

**A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF ÇANKIRI KARATEKİN UNIVERSITY**

**INVESTIGATIONS OF WATER FLOW BEHAVIOR OVER
DIFFERENT STEPPED SPILLWAY GEOMETRY**

**IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE DEGREE OF MASTER OF SCIENCE
IN
CIVIL ENGINEERING**

BY

AMJED ALAWADI

ÇANKIRI

2022

INVESTIGATIONS OF WATER FLOW BEHAVIOR OVER DIFFERENT STEPPED
SPILLWAY GEOMETRY

By Amjed ALAWADI

August 2022

We certify that we have read this thesis and that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science

Advisor : Prof. Dr. Ender SARIFAKIOĞLU

Co-Advisor : Asst. Prof. Dr. Salih YILMAZ

Examining Committee Members:

Chairman : Prof. Dr. Ender SARIFAKIOĞLU
Civil Engineering
Çankırı Karatekin University

Member : Prof. Dr. Zeliha SELEK
Civil Engineering
Gazi University

Member : Asst. Prof. Dr. Başak VARLI BİNGÖL
Civil Engineering
Çankırı Karatekin University

Approved for the Graduate School of Natural and Applied Sciences

Prof. Dr. İbrahim ÇİFTÇİ
Director of Graduate School

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Amjed ALAWADI

ABSTRACT

INVESTIGATIONS OF WATER FLOW BEHAVIOR OVER DIFFERENT STEPPED SPILLWAY GEOMETRY

Amjed ALAWADI

Master of Science in Civil Engineering

Advisor: Prof. Dr. Ender SARAFKIOĞLU

Co-Advisor: Asst. Prof. Dr. Salih YILMAZ

August 2022

In this project, the spillway effects have been investigated by various spillway design analysis using ANSYS CFD Fluent software. The slope of all the models is held constant for each case and two types of slops considered in this research. The inlet boundary condition of the stepped spillway model is represented by the discharge. The outlet boundary condition of the spillway is set by pressure, velocity and volume of friction. The results observed that it is feasible to improve the hydraulic behaviors of a pool configurations in stepped spillway. The pools in this study present an overcome of the flow stability and energy dissipation issues. The pressure value at the beginning of the step in the pooled configuration was large, while for the notched pool the maximum pressure values decreased near the step pool. Pool configuration (simple or notched) did not have a significant influence on the location of air entrainment. Also, the air entrainment location (inception point) moves downward when increases the discharge rate. This result observes the importance of the spillway steps shape to generate the pressure force based on height of each step.

2022, 56 pages

Keywords: Spillway slop design, Volume of friction, Inception point

ÖZET

FARKLI KADEMELİ DOLUSAVAK GEOMETRİSİ ÜZERİNDE SU AKIŞI DAVRANIŞININ İNCELENMESİ

Amjed ALAWADI

İnşaat Mühendisliği, Yüksek Lisans

Danışman: Prof. Dr. Ender SARAFKIOĞLU

Eş Danışman: Dr. Öğr. Üyesi Salih YILMAZ

Ağustos 2022

Bu projede, ANSYS CFD Fluent yazılımı kullanılarak çeşitli dolusavak tasarım analizleri yapılmış ve dolusavak etkileri araştırılmıştır. Tüm modellerde eğimi sabit tutulmuş olup, bu çalışmada iki tip eğim ele alınmıştır. Basamaklı dolusavak modelinin giriş sınır koşulu, deşarj ile temsil edilir. Dolusavak çıkış sınır koşulu basınç, hız ve sürtünme hacmi ile belirlenir. Sonuçlar, basamaklı dolusavak, havuz konfigürasyonlarının hidrolik davranışlarını iyileştirmenin mümkün olduğunu göstermiştir. Bu çalışmadaki havuzlar, akış kararlılığı ve enerji kaybı sorunlarının üstesinden gelmiştir. Basit havuz konfigürasyonunun başlangıcındaki basınç değeri büyükken, çentikli havuz için maksimum basınç değerleri havuza yakın kesimlerde düşmüştür. Havuz konfigürasyonunun (basit veya çentikli) hava sürüklenmesi konumu üzerinde önemli bir etkisi olmamıştır. Ayrıca, tahliye hızı arttığında hava sürüklenme konumu (başlangıç noktası) aşağı doğru hareket eder. Bu sonuç, her bir basamağın yüksekliğine bağlı olarak basınç kuvveti için dolusavak basamaklarının şeklinin önemini gösterir.

2022, 56 sayfa

Anahtar Kelimeler: Dolusavak eğim tasarımı, Sürtünme hacmi, Başlangıç noktası

PREFACE AND ACKNOWLEDGEMENTS

I would like to thank my thesis advisor, Prof. Dr. Ender SARIFAKIOĞLU, for her patience, guidance and understanding.

Amjed ALAWADI

Çankırı-2022



CONTENTS

ABSTRACT	i
ÖZET	ii
PREFACE AND ACKNOWLEDGEMENTS	iii
CONTENTS	iv
LIST OF SYMBOLS	vi
LIST OF ABBREVIATIONS	vii
LIST OF FIGURES	viii
LIST OF TABLES	x
1. INTRODUCTION	1
1.1 Spillways Types Design	2
1.2 Type of Flow in Spillways	2
1.3 Stream Kinds of Flow in Spillways	4
1.4 Problem Statement	5
1.5 Objective of Study	6
1.6 Thesis Organization	6
2. LITERATURE REVIEW	7
2.1 Stepped Spillway Flow Regimes	7
2.2 Energy Characteristics	8
2.3 Energy Dissipation	9
2.3.1 Hydrodynamic pressure	10
2.4 Stepped Spillway Designs	10
2.5 Flow Characteristics on Stepped Spillways	11
2.6 Discharge Capacity	12
2.7 Stepped Spillway Flow Model	13
2.8 Spillway Types	15
2.8.1 Ogee spillway	15
2.8.2 Chute spillway	18
2.8.3 Shaft spillway	19
2.8.4 Labyrinth spillway	20
3. MATERIALS AND METHODS	22

3.1 Model Development	22
3.2 Pre-Process Step	24
3.3 Summary	34
4. RESULTS AND DISCUSSION.....	36
4.1 Stepped Spillway Results.....	36
4.2 ANSYS Evaluation of Triangle Steps (Case 1).....	37
4.2.1 Triangle steps	38
4.3 ANSYS Evaluation of Rectangle Dam (Case 2).....	41
4.3.1 Rectangle spillway	41
4.4 Slop Effect on Inspection Point.....	49
5. CONCLUSION AND RECOMMENDATION.....	52
5.1 Future Work and Recommendation	52
REFERENCES	53
CURRICULUM VITAE.....	56

LIST OF SYMBOLS

A	Area of the opening
A_1	Half-major axis
B	Half-minor axis
C	Discharge coefficient
F	Froude
G	Gravitational acceleration
H	Head above the orifice opening centerline
H_1	Head above the orifice opening centerline
K	Constants
L	Crest length
V	Velocity
W	Width
X	Flow direction x
Y	Flow direction y
θ	Slop angle
Σ	Surface tension
Q	Flow rate

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
CFD	Computational Fluid Dynamic
D _c	Critical Depth
D _h	Comparable Diameter
H _d	Design Head
RCC	Roller Compacted Concrete
Re	Reynolds Number
USBR	US Reclamation Bureau
WES	The Waterways Experiment Station



LIST OF FIGURES

Figure 1.1	The main spillway at Mosul dam	2
Figure 1.2	The pressure effect on various step spillways (Jamel 2018).....	3
Figure 1.3	Criteria for stream kinds from various authors (Simões <i>et al.</i> 2012).....	4
Figure 2.1	Turbulent flow and energy dissipation (Salmasi and Samadi 2018).....	10
Figure 2.2	Flow regimes in stepped spillways (Van Alwon <i>et al.</i> 2017)	12
Figure 2.3	Control structure.....	13
Figure 2.4	Ogee spillway (Chatila and Jurdi 2004).....	16
Figure 2.5	Typical ogee crest profil (Chatila and Jurdi 2004).....	17
Figure 2.6	Chute spillway (Jahad <i>et al.</i> 2017)	18
Figure 2.7	Types contractions in chute spillways.....	19
Figure 2.8	Shaft spillway (Karlsson and Tallberg 2011).....	20
Figure 2.9	Typical labyrinth spillway (Karlsson and Tallberg 2011).....	21
Figure 3.1	Flowchart of the project	23
Figure 3.2	The CFD modelling sequence	24
Figure 3.3	Fluid flow (fluent) files	25
Figure 3.4	Generate the spillway shapes geometry	26
Figure 3.5	Spillway configuration	27
Figure 3.6	Meshing the geometry	28
Figure 3.7	ANSYS mesh application.....	28
Figure 3.8	Mesh quality control parameters	29
Figure 3.9	Mesh element quality values	29
Figure 3.10	Mesh metric outcomes skewness	30
Figure 3.11	Fluent launcher	31
Figure 3.12	General CFD settings	32
Figure 3.13	Model selection	33
Figure 3.14	Set up the boundary conditions	33
Figure 3.15	Run process	34
Figure 4.1	Velocity flow pattern of the triangle spillway simulation results	39
Figure 4.2	Velocity profile of the triangle spillway simulation results	40
Figure 4.3	Velocity flow pattern of the rectangle spillway Dam simulation results	42
Figure 4.4	Pressure profile of rectangular spillway Dam simulation results.....	43
Figure 4.5	The relationship between Froude surface roughness and discharge with slope 51.3°	44
Figure 4.6	The relationship between Froude surface roughness and discharge with slope 26.5°	45
Figure 4.7	The relationship between spillway step and ks	46
Figure 4.8	The relationship between spillway step and angle	46
Figure 4.9	The relationship between spillway step and angle	47

