Journal of Computational Applied Mechanics 2021, 52(2): 215-232
DOI: 10.22059/jcamech.2021.318239.594

RESEARCH PAPER

Determination of hydraulic characteristics of flow over a triangular sectioned weir by using experimental and numerical modeling.

Ali Yildiz[1,*], *Ali Ihsan Marti*[1], *Mustafa Gogus*[2]

[1] *Civil Engineering Department, Engineering and Natural Science Faculty, Konya Technical University, Konya, Turkey*

[2] *Civil Engineering Department, Faculty of Engineering, Cankaya University, Ankara, Turkey*

Abstract

The spillways of hydraulic structures transfer excessive water from dam reservoir to the downstream in a safe and controlled manner. A labyrinth or triangular weir is a flat spillway folded in plain view. The labyrinth weirs provide an increase in crest length for a given channel width and increase the flow capacity for a given weir load. As a result of the increased flow capacity, the labyrinth and triangular weirs require less space in the dam body than the flat weirs. In this study, experiments were carried out on the labyrinth weirs containing triangles of different heights and numbers by using 3 different weir heights (P=20cm, 30cm, and 40 cm) and 4 different weir shapes. Each experiment was repeated for 30 different discharge values. The effects of weir height and weir shape on the total head over the weir (H_T) and discharge (Q) were investigated. In addition, the numerical models of all experimental setups were created by ANSYS-Fluent program using Computational Fluid Dynamics (CFD). By comparing the results obtained from the numerical models with the physical models, the accuracy of the numerical models was tested. According to the results, as the number of the triangles (N) of the weir increases, the discharge coefficient (C_d) decreases. The weir height (P) does not have a major effect on the discharge.

Keywords: ANSYS-Fluent, CFD, Labyrinth weirs, Triangular sectioned weirs, Spillways.

Introduction

Spillways, used in canals, rivers, streams and dams, control the flow depth and discharge, as well as to measure the amount of discharge that will pass to the downstream [1]. They have important functions for the operation and stabilization of a dam besides its important place for the cost of the dam. The positions of the dam spillways are determined, and their hydraulic and static calculations are made. The most important factor in determining the location is the selection of the type of the spillway to be used. The most suitable spillway type can be determined in hydraulically manner; i.e. the spillway should be able to evacuate large floods without putting the dam body at risk. The type and location of the spillway should be planned, projected, constructed and operated reliably.

When the hydrology of the project is examined in some existing water structures, the need to increase the flood discharges may arise. The dam reservoir should be able to store this flood flow in the reservoir as much as possible, in the case of a high amount of rainfall. During the cases where the reservoir is insufficient, the excess water should be successfully discharged through the weirs and transferred to the downstream side. If the dam reservoir and spillway are not sufficient for the flood discharge, the dam body can be rehabilitated with labyrinth weirs to make them safer.

---

[*] Corresponding author: ayildiz@ktun.edu.tr

In recent years, the use of labyrinth weirs is an effective method to increase the spillway discharge capacity and the reservoir storage volume. By increasing the effective length of the spillway crest with labyrinth spillways, it will be possible to pass water to the downstream side with a smaller total head over the weir. The labyrinth and triangular weirs can be considered as the most advantageous alternative, especially for the reservoir level where the upstream area is limited or the topography limits the width of the spillway. In addition, the labyrinth weirs have easy construction and more reliable operating conditions compared to the sluice gate spillways. Also, the labyrinth and triangular weirs are used to increase the reservoir capacity of the dam, which decreases due to sedimentation.

Labyrinth weir is a useful type of weir to increase the reservoir capacity and regulate the flow; however there are some difficulties for the design processes due to their geometry and limited design data. The flow capacity over the labyrinth weirs are determined according to their net crest lengths (L_T) and crest shapes. Besides these advantages of the labyrinth weirs, they are good energy dissipaters and ventilation structures for water [2, 3].

Labyrinth type weirs have many geometric configurations; they are triangular (A) trapezoidal (B) and rectangular (C) (Figure 1). Among these 3 geometries, the triangular and trapezoidal labyrinth weirs provide the most efficiency[4].

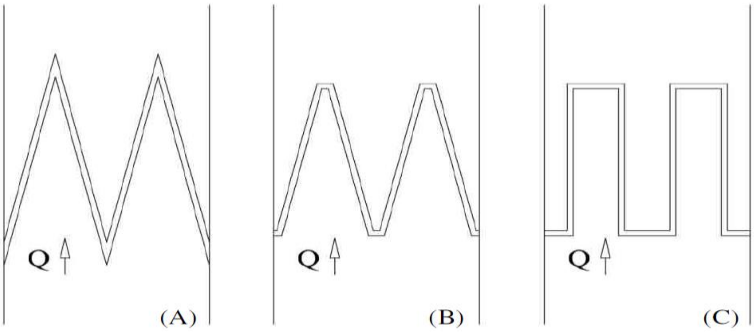


Figure 1. Different types of labyrinth weirs

Many studies have been conducted to determine the parameters affecting the design and optimization of labyrinth weirs. Gentilini [5] combined these oblique weirs to form a triangular form using his previous work on oblique weirs. At high weir loads, the results were found to be related with the w/P ratio, and the results are presented in relation to H_T/w. Kozak and Svap [6] conducted experiments on 11 different trapezoidal labyrinth weirs with a wall thickness of t_w = 6 mm. Kozak and Svan have clearly shown that a labyrinth weir discharges more flow than a flat weir of same width. Hay and Taylor [7] studied on 24 labyrinth weirs containing different numbers of triangular, trapezoidal and rectangular geometries in which the experiments were conducted for the range of $0.05<H/P<0.55$. Darvas [8] conducted hydraulic modeling studies on the design of labyrinth weirs, and the results obtained from these experiments made a significant contribution to the Woronora and Avon dams in Australia. Cassidy [9] conducted modeling studies for labyrinth weirs and obtained important results on the hydraulic performance of labyrinth weirs. Tullis et al. [10] conducted an experimental study on two different weir heights of 15 cm and 30 cm having different weir angles varying from 8^o to 35^o for labyrinth weirs. The most important part of these studies was studied in Utah Water Research Laboratory that the flow coefficient and C_d values were determined for different weir variables.

Although many studies have been performed on labyrinth weirs, and their hydraulic characteristics have been determined in detail, there exists less number of studies on triangular weirs that can be seen as hydraulically less efficient weirs compared to labyrinth weirs. However, they are easier to design and manufacture than the labyrinth weirs. Additionally, in this study, the numerical models of triangular weirs were formed by using a Computational Fluid Dynamics (HAD) based program named ANSYS-Fluent. HAD programs are frequently used in the design of hydraulic structures in recent years. Ghaderi [11] created a new design to improve the hydraulic performance of labyrinth weirs. In this design, the weir wall was notched and the weir crest edge was inclined. To simulate the flow over the weir CFD is used. Torres [12] conducted studies on labyrinth weirs to determine the most appropriate numerical model implementations also ANSYS and OpenFOAM programs are used for CFD models. Yildiz [13] used FLOW-3D for simulating flow over an ogee crested spillway and obtained consistent results between Numerical model and Experiment. Mohammadi [14] worked on behavior of rectangular graphene sheet under shear in-plane load numerically. The main purpose of this study is to determine the numerical model of triangular labyrinth weirs accurately. Sometimes the physical model experiments in the laboratory may not give correct results due to the scale effect. Therefore, real-dimensional analysis of models can be made by using numerical modeling programs whose accuracy will be tested by comparing the experimental data with the results obtained from the numerical models.

Material and Method

The main variables taken into account during the design process of the labyrinth spillways are the length of the weir wall (L), the width of the channel (W), the width of one cycle of weir (w), the length of weir side wall (B), the weir height (P), the side wall angle of the labyrinth weir (a), the number of labyrinth triangles (N), and the less effective variables are wall thickness (t), crest shape, and weir end form (Figure 2 and Figure 3). The most important hydraulic parameters are discharge (Q) and total head over the weir (H).

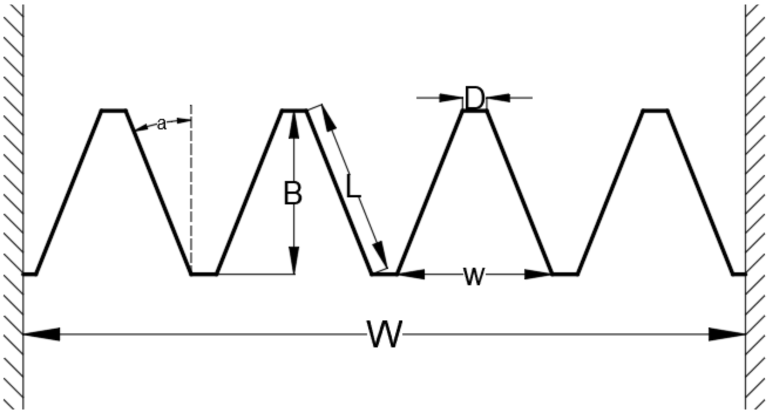


Figure 2. Geometrical parameters affecting the flow over a labyrinth weir

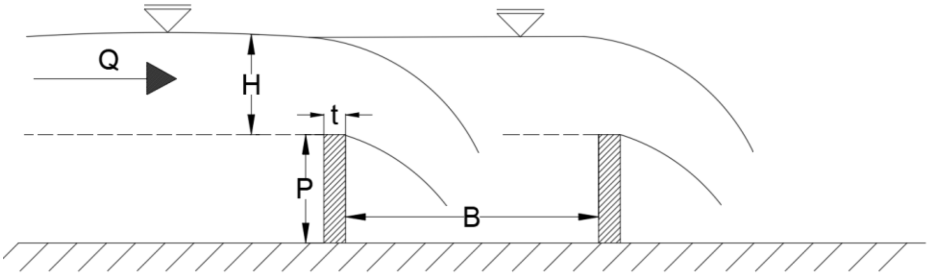


Figure 3. Hydraulic parameters affecting the flow over a labyrinth weir

The labyrinth spillways are generally used with U, V and trapezoidal plan shapes. These forms are used in the weir to produce different flow conditions. The flow over a labyrinth weir can be in 3 different ways, namely, no air entrainment, ventilated flow, and half ventilated flow (Figure 4).

