [8], it will be equal to the vapor pressure eventually.

$$\sigma = \frac{(P_{vp} - P_v)}{0.5\rho\, V_{vp}^2} \tag{2}$$

Cavitation erodes solid boundaries by removing material from the surface. Virtually any type of material, including metals, rubber, plastic, glass, silica, and concrete, can be damaged by this phenomenon [8].

If cavitation occurs far from the conduit walls, it poses no risk to any structures. However, if cavitation occurs near the walls, it may cause damage and put the structure at serious risk. Damage to the structure's surface begins at the downstream location where bubbles collapse. Over time, a significant recession develops on the surface. Due to the high-speed flow colliding with the downstream end of the cavity, the created recession becomes larger. The flow is capable of generating high pressures in the cracks beneath the structure. The pressure difference between the region where the flow impinges and the surrounding area has the potential to break sections of the structure and displace them due to the flow. [9]

Despite recent advances in the design and calculation of dams and related facilities using computational software, obtaining accurate results through theoretical modeling for many practical effects is still challenging due to the complexity and three-dimensionality of the flow pattern, mainly caused by the geometric complexity of the flow path in the vicinity of the gate. Therefore, due to the high costs of dam construction and related facilities, as well as the potential life and financial losses resulting from the potential failure of gates, hydraulic model testing for the discharge outlets of many high dams is practically necessary in our country. Therefore, models for outlets such as the Alborz Dam, Gavoshan Dam, Jagin Dam, Gatvand, etc., have been constructed and their performance has been evaluated.

II. MATERIALS AND METHODS

Sefidroud, or Sepidroud, is the third-longest river in Iran after Karun and Karkheh, with a length of 670 kilometers. It is the largest river in northern Iran, formed by the confluence of two rivers, Shahroud and Qizil Uzan, in the city of Manjil, and it flows towards the northeast into the Caspian Sea. The dam's geographical coordinates are approximately 49.3880° latitude and 36.7591° longitude. The dam is of the earth-fill type with a concrete core, and it has a reservoir volume of 1756 million cubic meters. The dam height from the riverbed is 106 meters, and its crest length is 425 meters.

Discharge outlets are one of the important issues for dam designers, and ensuring the capacity of water passage and proper performance of the conduit and related hydraulic and hydromechanical facilities, including gates, must be examined and evaluated.

For this purpose, a suitable model of this outlet was constructed and tested in the laboratory of the Soil Research Institute.

Experiments were conducted for four different heads. A metal tank with a height of about 18 meters was used to supply these heads. To adjust the head to the desired values, two pumps and one outlet pipe were used, and the research data used in this study were collected from the Regional Water Organization of Gilan Province.

Today, with the advancement of computer science, numerical methods have become a suitable alternative to laboratory models. Numerical methods in engineering sciences have gained more supporters due to cost and time savings. Among these software, the commercial software Flow 3D stands out, being one of the most powerful and practical CFD software in the world, developed and supported by Flow Science [10].

A four-block meshing approach was used for the mesh, where the reservoir, which is less important than other parts, was separated into a separate block and meshed with larger dimensions than other points in the flow path. Unlike the reservoir part, which is the main focus of the study and its results are highly important, in this part, mesh dimensions were reduced as much as possible, and as a result, a large number of meshes were assigned to this part.

Other parts of the conduit, i.e., upstream and downstream of the gates, were each introduced with a separate block and a similar meshing structure to the software.

The details of the geometry of the gate, which are important in this study, were modeled using the same software and the necessary changes were applied to these sections using a mesh to the computational grid to make them as similar as possible to the physical model.

Different parameters such as velocity, pressure, and air volume after the gate were evaluated in different conditions, and a comparison was made between the numerical and physical model results to verify the accuracy of the numerical model.

The details of the geometry of the gate, which are important in this study, were modeled using the same software and the necessary changes were applied to these sections using a mesh to the computational grid to make them as similar as possible to the physical model.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

Different parameters such as velocity, pressure, and air volume after the gate were evaluated in different conditions, and a comparison was made between the numerical and physical model results to verify the accuracy of the numerical model.

