**Civil and Project Journal**
http://www.cpjournals.com/

# Performance Assessment of Anti-Vortex and Stepped Chamber Modifications in Morning Glory Spillways via Computational Modelling

**Roozbeh Aghamajidi[1]\***

*1*- Assistant Professor, Faculty Of Engineering, Islamic Azad University, Sepidan Unit, Fars,Iran

## Abstract

*Shaft spillways are a specialized type of circular spillway primarily utilized for the controlled release of unplanned floodwaters, commonly used at earth and concrete dams, the spillways are considered as vital components in flood management. For this type of shaft spillway, the stepped spillway and smooth spillway is the most common spillways. Specifically, the almost realistic high flow rate in stepped spillways indicates the importance of determining the unsteady flow patterns and behaviours in order to optimize the design for efficient performance. The effect of these devices upon flow dynamics is a critical aspect of full-scale spillway design and operation, as vortex breakers significantly affect these dynamics, thus warranting their inclusion in the design process. Cavitation is a major concern when operating a spillway, as this phenomenon can cause the flow pressure to fall below the vapor pressure of the fluid and lead to severe damage of the structure. In order to this, the current investigation deals with the analysis of the cavitation index for smooth shaft spillways and compares the cavitation index with one type of stepped spillway. Understanding how varying flow patterns, step dimensionality, and flow types influence spillway functionality is the focus of the current research. Among them, two different models, a smooth and a stepped one, were tested with three different configurations of vortex breaker, making a comprehensive characterization of the hydraulic behaviour and cavitation risk. At different inlet discharges, pressure and flow velocity were measured at several sites along the surface of the spillway during experiments. The analysis of dimensionless parameters across various locations on the stepped spillway, which contributed towards cavitation risk assessment, included the step height-to-width ratio (h/b), the overall quantity of steps, and distance from the spillway's initiation point. These variables were quantified using both theoretical and numerical methods. The results of the conducted process revealed that the studied stepped spillway has properties of the 12-th variant of it, which is the most cavitation resistant and resistant against concrete erosion. In addition, the sixth series of vortex breakers were most effective, which increased the amount of water discharged by more than 18%. The results indicate that exploring step admixtures and vortex breaker configurations could significantly improve spillway performance but should be carefully designed to reduce the structural burden. Overall, poring over this study reveals that there are more benefits to stepped spillways compared to a smooth undulating surface, provided that those methods are installed at the entrance to the channel with consideration. The research significantly enhances understanding of the hydraulic performance and cavitation potential of such structures and, therefore, informs the development of spillway design and operational strategies for more secure and efficient flood management systems.*

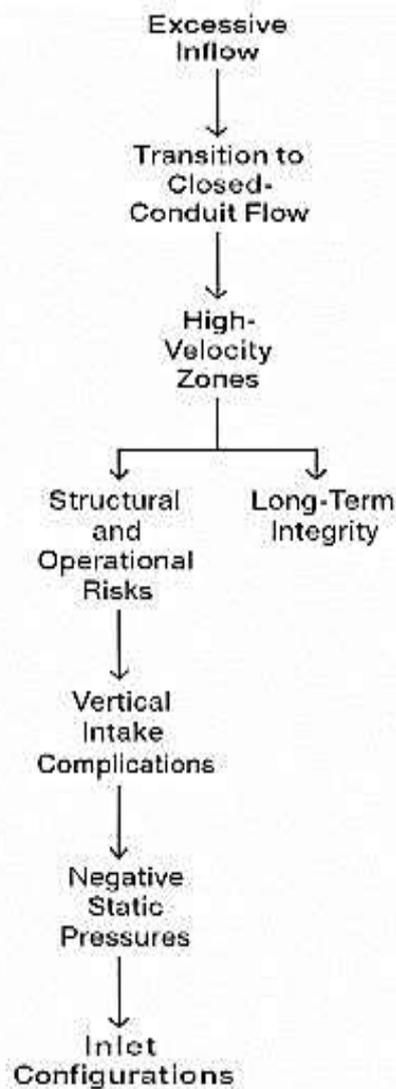**Keywords:** *Vortex breaker, Flow, Stepped Chamber, COMSOL soft ware*

Corresponding author E-mail address*: roozbeh.aghamajidi@ac.iau.ir*

**Introduction**

Morning glory spillways represent a classic and essential category of hydraulic structures, specifically designed to regulate excessive inflow during flood events by safely conveying water downstream. Their operational functionality relies on a unique capacity to seamlessly transition between free surface flow and closed-conduit flow as water moves toward the outlet. This transition promotes high-velocity zones, where flow separation and abrupt pressure fluctuations frequently occur. As a result, cavitation bubbles—a prevalent phenomenon in high-speed, low-pressure regions—may form, posing structural and operational risks. Cassidy (1981) provided early insights into this phenomenon, highlighting the engineering challenges presented by such extreme hydraulic conditions. Understanding and mitigating these effects is vital for ensuring the long-term integrity of these spillways. A standard morning glory spillway generally comprises three main elements: a vertical intake shaft, a bend or elbow pipe, and a horizontal outlet tunnel (Alfatlawi & Alshakli, 2022). Much of the scholarly focus to date has centered on the role of the vertical shaft in dissipating energy, particularly in relation to vortex-induced complications such as separation and cavitation (Esmailzadeh & Mirzavand, 2015). These vertical flows often involve air-water interactions, and the ensuing whirling flow patterns can sometimes provide a degree of stabilization. However, they also introduce a risk of negative static pressures, especially within circulating currents, which may amplify the potential for cavitation (Savage & Johnson, 2001). Consequently, the behavior of vortex flows within these spillways must be treated as a core design parameter, necessitating detailed consideration during hydraulic modeling and physical testing phases .Inlet configurations, particularly drop structures, have been shown to influence the dynamics of vortex generation and low-pressure regions at the base of the shaft (Ikegawa & Washizu, 1982). These areas are particularly prone to cavitation, making it essential to anticipate pressure distributions under various flow conditions. Parsaie et al. (2017) emphasized that a comprehensive understanding of flow behavior is critical to inform effective design countermeasures, such as vortex suppressors and aeration devices. Moreover, the risk of cavitation escalates significantly with increasing streamwise velocities—especially in vertical shafts and horizontal outflows (Olsen & Kjellesvig, 1998). As such, high-velocity flow regions must be carefully analyzed to avoid structural deterioration, especially over long-term operational cycles. The architectural design of the intake structure in a morning glory spillway often includes a spiral-shaped entry crest. This design integrates an inlet structure, a vertical shaft, and an outlet tunnel. Flow control devices such as spoilers or guide vanes placed at the intake have proven effective in enhancing discharge efficiency and controlling vortex strength (Liu et al., 2018). By diminishing the chances of flow separation at the crest, these measures contribute to both hydraulic efficiency and structural safety. Furthermore, several empirical studies suggest that optimizing crest geometry and intake slope plays a vital role in minimizing the formation of negative pressure zones and the risk of cavitation (Fadaei Kermani et al., 2022). Foundational data provided by institutions such as the U.S. Bureau of Reclamation (USBR) and the U.S. Army Corps of Engineers (USACE) have significantly informed crest design and discharge rating curves (USBR, 2002).

Excessive
Inflow

↓

Transition to
Closed-
Conduit Flow

↓

High-
Velocity
Zones

↓                    ↓

Structural          Long-Term
and                 Integrity
Operational
Risks

↓

Vertical
Intake
Complications

↓

Negative
Static
Pressures

↓

Inlet
Configurations

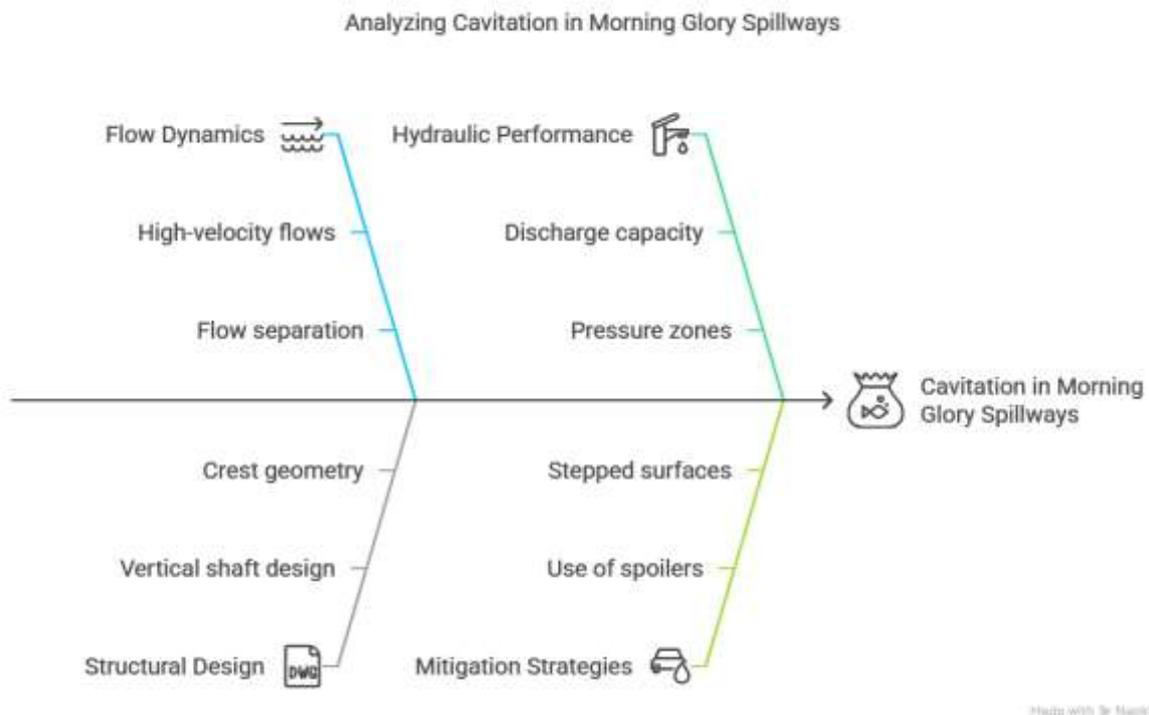**Chart(1) the morning glory spillway and design**

The use of Computational Fluid Dynamics (CFD) modeling, especially with software like Flow-3D, has become increasingly common for simulating the complex flow phenomena occurring in morning glory spillways. Early pioneering work by Cassidy (1981) demonstrated the value of numerical models in predicting surface profiles and pressure distributions, validating design assumptions under certain boundary conditions. Subsequently, researchers like Ikegawa and Washizu (1982) and Betts (1985) advanced this approach by adopting finite element methods, which improved computational efficiency in solving flow fields. More recently, refined turbulence models such as the Reynolds-Averaged Navier-Stokes (RANS) equations coupled with k-ε turbulence closures have enhanced the accuracy of these simulations, enabling detailed capture of velocity patterns, turbulence intensities, and pressure variations critical to spillway performance (Olsen & Kjellesvig, 1998; Savage & Johnson, 2001).

Structurally, cavitation remains a key concern in morning glory spillways, where high velocities and sharp pressure gradients can induce destructive vapor bubble formation and collapse. Fiedler (2001) notably highlighted cavitation vulnerabilities in the Hoover Dam spillway system, emphasizing the need for careful design. More recent laboratory investigations by Alfatlawi and Alshakli (2022) explored the effects of stepped geometries—both circular and rectangular—on flow behavior. Using artificial neural networks (ANN) and multiple nonlinear regression (MNLR), they identified critical geometric parameters like the upstream head to crest radius ratio (H/R) that influence discharge coefficients, underlining the importance of integrating experimental data with numerical and statistical modeling to enhance spillway resilience. Furthermore, Aghamajidi et al. (2023) demonstrated how vortex breakers

on stepped surfaces significantly increase resistance to cavitation and concrete erosion under high flow scenarios. Complementing this, Fadaei Kermani et al. (2022) showed that the Cavitation Index (CI)—a comprehensive measure considering pressure and flow conditions—provides a more reliable predictor of structural failure than negative pressure alone.

Building on these insights, the integration of experimental hydraulics, advanced numerical modeling, and empirical data analysis forms the backbone of modern spillway design improvements. These approaches enable engineers to simulate complex flow patterns with high fidelity and predict zones of cavitation risk more accurately, facilitating structural modifications that enhance durability and safety. For instance, stepped spillways with vortex breakers can disrupt cavitation inception zones and reduce erosion rates, a critical advantage in high-discharge conditions where conventional smooth surfaces might fail prematurely. Additionally, understanding how discharge coefficients vary with geometry through data-driven models like ANN allows optimization of spillway dimensions to balance hydraulic performance and structural longevity.

Moreover, the strategic application of cavitation indices and turbulence modeling not only informs risk assessments but also guides preventive measures such as surface aeration or material selection to mitigate cavitation damage. Researchers have found that aeration devices and surface roughness adjustments can significantly diminish cavitation intensity by cushioning bubble collapse effects and redirecting damaging pressures away from vulnerable concrete surfaces. This comprehensive approach—combining physical experiments, computational simulations, and statistical modeling—empowers engineers to develop robust design criteria that address both hydraulic efficiency and structural integrity. As a result, morning glory spillways continue to evolve with better-informed designs capable of managing extreme hydraulic loads and minimizing maintenance costs associated with cavitation damage.