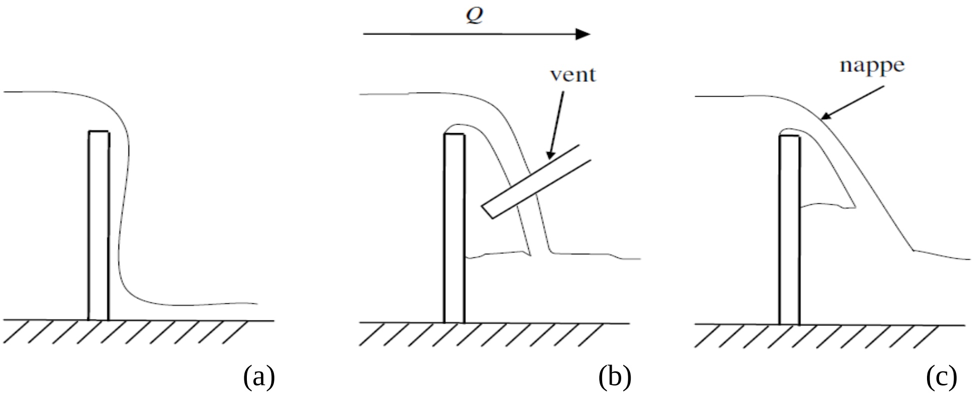


Figure 4. Flows formed according to the ventilation conditions of the downstream sides of the labyrinth weir (a) No air entrainment (b) Ventilated (c) Half ventilated

The basic equation of a linear weir given below, equation 2, can be used to calculate the discharge (Q) over a labyrinth weir [10].

$$Q=\frac{2}{3}*C_d*L_T*\sqrt{2*g}*H^{1.5} \tag{1}$$

Where, C_d is the Non dimensional discharge coefficient, L_T is the Net crest length, g is the Gravity, H is the Total head over the weir, Q is the Discharge

Experimental Study

The open channel system used for the physical modeling of triangular weirs has 650 cm length, 60 cm width and 50 cm height. The surfaces of the open channel are made of glass. (Figure 5)

Figure 5. Open channel system used in experimental setup

The flow in the open channel used for the experimental setup is provided by two pumps which are connected in parallel to the system. Therefore, the total discharge in the channel is equal to the sum of the flow coming from two pumps. The amount of flow that will be pumped to the channel is adjusted by the frequency changers in the panel to which the pumps are connected. Thus, the discharge range given to the system is between 1 and 45 lt/s. The discharge value is read by an ultrasonic flow meter placed between the pipes after the pumps. The water flowing in the open channel system is circulated from two reservoirs.

While conducting the experiments, two basic parameters were taken into account: the weir height (P) and the number of triangular (N) parts used in the weir. The effect of these two parameters on discharge was investigated. In the experiments, 3 different weir heights (P) were used; P = 20 cm, P = 30 cm and P = 40 cm. 4 different types of weir were tested, including 3 different number of triangles (N=2, N=3 and N=4) at each weir height and flat weir. In the experimental studies, a total of 12 experiments were conducted with triangular weirs and flat weirs. The triangular weirs were divided according to the number of triangles they contained. The general scheme of the experiments is shown in figure 6.

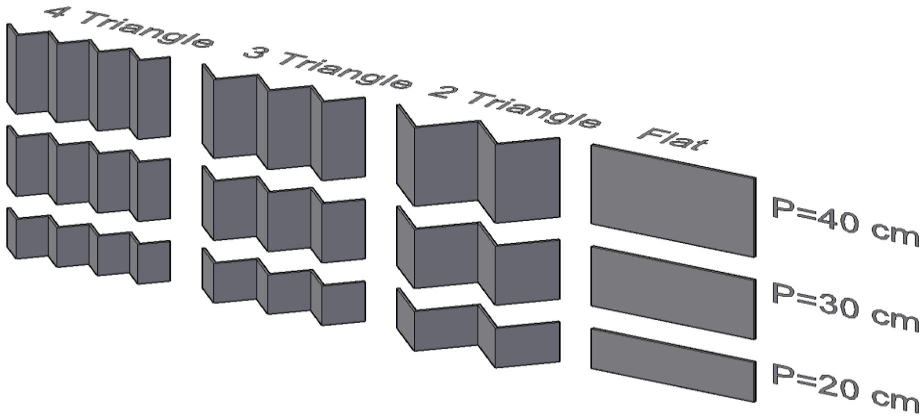


Figure 6. Weirs used in the experiments

CFD (Computational Fluid Dynamics)

In this study, the numerical solutions of the physical models were made by using the ANSYS-FLUENT program based on the finite volume method. The numerical methods investigating fluid motion within the computational fluid dynamics can be diversified as finite elements, finite volumes, boundary values and finite difference methods. The flow motion is represented by Navier-Stokes Equations [15]. CFD is a very efficient tool for solving these and similar problems that the core process of it is discretization. In this process, instead of solving many points infinite at all moments, it calculates the flow properties at a number of points and in certain time periods. Thus, the partial differential equations are transformed into a set of algebraic equations that can be solved with the aid of a computer. Discretization is the definition of the solution region as discrete points, elements and volumes by numerical methods [16]. Momentum is a set of correlated partial differential equations derived from the laws of energy and mass conservation. The unknowns in these equations are; flow rate, pressure, density, and temperature. The finite volume method enables to solve the partial differential equations derived from the conservation principles by converting them into discrete algebraic equations on finite volumes. The solution region is discretized by dividing it into finite volumes. The partial differential equations are integrated on each finite volume and converted into algebraic equations. Then the obtained algebraic equations are solved in order to calculate the dependent variable values of finite volumes [17].

ANSYS-FLUENT solves the conservation equations for mass and momentum.

The equation for the conservation of mass, or continuity equation 2, For 2D axisymmetric geometries, the continuity equation is given by equation 3, The conservation of momentum in an inertial (non-accelerating) reference frame is described by equation 4 as follows,

$$\frac{\partial\rho}{\partial t}+\Delta * x(\rho\upsilon)=S_m \tag{2}$$

$$\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x}(\rho\upsilon_x)+\frac{\partial}{\partial r}(\rho\upsilon_r)+\frac{\rho\upsilon_r}{r}=S_m \tag{3}$$

$$\frac{\partial}{\partial t}(\rho \upsilon)+\Delta(\rho \upsilon \upsilon)=-\Delta^{*}P+\Delta\tau+\rho g+F \tag{4}$$

Where; x is the axial coordinate, r is the radial coordinate, υ_x is the axial velocity, υ_r is the radial velocity, ρ is the density of fluid, P is the pressure, g is the gravity, F is the body force and τ is the stress tensor.

Numerical Modeling

A total number of 12 numerical models were formed depending on the performed physical experiments (Figure 7). In the experiments conducted, the flow rate was increased from 0 lt/s to 40 lt/s depending on the setup and the capacity of the weirs. Since it was not possible to give all flow rates in the numerical model, the analyses were carried out only by using certain flow rates, such as 5, 10, 15, 20, 25 and 30 lt/s. The weir loads corresponding to these flow rates were compared with the experimental data.

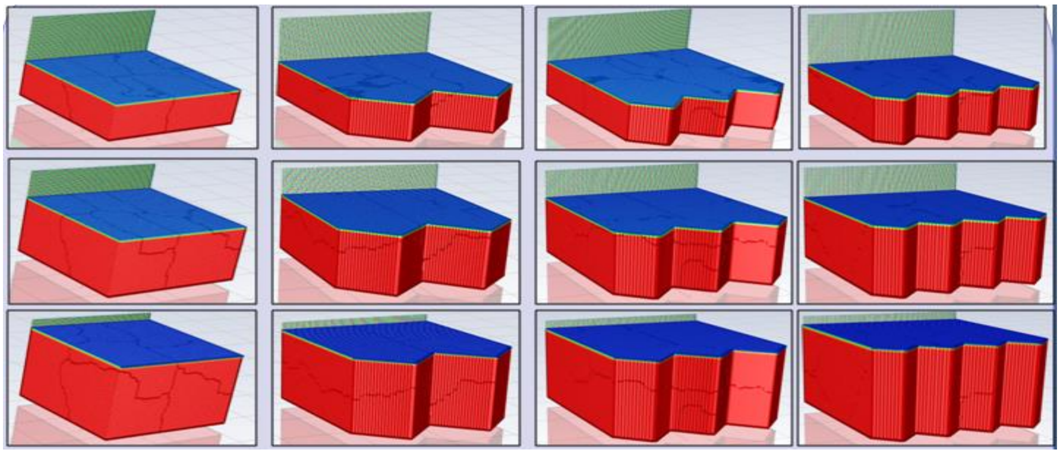


Figure 7: Triangular and flat weirs made of numerical models

The analysis in FLUENT module of ANSYS program consists of 4 stages; geometry, mesh, setup and solution. For the formation of the numerical models of the experiments on triangular weirs, the configurations done at each stage are as follows:

Geometry: In order to simulate the experiments in the FLUENT correctly, the geometries were formed in 3 dimensions, and the analyses were carried out in 3 dimensions. The geometry to be analyzed in FLUENT is not the geometry of the solid volume (Figure 8). The analyses are made by considering the geometry of the flow volume which is the volume where water and air are likely to be found, other than solid surfaces. Therefore, the size of the flow volume should be determined according to the user prediction. An oversized flow volume will unnecessarily lengthen the analysis. When determining the flow volume, it is necessary to determine in advance the situations such as how much the water will rise, from which points it will flow, whether it will jump or not. The flow volume was created by using CAD drawing programs and then transferred to ANSYS-FLUENT.