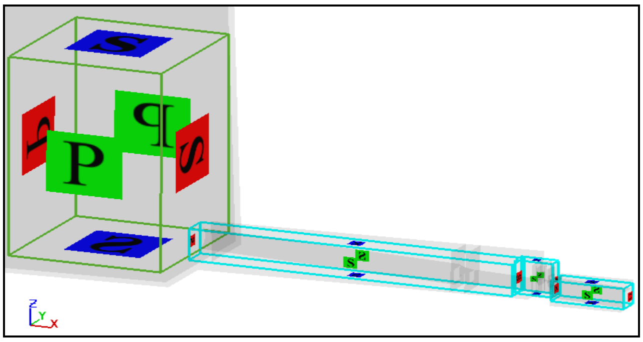

Fig 2 Representation of the Number and Location of Mesh Blocks and Their Boundary Conditions

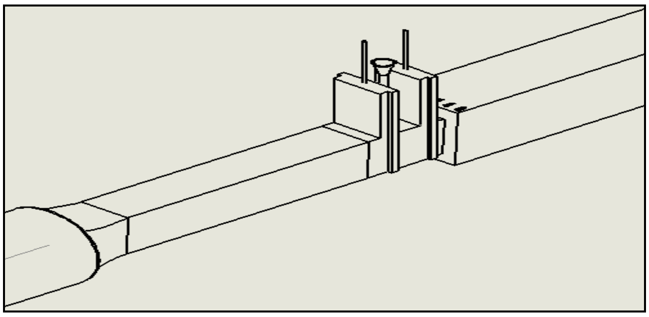

Fig 3 Three-Dimensional View of the Physical Model

After defining the boundary conditions, it is crucial to configure the parameters related to the initial conditions, a significant section of the software. In this section, the location of the flow entry point is determined to solve the problem effectively. Considering the reservoir head specified in the laboratory experiment, the numerical model simulates a reservoir filled with water up to this level. Additionally, to expedite velocity calculations, the channel upstream of the gates is also filled with fluid.

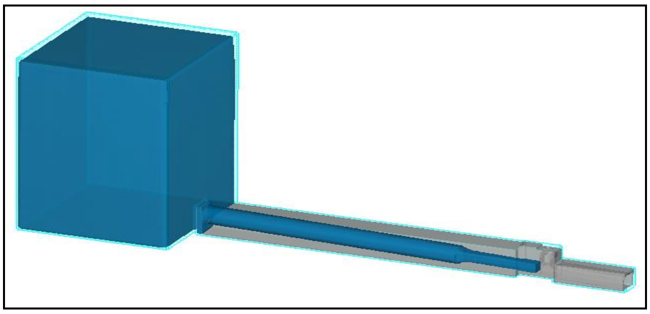

Fig 4 Initial Fluid Placement Before Starting the Analysis

One of the significant challenges affecting the precision of computations is determining appropriate boundary conditions. In the numerical model used, Outflow and Symmetry conditions were employed for flow outlets and walls, respectively. However, the Symmetry option could be replaced with Wall, considering that the walls are designed using SolidWorks software, making this substitution inconsequential to the computations.