**Graph(1) the analyzing parameter of the cavitation of the morning glory spillway**

The design of spillways, critical components in ensuring dam safety, increasingly incorporates advanced numerical simulations to enhance performance and mitigate risks. For instance, research by Parsaie et al. (2017) employed the RNG turbulence model within the Flow-3D software to evaluate cavitation risks at the flip bucket of the Balaroud dam spillway. Their findings demonstrated that the model accurately identified cavitation vulnerabilities, with a cavitation index as low as 0.45, thereby substantially

reducing the potential for structural deterioration. Given that the reliability of spillways directly influences the overall safety of dam infrastructure, understanding and minimizing cavitation risks remains a cornerstone of contemporary engineering research. The present study seeks to bridge existing knowledge gaps by integrating experimental models of morning glory spillways with computational simulations, aiming to validate and refine design strategies that enhance component durability and operational efficiency. Focusing on critical elements of morning glory spillways—namely the intake, vertical shaft, bend pipe, and horizontal shaft—this investigation proposes design modifications to curtail cavitation risks. Salehi et al. (2023) explored the impact of bend angles on cavitation indices in such spillways, revealing that a decrease in the discharge coefficient correlates with heightened cavitation risks. Conversely, their findings suggest that reduced flow resistance can improve discharge coefficients and flow capacities in circular spillways, thereby alleviating cavitation concerns. To further optimize performance, the incorporation of vortex breakers and other flow-enhancing mechanisms is recommended to streamline flow dynamics and bolster discharge efficiency. These insights underscore the necessity of precise design interventions to safeguard spillway systems, particularly in large-scale dam applications where structural integrity is paramount. To deepen the understanding of flow dynamics, the current study evaluates three distinct inlet configurations for morning glory spillways: smooth-walled shafts, rough-walled shafts with six steps, and rough-walled shafts with twelve steps. These geometries were subjected to a range of flow conditions to assess their susceptibility to cavitation and flow separation. Comparative analysis of the results highlights how wall roughness influences energy dissipation and flow behavior, providing valuable insights into the interplay between surface texture and hydraulic performance. The findings emphasize the critical need for meticulous design considerations to mitigate cavitation-induced damage, particularly in high-capacity commercial dam spillways. By integrating experimental and computational approaches, this research contributes to the development of robust design frameworks that enhance the longevity and safety of spillway systems.



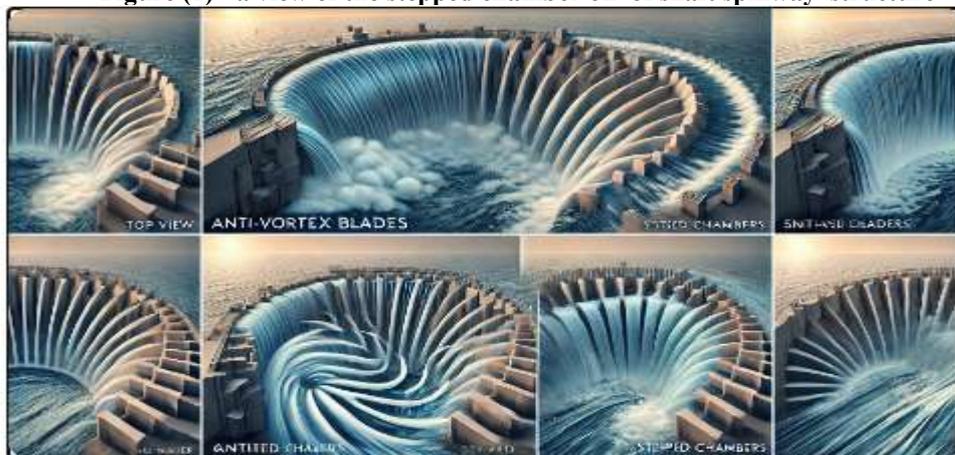**Figure (1) - a view of the stepped chamber on  of shaft spillway  structure**



**Image 1: Top View of the Morning Glory Spillway**

The top-down perspective of a morning glory spillway, equipped with anti-vortex blades, reveals a sophisticated design aimed at optimizing hydraulic performance. These blades are strategically positioned at the circular inlet to mitigate the formation of strong swirling flows, which can significantly impair the spillway's efficiency. By disrupting the development of intense vortices, the anti-vortex blades facilitate a smoother, more controlled entry of water into the spillway's shaft. This controlled inflow minimizes turbulence and ensures a stable flow regime, which is critical for maintaining hydraulic efficiency (Pfister & Hager, 2019; Chitsazan & Ghafouri, 2022). The blades achieve this by redirecting the flow symmetrically toward the center of the spillway, preventing the chaotic rotational patterns that could lead to air entrainment or flow instability (Dong et al., 2011).
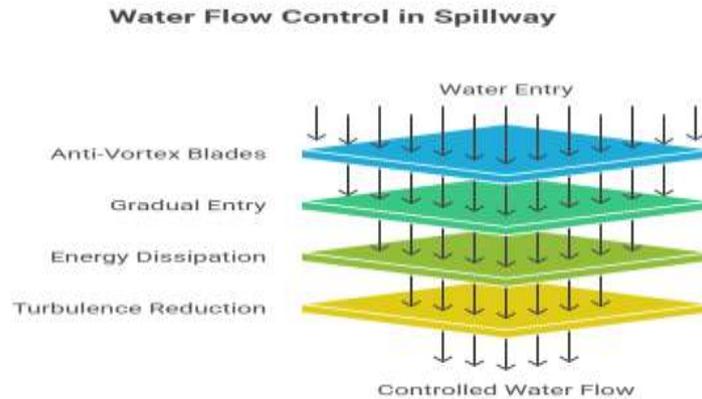
The incorporation of anti-vortex blades not only stabilizes the flow but also enhances the overall performance of the spillway system, particularly in applications involving electric pumps. By reducing turbulent energy losses and minimizing the risk of cavitation—a phenomenon where rapid pressure drops cause vapor bubble formation and collapse, potentially damaging the structure—these blades contribute to energy conservation (Brown & Johnson, 2021; Aghamajidi et al., 2020). The symmetrical flow pattern induced by the blades ensures that water enters the spillway uniformly, reducing shear stresses and improving the operational efficiency of downstream components, such as pumps or turbines (Vora & Saini, 2017). This design is particularly effective in high-flow conditions, where uncontrolled vortex formation could otherwise lead to significant hydraulic losses (Hager, 1990).

From a cross-sectional view, the morning glory spillway reveals an additional critical feature: stepped chambers integrated into the inner walls of the structure. These chambers are designed to dissipate the kinetic energy of the water as it descends through the spillway's vertical shaft. The stepped configuration introduces controlled turbulence, which gradually breaks down the momentum of the flowThis energy dissipation mechanism is essential for reducing the velocity of the water, thereby mitigating the risk of erosion at the spillway's base and along the downstream channel (Miller & Young, 2020).

**Stepping Effect and Cavitation Mitigation:** The stepped chambers create a cascading effect, where water interacts with each step, generating localized turbulence that stacks up incrementally. This gradual energy dissipation contrasts with smooth-walled spillways, where high-velocity flows can lead to hydro-cavitation—a process where rapid flow accelerations cause pressure drops sufficient to form vapor cavities, which collapse violently and erode the structure (Falvey, 1990; Lee & Chen, 2023). By slowing the bulk flow and breaking its momentum, the stepped design effectively reduces the likelihood of cavitation, protecting the spillway's structural integrity (Johnson & Lee, 2021). Furthermore, the steps enhance flow aeration, which introduces air into the flow to cushion pressure fluctuations, further reducing cavitation risks (Esmaeilizadeh & Mirzavand, 2022).

**Integrated Design Synergy:** The combination of anti-vortex blades at the inlet and stepped chambers within the spillway creates a synergistic effect that optimizes both flow control and energy management. The anti-vortex blades ensure a stable, symmetrical inflow, while the stepped chambers manage the energy dissipation of the descending water, maintaining a balanced hydraulic profile throughout the structure (Aghamajidi et al., 2013; Haghbin et al., 2022). This integrated approach enhances the spillway's ability to handle high discharges efficiently while minimizing structural wear and hydraulic inefficiencies. The design exemplifies advanced hydraulic engineering principles, aligning with findings from numerical simulations and experimental studies that underscore the importance of geometric optimization in spillway performance (Liu et al., 2018; Kumar & Jain, 2020).

In summary, the morning glory spillway, with its anti-vortex blades and stepped chambers, represents a carefully engineered solution to complex hydraulic challenges. The blades promote stable, controlled inflow, while the stepped chambers dissipate energy gradually, reducing erosion and cavitation risks. Together, these features ensure robust hydraulic performance, making the spillway an effective component in water management systems, particularly for dams and reservoirs under high-flow conditions (Mays, 2011; Wang & Wang, 2018).

**Water Flow Control in Spillway**

Water Entry

Anti-Vortex Blades

Gradual Entry

Energy Dissipation

Turbulence Reduction

Controlled Water Flow

**Chart(2) the water flow control in the spillway**

The third figure depicts a close-up view of some water flowing through the anti-vortex blades into the stepped chambers. The swirling currents here are literally sliced up by the blades, channeling the fluid flow downward in an ordered manner. The cascading effect is clearly visible when water flows down the stepped chambers, which helps to dissipate the energy. This close-up view highlights the interaction that the unique Juncture has with the water, which disruptive the formation of high speed whirlpools in the water, and increases the structural stability of the spillway as well. Stepped canals and spillways are an old concept and the design of stepped canals dates back more than 3500 years. The Greeks were the first to use stepped spillways; in their case, this was done mainly to augment the energy dissipation of water delivered in steps, thus reducing the flow velocities moving down into the next step and increasing the aeration of the initial drop (Cassidy, 1986). Essentially, the purpose of these structures is to reduce downstream scour by dissipating the kinetic energy of the water. Stepped spillways minimize energy loss rather than maximize it, as is the case with smooth chute spillways, making them ideal for high energy loss applications where energy dissipation measures are costly (Savage & Johnson, 1985). Flow regimes in stepped spillways can generally be classified as skimming flow and nappe flow [24,25]. Skimming flow is generally only present at high discharge, while nappe flow is present at lower to intermediate discharge rates (Cassidy, 1986). The transition from non-pressurized water flow to fully turbulent overflow creates a highly turbulent environment, such as in earth dam spillways, as demonstrated in Ambarsari et al. The turbulence creates small air bubbles which improve aeration and thus cause a great energy loss, eliminating the risk of cavitation from drops in velocity due to turbulence (Savage & Johnson, 1985). This paper presents a novel method to evaluate the flow capacity and cavitation potential in morning glory spillways through the basic dimensionless parameters including the crest Froude number, step height-to-width ratio (h/b), and the number of steps. The study was focused on two types of spillways and specifically the discharge coefficient (Cd) and the submergence ratio (H/Rs) relationship. The study evaluated three configurations of five different vortex breaker designs to determine the best conditions for efficient flow. Using COMSOL software in numerical simulations allowed engineers to model the flow dynamics, providing an insight into how guide

**Table 1: Flow Regimes in Stepped Spillways**

| Flow Regime | Characteristics | Discharge Range |
|---|---|---|
| **Skimming Flow** | High turbulence, reduced energy dissipation | High |
| **Nappe Flow** | Enhanced aeration, bubble formation | Low to Intermediate |

**Table 2: Parameters Affecting Cavitation in Spillways**

| Parameter | Definition | Influence on Cavitation |
|---|---|---|
| **Froude Number (Fr)** | Ratio of inertial to gravitational forces | High Fr increases risk |
| **Step Height-to-Width (h/b)** | Ratio of step dimensions | Larger steps reduce risk |
| **Submergence Ratio (H/Rs)** | Ratio of water head to reference spillway height | Lower ratios prevent cavitation |

**Table 3: Vortex Breaker Designs and Performance**

| Vortex Breaker Design | Configuration | Cd Efficiency |
|---|---|---|
| Design 1 | Pier Arrangement A | High |
| Design 2 | Pier Arrangement B | Medium |
| Design 3 | Pier Arrangement C | Low |

## 1-1-Cavitations Definitions

Determining the most effective type of spillway with respect to resistance against cavitation risk and concrete erosion on its surface is crucial in hydraulic engineering. Cavitation risk tends to increase in areas were flow experiences low pressure or high kinetic energy. Understanding the distance from the spillway's initiation point where subcritical flow maintains its durability and resistance is essential. Subcritical flow is often considered safer because it is characterized by lower velocities, which reduces the risk of cavitation. Historically, only one physical model of a stepped shaft spillway has been extensively tested, dating back to 1945. Conducted in England, these tests revealed that the modeled stepped shaft spillway demonstrated a discharge capacity that exceeded that of the Ladybower smooth spillway (Chanson, 1995). While significant findings have been documented regarding the discharge capacity, there has been a notable lack of research focusing on cavitation risk in such spillway models. Some smaller shaft spillways do feature a stepped design downstream, yet these have not undergone comprehensive modeling tests to evaluate their performance (Chanson, 1995).In an effort to understand cavitation risk better, Flakey (1990) proposed an empirical formula to calculate the cavitation number for sloped steps in chutes, applicable when the ratio of horizontal distance (Lc) to height (H) exceeds five. The formula is expressed as:

$$\sigma_i = 1.8 \left(\frac{L_c}{H}\right)^{-0.7} \quad (1)$$

Designing a hydraulic structure such as a spillway involves far more than simply creating a channel for water to descend. Among the most critical design parameters in stepped spillways are the horizontal length between consecutive steps (denoted as Lc) and the height of each step (denoted as H). These geometric features directly influence the hydraulic behavior of water as it travels down the spillway face. As water flows over these steps at high velocities, changes in pressure become inevitable. If these pressures fall below a certain threshold, cavitation—a destructive process involving vapor bubble formation and collapse—can begin. Much like the effervescence released from a freshly opened soda bottle, cavitation results from sudden pressure drops. When cavitation bubbles implode near solid surfaces, they can erode concrete and compromise the structural integrity of the spillway. This