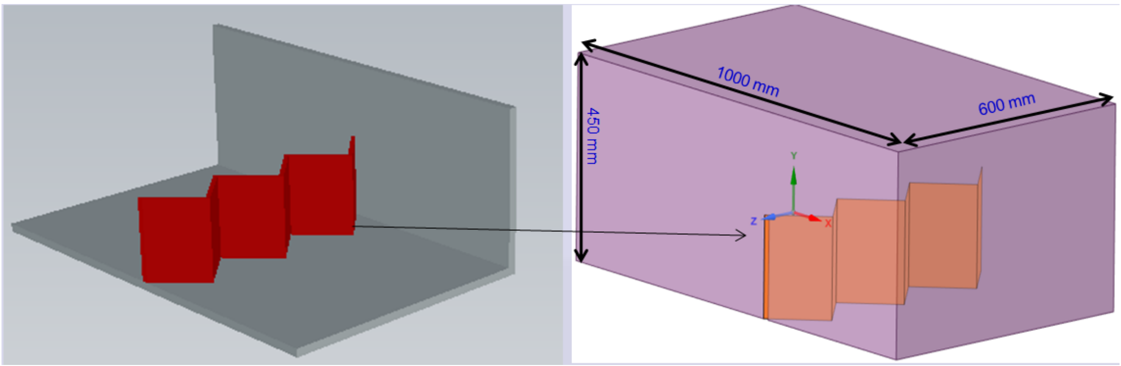
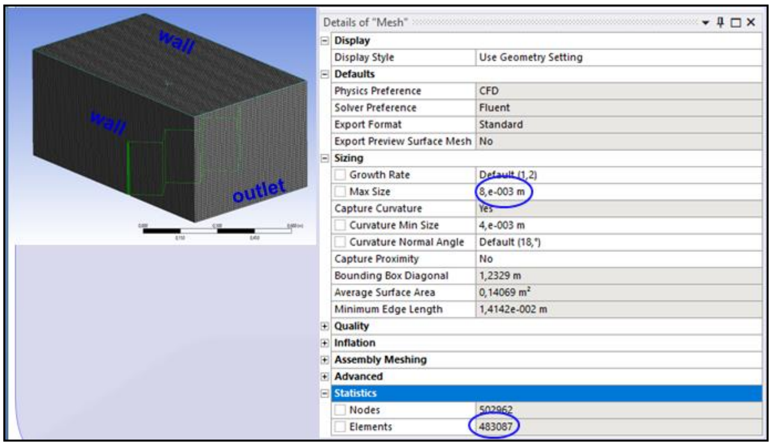


Figure 8: Solid model and flow volume

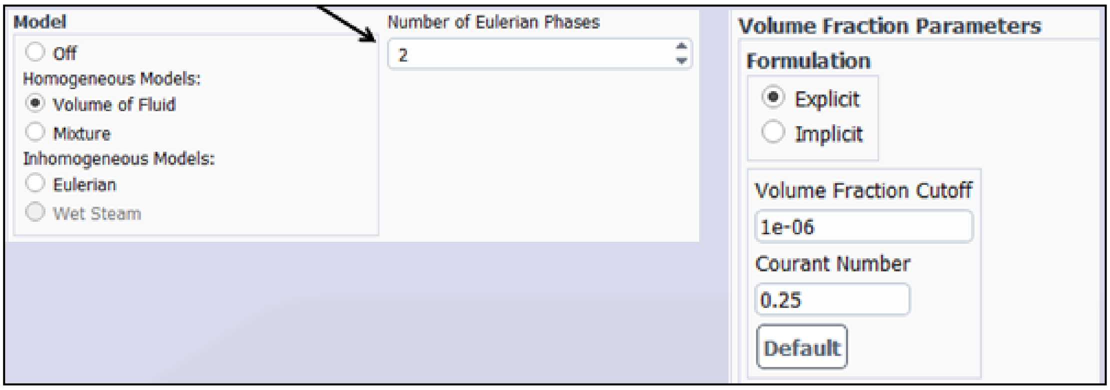
Mesh: Mesh cells are defined according to the flow volume. Since the program works with the finite volume method, the calculations were made in the cells to be formed. Mesh cells are generally defined depending on the general geometry of the flow volume. Since the geometry of the flow volume of the numerical model is a rectangular prism, the shape of mesh cells is chosen as cube. The shape of the mesh used in analysis has great importance. The shape of flow volume affects the analysis time, the total number of elements and the accuracy of the analysis. When solving in the finite volume method, the accuracy of the solution changes depending on which direction the fluid enters and leaves the volume. As the fluid enters the volume, an entry perpendicular to the surface will facilitate the calculations and increase the sensitivity of the solution. The properties of the surface of the flow volume i.e. boundary conditions are defined in the mesh section. The surface where water enters the volume is defined as "inlet", the surface where it leaves the flow volume as "outlet", and the remaining surfaces are defined as wall (Figure 9). The flow volume itself is defined as "interior". The edge size of the cube-shaped meshes used in flow volume is 0.008 m. The analysis was made with a total of 483000 elements.



Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
☐ Growth Rate	Default (1,2)
☐ Max Size	8,e-003 m
Capture Curvature	Yes
☐ Curvature Min Size	4,e-003 m
☐ Curvature Normal Angle	Default (18,°)
Capture Proximity	No
Bounding Box Diagonal	1,2329 m
Average Surface Area	0,14069 m²
Minimum Edge Length	1,4142e-002 m
Quality	
Inflation	
Assembly Meshing	
Advanced	
Statistics	
☐ Nodes	502962
☐ Elements	483087

Figure 9: Information for mesh size and total elements

Setup: In the experiments conducted in the laboratory, there was a free surface flow, that is, there was air above the water. Therefore, two-phase analysis was designed in such a way that both air and water were included in the flow volume. The multiphase model was chosen as the VOF (Volume of Fluid) method to supply the two-phase flow (Figure 10). For the solution to be more detailed, an explicit solution was made instead of an implicit solution. The program not only calculated the movement of water but also of air. The effect of air on water (free surface flow) was also taken into account in this way. (Figure 5)



Model
○ Off
Homogeneous Models:
● Volume of Fluid
○ Mixture
Inhomogeneous Models:
○ Eulerian
○ Wet Steam

Number of Eulerian Phases: 2

Volume Fraction Parameters
Formulation
● Explicit
○ Implicit

Volume Fraction Cutoff: 1e-06
Courant Number: 0.25

Figure 10: Multiphase model

K-epsilon RNG is used as a viscosity model, and PISO is used as the method. Since the water coming to the labyrinth weirs comes at a certain discharge through the channel, the “inlet” surface of the flow volume is defined as the "mass flow inlet". This means that there is a constant flow of water with a certain discharge. The “outlet” part of the flow volume is defined as the "pressure outlet". Air and water can freely leave the flow volume.

When the model is first started for the analysis, as there is no air entrainment to weir, water flows adhering to the downstream part of the weir. Since the analysis made in this way will yield different results from the experiments carried out in the laboratory. Therefore 3-stage solution has been implemented. Firstly, the downstream part of the triangular weir was started as “wall” condition. The water flowed adhering to this boundary. Then, this boundary was turned into an “outlet” condition, and this area was ventilated by providing a free flow. After ventilating, it was turned back to the “wall” condition and the same situation was obtained in the experiments (Figure 11).

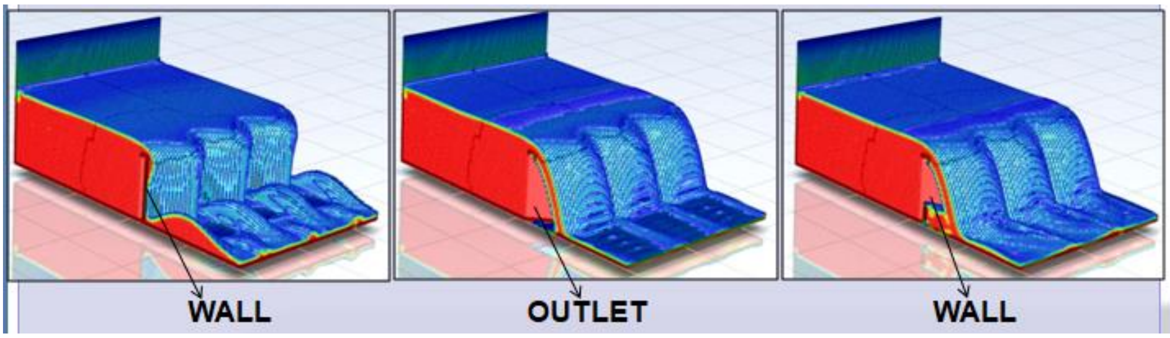


Figure 11. Changing the boundary condition of the downstream part of the weir to provide ventilation.