Figure 4.10	The relationship between spillway step and k_s	47
Figure 4.11	Comparison between triangle and rectangle steps	48
Figure 4.12	Volume of fraction for case 1	50
Figure 4.13	Volume of fraction for case 1, vel. 2	50
Figure 4.14	Volume of fraction for case 2	51



LIST OF TABLES

Table 3.1	Best numerical mesh results values.....	30
Table 4.1	Design parameters for numerical simulation of several tiered spillways.....	37
Table 4.2	Operating parameters	38
Table 4.3	Numerical results of triangle step spillway	48
Table 4.4	Numerical results of rectangle step spillway.....	49



1. INTRODUCTION

Water flowing over a spillway is an important field due to the capability to control the water release from the dam to its downstream. It has been studied in various research institutions and used to safely release excess floods. The flow over stepped spillways is a complex flow because of gas exchange derived from the turbulence of water flow and many other aspects (Niu *et al.* 2021). With global rules on the social impacts, many of dams are being considered out of the safety standard. The failure mode considered the water bathers such as retaining walls, spillways, saddle dams which represent a crucial issue in dam safety management process. To optimal method to avoid the failure mode is to design the hydraulic structure to discharge the waters safely. This issue is essential to prevent any structure damages. The flood release facility considered one of the main water bathers which is used to avoid damages of the dam structure from overtopping (Ali and Yousif 2019). The flow characteristics over a stepped drain stream can be specified in three main systems: nape, skimming and moving. In the engineering consideration, limning flow represents more relevant condition than the other two types. For tit reason, stepped spillways are usually designed as slimming flow condition depending on the conditions of Mechanics of Fluid and hydrology of arranged stepped drain stream (Uday *et al.* 2020).

The latest researches, conducted several studies on non-uniform and uniform stepped spillways have investigated water flow characteristics such as flow regime, flow pattern, and velocity field. They founded the position at which the turbulent condition of boundary level specifies the free surface and determined the energy loss in these spillways. The previous studies also contain many modifications of spillways structural reinforcement, layouts and devise innovative solutions to enhance the hydraulic performance. The next section will present briefly the main considerations and characteristics that can be utilized to investigate water flow behavior across a Stepped Spillway.

1.1 Spillways Types Design

Here are various spillways shapes and structures design depending on the suitability of site and other parameters. Spillways design can consider it as a part of the dam or separate from it. Also, it can be designed as a controllable part or uncontrollable part. The spillway with control reliability is provided with gates to control the water level and flow. The spillway consists of a terminal structure, control structure and an unloading channel. In the present study, the Spillways shape and geometry are considered the main issue to investigate the flow characteristics. In Figure 1.1 present on of the main spillways in Mosul Dam which represent the biggest dam in Iraq.



Figure 1.1 The main spillway at Mosul dam

1.2 Type of Flow in Spillways

Spillways are structures that can be used to evacuate overflowing of water flow dams. The high velocity represents the main important hydraulic problem in the field of water flow in spillways which causes cavitation at its downstream. For that, the flows along

stepped spillways considered an interesting concern in the previous studies. The researchers found that the napped regime in the flow over spillways observes three subtypes categorized by the creation of hydraulic jumps on the stair-beds. They also found that the second regime. The parameters of the three standards flow conditions may vary based on the engineering situations for entering the channel, size of the steps and conditions, the length of the channel based on region steps and finally the flow rates. It occurs by considering specific conditions such as the variations in steps length in relation to their height and bottom flows (related to scrape flow). The “transitional flow” among the backflow flows and the scrape flows is also defined as the area between these two “great” flows [S]. Numerical modeling such as computational fluid dynamic (CFD) can apply the specific related equations which are usually Navier-Stokes equations. These equations can be solved along the turbulence model. For that, the present work will investigate various spillways geometry which represent an essential factor in dam’s design, these illustrated in Figure 1.2.

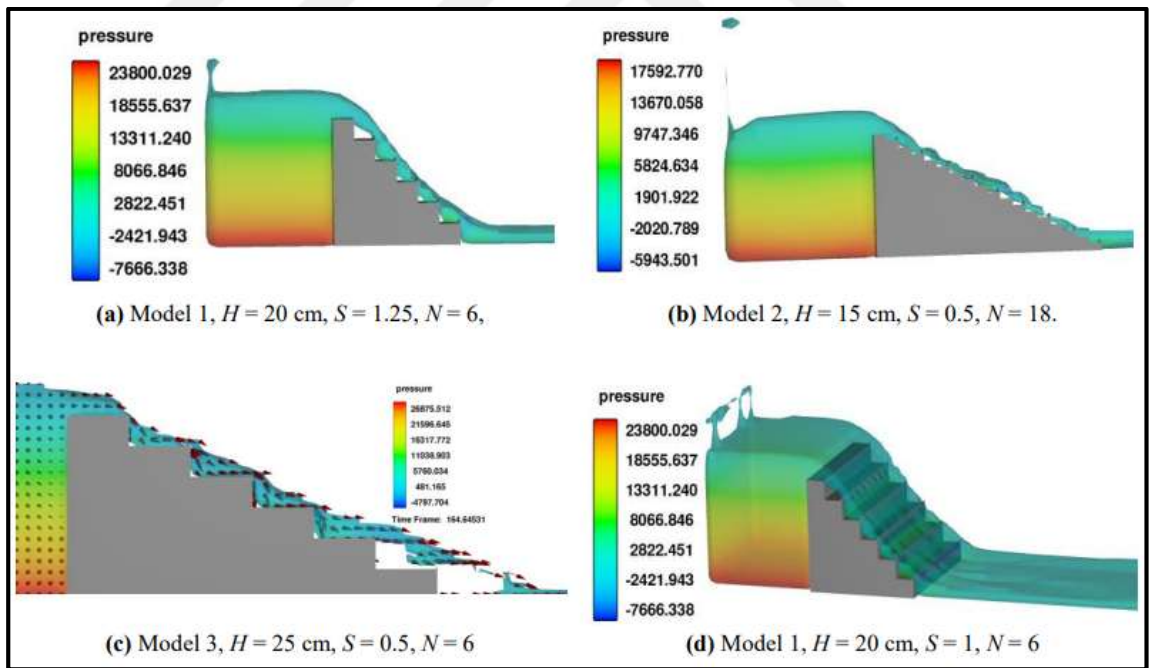


Figure 1.2 The pressure effect on various step spillways (Jamel 2018)

1.3 Stream Kinds of Flow in Spillways

Spillways are structures that can be used to evacuate overflowing of water flow dams. The high velocity represents the main important hydraulic problem in the field of water flow in spillways which causes cavitation at its downstream. For that, the flows along stepped spillways considered an interesting concern in the previous studies. The researchers found that the napped regime in the flow over spillways observes three sub-kinds categorized by the creation of hydraulic jumps on the bottom of the stairs. They also found that the second regime (the transition regime) which occurs when the flow regime changes from nappe to transition has no sub-division while the scrape flow system is sub-divided based on the dimension of the steps and the flow circumstances and observe various flow configurations in the fields near the steps as shown in Figure 1.3.

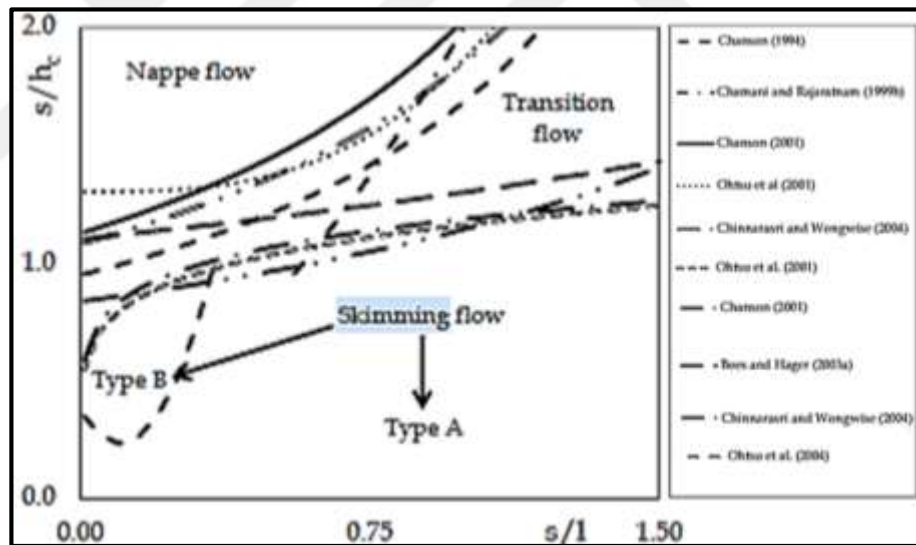


Figure 1.3 Criteria for stream kinds from various authors (Simões *et al.* 2012)

The parameters of the three standards flow conditions may vary based on the geometric conditions of entry into the channel, the steps size and conditions, the length of the channel based on region steps and finally the flow rates. It occurs by considering specific conditions such as the variations in steps length in relation to their height and bottom flows (related to scrape flow). The “transitional flow” between the backflow flows and the scrape flows is also known as the area among these two “extreme” flows.