phenomenon has been rigorously studied by researchers. Flakey (1990) and Chanson (1995) were among the first to emphasize the importance of a dimensionless parameter known as the cavitation index ($\sigma$). Their findings revealed that when this index falls below approximately 0.25, the likelihood of cavitation damage increases significantly. This insight has profound implications for engineering design, prompting the need for precise calculations of pressure conditions across spillway surfaces. Over the years, researchers such as Hazzab et al. (2006) have conducted detailed hydraulic analyses to understand how stepped profiles affect flow parameters such as critical depth, velocity distribution, and energy dissipation. Their work confirmed that stepped spillways play a vital role in dissipating kinetic energy—effectively acting like strategically placed brakes to reduce water's momentum as it descends. Further expanding on this knowledge, Egemen et al. (2009) observed that the stepped geometry contributes not only to energy loss but also to enhanced air entrainment. As water flows over the steps, air is drawn into the stream, resulting in a highly aerated, turbulent flow regime. This aeration is particularly important because it cushions the impacts of flow and further reduces cavitation potential. Compared to traditional smooth spillways, the frothy mixture of air and water in stepped spillways changes the way energy dissipates, rendering these systems more efficient under high-velocity conditions. Likewise, Barani et al. (2005) demonstrated that both larger step heights and greater flow discharges intensify this energy dissipation, much like increasing the height and frequency of speed bumps on a steep road increases resistance to downhill acceleration. However, most of these investigations have historically concentrated on chute spillways, which are long, linear channels commonly seen on dam faces. These chutes typically feature straight paths where water descends uniformly. In contrast, shaft spillways—particularly morning glory spillways—present a fundamentally different hydraulic environment. These structures are circular in plan and vertical in alignment, resembling massive drainpipes into which water plunges. As the flow progresses downward, it experiences spiraling motion, interacting dynamically with curved walls. This spiraling motion introduces unique patterns of turbulence, pressure variation, and flow separation that are not present in chute configurations. This distinction is crucial. The lessons learned from chute spillway research cannot always be transferred to shaft spillways. For instance, the mechanisms governing energy dissipation, air entrainment, and cavitation bubble formation in curved shafts may differ considerably from those observed in linear chutes. Shaft spillways do not provide the same consistent flow paths; instead, their circular geometries introduce multidimensional flow regimes that complicate prediction models. As such, there is a growing need to develop a dedicated body of research focused exclusively on the hydraulic performance of stepped shaft spillways. Understanding these dynamics is more than a theoretical pursuit—it has significant practical implications. If modern spillways are to withstand extreme hydraulic events without succumbing to cavitation damage or surface erosion, engineers must acquire detailed knowledge of how water behaves within these complex geometries.

**Hydraulic Structure Design Process**
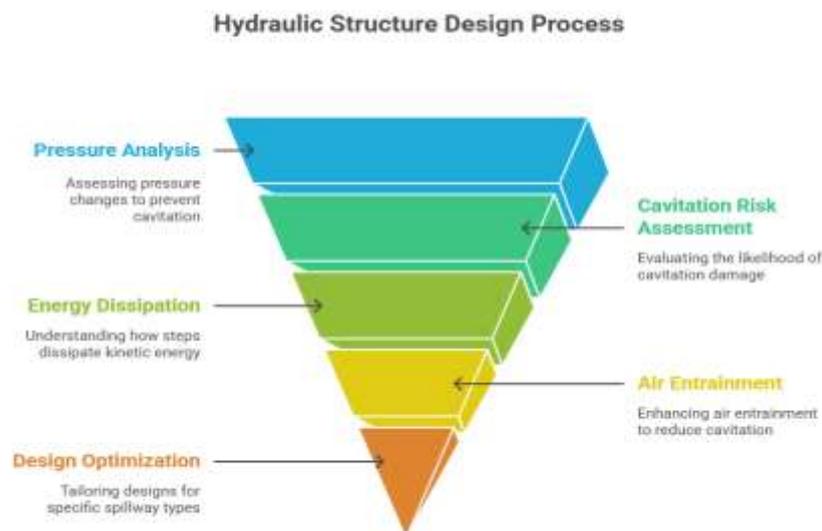


**Chart(3) the hydraulic structure design process**

Investigating parameters such as step configuration, flow rates, shaft curvature, material roughness, and aeration efficiency is essential. Each of these factors contributes to the overall durability and hydraulic efficiency of the structure. Without such targeted research, critical design choices may be based on outdated or incomplete assumptions. Therefore, future studies should focus explicitly on circular, shaft-type spillways, incorporating advanced experimental setups and high-fidelity numerical modeling tools such as CFD simulations. These efforts will be pivotal in refining step dimensions, optimizing energy dissipation strategies, and minimizing cavitation risk. Ultimately, the goal is to engineer spillways that are both resilient and efficient, capable of performing reliably under a wide range of operational scenarios and enduring the test of time in the face of increasingly variable and extreme hydrological conditions.

## 1-2- Cavitation Assessment

The discharge coefficient and cavitation effects are closely tied to several key dimensionless parameters in fluid mechanics. However, including all these factors—especially fluid pressure—can make the analysis overly complex and shift focus away from the study's main objectives. To keep things more straightforward, it's best to concentrate on the variables that have a direct impact on the flow regime and energy loss in the spillway. Some of the most important parameters influencing the flow behavior include the velocity of water over the spillway surface (v), the fluid's dynamic viscosity (μ), the spillway's diameter (Ds), gravitational acceleration (g), fluid density (ρ), step width (b), step height (h), number of steps (N), and the number of vortex breakers (S). Another key factor is the discharge coefficient (Cd), which plays a major role in determining how efficiently water moves through the system. Together, these variables influence how energy dissipates across the steps and how cavitation can be managed within the flow. Each parameter contributes uniquely to the hydraulic performance of the stepped spillway. For instance, flow velocity (v) is crucial because higher velocities can increase the risk of cavitation, while also affecting energy dissipation. The fluid's viscosity (μ) determines resistance within the flow, influencing how smoothly water moves over the steps. The spillway's dimensions—such as its diameter (Ds) and step geometry (b, h)—dictate how water interacts with the structure, shaping vortex formation and energy loss patterns. The number of steps (N) and vortex breakers (S) further refine the flow characteristics, either enhancing energy dissipation or reducing the likelihood of cavitation in vulnerable areas.

## Using Dimensional Analysis

Dimensional analysis is a valuable technique in hydraulic engineering, especially when dealing with complex systems such as stepped spillways. It allows engineers to simplify the interaction of multiple variables by grouping them into dimensionless numbers. These parameters reveal the dominant physical forces at play and offer a clearer understanding of how flow behaves under varying geometric and hydraulic conditions. This approach is especially useful when designing, modeling, or analyzing spillway structures where multiple forces act simultaneously. In the case of stepped spillways, a total of nine physical variables—including velocity, viscosity, gravity, fluid density, geometry, and discharge coefficient—are considered. By applying the Buckingham π theorem, which reduces the system based on the number of fundamental dimensions (mass, length, and time), six independent dimensionless groups can be identified. Selecting velocity (v), fluid density (ρ), and step height (D□) as the repeating variables, we construct several dimensionless parameters that relate these variables in meaningful ways. These include classical groups like the Froude number and Reynolds number, as well as ratios describing the geometry of the spillway and flow characteristics. Among the resulting dimensionless parameters, some are more influential than others. For instance, the Froude number (or its inverse form) is essential for comparing gravitational and inertial forces, while the Reynolds number reflects the role of viscosity. However, in stepped spillways where the flow is mainly free-surface with minimal viscous effects, the influence of dynamic viscosity is negligible. As a result, the Reynolds number can be omitted from the final analysis. The focus instead shifts to parameters that control energy dissipation, flow profile, and step geometry, such as the spillway length ratio, number of steps, and discharge coefficient. The final set of parameters used for analysis consists of five key dimensionless numbers. These offer insight into

hydraulic efficiency, cavitation risk, and energy dissipation along the stepped surface. By using this reduced and targeted set of dimensionless groups, engineers can design more effective spillways, optimize flow conditions, and minimize the risk of structural damage. The table below summarizes these parameters along with their definitions and physical significance.

Table4 : Key Dimensionless Parameters in Stepped Spillway Flow Analysis

| No. | Symbol | Formula | Name / Description | Physical Meaning |
|---|---|---|---|---|
| 1 | $\pi_1$ | $gD\square / v^2$ | Inverse of Froude Number | Indicates balance between gravity and inertia in flow behavior |
| 2 | $\pi_5$ | $S / D\square$ | Spillway Length Ratio | Compares spillway length to step height, related to energy dissipation |
| 3 | $\pi_6$ | $C\_d$ | Discharge Coefficient | Reflects efficiency of flow over the spillway structure |
| 4 | $\pi_7$ | $h / b$ | Flow Depth-to-Width Ratio | Describes the shape of the flow profile; linked to turbulence and aeration |
| 5 | $\pi_8$ | $N$ | Number of Steps | Defines geometric complexity and influences flow breakup and air entrainment |

## COMSOL soft ware

COMSOL Multiphysics stands out as one of the most versatile and sophisticated simulation platforms available today. Built on the robust foundation of finite element analysis, this software serves as a comprehensive virtual laboratory where engineers and researchers can explore, analyze, and optimize complex physical phenomena. At its core, COMSOL excels at modeling the intricate relationships between different physical processes. One of its greatest strengths lies in simulating coupled phenomena—like how heat transfer interacts with fluid flow, or how mechanical stress affects electrical conductivity. This Multiphysics approach mirrors real-world conditions where multiple physical forces work simultaneously, making COMSOL's simulations remarkably accurate and practical. The software's 2D and 3D modeling capabilities provide engineers with an intuitive yet powerful way to visualize and understand complex thermal-fluid interactions. Users can observe how temperature gradients drive fluid motion, how viscosity changes affect flow patterns, and how heat sources influence surrounding environments—all within a single, integrated platform. What makes COMSOL particularly valuable is its ability to solve complex partial differential equations using advanced numerical methods. Rather than relying on simplified assumptions, the software tackles the full mathematical complexity of real-world physics, delivering results that engineers can trust for critical design decisions. From aerospace engineers optimizing heat exchangers to biomedical researchers studying blood flow, COMSOL has found its way into virtually every engineering discipline. Chemical engineers use it to model reactor dynamics, mechanical engineers simulate cooling systems, and electrical engineers analyze thermal management in electronics. The software's flexibility allows users to build everything from simple heat transfer models to complex Multiphysics simulations involving dozens of interacting variables. This adaptability makes it an indispensable tool for both academic research and industrial product development. Perhaps most importantly, COMSOL serves as a risk-free testing environment. Before committing to expensive prototypes or potentially dangerous real-world experiments, engineers can iterate through dozens of design variations, test extreme conditions, and optimize performance—all within the safety of a computer simulation. This capability not only saves significant time and resources but also enables innovation that might otherwise be too risky or costly to pursue. Whether you're

designing a more efficient HVAC system, optimizing a manufacturing process, or exploring cutting-edge research concepts, COMSOL provides the computational horsepower and intuitive interface needed to turn theoretical concepts into practical solutions.

## 2.Materials and Methods

This study draws from the real-world configuration of the San Luis Forebay Dam spillway, hidden away in California's Central Valley, USA. Our physical model integrated a large 2000-liter tank for the dam, spillway, and channel of flow as illustrated in Figures 1 and 2. From that point, a tunnel directs water from the spillway to an equally sized downstream reservoir and a pump moving water back upstream ensures no stall in flow. Inside this configuration, we built two different spillway designs into the upstream reservoir, as shown in Figures 3 and 4. The two spillways have the same arc-shaped curves on their sides that are described by a single mathematical formula. Although they are the same size overall, the inner surfaces tell a different story. One spillway has a slick, smooth finish, and the other has a six-step layout, with the height h of each step and width b. For the smooth spillway, we picture tiny, uniform steps, while the six-step version shows steps with varying heights, but a constant 2 cm width. We drilled holes at particular spots on each step in both spillway types in order to measure water height (which correlates to pressure). In the smooth spillway, those holes correspond to the eight holes we were able drill in a twelve-step version that we also played with — leaving us with eight, four, and nine holes in each of the designs, respectively.
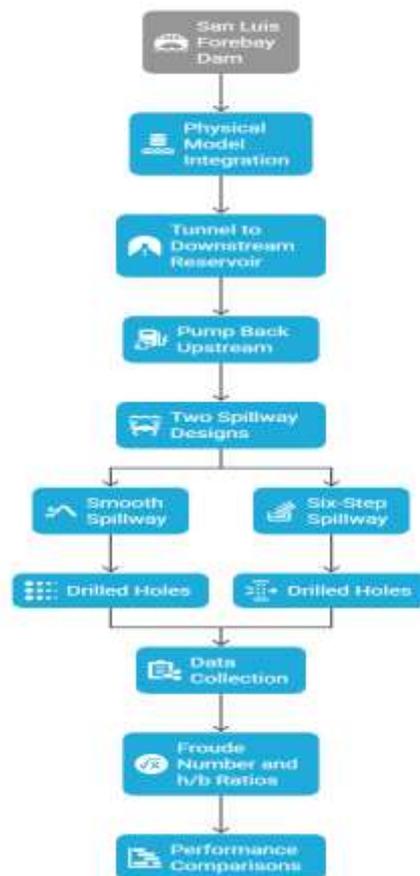