In the analysis, "time-step size" changes between 0.001 seconds and 0.005 seconds depending on the height of the weirs and the flow rate. In some cases, the "time-step size" was reduced to 0.0001 seconds. In the analyses, a water volume was defined as the upstream of the labyrinth weirs to speed up the solution (Figure 7). In this way, the analysis time was reduced without waiting for the upstream part of the triangular weir to fill with water.

Results and Discussion

The aim of the experiments is to obtain the best flow rate in the same channel width. In other words, it is to increase the flow-discharge capacity of weir by providing more flow pass with the same total head (H). The triangular labyrinth weirs provide greater crest length than the flat weirs with the same channel width. Increasing the net crest length (L) had a direct positive effect on the discharge. The results obtained from the experiments were examined in 2 parts as the effect of the weir height (P) and the shape of the weir depending on the number of triangles (N) it contains.

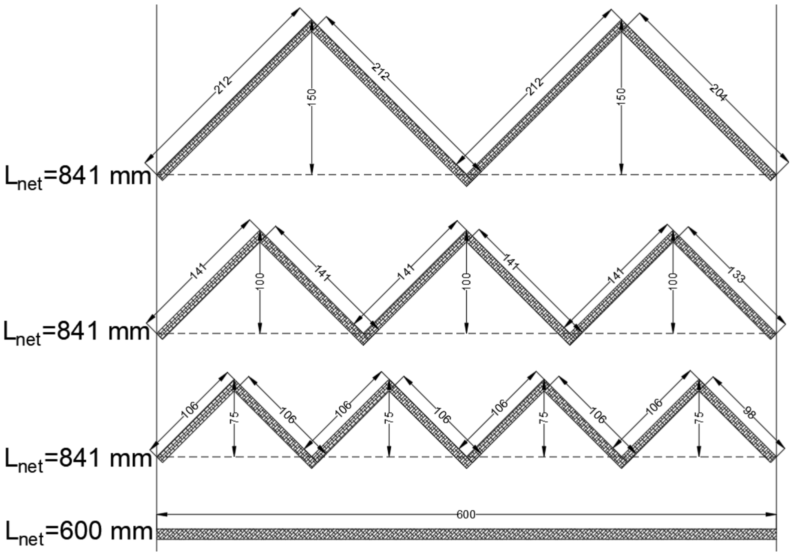


Figure 12. Top view of the triangular weirs containing N=2, N=3 and N=4 triangles.

Firstly, the effect of weir shape on the discharges (Q) was investigated. This was done by comparing the different types of weirs at the same weir height (P). In order to understand the efficiency of the weirs more accurately and compare their performance, the discharge coefficients (C_d) were examined (Figure 13, figure 14 and figure 15). The purpose of comparing the discharge coefficients (C_d) is to see the net crest lengths (L_{net}) of the weirs as equal. The discharge coefficients (C_d) and the dimensionless total head (H_T/P) were all shown on the same graphs. The total crest length was L_{net} = 84.1 cm for all cases including the triangle (2 triangles, 3 triangles and 4 triangles) regardless of the number of triangles used. In flat weir, the net crest length was L_{net} = 60 cm. The total lengths of the weirs and the channel width can be seen in figure 12.