Numerical modeling such as computational fluid dynamic (CFD) can apply the specific related equations which are usually Navier-Stokes equations. These equations can be solved along the turbulence models.

1.4 Problem Statement

Stepped spillways can be defined as a hydraulic structure that have regained significant interest for dam engineers and researchers, especially because of the technological progress in building dams (Amador *et al.* 2014, Varagilal 2018). Placing concrete in the elevators allows steps to build a rapid drain on the facades of the embankment downstream. In addition to construction economy, it means higher energy dissipation along the channel compared to a traditional drain channel. This phenomenon causes a decrease in the dimensions of the static basin at the end of the water drain (Aal *et al.* 2018, Parsaie and Haghiabi 2019). At high flow rates, potential structural damage can occur due to the dynamic loads caused by the flow. These damages are thought to be mainly due to cavitation effects. Therefore, a better view of the complex flow behavior is wanted to improve the design of graded outflow paths.

For that, to achieve the function of a downstream stream, the passage of water from the tank to the downstream must be designed in a specific way. The evaluation and prediction of the discharge leads to high efficiency and accuracy (Niu *et al.* 2021). Also, probability of reducing the discharge that can cause serious damage in the system should be minimalized (Ali and Yousif 2019, Meireles 2015). This solution provides a reduction of the downstream dissipation structure. The main problem is the correlation among the water flow characteristics and the spillway shape and geometry (Raza *et al.* 2021, Barani *et al.* 2005, Saleh and Husain 2020). The optimum spillway shape and dimensions can present an optimum spillway design to increase the dam safety.

1.5 Objective of Study

For achieving present problem solution the investigation objective can be specified with the two points:

- i) To investigate the state of the art stepped spillways types and conditions and to find the main function parameters of a spillway.
- ii) Build a simulation model to investigate various types of spillways geometry effects, the flow patterns and regimes by using ANSYS FLUENT program.

1.6 Thesis Organization

The present thesis involves five chapters. Chapter one includes an introduction to spillways types and flow behavior. Chapter two presents a general view of spillways dam, effects of water flow behavior using various studies collected from vary researches.

The experimental software program, involving the effective studied parameters, the work methodology, and testing of CFD model are presented in Chapter three. Chapter four discusses the result and the calculation of the work. Chapter five is the conclusion.

2. LITERATURE REVIEW

The tiered spillway plan is used in hydraulic constructions that have a crucial application. Graded flow ducts are used as an exit to the overflow or flood discharge where gravity dams and dams with various types of channels and slopes are employed to allow safe flood water discharge and to prevent erosion of the water body downstream. The construction of small graded drains is suitable for the simple gabion technique. Gradient drops are generally used with uniform step height and flat horizontal steps while in some conditions the steps may be slanted or aggregated down or up. Step heights or channel slope can present various lengths to the spillway (Ali and Yousif 2019).

2.1 Stepped Spillway Flow Regimes

Backflow, transition flow, and scraping flow systems with rising discharge for a particular waterfall dimension can all be classified as stepped drain flows. The discharge channel setup, which comprises channel slope, step height, and discharge, determines how flow systems change. Finally, the flow system affects the amount of electricity dissipated. The waterfalls descend one by one on the steps at a laminar flow rate, and the rays fall one by one into the rear flow system. It can be divided into three types of breach flow subsystems (Felder and Chanson 2013). The transition flow regime in intermediate discharges exhibits perplexing behavior, with heavy scattering and start point. The present system of flow initially detected and investigated by Ohtsu & Yasuda in 1995, also it was explained by Elviro and Mateos in 1995 and Chanson in 1996 (Meireles 2015). Limited investigational studies have been made to characterize particular transient flow characteristics. Chanson and Toombes defined two transient flow subsystems in 2004 that differed in empty fraction distribution profiles.

Some researchers investigated the instability of the transient flow system for combined stepped spills (Jahad *et al.* 2017). The energy of the water flow is wasted in the transmission of shear stress from the prevailing fluxes. In general, drains are intended for excessive discharge in a scraping flow system, which is why the majority of exploratory research for this system has been completed. The flow of air and water can

be fully advanced at the lower end, and the concentration, speed, and depth of the air will attain a constant value in these uniform equilibrium settings.

2.2 Energy Characteristics

The substantial percentage of energy distribution alongside the chute is a crucial feature of stepped spillways. When compared to smooth drainage, drainage steps act as big features that increase flow resistance and result in more energy loss. Only cage and transmission flow systems can achieve the highest energy dissipation performance; nevertheless, flows instability can occur, which is why graduated flow steps are intended to enhance the discharge in a scraping flow system. Flow processes in air-scraping water flows effect energy dissipation, which includes water motions in step cavities that are kept from the prevailing flow by shear (Ashoor and Riazi 2019). The breakdown of air bubbles, the creation of water droplets, and the collision of droplets and bubbles are all dynamic processes came from the water flow interaction and the surface of the path which cause an energy dissipation in skimming flows has been estimated in numerous investigations in stepped spillways (Jamel 2018, Niu *et al.* 2021). The design guidelines for the maximum efficient power dissipation are based on the discharge and channel slope, a stream of water may dissipate up to 90% of the overall energy. The researchers suggested that is a consideration of the residual energy can be more appropriate than the rate of power dissipation in engineering uses.

A comparison of the different channel droplets for moderate stepped discharges revealed that the bigger moderate channel droplets had lower specific residual energy (Saleh and Husain 2020). The results were independent of the step altitudes, indicating that the channel slope can be changed to get the best energy dissipation action. In the current thesis, the energy dissipation action of graded ducts with slopes ranging from 3.4 to 26.6 degrees is compared (Felder and Chanson 2013). The flow resistance is inextricably tied to the power dissipation. Scrape flow resistance is dominated by recycling and unstable momentum exchanges between dominant and cavity flows. Scrape flows are characterized by high flow resistance, which is mainly due to the shape losses. Factor of friction is utilized to measure the flow resistance. The flow resistance

of the graded channels depends on the dimension of the channel and is constantly higher than that of the smooth return channels.

2.3 Energy Dissipation

The upstream reservoir of water body in the high gravitational potential energy, under the released of downstream, the possible water power will be transformed to kinetic power. If the kinetic power is not dissipated to a significant level, the dam itself and its downstream river course will be subjected to potential damages by the high speed flows. Therefore, it is essential to take engineering requirements to reduce the flow velocity. The state of energy dissipater can achieve the goal, in which the energy is dissipated through diffusion, impact, turbulence, and internal friction (Uday *et al.* 2020). A flow of water in case of hydraulic jump stilling basin is one of the most used energy dissipation condition as shown in Figure 2.1. A hydraulic jump is a phenomenon where the flow undergoes a sudden change from very critical to a low critical. This is characterized by turbulence condition and mixing the air with water, within a large portion of the kinetic energy is dissipated (Parsaie and Haghiabi 2019). A stilling basin is a pool that facilitates the formation of hydraulic jump (Johansson 2007).

A trajectory bucket considered an effective energy dissipation condition. This structure generates a water jet in the air. Resistance of air and entrainment dissipate part of the energy. In addition, the dissipation also takes place through impact and pool diffusion. This type of organization is usually subjected at the spillway toe, for which a sound riverbed is required, in this case the scour is a major concern. The energy dissipater is the third type of solid roller bucket. It is suitable for a site with large tail water depth. The bucket is submerged and produces a bucket roller followed by a ground roller. The flow interaction between the rollers is the main source of energy loss. For shaft and tunnel spillways, hydraulic frictions and swirling devices are two options (Jahad *et al.* 2017).



Figure 2.1 Turbulent flow and energy dissipation (Salmasi and Samadi 2018)

2.3.1 Hydrodynamic pressure

Hydrodynamic pressures are caused by the flow, it would lower the discharge capacity and produce cavitation erosion. In crest shape condition, there should be exactly the same as the lower nape of the flow, so that the value of pressure on the crest is zero. The ideal profile is not possible to recognize for various heads. It is usually enhanced which can provide low pressures to develop. Due to the design discharge, the effective pressure is high at a low head, while it becomes lower at a high head. For the perspective of hydrodynamic pressure, turbulence flow should be avoided, as any pressure disturbance field, the result is in large pressure fluctuations, the driven effective forces of cavitation damage and scour (Zeng *et al.* 2022).