The crucial part influencing the computational accuracy is the specification of the boundary conditions for the model's inlet section, which is the reservoir supplying the head. Given that the outflow from the model's end reduces the water volume inside the reservoir, preventing the replication of experimental conditions, the Specified Pressure condition was used in two ways. In the first scenario, the pressure equivalent to the height of water in the reservoir was calculated and applied to the upper surface of the water to maintain a constant pressure and water level inside the reservoir at all times. Initially, this method was employed for analysis, but discrepancies were observed in flow rates and pressures at various points in the model compared to the laboratory. To mitigate this, a second approach using Fluid Elevation was applied. In this method, a pressure equivalent to the height of water in the reservoir (670 centimeters) was applied to the reservoir walls, resulting in significantly reduced errors in flow rate and pressure compared to the first method. The maximum errors for flow rate and pressure occurred before the emergency gate, reaching a maximum of 15% difference between the two methods.After defining the boundary conditions, the point of flow entry for problem-solving needs to be specified. Considering the designated reservoir head in the laboratory, the numerical model filled the reservoir up to this level so that water, influenced by its weight, could flow through all points in the channel. However, due to the extended time required for water to reach all parts of the channel and achieve a stable state, the section upstream of the gates was also filled with fluid to expedite the analysis. This approach significantly reduced the analysis time by up to 20% (approximately 6 hours). In Figure (4), the parts of the model filled with water can be observed.

III. RESULTS AND DISCUSSION

In this section, the experimental results are compared with the numerical results obtained from the Flow3D software for a head of 670 centimeters. The maximum error in each section is then detailed. Following that, a brief discussion is provided regarding the discharge coefficient and the Froude number, which serve as the basis for result comparison.

➢ *In the Single-Gate Performance Mode:*

Table 1 Comparison of Permeability Coefficient between Experimental and Numerical Results

Opening Percentage	Experimental	Numerical	Error (mm)
80	0.78	0.8	2.56
60	0.76	0.78	2.63
30	0.73	0.74	1.36

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

Table 2 Comparison of Water Flow Rate between Experimental and Numerical Results

opening Percentage	Experimental Flow rate (m^3/s)	Numerical Flow Rate (m^3/s)	Error (mm)
80	90.80	94.17	2.58
60	68.85	78.04	13.34
30	34.43	40	16.17

Table 3 Comparison of Froude Number between Experimental and Numerical Results

Opening Percentage	Experimental	Numerical	Error (mm)
80	8.9	10	13.36
60	9.89	11.5	16.5
30	13.79	16.5	19.65

➢ *In Simultaneous Operation Mode:*

Table 4 Comparison of Permeability Coefficient between Experimental and Numerical Results

Emergency	Service	Experimental	Numerical	Error (mm)
78	80	0.71	0.8	12.86
57	60	0.6	0.69	15
29	30	0.75	0.86	14.66

Table 5 Comparison of Water Flow Rate between Experimental and Numerical Results

Emergency	Service	Experimental Flow Rate (m^3/s)	Numerical Flow Rate (m^3/s)	Error (mm)
78	80	75.66	78.01	3.11
57	60	51.42	63.2	21.9
29	30	51.42	59	14.74

Table 6 Comparison of Froude Number between Experimental and Numerical Results

Emergency	Service	Laboratory Test	Numerical	Error (mm)
78	**80**	**8.04**	**7.51**	**6.55**
57	**60**	**10**	**10.74**	**7.4**
29	**30**	**20.57**	**23**	**11.8**

➢ *Permeability Capacity*

By comparing the results, it is observed that the flow rate obtained from the software has a relatively acceptable accuracy compared to the experimental results. However, in all cases, the flow rate obtained from the software is higher than the flow rate recorded in the laboratory. The error percentage ranges from 2 to 16 percent in the single-gate mode and from 3 to 15 percent for simultaneous operation. The highest errors are related to the minimum opening, i.e., a 30% opening of the service gate. This error is due to the presence of powder flow at low openings, causing more significant discrepancies in the software results.