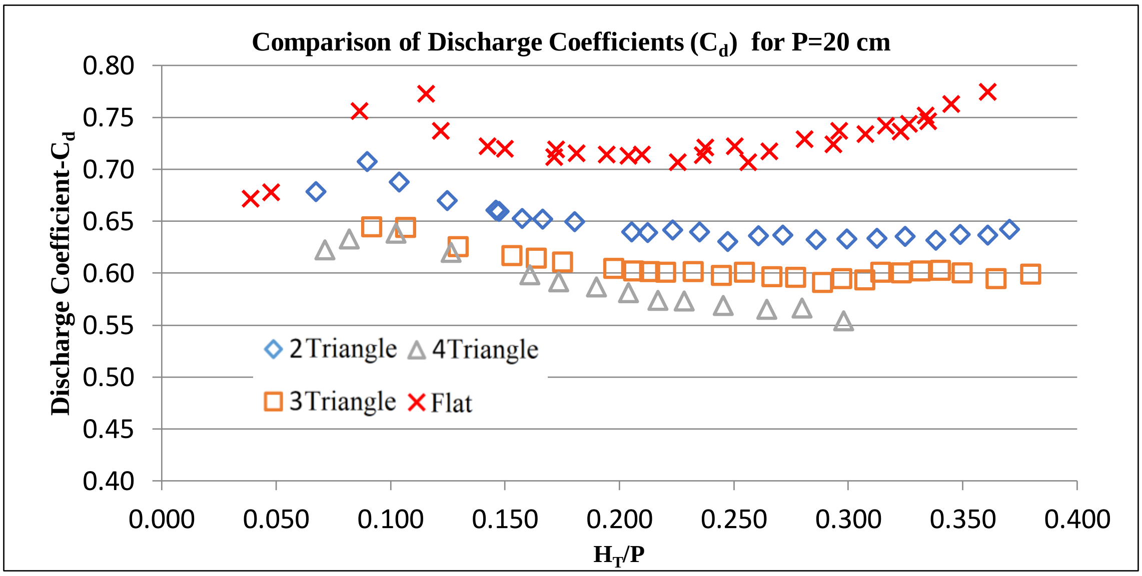


Figure 13. Comparison of Discharge Coefficients (C_d) for P=20 cm

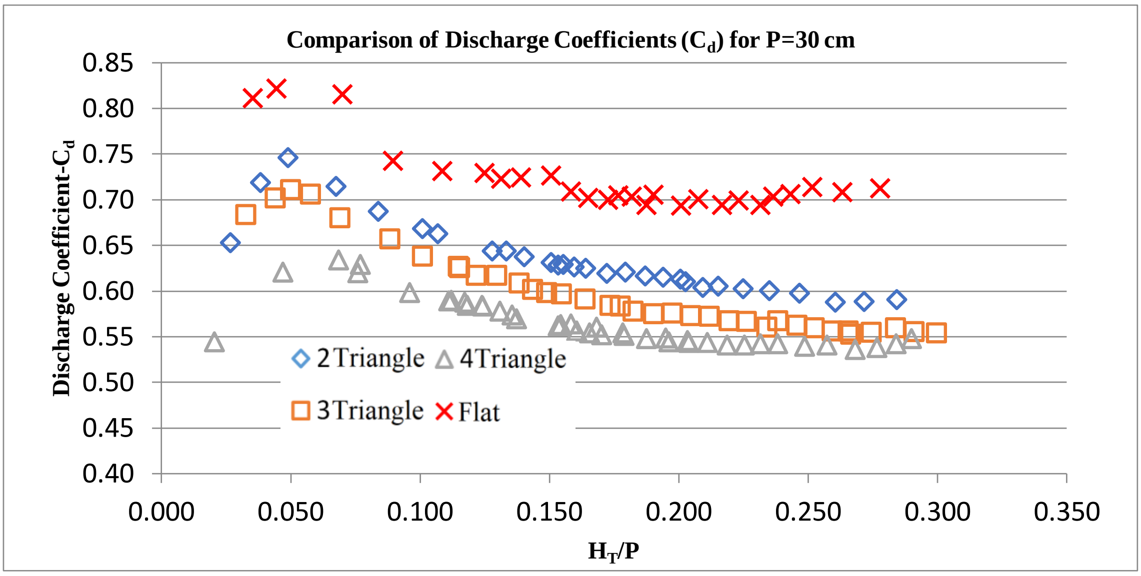


Figure 14. Comparison of Discharge Coefficients (C_d) for P=30 cm

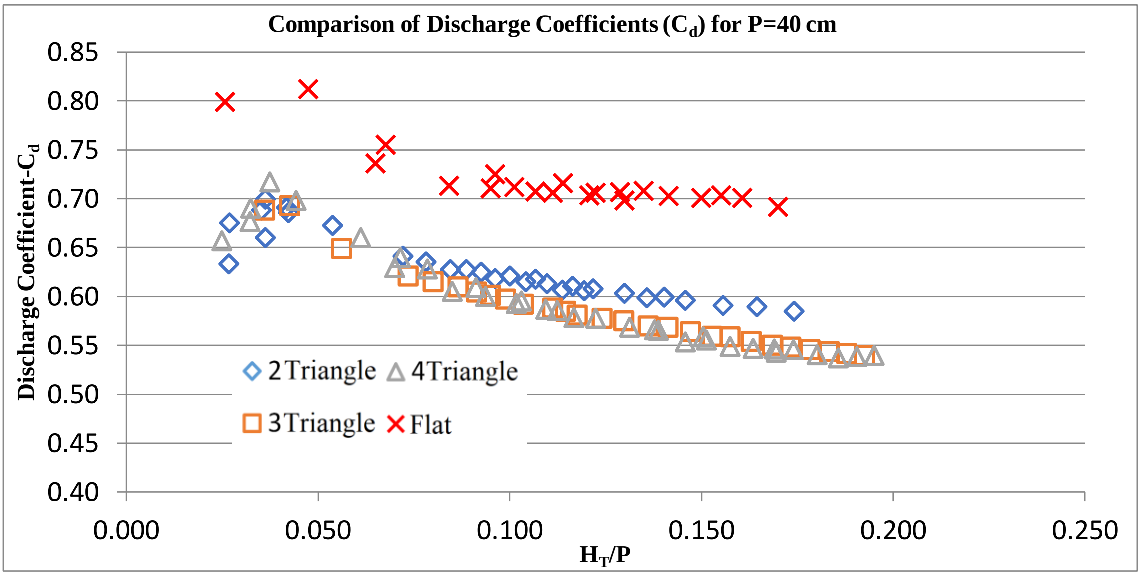


Figure 15. Comparison of Discharge Coefficients (C_d) for P=40 cm

When the results are examined, it is seen that the weirs including N = 2 triangles (H) had more flow than the other triangular weirs at the same total head for the same weir height (P). After the weirs including N=2 triangles, the weirs including N=3 triangles and N=4 triangles take place in terms of flow efficiency. To compare the discharge coefficients (C_d), the sharp-edged weir formula, equation 1, was used. When the discharge coefficients (C_d) were compared, the flat weir gave the best discharge (Q) performance. In terms of weir coefficients, the C_d values were listed as $C_{d\ Flat} > C_{d\ N=2\ Triangle} > C_{d\ N=3\ Triangle} > C_{d\ N=4\ Triangle}$ (Figure 13 - Figure 15.). This is valid for all weir heights such as P = 20 cm, P = 30 cm and P = 40 cm. The discharge coefficient (C_d) of the flat weir was higher than the triangular weir. This difference was due to the difference in the approach angle of the water to the weirs. While the straight weir meets the water directly at about 90^o angle, this angle decreases in triangular weirs, and this reduces the discharge coefficient (C_d). If the discharge coefficient (C_d) of the flat weir is higher than the triangular weir, it will not make flat weirs more efficient. Because in triangular weirs, the same channel width is used more efficiently, and longer net crest length (L_{net}) is provided. The fact that the weirs including N = 2 triangles give better results than the other triangular weirs (including N = 3 and N = 4 triangles) is due to the less interference zones (Figure 16). It means that, it becomes $Q_{N=2\ Triangle} > Q_{N=3\ Triangle} > Q_{N=4\ Triangle} > Q_{Flat}$ with same total head over the weir (H).

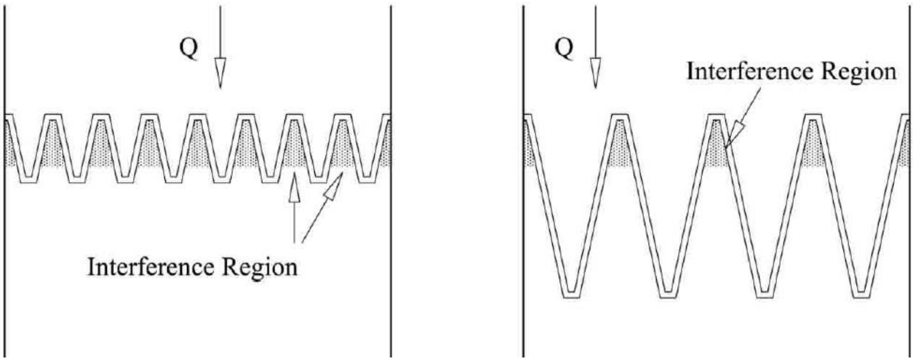


Figure 16. Interference regions

Secondly, the weirs of same type having different weir heights (P) were compared in terms of discharge coefficient (C_d) and the total head over the weir (H). The purpose of this comparison was to examine the effect of weir height (P) on the discharge coefficient (C_d) (Figure 17, figure 18, figure 19 and figure 20). When figure 17 is examined, it is seen that the weir height (P) does not have a significant effect on the discharge coefficient (C_d) for flat weirs. However, it is observed that as the weir height (P) decreases the discharge coefficient (Cd) increases slightly. The reason for this, as the height of the weir (P) decreases, the ventilation on the downstream side of the weir decreases, and more vacuum effect occurs. With the effect of vacuum, more water is drawn and the discharge (Q) increases and the total head over the weir (HT) decreases (Figure 18, figure 19 and figure 20).

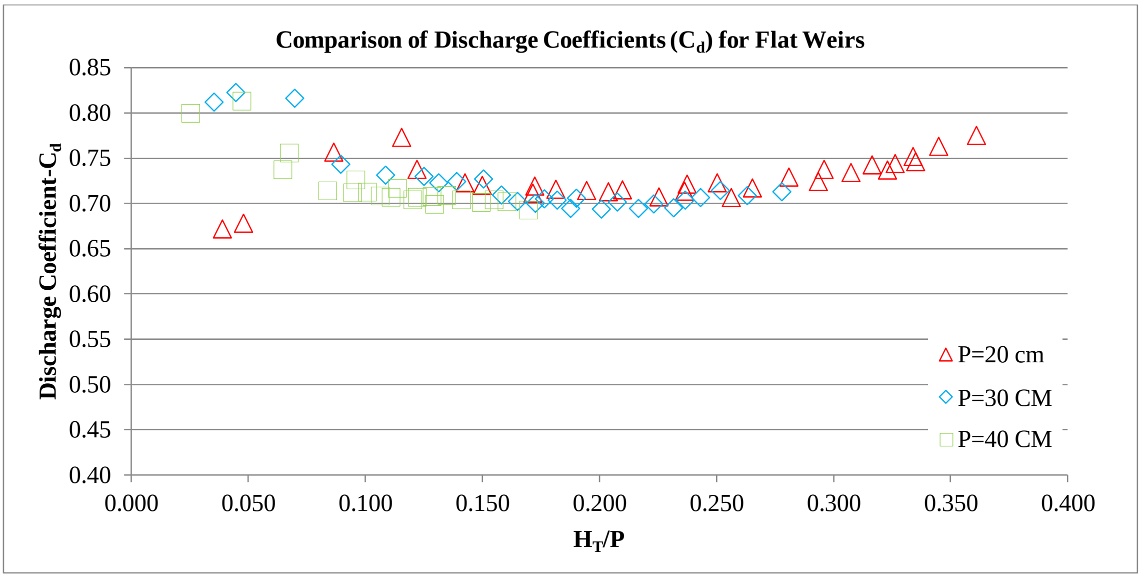


Figure 17. Comparison of Discharge Coefficients (C_d) for Flat Weirs

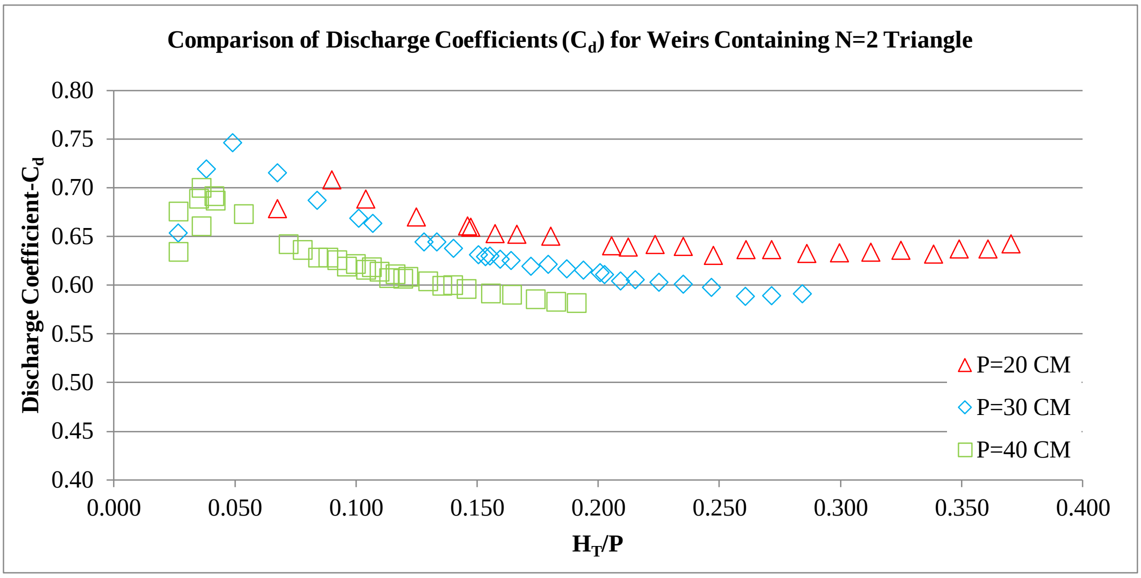


Figure 18. Comparison of Discharge Coefficients (C_d) for Weirs Containing N=3 Triangle