2.4 Stepped Spillway Designs

Consideration is given to design of stepped water drains based on a scraping flow system for greater drainage capacities; unstable flows should be avoided in the transitional flow system. The instructions can provide a significant information about

the flow scraping, air trapping location based on start point L1, the net water flow depth in two cases and the mean flow speed U_w . The other parameters also will be specified such as the equal fraction factor, flow rate, average air consideration and properties of flow depth. The watercourse design will observe the stream height, width, step height and slope. Likelihood of estimating the power dissipation along the gradient stream and determining the remaining head at the stream's end for an acceptable pour-sump design is based on this data (Jahad *et al.* 2017, Van Alwon *et al.* 2017). The safety factor can also be used to determine the size of the graded drainage sidewall. Several principles have been proposed based on various experimental results that provide adequate clues for graded channels in a scarping flow system for a specific set of slopes. The high number of design criteria limits the accuracy of pre-design calculations, and a practical trial in a massive size specimen yields more precise results for the entire design. Furthermore, the design standards only apply to flat steps, and varied step layouts necessitate further testing. In the current study, several configurations of uniform steps, irregular steps, and varied configurations of mixed steps are discussed for graded spillways (Niu *et al.* 2021, Morovati *et al.* 2016).

2.5 Flow Characteristics on Stepped Spillways

A spillway having steps on the chute is known as a stepped spillway. The past decades have seen a growing interest in it. This is due to the progresses in the Roller Compacted Concrete (RCC) technique for dams construction. Steps are also used on a downstream slope of embankment dams for passing floods. The steps perform as roughness components and heighten the bottom friction. As a result, a substantial amount of the kinetic power of the flow is wasted, leading to a reduction in the dimensions of the downstream energy dissipater (Zeng *et al.* 2022). In flow conditions and step geometries, three kinds of flow regimes can happen in a graded spillway. When the flow rate is small or the channel slope is relatively flat, a nappe flow is observed as in Figure 2.2. In this regime, a succession of free falling jets impinges on the steps, and a partial or full hydraulic jump might be formed on the steps. Energy loss is achieved by jet break up, flow mixing and hydraulic jumps (if formed). As the discharge increases, a transitional flow occurs, the cavity under the nappe is unobvious. The jets still hit the

step surfaces, causing a flow stagnation. Splashing and chaotic appearance are the primary phenomenon of this type of flow regime. When the discharge is large enough, the cavity completely disappears and the entire steps are filled with water. The flow skims over the steps as a coherent stream. This is called skimming flow. Stable recirculating vortices are developed under the artificial bottom made by the step ends. This regime is the most relevant one in engineering applications (Ghaderi and Abbasi 2021).

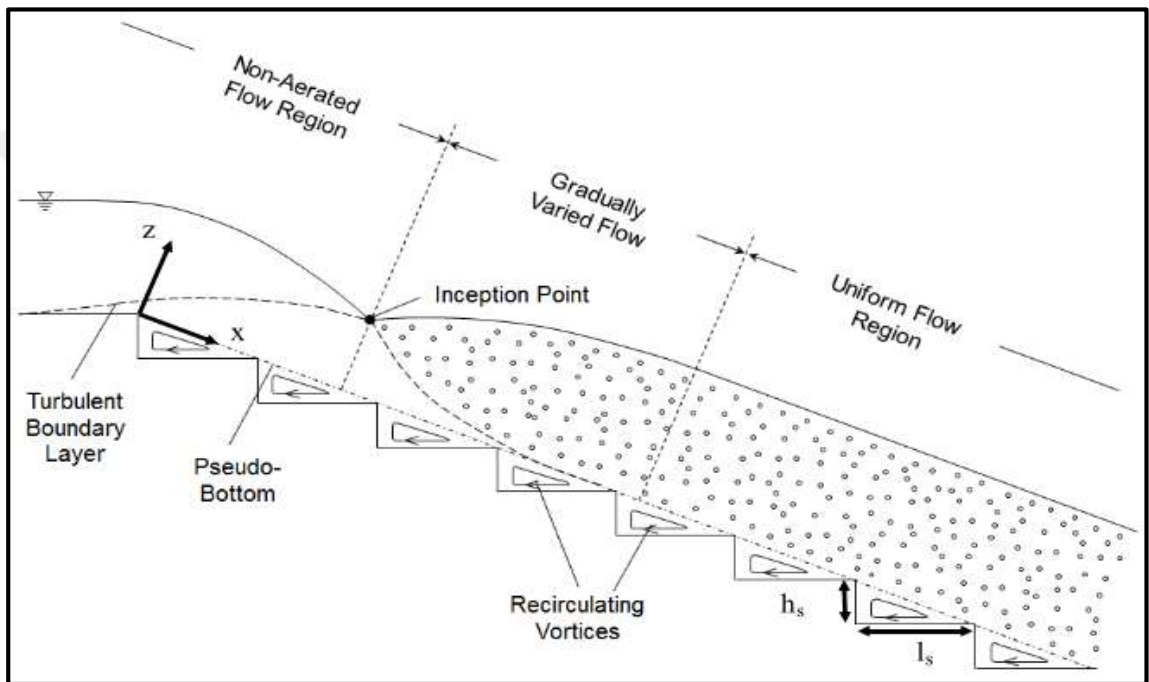


Figure 2.2 Flow regimes in stepped spillways (Van Alwon *et al.* 2017)

2.6 Discharge Capacity

Analytical methods or physical models are generally used to estimate the discharge of a spillway. The discharge prediction is based on the types of control structures. For example, if the flow uncontrolled and has a free surface as in Figure 2.3a, the discharge is determined by (Bautista *et al.* 2003), also as shown in Equation (2.1).

$$Q = CLH^{1.5} \quad (2.1)$$

where Q = flow rate, H = water head above the crest, L = crest length, and C = discharge coefficient. If the flow is controlled, a constriction of the wetted area is created, leading to changes in the pressure and velocity as in Figure 2.3b. The discharge is calculated using the Equation (2.2) (Morovati *et al.* 2016).

$$Q = CA \sqrt{2gH} \quad (2.2)$$

in which A = area of the opening, H = head above the orifice opening centerline, and g = gravitational acceleration.

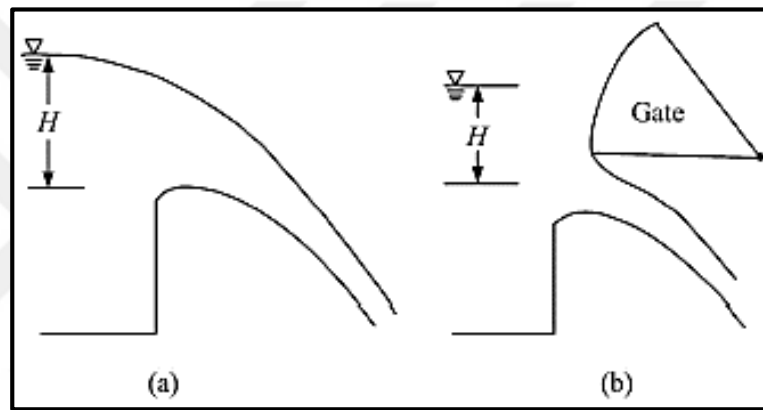


Figure 2.3 Control structure

2.7 Stepped Spillway Flow Model

Due to the high capability of computers, there has been a dramatic increase in interest in numerical modeling in the fields of hydraulic engineering and fluid mechanics. Today, many complex flows and turbulent flows can be calculated. But numerical modeling of graded flows is remaining in its infancy in spite of initial attempts. Several successful investigational investigations have enhanced our understanding of graded flows, and physical modeling is still the only way to better comprehend the micro and macro aspects of water and air flow (Guenther *et al.* 2013).

Graded spillway flows have a number of different restrictions that can't be fully modeled without working at full size. The maximum significant air-water flow

characteristics can be classified using a geometric evaluation of the corresponding limits (Guenther *et al.* 2013).

$$C, \frac{v}{\sqrt{g \times h}}, \frac{u}{v}, T_{int} \times \sqrt{\frac{g}{h}}, \frac{L_{xz}}{h}, \frac{d_{ab}}{h}, \dots =$$

$$F_1 \left(\frac{x}{h}, \frac{y}{h}, \frac{q_w}{\sqrt{g \times h^3}}, p_w \times \frac{V \times D_H}{\mu_w}, \frac{g \times \mu_w^4}{\rho_w \times \sigma^3}, \frac{W}{h}, \theta, \frac{K_s}{h}, \dots \right) \quad (2.3)$$

The water discharge for each unit width is known as q_w , the similar diameter of the pipe or hydraulic diameter is known as D_H , and the water density and dynamic viscosity are known as ρ_w and μ_w , respectively as shown in Equation (2.3). K_s is the covering roughness elevation, W is the descending width, h is the step elevation, θ is the angle between the artificial end and the horizontal end, and ϕ is the angle between the artificial end and the horizontal end. On the left, engineering factors are merged with the Froude Fr , Reynolds Re , and Weber We numbers, while on the right, engineering factors are combined with the Froude Fr , Reynolds Re , and Weber We numbers as shown in Equation (2.4). It is possible to use the Morton number Mo as a substitute:

$$Mo = \frac{We^3}{Fr^2 \times Re^4} = \frac{g \times \mu_w^4}{\rho_w \times \sigma^3} \quad (2.4)$$

Nonetheless, the Froude and Reynolds numbers in model and prototype cannot be instantly matched until used at full scale, therefore identical dynamic likeness is not attainable. Because the gravitational influence is the most complex, studies in free-flowing streams, which include graded flows, are frequently based on Froude simulations. The air introduction procedure, on the other hand, is influenced by the huge scale in tiny size samples. To eliminate and define these scale effects, precise measurement requirements in the model structure and regular observation of the flow structure are required. Previous experiments utilizing Froude, which were carried out analytically in similar geometric graded spillway models, revealed significant scale effects. In studies based on undistorted Reynolds similarity, Felder and Chanson in 2013 highlighted the scale impacts. The current study was based on a Froude

simulation, and Equation (2.5) may be reduced if the fixed parameters were taken into account:

$$C, \frac{v}{\sqrt{g \times d_c}}, \frac{u}{v}, T_{int} \times \sqrt{\frac{g}{d_c}}, \frac{L_{xz}}{d_c}, \frac{d_{ab}}{d_c}, \dots = F_2 \left(\frac{x}{d_c}, \frac{y}{d_c}, \frac{d_c}{h}, Re \right) \quad (2.5)$$

Where $d_c = q^2/3g$ is the critical depth, and it shows the Froude number. In the Froude similitude, this phrase refers to the discharge's dimensionless shape.