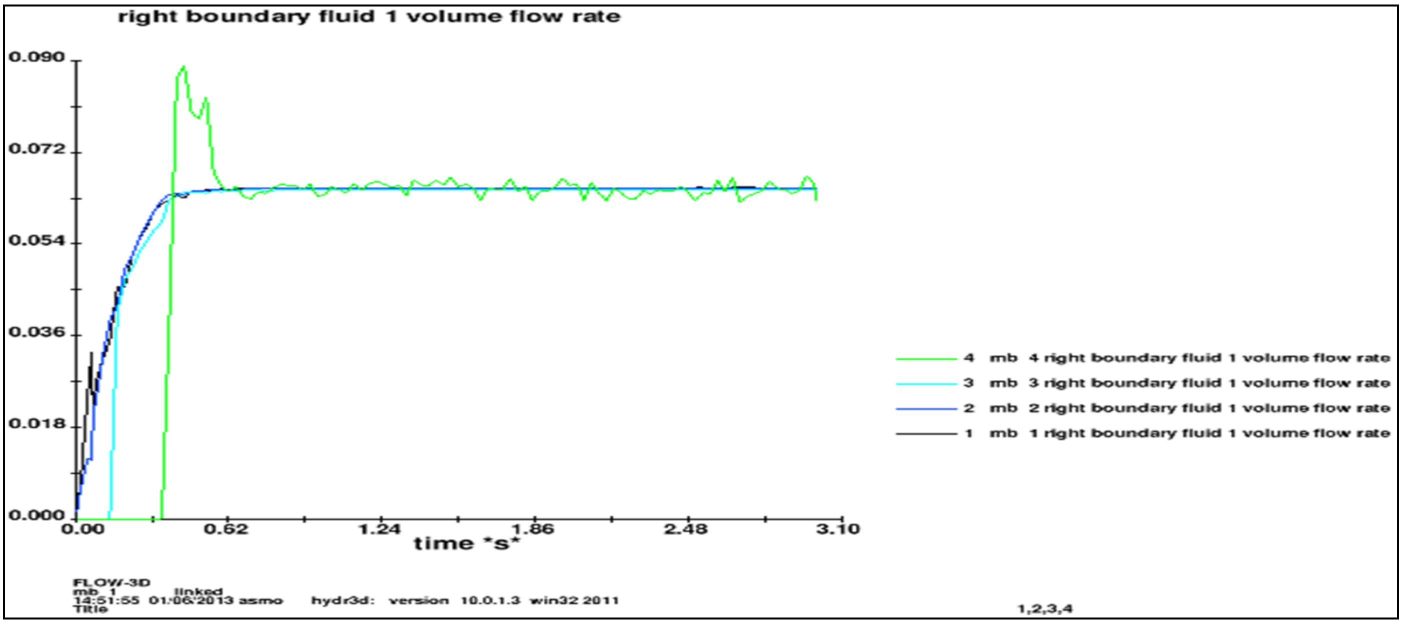

Fig 5 Software Output: Flow Rate at 60% Opening in a 670 cm Head

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

➢ *Friction Factor*

In this section, a comparison is made between the friction factors obtained from experimental results and numerical data. Similar to the flow capacity discussed in the previous section, here too, a relatively large difference is observed in smaller openings compared to other openings. In larger opening ranges, the error has reduced by up to 6%. For example, in the case of a 30% opening in single-gate mode, the error has decreased by about 10%, and in combined operation mode, it is approximately 4.5%. This indicates that the software accuracy is lower at smaller openings due to the presence of low powder flow. Additionally, since a similar meshing has been used for all openings (for computational speed increase), in smaller openings, there are fewer cells in the flow path, resulting in reduced accuracy. Figures (6) and (7) show the variations of the friction factor along the length of the channel for a 60% opening.

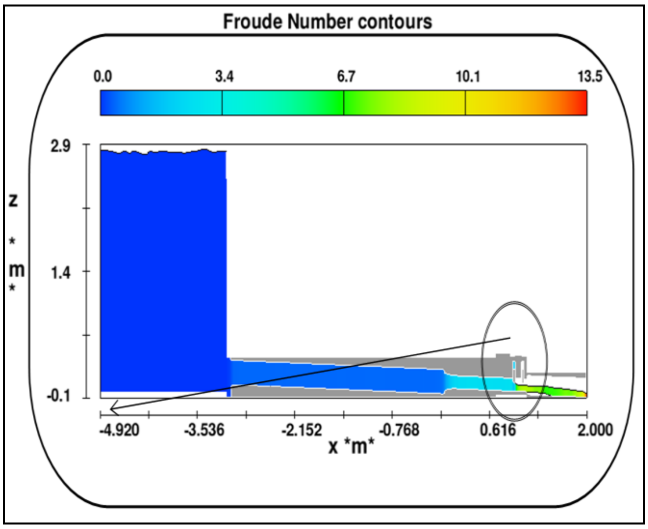

Fig 6 Changes in the Friction Factor along the Length of the Channel for a 60% Opening.