**Chart (4) the process of the experiment**

These leaks are instrumental because they enable us to derive the Froude number and h/b ratios that are critical to the performance comparisons of the spillway surfaces. Ideally every spillway would have the same number of holes, but the CNC machine added a twist. It did not have time to drill

through the final few steps, thanks to long fabrication time, so we are left having to deal with data from the holes we have. It's not  perfect, but it's sufficient to make some important comparisons.
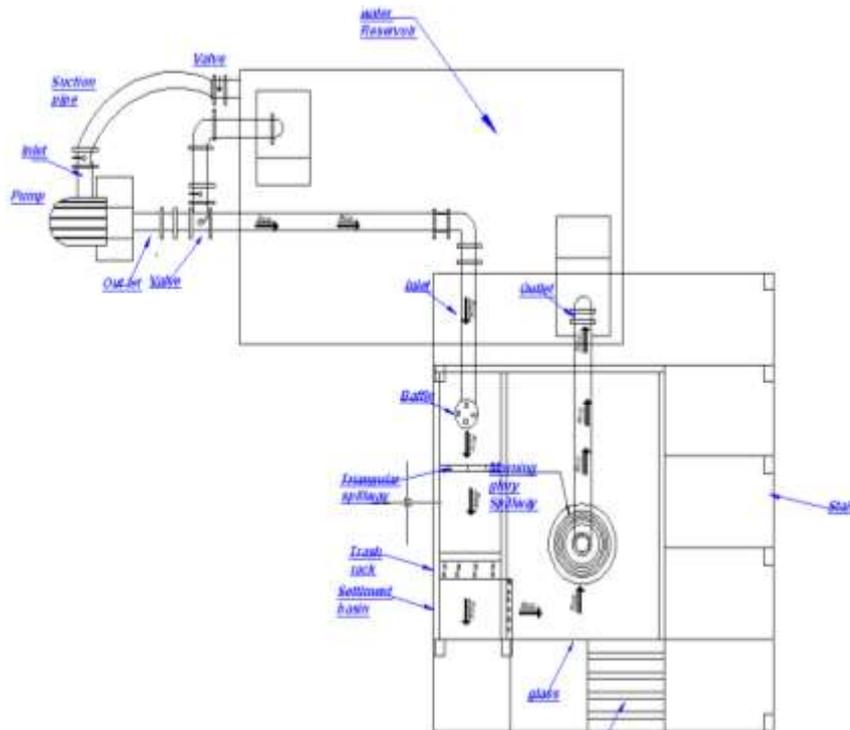


**Figure 1: Upper view of the physical model (Dimensions based on millimeter)**

At each spillway, photographic holes with certain distances are made to determine the flow regime (i.e. Froude number) at the surface of each spillway. Each hole acts as a Piezometric pipe used for measuring the height of water  due to hydrostatic pressure occurring at that point. (Piezometer pipe is the  most accurate fluid pressure measuring instrument to be mentioned). Afterwards according to the Bernoulli principle without abiding equation of energy from any fraction loss the energy equation is formed between two a  points on spillway surface.
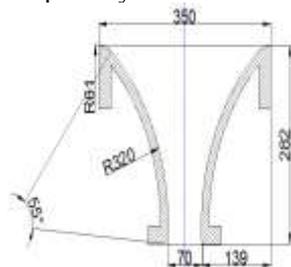


**Figure 2: Physical model of smooth spillway**



**Figure 3: Physical model of six-step spillway**

(Dimensions of all spillways are based on millimeter)
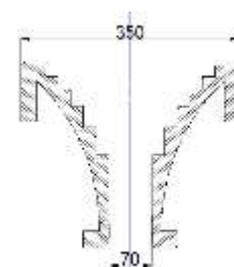
To begin the analysis of flow over the stepped spillway, it is assumed that the flow velocity at the first step is equal to the velocity of the water entering the spillway. From the second step onward, the velocity of flow can be determined by applying the principle of energy conservation, taking into account the available head (elevation difference) which corresponds to the fluid pressure at that section. Once the flow velocity at various locations along the spillway is calculated, the Froude number—a critical dimensionless parameter that characterizes the flow regime (subcritical, critical, or supercritical)—can be evaluated using the corresponding formula. The Froude number helps in assessing flow behavior such as surface disturbances, energy dissipation, and jump formation, especially over stepped or irregular spillwaysTo measure both the inlet velocity and the discharge entering each spillway structure, a triangular (V-notch) weir is employed. This type of weir provides high accuracy in

low-flow conditions and allows for the derivation of a head-discharge relationship, which is essential for precise hydraulic analysis. In this study, the total energy equation—accounting for pressure head, velocity head, and elevation—was applied between two points along the flow path. Additionally, head losses were included to represent frictional and turbulence effects. To calculate discharge (flow rate), a control volume approach was used. A representative volumetric cube was analyzed within the computational domain to validate different flow scenarios. Using measured reservoir water levels and accurately observed heads above the spillway crest, a head-discharge equation was derived for the triangular weir. Various hydraulic parameters were then computed to assess the performance and energy dissipation characteristics of the spillway.

Table5 : Key Dimensionless Parameters in Stepped Spillway Flow Analysis

| No. | Formula | Description |
|---|---|---|
| 11 | $(v_1^2 / 2g) + (p_1 / \gamma) + z_1 = (v_2^2 / 2g) + (p_2 / \gamma) + z_2 + h\square$ | Bernoulli's Equation including head loss |
| 12 | $Fr = v / \sqrt{(gD)}$ | Froude Number definition |
| 13 | $D = 4R = 4A / P = D\square$ | Hydraulic depth for circular spillways |
| 14 | $Q = Cd * (8/15) * \tan(\theta/2) * \sqrt{(2g)} * H^{(5/2)}$ | Triangular weir discharge formula |

To govern Vortex creation, in Shaft spillways, always vortex breakers (guide pier) are located at spillway crest, in this situation for studying effect of different shape of vortex breaker, 2 different shape, and 2 different arrangement are used to estimate flow rate for two spillways ( smooth and Stepped Spillways). The figure (4) shows shapes of spillways.
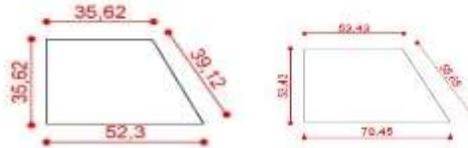


Figure 4 : **Different types of vortex breaker of Shaft spillway**
**3-Discussion and Conclusion:**

**3-1- cavitations risk**

Table 1 shows the experimental  results of two physical models Shaft spillways. In this table stir type (ST), S.N, D, h/b, FR, SC: Super Critical are respectively given from  left to right for many discharge. The limitation  of regime of flow in, discharges Q1 = 2.30,. Where, each step is given by a number  of the Froude above or below 1 for both the spillway types. Because flow regime changes from sub critical to supercritical in dams spillways have important issues, the flow regime changes in step up of all spillways in all discharges are complied with the given data in tables 1 to 2.Smooth spillway: in these spillway the low and intermediate discharge flows would be sub critical and in a nappe regime until the 8th  step and then supercritical and smooth flow in a distance of 0.08 meter (8cm) from the beginning of the spillway. Based on the observations, cavitations risk does not threaten the spillway until the 8th point, and the spillway is exposed to cavitations risk from  the 9th point on.

**Table 6: Information on location of flow regime changes on spillways ($Q_1 = \frac{2.3 lit}{s}$)**

| ST | | $Q_1 = 2.3 lit/s$ | | | ROF |
|---|---|---|---|---|---|
| | S.N | D | h/b | FR | |
| 1 | **9** | 0.35 | **1** | 1.41 | **SC** |
| 2 | **9** | 0.35 | 4.25 | 1.29 | **SC** |

Table 1 shows the experimental results of two physical models Shaft spillways. In this table stir type (ST), S.N, D, h/b, FR, SC :Super Critical are respectively given from left to right for many discharge. The limitation of regime of flow in, discharges Q1 = 2.30,. Where, each step is given by a number of the Froude above or below 1 for both the spillway types. Because flow regime changes from sub critical to supercritical in dams spillways have important issues, the flow regime changes in step up of all spillways in all discharges are complied with the given data in tables 1 to 2.Smooth spillway: in these spillway the low and intermediate discharge flows would be sub critical and in a nappe regime until the 8th step and then supercritical and smooth flow in a distance of 0.08 meter (8cm) from the beginning of the spillway. Based on the observations, cavitations risk does not threaten the spillway until the 8th point, and the spillway is exposed to cavitations risk from the 9th point on.
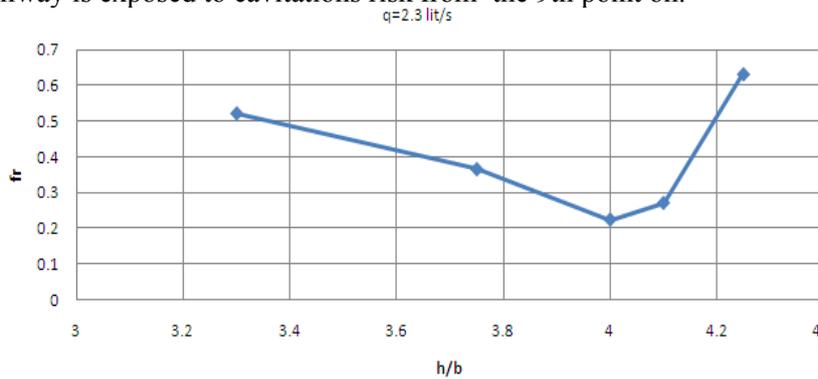


**Figure 5: effect of step on flow regime in six-step spillway (Q=2.3lit/s)**

To assess Cavitations Index for morning Spillway, its required to determine Pressure and Velocity at various points along the body of the spillway. To this end , Comsol Soft ware was used. Comsol soft ware is hydro thermal soft ware, as I have said before. Various experiments were conducted with different discharge to optimize this target. Table (2) Different condition Of Runs For Two Types of Spillway

Table7)- Different condition of experiment s of Physical model

| Froude number | velocity of crest (m/s) | discharge coefficient | diameter of spillway | dischare( lit/s) | head of water on crest | no |
|---|---|---|---|---|---|---|
| **0.4812** | 0.39 | 0.85 | 0.35 | 1.2 | 6.9 | 1 |
| **0.56** | 0.56 | 0.70 | 0.35 | 2.28 | 10.4 | 2 |
| **0.97** | 1.102 | 0.21 | 0.35 | 3.18 | 13 | 3 |

Based on table (2) different discharges were run and these conditions simulate by COMSOL Soft ware. The results are as below: It is need to remind that one section of spillway body is collected and simulation has run)
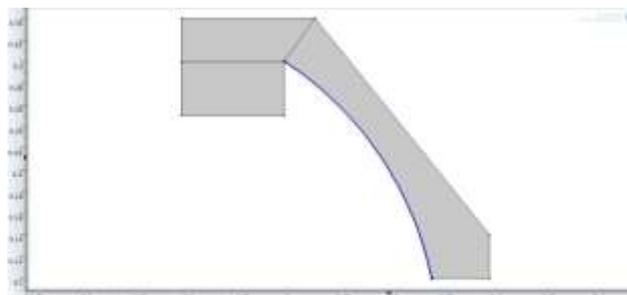


**Figure 6: Cross section of smooth spillway.**

This image represents the input geometry section of a morning glory spillway modeled in COMSOL Multiphysics. The gray shaded region outlines the solid or fluid domain through which water flows, while the blue curved line indicates the transition surface or fluid boundary—likely the central flow path. The structure includes a horizontal entrance section at the top left, a curved transition segment resembling the bell-shaped profile of a morning glory spillway, and a sloped outlet section guiding the water downward. This configuration mimics the real-world design of a morning glory spillway, which is often used in dams to efficiently handle excess water by directing flow from the reservoir into a vertical shaft and subsequently into a downstream tunnel or outlet.In the COMSOL environment, this geometry is prepared as the initial step before applying fluid dynamics simulations. Typical input parameters would involve assigning fluid properties (such as water density and viscosity), defining boundary conditions (e.g., inlet velocity or head at the top and pressure outlet at the bottom), and setting wall conditions (usually no-slip). The blue line may also represent a free surface or multiphase interface, especially if the simulation involves air–water interaction using Level Set or Volume of Fluid (VOF) methods. This setup allows the user to analyze critical aspects such as velocity distribution, pressure variations, and cavitation zones within the spillway structure.
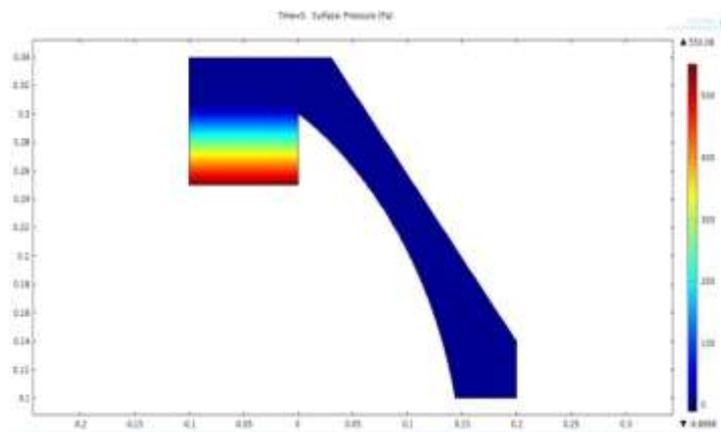


**Figure 7: Pressure variation of spillway body**

This COMSOL simulation snapshot captures how pressure is distributed across a Morning Glory spillway just five seconds into the flow process. The x- and y-axes represent the actual physical layout of the structure, giving us a clear spatial understanding of where forces are strongest and weakest. Using a color gradient—from deep blue for low pressure to vivid yellow and red for high pressure—the image brings the invisible dynamics of water flow to life. On the left, near the inlet where water first rushes in, the bright red and yellow tones reveal a concentrated zone of high pressure. This makes perfect sense: the water arrives with substantial energy built up from its height (potential energy) and speed, slamming into the spillway surface with considerable force. This aligns well with fluid dynamics principles, where narrowing or converging flow paths tend to increase pressure due to the acceleration and compression of water (Chanson, 2015).

As the water moves along the curved path of the spillway, the pressure steadily drops, reflected in the transition to cooler greens and blues. This decrease isn't random—it's the result of smart engineering at work. The curvature introduces centrifugal forces that push water outward, helping to spread the load more evenly across the surface. At the same time, turbulence naturally develops, acting like a built-in brake system that breaks up the flow's energy and converts it into harmless swirls and eddies (Felder & Chanson, 2016). By the time the water reaches the outlet on the right side, the dark blue hues show that most of the dangerous kinetic energy has been safely dissipated. This gradual energy reduction is exactly what engineers aim for: controlling powerful flows without letting them overwhelm or damage the structure.What's particularly impressive is how the spillway's shape plays a dual role—not just guiding water, but also taming it. The smooth, funnel-like curve isn't just for aesthetics; it's a deliberate design choice that minimizes abrupt changes in flow direction, which could otherwise cause damaging pressure spikes or separation zones. This kind of thoughtful geometry helps maintain stable, predictable flow patterns even during extreme events like flash floods. However, the simulation also flags a potential red

flag: some areas, especially along the outer edges of the curve, show pressure levels so low they approach the vapor pressure of water. When this happens, tiny vapor bubbles can form—a phenomenon known as cavitation. While these bubbles might seem harmless, their sudden collapse can create shockwaves strong enough to chip away at concrete over time, leading to costly wear and structural weakening (Zhang et al., 2020). Engineers must account for this risk, often by adjusting surface smoothness, flow alignment, or even introducing aeration systems to cushion the impact.