2.8 Spillway Types

Spillways are normally classified according to their geometrical layouts (ogee, chute, stepped, shaft, etc.), functions (service, auxiliary and emergency) and control structures (gated, ungated and sluice). Vischer and Rutschmann develop a systematic classification approach considering all the essential elements of the channel, such as entrance, regulation or control, discharge transporter, and opening. According to Vischer and Hager, spillways are classified in a simple way: frontal spillway, side spillway and shaft spillway (Li 2021).

2.8.1 Ogee spillway

One promising solution is to think of a graded drain instead of the conventional, even-flow drain, where a series of droplets are reflected from the top to the toe. Under a varied collection of working conditions, graded drain construction is estimated to do important power loss to the lower structure itself, decreasing the requirement for a more expensive stilling basin. The steps perform as roughing elements to decrease the acceleration of the flow and thus the final speed. Furthermore, too much turbulence brought by the steps helps speed the advance of a boundary layer and air intake to increase flow volume. Together the decreased speed and the cushioning influence of the entrained air therefore decrease the cavitation potential as shown in Figure 2.4 (Chatila and Jurdi 2004). An ogee spillway has a high foundation requirement and is often used

in the main river channel. The structure consists of a crest, a rear slope and a toe (Karlsson and Tallberg 2011).

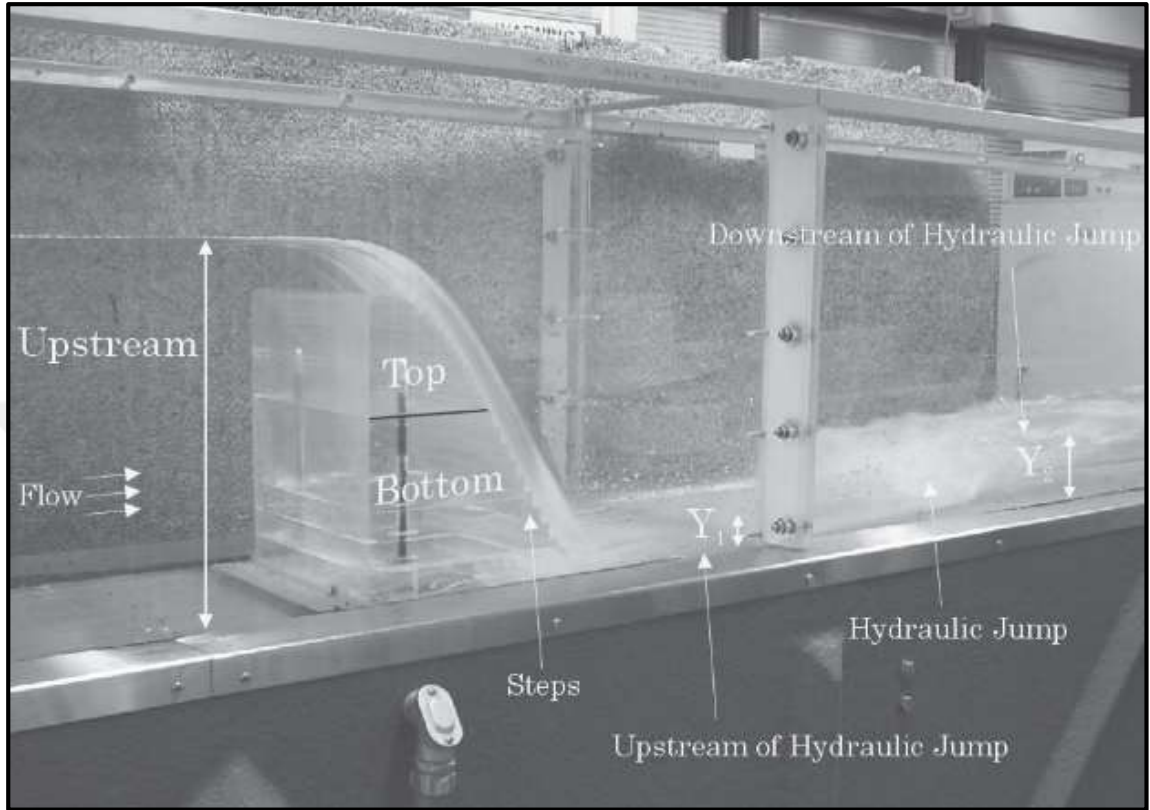


Figure 2.4 Ogee spillway (Chatila and Jurdi 2004)

The design of the crest shape is one of the main concerns. Extensive experimental tests are performed in the US Reclamation Bureau (USBR) laboratories to optimize its design. The crest profile is described in Equation (2.6) below (John LaBoon 2014).

$$\frac{Y}{Hd} = \left(\frac{X}{Hd}\right)^n \quad (2.6)$$

in which Hd is design head, K and n are constants dependent on upstream face inclination and approach velocity, shown in Figure 2.4. The Waterways Experiment Station (WES) at the US Army Corps of Engineers proposes several standard spillway profiles. They are designed based on the Equation (2.7).

$$X^n = KH_d^{n-1} Y \quad (2.7)$$

Where K and n are limits that are dependent on the slope of the upstream direction, and X and Y are coordinates as in Figure 2.5. However, if the approach flow is too deep or shallow, there is no universal design procedures to follow. To address the discontinuity issue at the intersection of the crest and the upstream face, a revised design is developed by Murphy. The upstream quadrant is in an elliptic shape defined by Equation (2.8):

$$X^2/A^2 + X^2/B^2 = 1 \quad (2.8)$$

in which A = half-major axis and B = half-minor axis. The downstream crest shape is constructed according to the Equation (2.9).

$$X^{1.85} = KH_d^{0.85} Y \quad (2.9)$$

where K is a constant depending on the approach water depth and design head. For $P/Hd \geq 2$, $A = 0.28Hd$ and $B = 0.164Hd$.

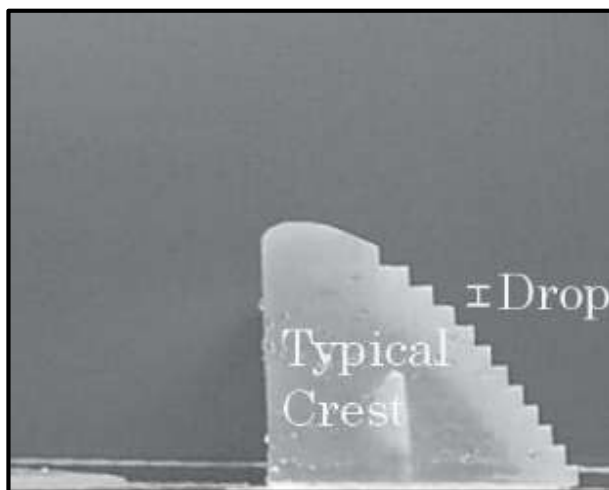


Figure 2.5 Typical ogee crest profil (Chatila and Jurdi 2004)

2.8.2 Chute spillway

A chute is a discharge channel downstream of a control structure. Typically, a chute spillway is composed of an approach channel, a chute, and a tail water section, as shown in Figure 2.6. The dimension and profile of the chute are mainly dependent on the hydraulic requirements. The geological and topographical features also have an impact on their selection. A chute spillway can be either straight or curved. Its cross-sectional profile can be a rectangular or trapezoidal shape. Ideally, it should be straight, as the flow in it is usually supercritical. Any constriction, expansion, or curvature would lead to unfavorable flow behaviors, such as shock waves. Nevertheless, owing to geographical restrictions or for energy dissipation purposes, contractions or expansions in width might be applied to a chute spillway (Amador *et al.* 2004).