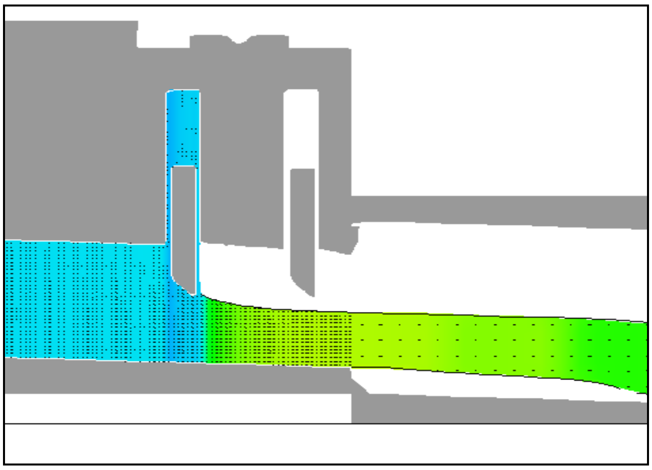

Fig 7 Changes in the Friction Factor in the Range between Two Gates for a 60% Opening.

In the case of full opening of the emergency and service gates, the measured pressure changes at all points in the channel are positive. In the series of diagrams from the previous section, the lowest pressures are related to the piezometers located in the groove range of the service gate, where these pressures are positive but close to the critical state. With decreasing openings, the measured pressure values increase and move significantly away from the critical range.

To evaluate the occurrence of cavitation, the cavitation index parameter (σ), defined as the ratio of absolute pressure to hydrodynamic pressure, was used. Based on this, the cavitation index is calculated using the measured pressures and flow velocities at each gate opening, and it is compared with the critical cavitation number to assess the likelihood of cavitation occurrence.

These tables represent the cavitation index at various points along the channel for different gate openings, respectively, in the two functions of the service and emergency gates at the maximum reservoir level. This is the most critical hydraulic performance condition of the channel, and if the results obtained in this condition are acceptable, a rational evaluation of the channel performance will be possible. To prevent damage from cavitation, it is necessary for the index not to be less than a critical limit (0.25 for the channel and 0.20 for the gate groove).

By observing the tables, it is noted that in the case of full gate opening and different gate openings from 2% to 100%, the measured pressures at critical points in the channel are positive. Considering the significant pressure drop in the groove of the service gate and before the constriction in the channel walls, this problem can be addressed by modifying the geometry, such as adding a projection to the flow ceiling. As seen, in the case of full opening of the emergency and service gates, the changes in the measured pressures at all points in the channel are positive. The recorded negative pressures are related to the region after the service gate, where complete air bleeding has occurred, and therefore, the pressure drop is not significant within the flow.

By studying the series of mentioned tables in the fully opened positions (100% opening), the cavitation index values were evaluated at critical points in the channel. Based on the observations, the minimum index around the service gate and inside the groove and after the groove has been recorded. Tables (7) and (8) show the cavitation index values in the critical groove range of the service gate and the emergency gate at several gate openings. According to the table below, the minimum index in the channel at 100% opening, in Piezometer 67 (service gate groove), is 0.62, which is above the critical range, and Piezometer 47 is recorded at 0.43, which is outside the critical index, indicating a negative likelihood of cavitation occurrence in the channel.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

Table 7 Cavitation Index Values in the Critical Range of the Service Gate Groove at 100% Gate Opening