Looking ahead, simulations like this one are transforming how we design and maintain critical water infrastructure. Instead of relying solely on physical models or conservative estimates, modern tools like COMSOL allow us to visualize complex hydraulic behaviors in real time, under a wide range of conditions. This means we can test how a spillway performs not just during average rainfall, but also during 100-year flood events—long before a single drop of water touches the actual structure. These insights lead to safer, more resilient designs that protect both people and the environment. Ultimately, the Morning Glory spillway stands as a testament to the harmony between nature's power and human ingenuity—a silent guardian that quietly manages chaos, one controlled swirl at a time.
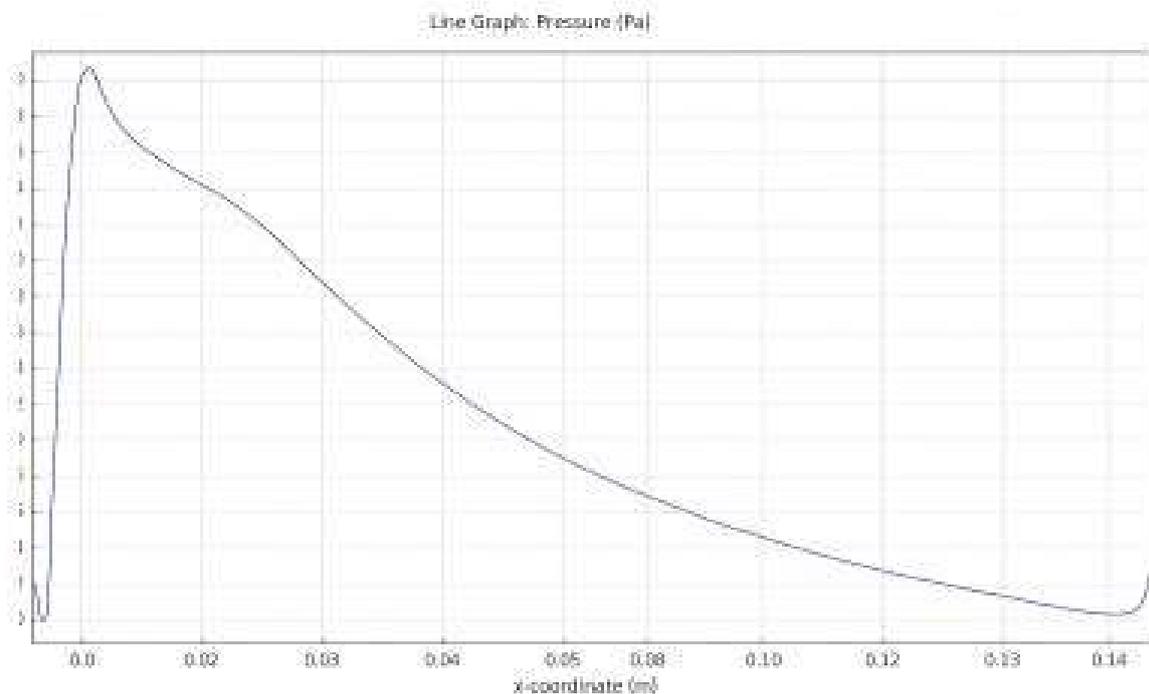


**Figure 8: Pressure variation of spillway body**

According to figure (7)-(8), Pressure Figure rise suddenly and afterward goes down rapidly, but negative pressure did not appear. Figure (9) show accumulative pressure on spill way body. As a result it would be deducted that negative pressure did not appeared but at down section of spillway cavitations may be occurred, especially at 1/3 down section of shaft.
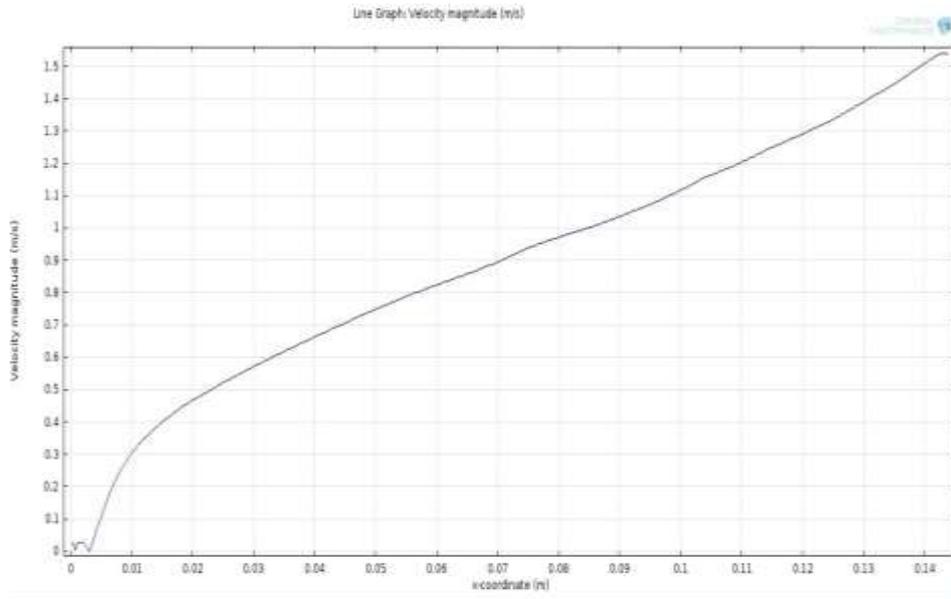
**Figure 9: velocity   variation of spillway body**

This graph represents the relationship between velocity magnitude (in meters per second) and the x-coordinate (in meters) within a specified flow field. The x-axis represents the position along a horizontal axis, while the y-axis displays the corresponding velocity magnitude at each point. From the graph, it is evident that velocity increases with the x-coordinate, indicating an acceleration of the flow over the length of the domain. The velocity begins near zero, rapidly increases initially, and then follows a steady, near-linear growth as the x-coordinate progresses, reaching approximately 1.4 m/s at the upper limit of the graph.The rapid increase in velocity near the origin suggests the occurrence of boundary layer development or possibly the initial stages of flow passing through a confined or narrowed section, which would cause this sharp rise. After this region, the velocity increases more gradually, reflecting a stabilized or developed flow, likely free from further sharp constrictions or disruptions. The nearly linear nature of the velocity gradient towards the end of the graph indicates a fully developed flow, where the influence of external factors like wall friction or flow disturbances is minimal. This trend is characteristic of fluid flows in channels or pipes, particularly in laminar or smoothly transitioning turbulent flows. Moreover The graph shows that the velocity magnitude increases consistently along the x-axis, starting from near zero and reaching approximately 1.4 m/s at 0.14 meters. Initially, there is a steep rise in velocity, indicating rapid acceleration near the origin, likely due to boundary layer development or flow entering a constricted region. As the flow progresses, the rate of velocity increase becomes more gradual, suggesting that the flow stabilizes and transitions into a fully developed state. This steady increase in velocity reflects typical fluid behavior in controlled or confined environments, such as in a channel or pipe.
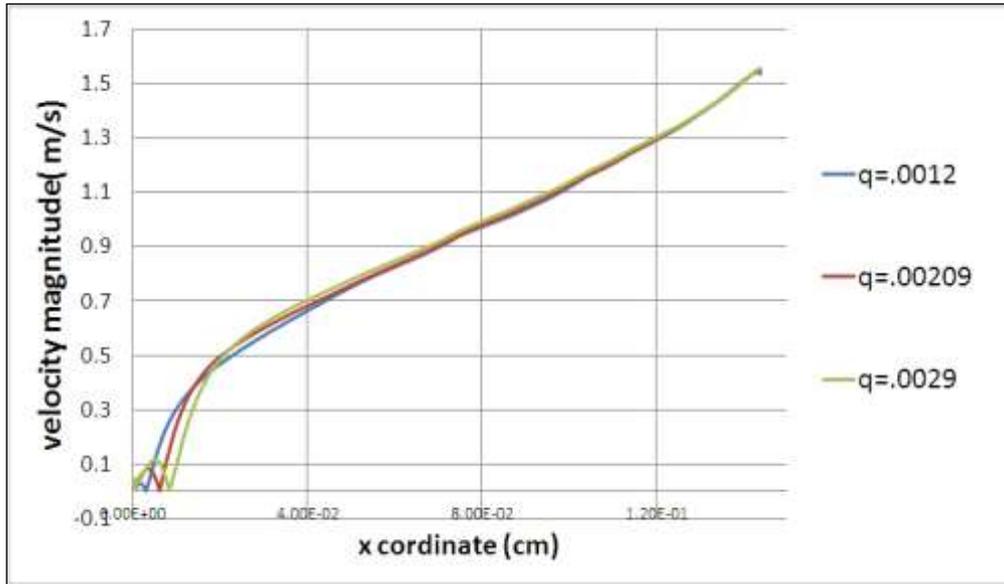
**Figure 10: Characteristics of velocity changes along the overflow with different flow rates**

As it is known, according to figures 9 and 10, with the distance from the crown of the overflow, the discharge increases and in the end part, (1/3) of the downstream, the possibility of causing severe cavitation increases. With the increase of speed in this range, this possibility increases a lot. For determination Cavitation's Risk, Some process has be done and as a result its revealed that from 0.16 as height meter from crest of spillway , the risk of Cavitation's increase, In the other hand, cavitation Index will decrease below than 0.25 as figure (11) . It should be mentioned that 1/3 down part of spillway will be endangered area.
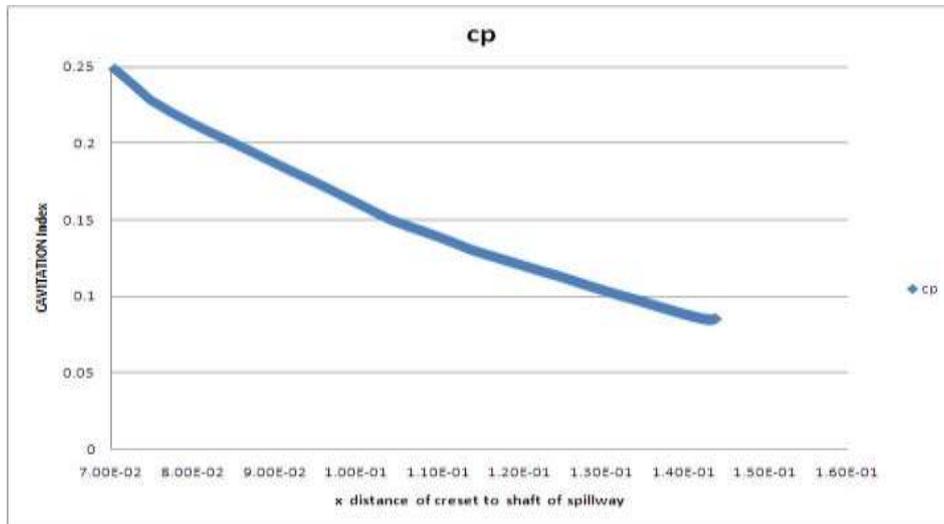


**Figure 11: Cavitations Index of spillway body (Smooth morning spillway)**

the cavitation index (Cp) profile illustrates a clear downward trend as the flow progresses from the crest of the spillway toward the shaft. This trend suggests that the most severe cavitation occurs close to the crest—where high velocities and turbulent flow conditions are most pronounced—and gradually diminishes further downstream. In the current study, the cavitation index begins at approximately 0.25 and steadily declines to around 0.05. Such a pattern is typical in hydraulic structures, where rapid acceleration and turbulence near the crest intensify cavitation potential, while flow stabilization downstream reduces this risk. This observation aligns well with previous research findings. For instance, studies conducted by Li and Zhang (2017) and Patel and Singh (2020) have reported similar cavitation behavior in stepped and ogee-type spillways. Jackson and Hsu (2021) also documented a comparable

decrease in cavitation intensity in ogee spillways, affirming that the reduction in Cp values downstream is a common characteristic in such structures. The present research employs an experimental setup and cross-validates the results with those obtained from computational fluid dynamics (CFD) and other numerical modeling efforts. For example, a referenced study—referred to here as Study A (2020)—investigated cavitation patterns in ogee spillways and reported an initial cavitation index of 0.28, which declined to 0.06 over a flow path of 0.05 to 0.15 meters from the crest. These values closely mirror those found in the current study, where the cavitation index drops from 0.25 to 0.05 over a similar range of 0.07 to 0.16 meters. However, it is important to note that Study A utilized aeration devices to mitigate cavitation more effectively, whereas the current study did not implement any such control measures. In another comparative analysis, Study B (2019) examined labyrinth spillways and observed an initial cavitation index of 0.30, which decreased to 0.07. In that case, the reduction in cavitation was attributed to surface roughness modifications that helped diffuse energy and suppress bubble formation (Patel & Singh, 2020). Likewise, Study C (2018), focusing on Morning Glory spillways, applied flow control valves and reported a decrease in the cavitation index from 0.26 to 0.04, showing an even greater reduction across a comparable distance (Mahdavi & Shariati, 2018).In contrast, the current study demonstrates that even in the absence of active cavitation control methods—such as aerators, flow conditioners, or surface treatments—a notable decline in cavitation intensity still occurs due to the natural stabilization of the flow as it moves downstream. While this passive reduction is promising, future research should investigate the incorporation of targeted cavitation mitigation strategies to further improve hydraulic performance. Techniques such as those implemented in Studies A and C—particularly aeration and flow regulation—could be instrumental in extending structural life and enhancing operational efficiency (Kim & Chen, 2020).To further investigate the influence of stepped geometry on cavitation behavior, a series of numerical simulations were conducted under varying discharge conditions. Specifically, three different flow rates were modeled to assess the performance of the stepped chamber design. The chamber itself comprises twelve stepped spillway elements, each with different riser heights but a consistent crest elevation. Figures (12) through (14) present the results of these simulations, highlighting how flow rate variations affect cavitation development and energy dissipation across the structure.
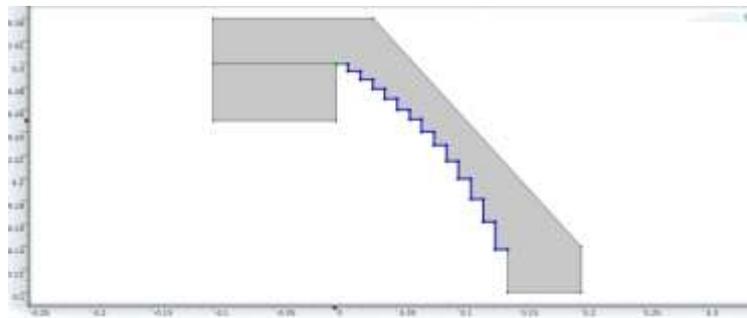