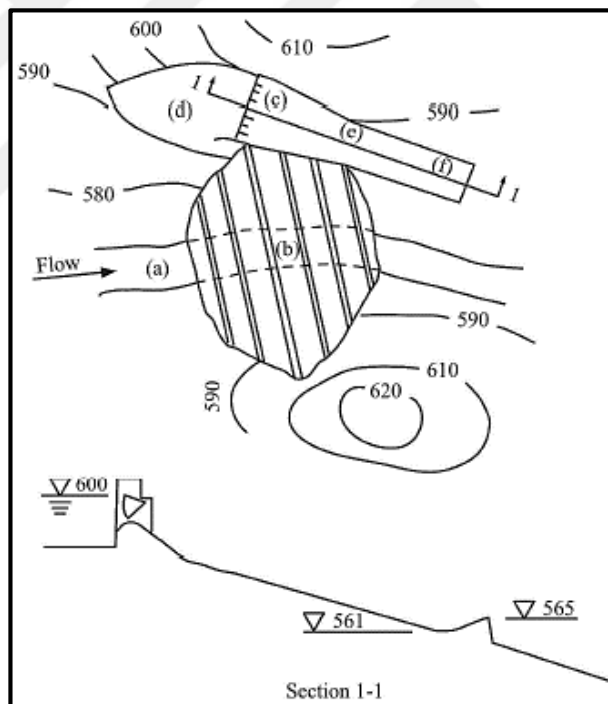


Figure 2.6 Chute spillway (Jahad *et al.* 2017)

Figure 2.7 shows three types of contractions in chute spillways. It should be noted that cross-wave is a potential risk resulting from the deflection the supercritical flow. It can cause uneven water surface and high air entrainment. In a prototype, engineering measures should be taken to achieve flows as uniform as possible.

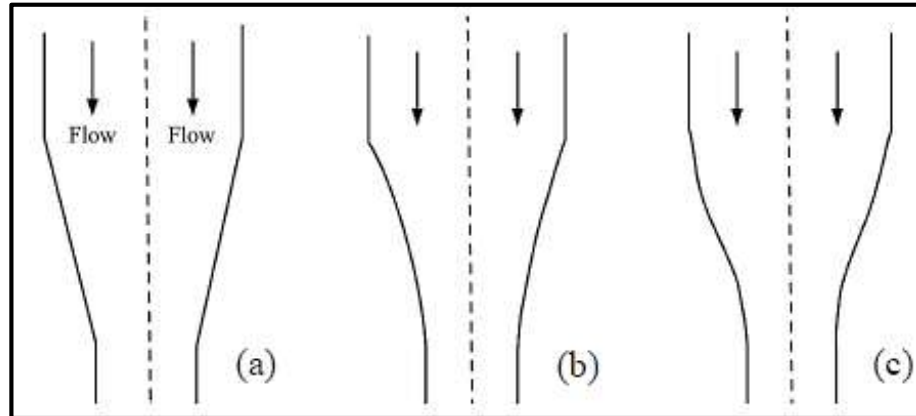


Figure 2.7 Types contractions in chute spillways

2.8.3 Shaft spillway

A shaft spillway is a closed conduit transferring water from a high to low elevation. A water drainage canal can be constructed in several methods. It usually consists of a vertical shaft or, in some situations, an inclined shaft. The drain duct is very useful when discharging a large amount of water from high tanks (Karlsson and Tallberg 2011). It is widely used in dam engineering, erosion-control structures, and highway culverts. Due to its vertical arrangement, this type of spillway is advantageous in narrow canyons with steep abutments (Figure 2.8). In addition, a shaft spillway is capable of operating at its near-maximum capacity even at low heads, resulting in a significant attenuation of floods. Air entrainment is the most important and complex issue in a drop shaft. It occurs when the flow from the intake hits the water column in the shaft. Its occurrence is closely linked to the flow regimes explained above. With the increase in water head, the air discharge first climbs and then drops to zero when it is submerged (Li 2021).

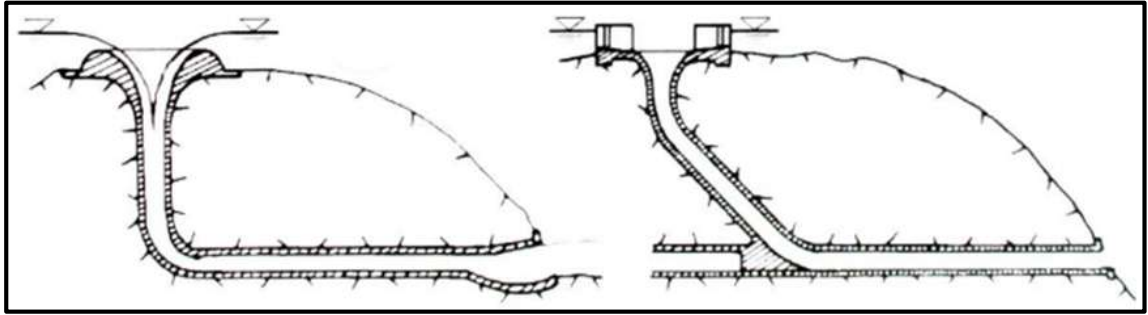


Figure 2.8 Shaft spillway (Karlsson and Tallberg 2011)

2.8.4 Labyrinth spillway

A labyrinth spillway features a number of slender walls. Its crest is prolonged with a weir in triangular, trapezoidal or rectangular shape in plan, these shown in Figure 2.9. It is designed particularly for conveying large floods at low heads, as the water can still flow over the entire crest length even at a low head. The stepped drainage has some long length steps comparing to the regular graded fall and also has a wider effective width. The proof of the numerical prototypes is founded on investigational outcomes of water surface, compression measurements and relative power waste. Simulations extend the understanding of flow phenomena over normal fall cycles and graded fall and make the flow performance more explicit (Saleh and Husain 2020). For instance, if a higher design flood is introduced, a labyrinth structure is an effective measure to upgrade the existing spillway in a narrow waterway. Recent successful applications of labyrinth spillways show their potential in improving discharge characteristics. However, as the water level increases, a labyrinth spillway suffers from limited discharge, which is due to the interactions of the flows from the upstream apex and the sidewalls. If the head is large enough, its discharge capacity will further drop close to that of a straight weir with a same channel width. Depending on the head, the flow patterns are classified as fully aerated, partially aerated, transitional, and suppressed flow.

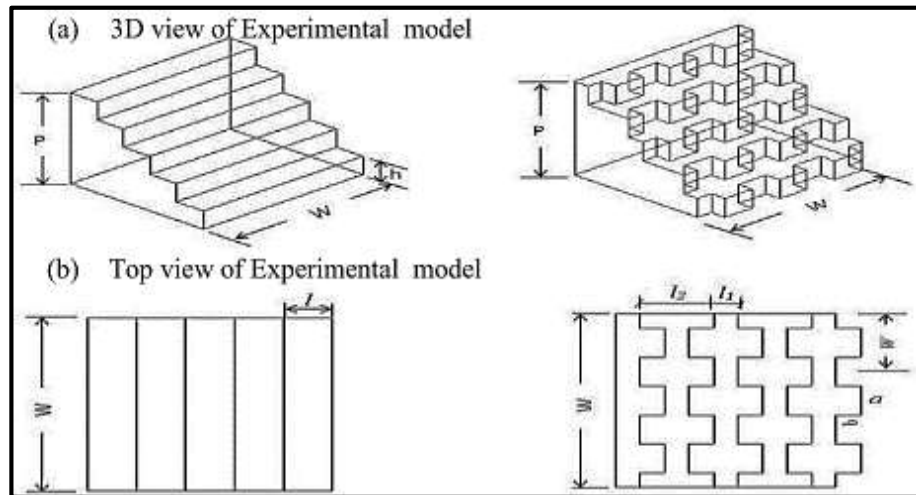


Figure 2.9 Typical labyrinth spillway (Karlsson and Tallberg 2011)

3. MATERIALS AND METHODS

This chapter presents the procedures of the project in order to investigate the water over stepped spillway behavior using computational fluid dynamics technique. Numerical methods correlated with computerized algorithms are used to explain and analyze problems involving water flow. The present study purposes to create a numerical model can investigate the water over stepped spillway characteristics in simulated spillway channel in order to find out the spillway shape effect based on water forces, water flow effect correlated with the spillway geometry. The calculations required for simulating the interaction of the dimensions of the drain stream with several effective parameters, and first conditions are performed by CFD. The approach technique of current work will discretized the model by partial variously equations in a limited range of control sizes. The solver will prepare the overall preservation equations for mass, momentum and power using CFD Analyzer. Current chapter presents all the experimental data and assumptions such as dimensions and the values of water flow throughout the project using the traditional methodology procedure as shown in the flow chart in Figure 3.1.

3.1 Model Development

This section will present the model implementation procedure. The ANSYS FLUENT software is used to solve CFD problems based on the selected model. In this section, the researcher designs the steps to create the simulation model based on the three spillway shape design. The symmetric type will be used as a baseline to investigate the water over stepped spillway behavior of the other two types (Figure 3.2).

The present steps contain the tasks of preparing the geometrical shape by sequins of operations starting with pre-processing step which is constructing to generate the mesh, followed by the physical model fix, boundary conditions, initial conditions, and additional suitable parameters that are defined in the modelling and solving step, and finally and post-processing step. The experiments can be used to obtain outcomes through the computational range in the post-processing step.