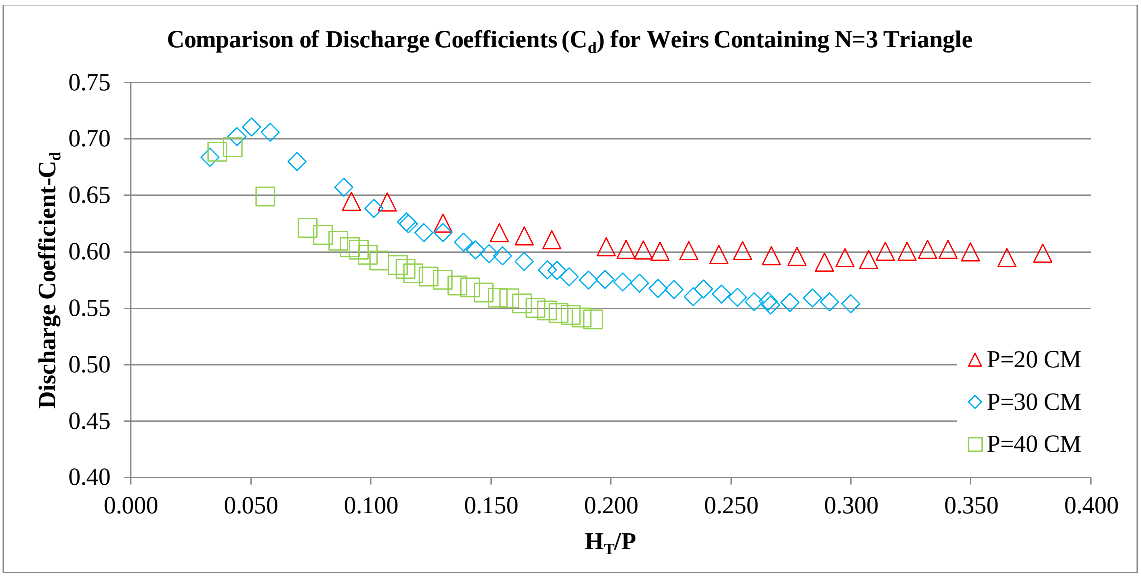


Figure 19. Comparison of Discharge Coefficients (C_d) for Weirs Containing N=3 Triangle

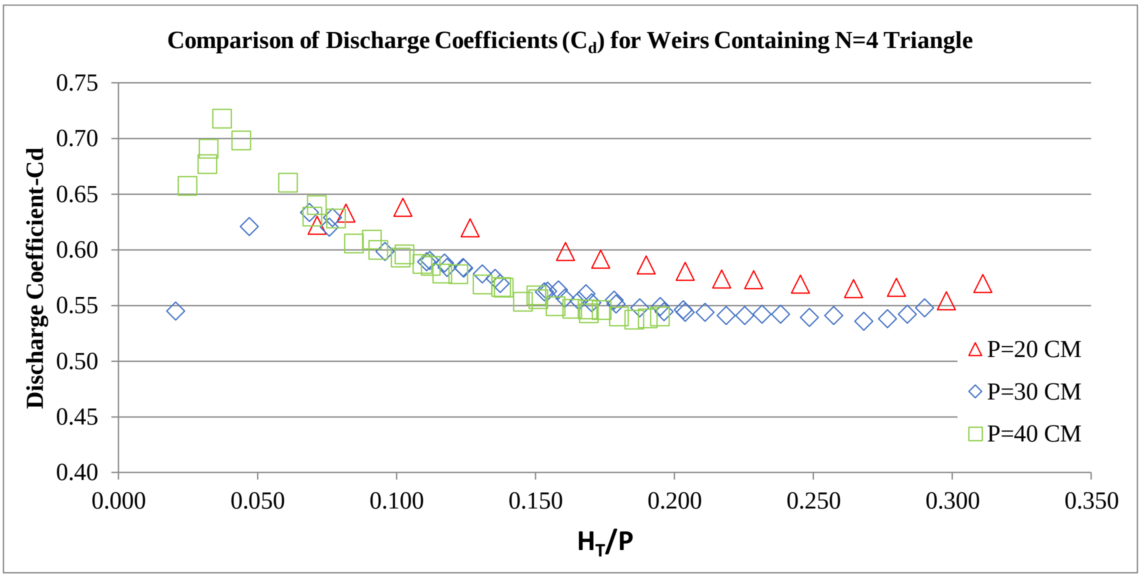


Figure 20. Comparison of Discharge Coefficients (C_d) for Weirs Containing N=4 Triangle

The obtained discharge coefficient (C_d) results were also compared with the previous studies. One of the most important studies on labyrinth weirs has been performed by Tullis [10]. Tullis conducted experiments with labyrinth weirs at different angles and shared the results of the weir coefficient obtained from these experiments with dimensionless parameters Cd and H/P. Tullis used maximum 35^o side wall angles in his studies. The side wall angle of the triangular weirs in this study is 45^o degrees. The discharge coefficients (C_d) obtained for different weir heights (P) from the N = 2 triangle weir show consistency when compared with the graphic.

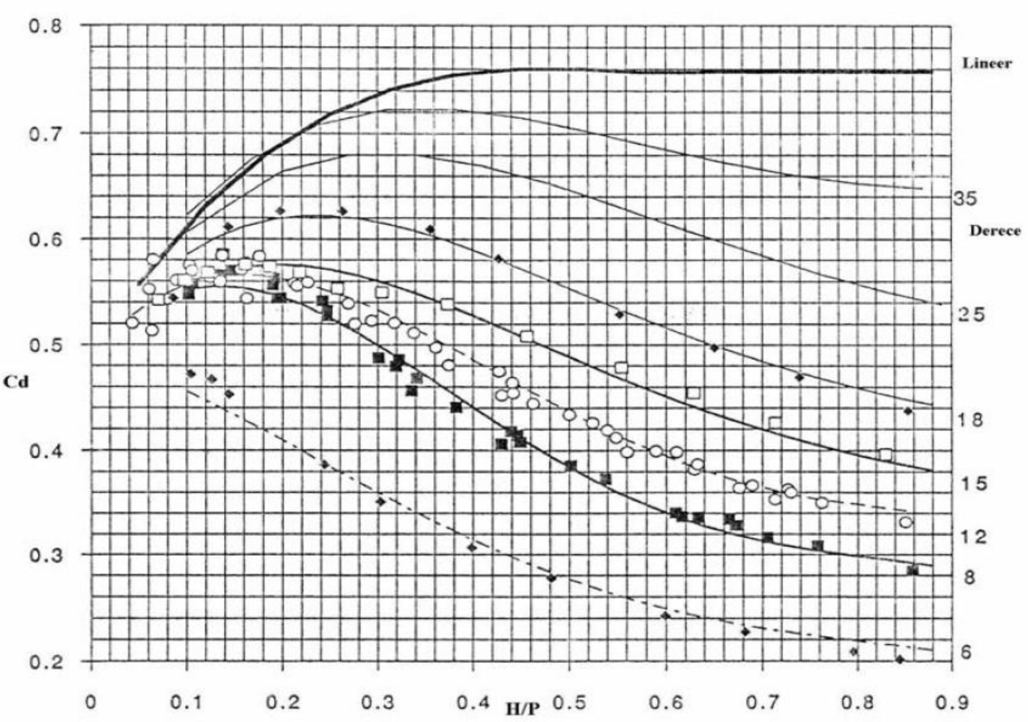


Figure 21. Crest coefficient for labyrinth spillways [18]

Comparison of Numerical Model Experimental Study

The results obtained from the numerical model were compared with the experimental results in two terms. These are visual comparison and numerical comparison. In visual comparison, the 3D images obtained from the numerical model were compared with the experiments carried out in the laboratory. In visual comparison, the water surface profiles, the way the water is poured over the weirs depending on the characteristics of the weirs, and the turbulence they cause in the downstream part of the weirs were compared. When the 3D images obtained from the numerical model were compared with the experiments, it was seen that the water surface profiles and the flow characteristics were very similar. In both models, the water behaved like a reservoir behind the weirs and flowed over the weirs with a similar profile. There was no fluctuation or turbulence in the reservoir that this means that the assigned boundary condition worked successfully, and water flowed freely over the weirs. When water flowed over the weirs, the flow was semi-ventilated on the downstream side. By gradually changing the downstream surface of the labyrinth weir, semi-ventilated flow was successfully achieved. The comparison of the 3D images obtained from the numerical model and the experiment pictures are given below (Figure 22).