Gate Opening (%)	**Piezometer Number**					
	47	55	65	66	67	68
100	0.43	0.47	0.66	0.65	0.62	0.59
90	0.43	0.47	0.65	0.66	0.62	0.60
80	0.83	0.72	1/070	1.050	1.010	1

Table 8 Cavitation Index Values in the Critical Range of the Emergency Gate Groove at 100% Gate Opening Performance

Gate Opening (%)	**Piezometer Number**					
	45	53	61	62	63	64
100	0.88	0.90	0.94	0.94	0.92	0.92
90	0.89	0.92	0.96	0.96	0.96	0.94
80	1.610	1.690	1.710	1.720	1.720	1.720

For the simultaneous operation mode at a head of 670 centimeters, the following tables are provided to show the results of pressures and cavitation indices. As seen, for this mode as well, the changes in measured pressures at some points are negative, and the cavitation index values in sensitive points of the channel are in a range close to critical, indicating the possibility of cavitation occurrence.

Table 9 Pressure Changes in the Slot Range of the Service Gate in the Simultaneous Operation Mode

Gate Opening (%)	**Piezometer Number**						
	47	55	60	65	66	67	68
100S 100 ،E	6.45	2.22	5.12	6.05	6.1	5.87	4.4
90S 90 ،E	5.18	1.5	0.45	5.6	5.05	4.07	3.65
80S 78 ،E	5.18	1.35	0	4.25	3.55	1.55	0.73

Table 10 Changes in pressure in the Range of the Emergency Gate Groove in the Simultaneous Operation Mode.

Gate Opening (%)	**Piezometer Number**						
	45	53	58	61	62	63	64
100S 100 ،E	12.89	13.02	16.22	15.55	15.55	14.57	14.75
90S 90 ،E	18.89	14.82	0.15	19.45	28.8	27.62	21.05
80S 78 ،E	23.63	17.67	-2.61	32.23	32.20	29.79	26.15

Table 11 Changes in Cavitation Index in the Slot Range of the Service Gate in Simultaneous Operation Mode.

Gate Opening (%)	**Piezometer Number**						
	45	53	58	61	62	63	64
100S 100 ،E	0.28	0.28	0.25	0.28	0.29	0.3	0.26
90S 90 ،E	0.5	0.42	0.69	0.7	0.7	0.66	0.68
80S 78 ،E	0.33	0.27	0.31	0.28	0.29	0.23	0.21

Table 12 Changes in Cavitation Index in the Slot Range of the Emergency Gate in Simultaneous Operation Mode.

Gate Opening (%)	**Piezometer Number**						
	47	55	60	65	66	67	68
100S 100 ،E	0.25	0.3	0.25	0.3	0.27	0.26	0.26
90S 90 ،E	0.25	0.25	0.26	0.25	0.25	0.25	0.25
80S 78 ،E	0.79	0.55	0.35	1.08	1	1	1.66

As seen in the above tables, the most critical condition is observed at 80% opening of the service gate and 78% opening of the emergency gate. In these cases, the cavitation index in the service gate slot reaches 22.0, which is again higher than the critical state, practically indicating the occurrence of cavitation after negative aeration.

Numerical modeling was performed using Flow3D software, and the geometry model was created using SolidWorks. Calibration of the software was carried out using experimental results related to a 670 cm head, where the main criteria for comparing numerical and experimental results were the flow rates and Froude numbers. The most critical factors for improving accuracy in software results are the number and size of meshes, with the type of turbulence being a secondary consideration. Optimal mesh refinement was achieved, particularly in the area between the two gates, which improved the acceptability of the results.

Considering the results obtained from the software, good agreement was observed at larger gate openings, while the most significant differences were observed at smaller gate openings due to the presence of powder flow, revealing the software's weakness in providing data for this type of

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24MAY2113

flow. Some of these differences are also attributed to the experimental data collection, where the reduction in flow rates and the fact that the level gauge has a certain error in determining the water surface contribute to an increase in the error percentage.