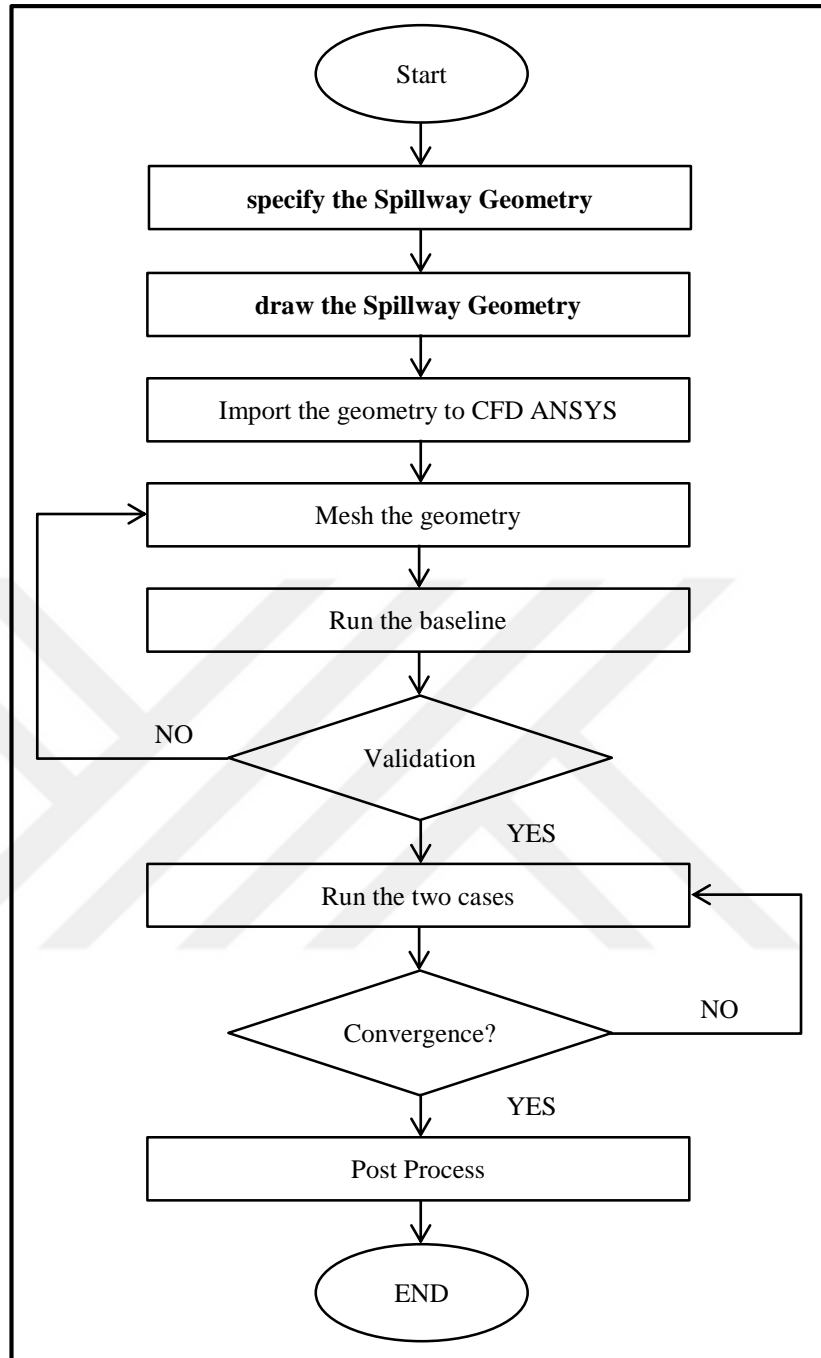


Figure 3.1 Flowchart of the project

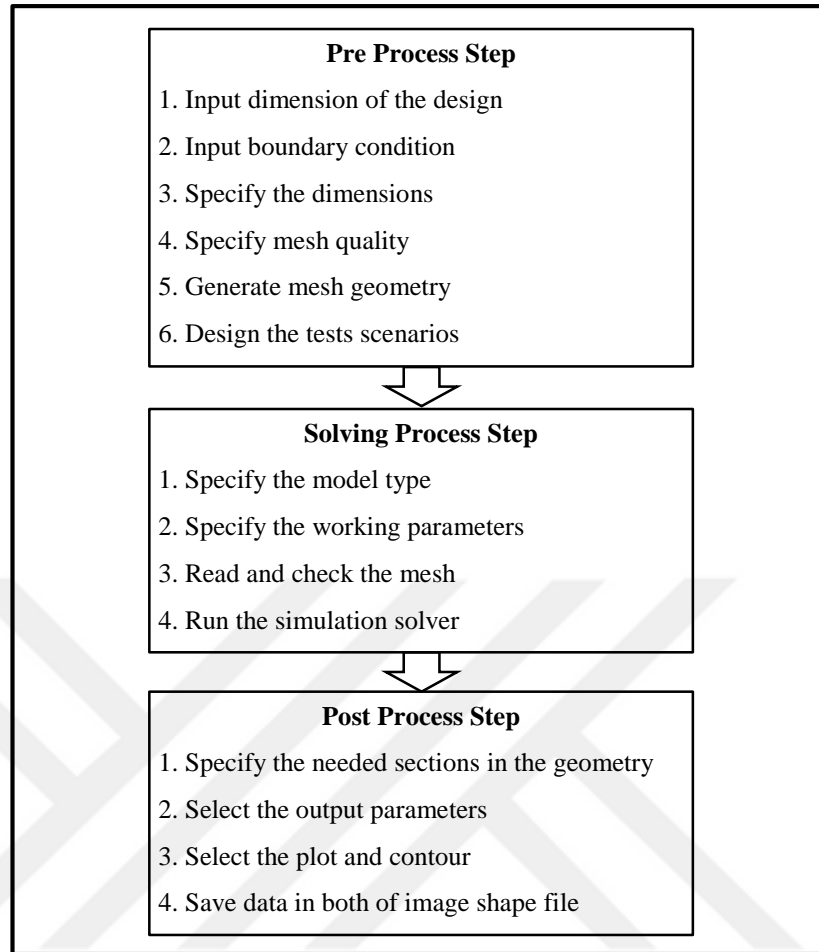


Figure 3.2 The CFD modelling sequence

3.2 Pre-Process Step

In order to prepare the specific ANSYS software, need to select a new Fluent fluid flow analysis system by double-clicking the fluid flow choices related to the analysis systems as presented in the Figure 3.3.

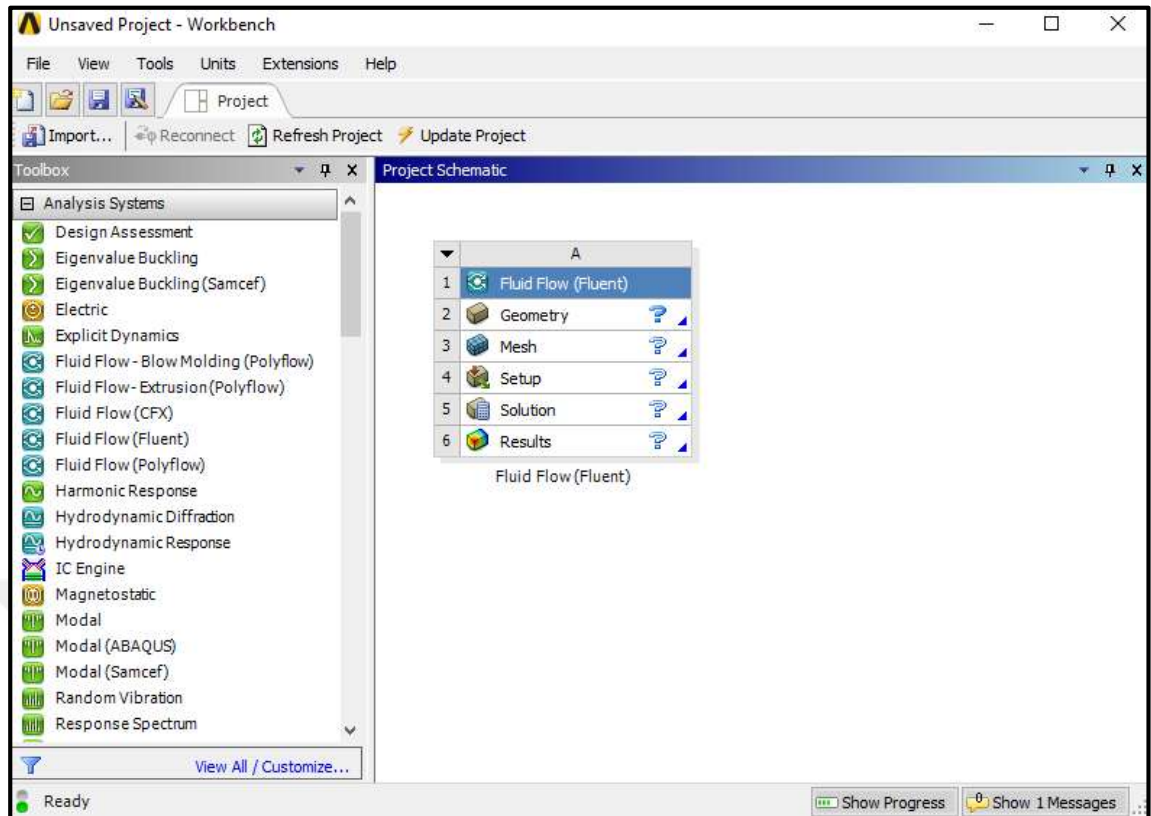


Figure 3.3 Fluid flow (fluent) files

The fluid flow analysis system consists of different cells (geometry, mesh, etc.) that characterize the workflow for accomplishment of the analysis. ANSYS Workbench consists of numerous data-combined and native uses in a single, seamless project process, where each cell can retrieve data from other cells and offer data to additional cells. Next step concern with creating the geometry.