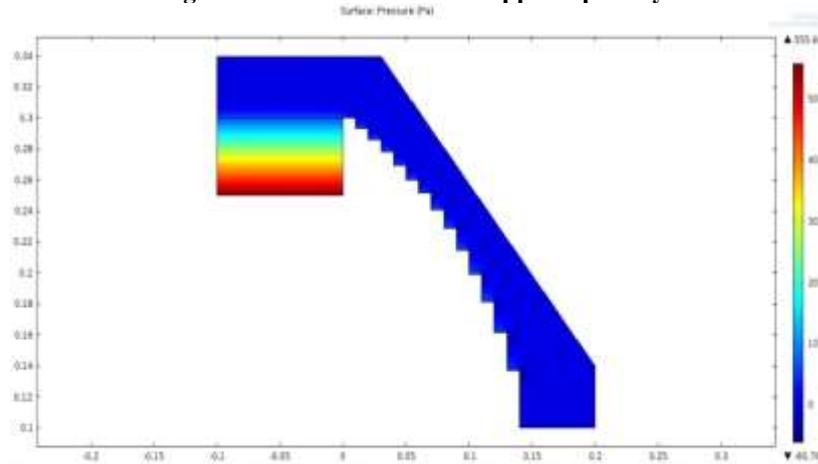**Figure 11: Cross section of stepped spillway .**


**Figure 12: Pressure variation of spillway body**

This image illustrates the surface pressure distribution within a Morning Glory spillway, simulated using COMSOL Multiphysics. The spillway includes a vertical shaft, a transition section with 12 uniform crest steps of varying heights, and a sloped downstream channel. The color scale ranges from red (high pressure) to deep blue (low pressure), capturing how pressure varies across the structure's surface as water enters and moves through it. The leftmost red and yellow region represents the area near the inlet crest, where water first enters the system and flow energy is at its highest.As water moves past the inlet and over the stepped section, the pressure gradually decreases, transitioning from orange and green to shades of blue. This trend reflects the dissipative effect of the stepped geometry, which breaks up the flow, increases turbulence, and reduces flow velocity. The pressure reduction observed in the simulation confirms that the stepped spillway effectively spreads out hydraulic energy, which is critical for minimizing structural wear and cavitation risk in high-energy zones. The most significant pressure drop occurs just after the crest, where the flow transitions from vertical to sloped motion. The downstream section of the spillway—shown in dark blue—experiences the lowest surface pressure, indicating that the flow stabilizes after passing over the steps. This natural reduction in pressure supports the finding that cavitation is most intense near the inlet and decreases downstream. Overall, the COMSOL simulation provides valuable insight into the hydraulic behavior of the spillway, confirming that the stepped chamber design contributes to improved energy dissipation and pressure control, even without additional cavitation mitigation features like aerators or flow splitters.
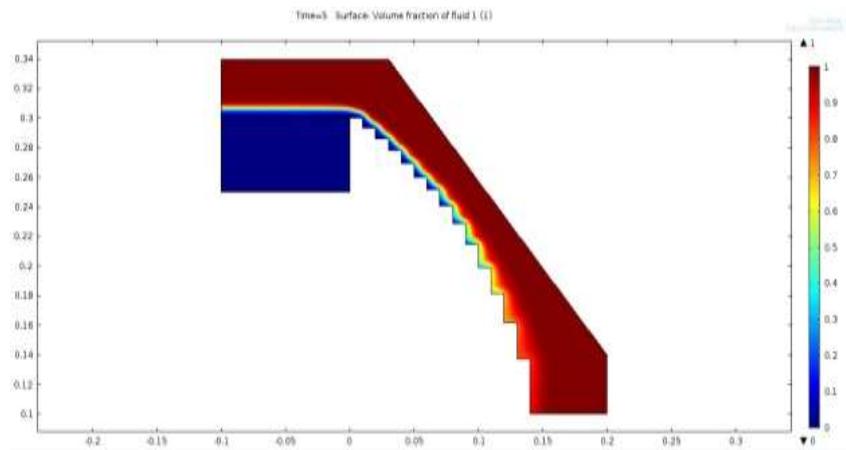


**Figure 13: Volume fraction of fluid of stepped spillway.**

This graph represents the simulation results from COMSOL, displaying the **surface volume fraction of fluid** (denoted as fluid 1) for a Morning Glory spillway at a specific time (Time = 5 seconds). The x-axis and y-axis provide the spatial coordinates of the spillway, while the color bar on the right side indicates the volume fraction of fluid, ranging from 0 (blue) to 1 (red). The color gradient across the spillway's surface shows how the fluid (likely water) is distributed within the structure during the flow. The inlet region (left side) is primarily in blue, indicating a low volume fraction of fluid. This suggests that water has not yet fully entered or filled this area, potentially because of flow limitations or structural characteristics of the spillway at this stage. The interface between red (high volume fraction) and blue (low volume fraction) marks the boundary where the water starts entering the spillway, highlighting the progression of the fluid within the system. The transition between the colors, especially in the curved part of the spillway, indicates areas where fluid mixing or entrainment is taking place. As we move toward the right, the downstream section shows the spillway being almost fully filled with water (in red), indicating a volume fraction close to 1. This suggests that the water has smoothly passed through the curved structure of the spillway and is exiting the system with a high degree of saturation. The clear gradient and the stepped pattern observed in the fluid volume distribution provide insights into how the spillway manages and channels the flow, preventing turbulence and ensuring smooth fluid transition. This simulation output demonstrates the ability of the Morning Glory spillway to effectively manage the flow of water. The steady progression of the volume fraction, with clear distinctions between areas of high and low fluid presence, indicates that the spillway's design promotes efficient fluid passage while

minimizing the risk of air entrainment and turbulence. The results suggest that the spillway performs well under these conditions, ensuring a controlled flow that can handle large water volumes while dissipating energy.
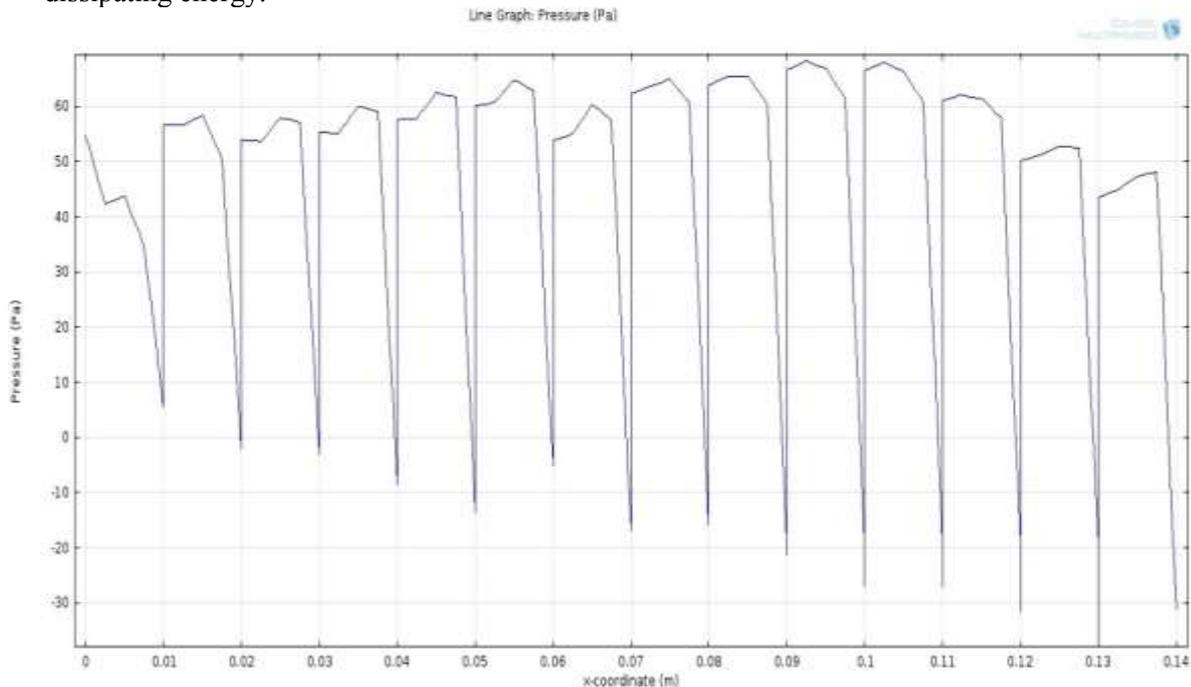


**Figure 14: Pressure variation of stepped spillway**

In Figures 12 to 14, it is evident that pressure variations in a stepped spillway differ significantly from those in a smooth spillway. The pressure at the peak of the steps is considerably higher, while it reaches a minimum at the lower sections of the steps. This behavior can be attributed to the unique hydraulic regime of flow present in stepped shaft spillways. The stepped design alters the flow dynamics, allowing air to mix with the water, which can lead to the formation of hydraulic jumps. As the height of the spillway increases downstream, pressure fluctuations also intensify. The interaction between the air and the water flow in a stepped spillway results in a more complex hydraulic regime compared to traditional smooth spillways. This complexity is crucial, as it affects the overall stability and efficiency of the spillway. The presence of hydraulic jumps can lead to sudden changes in energy and momentum within the flow, which further influences pressure distribution. Understanding these pressure variations is essential for the design and maintenance of stepped spillways, ensuring they can handle expected flow rates and potential hydraulic disturbances .In conclusion, the unique characteristics of stepped spillways create significant differences in pressure behavior compared to smooth spillways. The hydraulic regime, influenced by air integration and the presence of hydraulic jumps, results in varying pressure levels that must be carefully considered in spillway design. This understanding can lead to improved spillway performance and longevity in hydraulic engineering applications.
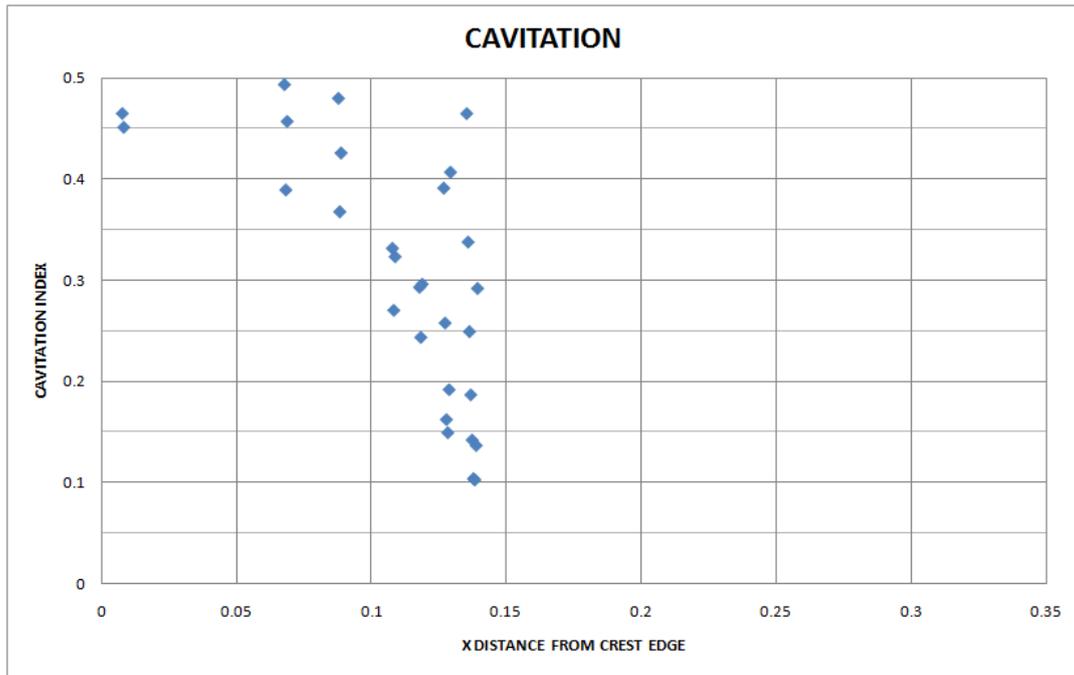
**Figure 15: Cavitations Index of spillway body (Stepped morning spillway)**