Based on the results of the experiments and the information presented in this report, the following recommendations are made:

Conduct experiments on more heads to obtain more accurate results, especially for simultaneous operation, which is crucial.

Due to the significant impact of the downstream tunnel roughness coefficient on flow cavitation and air entrainment into the channel, minimizing this coefficient during model construction operations, such as smoothing, is recommended.

Forces on the gates and the phenomenon of their vibration are important factors not addressed in this research, but they are among the essential factors in spillways.

A detailed examination of the performance of air vents, optimizing them using fuzzy systems, which are effective in diameter and the number of vent pipes, is also an important factor in spillways and could be a direction for future research.

Numerical solution of this model using two-phase mixture methods in software such as Fluent, Ansys, etc., and comparing them with experimental results.

Finally, modeling these data using software such as Fluent, Ansys, etc., to investigate the capabilities of these software for ensuring the accuracy of the modeling process, providing confidence to researchers in the modeling

.

REFERENCES

[1]. Hosseini, S. M., & Abrishami, J. (2008). "Open Channel Hydraulics: Hydraulic." 18th edition, Imam Reza University Press, Mashhad.

[2]. Marston, L. and Cai, X. (2016), An overview of water reallocation and the barriers to its implementation. WIREs Water, 3: 658-677. https://doi.org/10.1002/wat2.1159

[3]. Emamgholizadeh, S., Borojeni, H. S., & Bina, M. (2005). The Flushing of the Sediments near the Power Intakes in the Dez Reservoir. Transactions on Ecology and the Environment, 83, 621-630.

[4]. Emami, M. K., Kavyanpour, M. R., & Roshan, R. (2010). "Numerical Investigation of Velocity Distribution in Turbulent Flow in Submerged Discharge Outlets (Case Study: Submerged Discharge Outlet of Sefidrood Dam)." 9th Iranian Hydraulic Conference, Tarbiat Modares University, November 2009, Tehran.

[5]. Nezhad Nadari, et al. "Simulation of Flow in the Bottom Outlet of the Narmashir Dam." First National Conference on Dams and Hydropower Plants, 2011.

[6]. Nakhei, M. (1999). "Pressure Fluctuations in Downstream of Submerged Gates." Master's thesis, K.N. Toosi University of Technology, Tehran.

[7]. Naudascher, E., & Rockwell, D. (1994). "Flow-induced Vibration, IAHR Design Manual No.7." Balkema.

[8]. Jin, T. (1990). "Cavitation in Hydraulic Structures."

[9]. Najafi, M.R., Roshan, R., Zarrati, A.R., Kavianpour, M.R. (2009). Numerical Modeling of Flow Condition in a Bottom Outlet. In: Advances in Water Resources and Hydraulic Engineering. Springer, Berlin, Heidelberg. https://doi.org/10.1007/978-3-540-89465-0_313.

[10]. Bowers, E., & Toso, J. (1988). "Karnafuli Project, Model Studies of Spillway Damage." Journal of Hydraulic Engineering, ASCE, Vol. 114, No. 5.

[11]. W. T., & Dahleh, M. D. (1998). "Theory of Vibration with Applications."

[12]. Qasemzadeh, F. (2015). "Simulation of Hydraulic Problems in Flow-3D." Nashravaran Publications, 2015.

[13]. Amirsayafi, P. Measures for Success in Dam Bottom Outlet Design. *GSTF J Eng Technol* **3**, 30 (2015). https://doi.org/10.7603/s40707-014-0030-2

[14]. Pearsall, I. S. (1974). Cavitation. Chartered Mechanical Engineer, 21(7).

[15]. J.-P. Franc and J.-M. Michel, "Attached cavitation and the boundary layer: Experimental investigation and numerical treatment," J. Fluid Mech. 154, 63–90 (1985).

[16]. S. R. Gonzalez-Avila, F. Denner, and C.-D. Ohl, "The acoustic pressure generated by the cavitation bubble expansion and collapse near a rigid wall," Phys. Fluids 33(3), 032118 (2021)