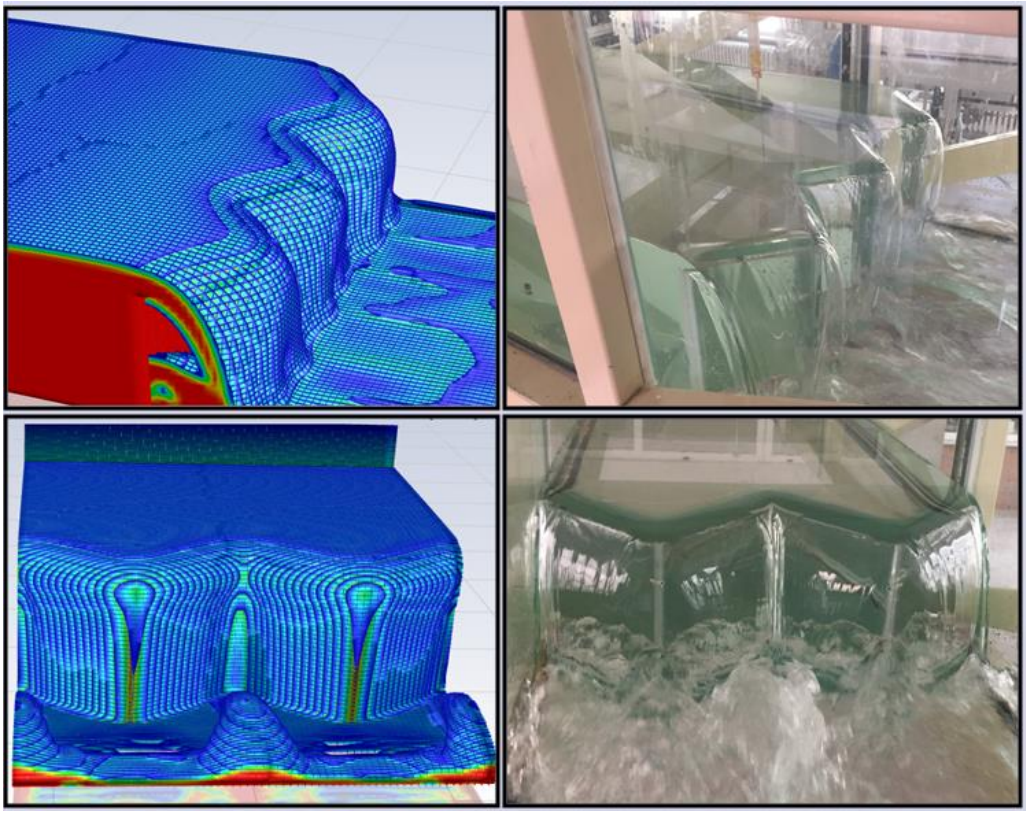


Figure 22. Visual comparison of numerical model and experimental setup

In the numerical comparison, the total heads obtained from the numerical models and experiments were compared. A total number of 12 numerical models were formed (Figure 7). In the experiments, the discharge was increased from 0 lt/s to 40 lt/s depending on the setup and the capacity of the weirs. Since it was not possible to give all the flow rates in the numerical model, the analyses were carried out only for 7 discharge values of 5-10-15-20-25-30-35 lt/s. The total heads corresponding to these discharges were read and compared with the experimental data. The analysis time performed on each discharge value approximately took 4 hours.

The results obtained from the numerical models and the experiments were compared in terms of total heads (H) and discharges (Q). In both models, the total heads (H) corresponding to discharges (Q) were compared graphically. The purpose of comparing graphically is that the flow values given in the numerical model were sometimes not found in the experiments. Each experiment was repeated approximately for 20-30 different flow rates; however, since it is not possible to do all of these in the numerical model, the analyzes were made using the flow rates at certain intervals. While the weir load can be clearly read by the limnimeter in the experiments, this situation differs slightly in the numerical model. In the numerical model, weir loads were read depending on the filling rate of water and air. For this, a linear graph was drawn, and the weir load was calculated by looking at the starting coordinate and the end coordinate of the water in this graph. The end of the water was not clear, as the program worked according to the finite volume method. The occupancy of the cells took values between 0 and 1.

The graphical comparisons of the results obtained from the numerical model and experiments are given in figure 23, figure 24 and figure 25 The comparisons are given separately depending on the weir height (P) and the number of triangles (N) existing in the weir. As mentioned above, all the discharge values (Q) were not used in the numerical model. When the obtained results are examined, an average of 93% similarity was observed between the numerical model and the experiments. The numerical model gave very consistent results with the results of the experiments. However, in the weir with P = 20 cm height and N = 3 triangles, there are some differences between the numerical model and the experimental data at 25 m^3/s and 30 m^3/s flow rates. The numerical model presented low total head (H) at the same discharge (Q) values in comparison to the experimental data. The main reason for these differences was due to the fluctuations in the water surface of the numerical model. In the numerical model, when the analysis is stopped during the measurement process, if a small wave occurs on the water surface at the measurement point, this will affect the result. Moreover, the size of the mesh cells was supposed to affect the measurement results. As a result, the numerical model gave consistent results both visually and numerically with the experimental data.

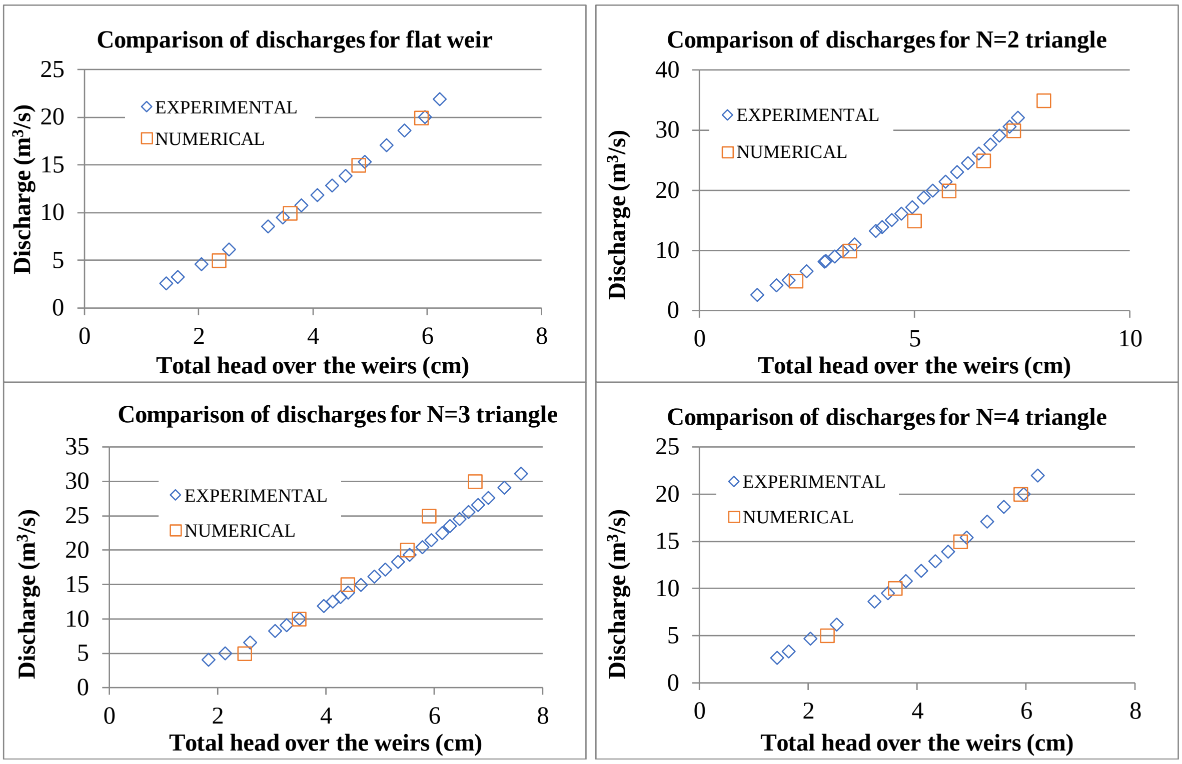
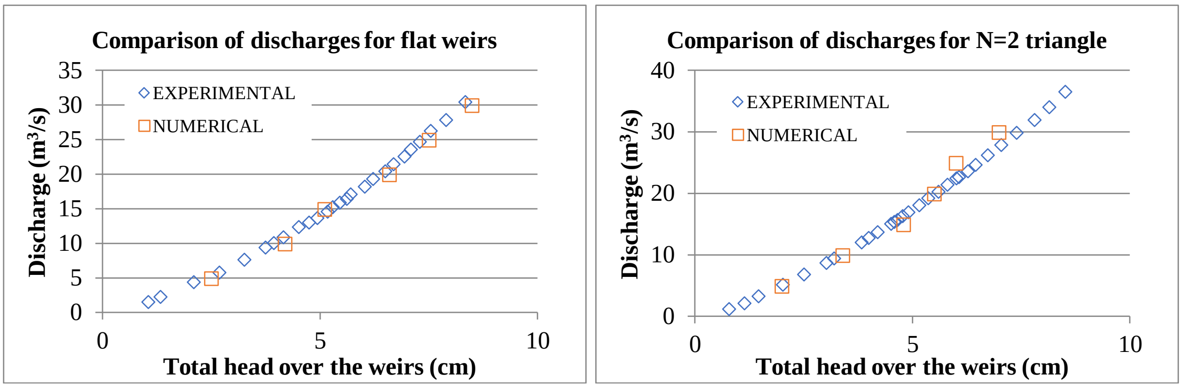


Figure 23. weir height P=20 cm, the comparison of discharge values obtained from the numerical model and experiments for flat weir and triangular weirs that contain N=2, N=3 and N=4 triangles



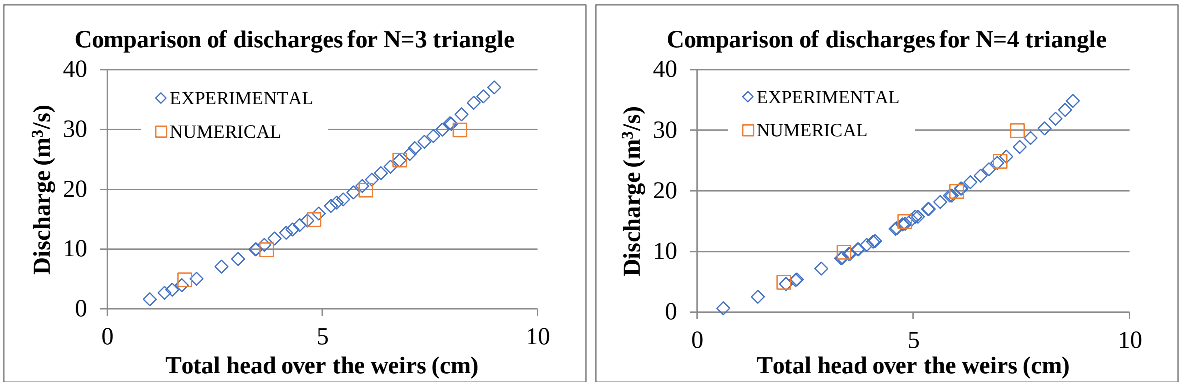


Figure 24. For weir height P=30 cm, the comparison of discharge values obtained from the numerical model and experiments for flat weir and triangular weirs that contain N=2, N=3 and N=4 triangles