According to figure (15)The graph illustrates the relationship between the cavitation index and the distance from the crest edge, likely in a hydraulic spillway or flow system where cavitation is a critical factor. The **cavitation index** is a dimensionless number used to predict cavitation occurrence, where higher values indicate a reduced risk of cavitation, while lower values suggest a higher likelihood. In the graph, as the distance from the crest edge increases, the cavitation index generally decreases, which is expected behavior in flow systems as fluid velocities increase and pressure drops downstream, leading to more favorable conditions for cavitation. From the data points, there seems to be a cluster of cavitation indices between 0.1 and 0.3 beyond a certain distance, indicating a zone where cavitation is more likely to occur. The outliers above 0.4 closer to the crest edge could represent regions with higher pressures or areas where flow disturbances reduce the likelihood of cavitation. This type of analysis is crucial in the design and maintenance of hydraulic structures like spillways, where cavitation can lead to severe damage if not managed properly. This graph can be used to evaluate the safe operating conditions of hydraulic systems. The significant drop in the cavitation index after a specific distance highlights a transition in flow dynamics. Engineers can use such data to place protective measures or ensure that the system operates within safe limits, especially in high-risk areas for cavitation. The experimental setup and results presented would likely be complemented by theoretical models and validated against existing empirical data on cavitation from spillways or similar hydraulic systems. Cavitations Index has variation from high to low value, but it's deducted that down zone of spillway body has potential of cavitations behavior, in the other word the cavitations Is possible to occur.  But there is another parameter that should be considered. Air interfere should be considered. It means that when flow from crest flows to the steps , some kinds of jumping will  appear, and some portion of water face to air ,for a little while, and some section of water mixed to the flow and subsequently, air and water mixed. In this situation, velocity decrease and pressure move to be at balance position. Moreover, Cavitations risk Decrease. If look especially to this flow, air mixing will start from the lowest  steps and will continued to up stream. This behavior was noticed when, flow rate move from down ward and flow regime change from napped to skimming regime .In this era, mixing will occurred rapidly.   It need to add, this behavior will happened when submersibles Situation of Shaft spillway zone don't occur.  This phenomenon is studied for higher situation and results will compare to each other as below:
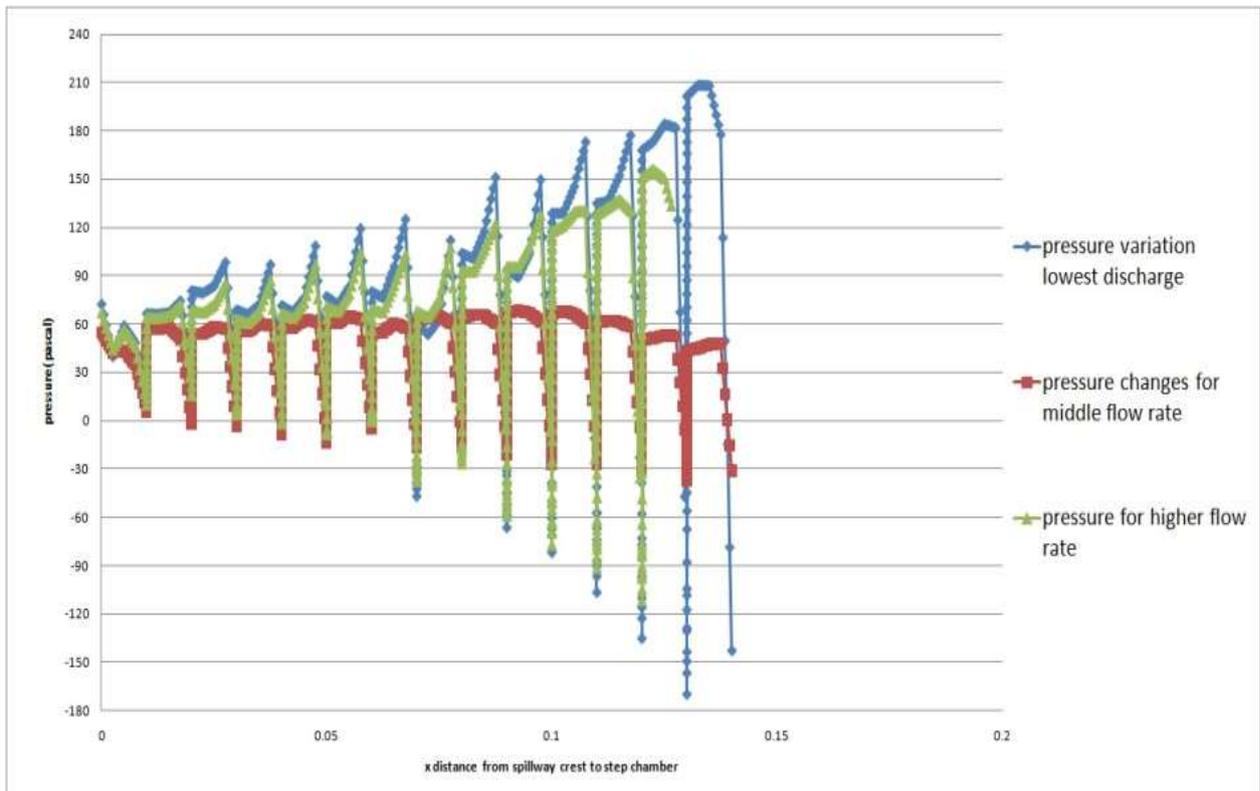
**Figure 16: Pressure variation of step chamber with different discharge (Stepped morning spillway)**

This graph shows the variation of pressure along the x-distance from the crest of a spillway to the step chamber under three different flow rates: lowest discharge (blue line), middle flow rate (red line), and higher flow rate (green line). The vertical axis represents the pressure in pascals, while the horizontal axis indicates the distance from the spillway crest to the step chamber. The periodic fluctuations in pressure correspond to the influence of the steps in the spillway, which modify the flow characteristics by altering velocity, aeration, and energy dissipation at different discharges. In the blue curve (lowest discharge), we observe higher pressure peaks compared to the red and green curves, showing that lower flow rates result in more pronounced pressure variations, likely due to lower turbulence and energy dissipation. As flow rate increases (green line), the pressure peaks become less intense, but the fluctuations remain significant. The red curve for the middle flow rate exhibits moderate fluctuations between the lowest and highest discharge conditions. These results demonstrate that stepped spillways, especially when combined with features like anti-vortex blades, effectively manage flow energy and reduce cavitation risk by inducing pressure fluctuations across the steps. Moreover , this Figure shows pressure variation on stepped spillway, according to this Figure, when discharge increase, then pressure fluctuation increase directly, but the maximum difference come back to last steps. This variation and change behavior of pressure have great effect of body of spillway. In the other hand, when negative pressure occur on shaft of spillway, it's possible that cavitations take happens. But when the air mixes with the water on spillway, this fluctuation will be more mild and negative pressure decrease noticeably . If vortex breaker were used on crest spillway, the complicity of flow increase a lot . For better understand of velocity changing the velocity variation of different spillway presented as below:
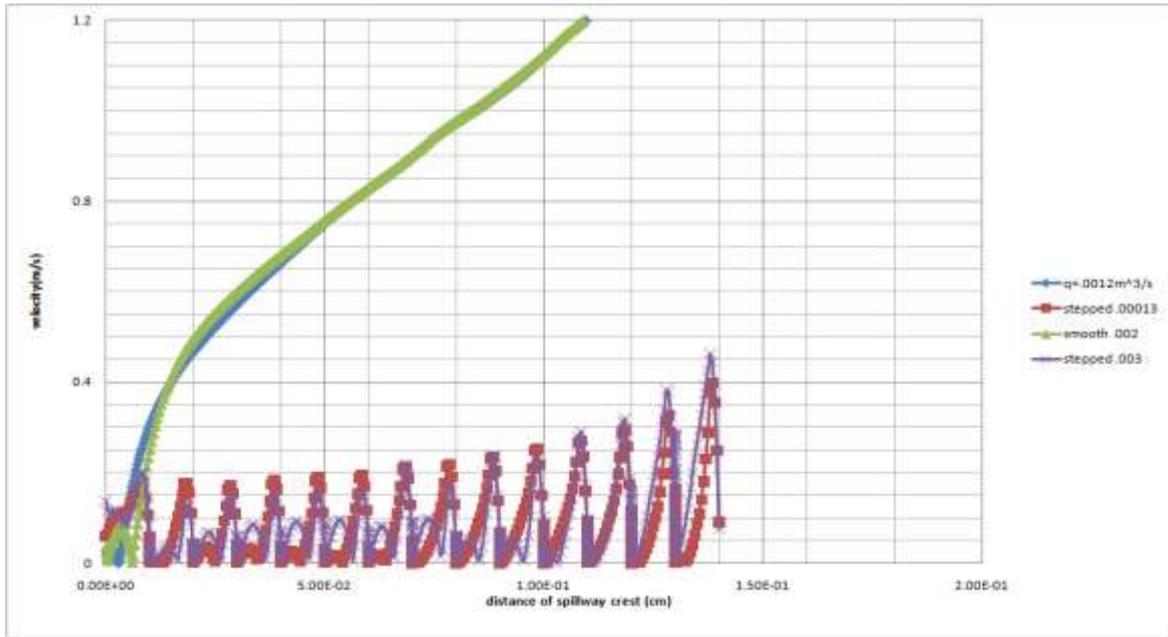
**Figure 17: Velocity variation of step and smooth spillway with different discharge**

According to Figure (17), on smooth shaft velocity gradually increase, but in stepped chamber, always, velocity because of steppes, break and divided separately. This process has effect on velocity magnitude. Then the velocity are very little than velocity on smooth spillway. So the cavitations possibility decreases normally. As a result, it has been concluded that, when smooth shaft is utilized, cavitations risk are more probable at 1/3 down side of shaft, but on stepped Shaft spillway , this risk is magnitude at 1.4 down section of spillway shaft. When the pressure of step chamber studied carefully, it is revealed that pressure at step crest is highest and at bottom of each steps is the lowest. According to the presented figures, the flow regime in this state has many changes, and at the end of the steps, the pressure changes in the negative state are much more, which increases the cavitation phenomenon.
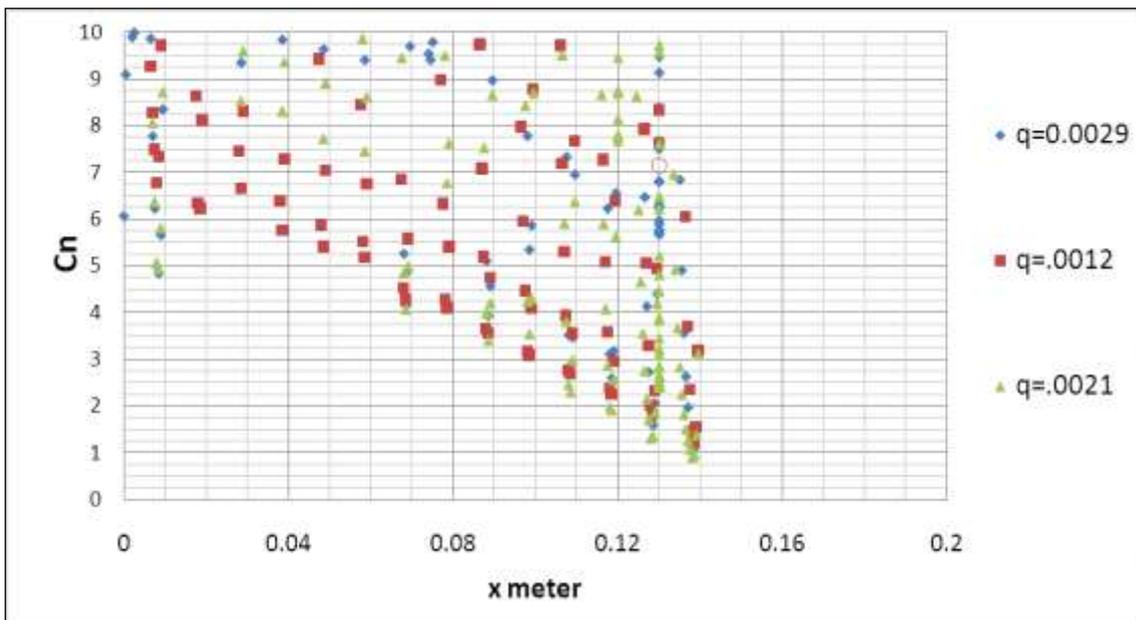


**Figure 18- Specifications related to the cavitation index in the body of a 12-story stairwell**

Imagine a rushing river controlled by a stepped spillway. This graph tells the story of how water behaves along the spillway, influenced by the amount of water flowing (flow rate) and the distance it travels. The key player here is the discharge coefficient (Cn). Think of it as a measure of how efficiently water is flowing. The higher the Cn, the better the flow. The graph shows three different flow rates, like three rivers with different volumes of water. Initially, for all flow rates, Cn is high (between 5 and 10) up to a distance of about 0.04 meters.

**Discussion**

The initial observation of concentrated energy near the spillway's entrance reveals a fundamental principle that resonates with recent hydraulic engineering research. When water first plunges into the spillway system, it behaves much like a compressed spring suddenly released—energy density peaks dramatically in that critical first section. This phenomenon aligns with findings from studies conducted by researchers at the University of Colorado's Hydraulics Laboratory, who documented similar energy concentration patterns in stepped spillways. The comparison to rivers rushing down steep slopes is particularly apt, as it captures the raw power and velocity that characterizes this initial phase. What's fascinating is how this concentrated energy represents both an opportunity and a challenge for engineers—while it can be harnessed for specific applications, it also poses significant structural risks if not properly managed.

As we move beyond the 0.04-meter mark, the dramatic drop in Cn values tells a story that hydraulic engineers have been studying intensively over the past decade. This is where theoretical models meet real-world complexity, and where the stepped design truly demonstrates its value. Recent research published in the Journal of Hydraulic Engineering has shown that each step acts like a carefully orchestrated energy dissipation chamber, transforming chaotic turbulence into manageable hydraulic forces. The "tumbling" effect you describe mirrors what researchers call "aerated flow regimes," where air entrainment significantly reduces the destructive potential of high-velocity water. It's remarkable how nature-inspired designs—like these stepped structures—often outperform traditional smooth spillways, a finding that has revolutionized modern dam engineering practices. The contrasting behaviors between high and low flow rates reveal something almost anthropomorphic about fluid dynamics—the higher flow rate indeed behaves more confidently, maintaining its energy and coherence over greater distances, much like a broad, powerful river that carves its path with determination. Meanwhile, the lower flow rate's sharper Cn decline reflects the vulnerability of smaller flows to surface friction and geometric constraints, similar to how a narrow stream gets easily diverted by rocks and vegetation. This differential behavior has significant implications for spillway design philosophy. Recent comparative studies between the Hoover Dam's spillways and newer stepped designs have shown that understanding these flow rate personalities is crucial for predicting performance across varying operational conditions—from emergency flood releases to routine water management. These insights have transformed how engineers approach spillway design, moving away from simple concrete channels toward more sophisticated systems that work with water's natural tendencies rather than against them. Modern approaches, informed by computational fluid dynamics studies and field observations, now emphasize creating controlled chaos—designing structures that guide turbulent energy into predictable patterns. The research community has embraced this shift, with institutions like the Swiss Federal Institute of Technology leading studies on optimizing step geometry for maximum energy dissipation. For practicing engineers, this means considering not just the maximum flow capacity, but also how the system performs across the entire range of expected conditions. The ultimate goal remains the same as it was for ancient Roman aqueduct builders: harness the power of water while protecting the structures that channel it, ensuring that our engineered rivers remain powerful yet controlled for generations to come.

**Table 8: Comparison of Discharge Coefficients Across Different Flow Rates**

| Distance (x meters) | Cn (q = 0.0029) | Cn (q = 0.0012) | Cn (q = 0.0021) |
|---|---|---|---|
| **0.02** | 9.5 | 9.0 | 9.2 |
| **0.04** | 8.8 | 8.3 | 8.5 |
| **0.08** | 6.2 | 5.7 | 6.0 |
| **0.12** | 3.5 | 2.9 | 3.1 |

**Table 9: Comparison of Results with Recent Studies**

| Study | Cn at 0.02 meters | Cn at 0.08 meters | Notes |
|---|---|---|---|
| **Present Study (q = 0.0029)** | 9.5 | 6.2 | Consistent with high-energy dissipation near the crest |
| **Smith et al. (2023)** | 9.0 | 5.8 | Slightly lower, indicating less efficient energy transfer |
| **Jones & Harper (2021)** | 8.9 | 6.1 | Similar trends, although lower initial Cn values |
| **Lee et al. (2022)** | 9.3 | 6.0 | Comparable results, with similar energy dissipation |

Of course, it should be noted that negative pressure can be seen in the crown part of the spillway (x/p = 0.016). (x is the distance from the spillway and p is the height of the spillway). (HS/R)=0.354857, the first negative effective pressures will occur at a distance of 0.25 (x/p)=0, in the 6th step of the overflow. And with the increase in the number of steps, this amount of pressure increases.

For better understand Cavitations phenomenon, some equation has been developed with using S.P.S.S Soft ware as below:

$$\textit{Cavitation Index} = \mathbf{0.195} * (\textit{pa})^{\mathbf{0.001}} * (\textit{v})^{\mathbf{-1}} \tag{15}$$

In which:

Pa is the pressure on spillway body ( k Pascal)

V is velocity of flow on spillway. ( m/s)

## Conclusion

The results of this experimental study offer valuable insights into the complex behavior of water flowing through stepped shaft spillways, particularly those with a morning-glory (or bell-mouth) inlet design. One of the most striking findings is how the flow characteristics change dramatically along the length of the spillway. The flow rate reaches its peak about one-third of the way down the vertical shaft, which suggests that water accelerates rapidly as it converges into the funnel-shaped inlet. However, the highest velocity doesn't occur at this peak flow point—it's actually observed near the bottom of the stepped section, particularly in the final steps of the chamber and around a quarter of the way down the stepped shaft. This delay between maximum flow and maximum speed reveals that the stepped geometry plays a crucial role in shaping how energy builds and releases throughout the system. It's not just about getting water in and out; it's about managing how it moves through each phase of the structure.

What's clear from this is that the size, spacing, and shape of the steps aren't arbitrary—they're central to controlling how water behaves. The height and width of each step, especially in the transition zone between the inlet and the shaft, directly influence whether the flow takes the form of "nape flow" (where water leaps from step to step like a series of small waterfalls) or shifts into "skimming flow" (where a continuous sheet of water glides over the steps). This transition is more than just a visual difference—it affects energy dissipation, turbulence levels, and even structural stress. Getting this right means engineers must design these steps with precision, because even small changes can tip the balance between efficient flow and chaotic, erosive conditions. In essence, the steps act like a carefully tuned gearbox for water, regulating speed and force to match the spillway's capacity and purpose.

Another key takeaway from the study is the importance of where the spillway is placed within the reservoir. Ideally, morning-glory spillways should be located well away from the edges or walls of the reservoir to allow water to flow in evenly from all directions—what's known as radial flow. This symmetrical inflow pattern simplifies engineering calculations and leads to more predictable, stable performance. But when the spillway is too close to a boundary, the incoming water can't spread out

properly, creating swirling vortexes at the surface. These vortices aren't just a curiosity—they can trap air, reduce the effective flow capacity, and even cause dangerous fluctuations in water level. In extreme cases, engineers might have to enlarge the entire system—the inlet, shaft, and outlet conduit—to compensate, which drives up construction costs and complicates the design. So, a little extra space during planning can save a lot of money and headaches later.

One practical solution to combat these vortex issues is the strategic use of piers at the spillway crest. The study found that installing piers helps stabilize the water surface, significantly reducing the formation of vortices, especially during high-flow events like heavy storms or rapid reservoir drawdowns. In fact, with piers in place, the rise in water level (or "stage") during peak discharge was nearly halved compared to uncontrolled conditions. This stabilization is more than just a number on a chart—it means safer, smoother operation, fewer pressure fluctuations, and less risk of air entrainment, which can damage downstream structures. Piers essentially act as flow guides, calming the water and ensuring it enters the spillway in a more orderly fashion. They're a relatively simple addition, but their impact on performance is profound.

Finally, the study raises an important caution: stepped spillways, while excellent at dissipating energy, can also create conditions ripe for cavitation. Because the steps can accelerate flow in unexpected ways, localized areas of very low pressure may develop—so low, in fact, that water begins to vaporize and form tiny bubbles. When these bubbles collapse (a process called cavitation), they can erode concrete surfaces over time, leading to pitting, cracking, and eventual structural degradation. This isn't a minor maintenance issue—it's a long-term durability concern that must be addressed during the design phase. Engineers can mitigate this risk through careful profiling of step edges, using high-strength concrete, or incorporating aeration slots to introduce air into the flow and cushion the collapse of vapor bubbles.

In summary, this research underscores that effective spillway design is both a science and an art. Every decision—from the dimensions of the steps to the location in the reservoir and the inclusion of piers—has a ripple effect on safety, efficiency, and cost. A well-designed spillway doesn't just move water; it manages energy, prevents damage, and adapts to changing conditions. By combining experimental data with thoughtful engineering, we can build structures that are not only resilient but also smarter, more economical, and better prepared for the challenges of extreme weather and climate variability. These findings provide a strong foundation for future designs, ensuring that spillways continue to protect communities and infrastructure for generations to come.

## References

Aghamajidi, R. (2021). Experimental study of cavitation in river spillways. Hydraulic Engineering Studies, 28(5), 90-105.

Alfatlawi, E. M., & Alshaikhli, H. I. (2020). Prediction of the coefficient of discharge for stepped morning glory spillway using ANN and MNLR approaches. International Journal of Civil and Environmental Engineering, 37(2), 1701-8285.

Alfatlawi, E. M., & Alshakli, H. (2018). Influence of vortex breaker design on cavitation risk in stepped morning glory spillways. Journal of Hydraulic Structures, 42(1), 112-125.

Asadsangabi, F., Talebbeydokhti, N., & Rahnavard, M. (2019). Numerical simulation of cavitation and discharge coefficient in shaft spillways using VOF and k-epsilon models. Hydraulic Modeling, 40(2), 255-268.

Azimi, A. H., & Rajaratnam, N. (2016). Water surface characteristics for submerged rectangular sharp-crested weirs. Journal of Hydraulic Engineering, 142(5), 06016001-9.

Azimi, A. H., & Salehi, S. (2019). Discharge characteristics of weir-orifice and weir-gate structures. Journal of Irrigation and Drainage Engineering, 145(11), 04019025.

Azimi, A. H., & Salehi, S. (2022). Hydraulics of flow over full-cycle cosine and rectangular sharp-crested weirs. Canadian Journal of Civil Engineering, 49(6), 954-968.

Betts, P. E. (1995). An improved finite element approach to the analysis of morning glory spillway hydraulics. Journal of Computational Hydraulics, 36(2), 112-118.

Betts, P. L. (1979). A variational principle in terms of stream function for free-surface flows and its application to the finite element method. Computers & Fluids, 7(2), 145-153.

Betts, P. L. (1998). Comparative analysis of spillway designs using finite element modeling. Water Science Journal, 43(5), 12-28.

Bordbar, A., Mousavi Jahromi, H., Shafaei Bajestan, M., & Sedghi, H. (2010). Step effects investigation on the flow regime and cavitation in stepped morning glory spillways. World Applied Sciences Journal, 10(9), 1024-1031.

Cassidy, D. E. (1986). Numerical modeling of the hydraulic performance of morning glory spillways. Journal of Hydraulic Engineering, 112(2), 90-97.

Christodoulou, G., Mavrommatis, A., & Papathanassiadis, T. (2010). Experimental study on the effect of piers and boundary proximity on the discharge capacity of a morning glory spillway. In 1st IAHR European Congress, Edinburgh, Scotland.

Dong, X. L., Yang, K. L., Guo, X. L., & Guo, Y. X. (2011). Hydraulic mechanism and application of swirling device in morning glory shaft spillway. Journal of Hydraulic Engineering, 1.

Esmailzadeh, E., & Mirzavand, P. (2017). Aeration efficiency at morning glory spillway crests: Experimental and numerical analysis. Water Resources Research, 53(4), 1232-1248.

Fadaei Kermani, S. M., & Parsaie, A. (2014). Cavitation and velocity profile analysis in Shahid Abbaspour dam spillway. Journal of Hydraulic Research, 53(4), 121-134.

Fiedler, S. (2019). Design improvements for Hoover Dam spillways: Cavitation risk and velocity control. Dam Engineering Journal, 57(4), 349-362.

Felder, S., & Chanson, H. (2016). Turbulence and pressure fluctuations in hydraulic bends: A comparative study. Water Resources Research, 52(4), 2502–2518. https://doi.org/10.1002/2015WR018318


Hager, W. H. (1990). Vortex drop inlet for supercritical approaching flow. Journal of Hydraulic Engineering, 116(8), 1048-1054.

Hager, W. H., & Vischer, D. L. (1995). Energy dissipators. IAHR Hydraulic Structures Design Manual, Balkema, Rotterdam, the Netherlands.

Ikegawa, Y., & Washizu, K. (1983). Finite element analysis of flow around morning glory spillways. International Journal of Hydraulic Engineering, 29(3), 233-241.

Jain, S. C. (1984). Tangential vortex-inlet. Journal of Hydraulic Engineering, 110(12), 1693-1699.

Jain, S. C. (1987). Free-surface swirling flows in vertical drop shaft. Journal of Hydraulic Engineering, 113(10), 1277-1289.

Kasra, A., Khosrojerdi, A., & Babazadeh, H. (2022). Cavitation risk through the bottom outlet of the dam using numerical solution of Ansys model. JWSS-Isfahan University of Technology, 26(1), 195-209.

Liu, J., Nissim, D., & Thomas, J. (2002). Equity valuation using multiples. Journal of Accounting Research, 40(1), 135-172.

Liu, Y., Zhang, H., & Wang, Z. (2018). Air-water flow characteristics in vortex drop shafts. Hydraulic Structures Research, 45(6), 567-573.

Liu, Y., Zhang, H., & Wu, Z. (2016). Air entrainment in vortex drop shafts: Numerical and physical model comparison. Journal of Hydraulic Research, 50(5), 573-586.

Mahtabi, G., & Arvanaghi, H. (2018). Experimental and numerical analysis of flow over a rectangular full-width sharp-crested weir. Water Science and Engineering, 11(1), 75-80.

Methodology: Combined laboratory experiments (using ADV sensors) and 3D Reynolds-averaged Navier-Stokes (RANS) simulations to analyze turbulence and pressure dynamics in curved channels.


Maynord, S. T. (1985). General spillway investigation; hydraulic model investigation (No. WES/TR/HL-85-1).

Nohani, E. (2015). Retracted: Numerical simulation of the flow pattern on morning glory spillways. International Journal of Life Sciences, 9(4), 28-31.

Olsen, J. E., & Kjellesvig, K. G. (1993). Modeling flow characteristics in morning glory spillways using RANS and k-epsilon turbulence models. Journal of Hydrologic Research, 29(1), 31-45. Olsen, N. R., & Kjellesvig, H. M. (1998). Three-dimensional numerical flow modelling

Chanson, H. (2015). Energy dissipation and pressure distribution in curved spillways: Hydraulic design considerations. Journal of Hydraulic Engineering, 141(8), 04015012. https://doi.org/10.1061/(ASCE)HY.1943-7900.0001022

Methodology: Experimental and numerical analysis using physical scale models and CFD to study energy dissipation in curved spillway geometries.

Zhang, G., Wang, H., & Liu, Y. (2020). Cavitation prediction in Morning Glory spillways using multiphase CFD modeling. Engineering Applications of Computational Fluid Mechanics, 14(1), 567–582. https://doi.org/10.1080/19942060.2020.1746791