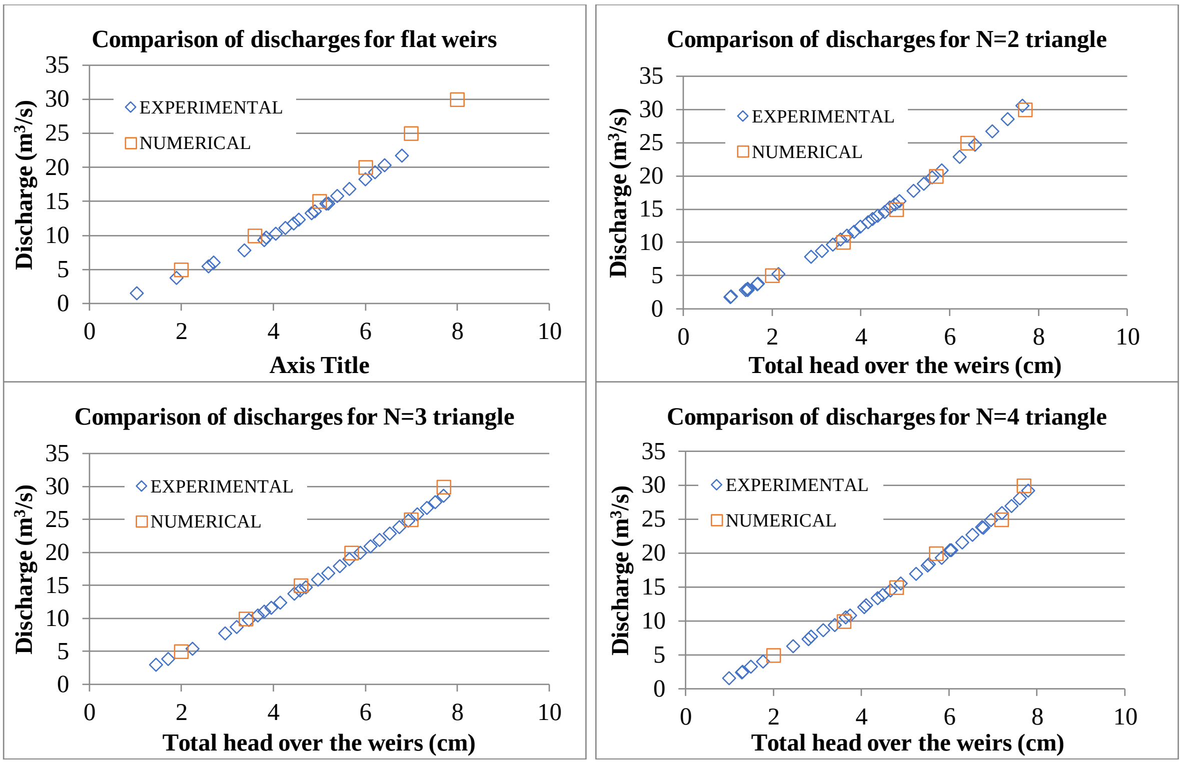


Figure 25. For weir height P=40 cm, the comparison of discharge values obtained from the numerical model and experiments for flat weir and triangular weirs that contain N=2, N=3 and N=4 triangles

Conclusion

In this study, experiments were carried out with labyrinth weirs having a triangular sectioned form and numerical models of these experiments were created. These triangular sectioned weirs have different weir heights (P) and triangular numbers (N). The aim of this study is to investigate the effect of weir height (P) and number of triangles (N) contained in the weir on discharge coefficient (C_d). In addition, the results obtained from the numerical models of the experimental setups modeled using by ANSYS-FLUENT program were compared with the experimental data. According to the results obtained,

- At lower total heads (H_T), as the weir height (P) increases, the discharge coefficient (C_d) also increases, but at higher total heads (H_T), the effect of the weir height (P) on the discharge (Q) decreases.
- As the number of triangles (N) contained in the weir increases, the discharge coefficient (C_d) decreases. The most important reason for this situation is that as the number of triangles (N) increases, the interface regions (Figure 16) to which the current will be exposed also increase. These areas prevent the flow from passing easily over the weir.
- The numerical models show highly consistent results with experiments. The main reason for this is the method applied while conducting analyzes. The boundary condition of the downstream side of the triangular sectioned weirs in the numerical model was changed to obtain the most realistic flow conditions (Figure 11).

In addition, the accuracy of the experiments was tested by comparing the experimental results with the data in the literature. The experiments gave results consistent with previous studies. The most important part of this study is the successful simulation of triangular sectioned weirs with numerical modeling program. The parameters and methods used in the creating numerical models will help the studies on this subject.

References

[1] Ş. Y. Kumcu, and M. Uçar, "Effect of Experimental and Mathematical Modeling of Spillway on Dam Safety," Advances in Safety Management and Human Factors. pp. 296-305.

[2] P. R. Wormleaton, and E. Soufiani, “Aeration performance of triangular planform labyrinth weirs,” Journal of Environmental Engineering-Asce, vol. 124, no. 8, pp. 709-719, Aug, 1998.

[3] P. R. Wormleaton, and C. C. Tsang, “Aeration performance of rectangular planform labyrinth weirs,” Journal of Environmental Engineering-Asce, vol. 126, no. 5, pp. 456-465, May, 2000.

[4] B. M. Crookston, “LABYRINTH WEIRS,” Civil and Environmental Engineering, Utah State University, Logan, Utah, 2010.

[5] B. Gentilini, “Stramazzi con cresta a pianta obli-qua e zigzag,” Mem. e Stud. del Reg. Politec. di Milano, vol. 48, pp. 1-12, 1941.

[6] M. Kozák, and J. Sváb, “Tört alaprajzú bukók laboratóriumi vizsgálatatle,” Hidrológiai Közlöny, vol. 5, pp. 376-378, 1961.

[7] N. Hay, and G. Taylor, “Performance and Design of Labyrinth Weirs,” Journal of the Hydraulics Division, vol. 96, no. 11, pp. 2337-2357, 1970.

[8] L. A. Darvas, “Discussion of “Performance and Design of Labyrinth Weirs&#x201d,” Journal of the Hydraulics Division, vol. 97, no. 8, pp. 1246-1251, 1971.

[9] J. J. Cassidy, C. A. Gardner, and R. T. Peacock, “Boardman Labyrinth—Crest Spillway,” Journal of Hydraulic Engineering, vol. 111, no. 3, pp. 398-416, 1985.

[10] J. P. Tullis, N. Amanian, and D. Waldron, “Design of Labyrinth Spillways,” Journal of Hydraulic Engineering-Asce, vol. 121, no. 3, pp. 247-255, Mar, 1995.

[11] A. Ghaderi, R. Daneshfaraz, S. Abbasi et al., “Numerical analysis of the hydraulic characteristics of modified labyrinth weirs,” International Journal of Energy and Water Resources, vol. 4, no. 4, pp. 425-436, 2020.

[12] C. Torres, D. Borman, A. Sleigh et al., “Application of Three-Dimensional CFD VOF to Characterize Free-Surface Flow over Trapezoidal Labyrinth Weir and Spillway,” Journal of Hydraulic Engineering, vol. 147, no. 3, pp. 04021002, 2021.

[13] A. Yildiz, A. Yarar, S. Y. Kumcu et al., “Numerical and ANFIS modeling of flow over an ogee-crested spillway,” Applied Water Science, vol. 10, no. 4, Mar 13, 2020.

[14] M. Mohammadi, A. Farajpour, M. Goodarzi et al., “Numerical study of the effect of shear in-plane load on the vibration analysis of graphene sheet embedded in an elastic medium,” Computational Materials Science, vol. 82, pp. 510-520, Feb, 2014.

[15] C. W. Hirt, and B. D. Nichols, “Volume of Fluid (Vof) Method for the Dynamics of Free Boundaries,” Journal of Computational Physics, vol. 39, no. 1, pp. 201-225, 1981.

[16] J. H. Ferziger, and M. Perić, "Solution of the Navier-Stokes Equations," Computational Methods for Fluid Dynamics, pp. 157-216, Berlin, Heidelberg: Springer Berlin Heidelberg, 2002.

[17] F. Moukalled, L. Mangani, and M. Darwish, "Erratum to: The Finite Volume Method in Computational Fluid Dynamics," The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab, pp. E1-E1, Cham: Springer International Publishing, 2016.

[18] B. M. Crookston, and B. P. Tullis, “Hydraulic Design and Analysis of Labyrinth Weirs. I: Discharge Relationships,” Journal of Irrigation and Drainage Engineering, vol. 139, no. 5, pp. 363-370, 2013.