Step 1: Creating the Geometry

This step is the first step to create the computational Spillway geometry. This step required the standard data set of the Spillway shape which is available in (Saqib *et al.* 2022).

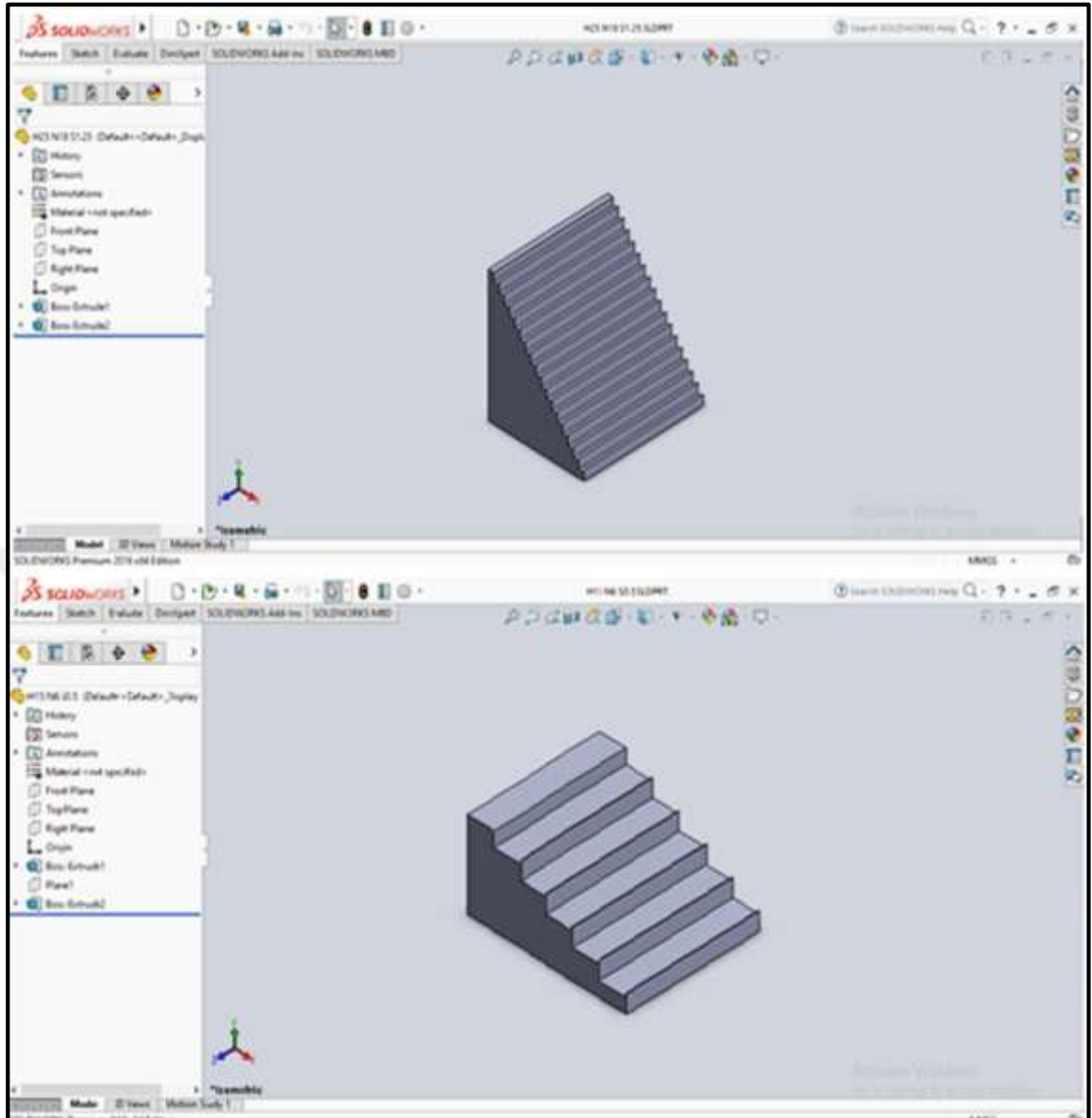


Figure 3.4 Generate the spillway shapes geometry

When the geometrical dimensions uploaded to the ANSYS Design Modeler, the spillway dimensions have specified. It is seen in Figure 3.4 the dimensions specified in two various spillway models. These dimensions represent the boundary conditions of the water over stepped spillway (Figure 3.5).

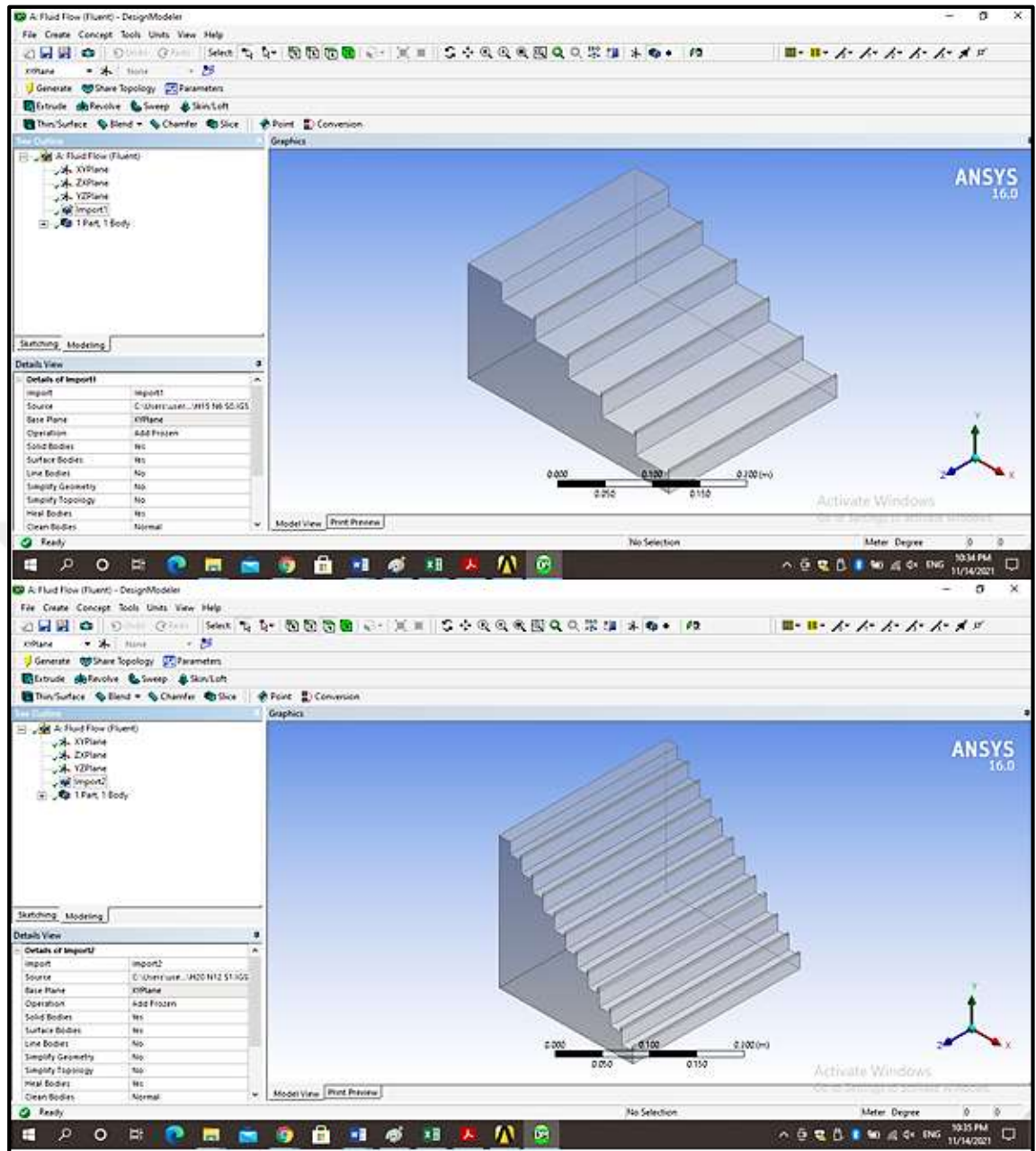


Figure 3.5 Spillway configuration

Step 2: Meshing the Geometry

Meshing the shape is divided into innumerable small finite number of elements. Meshing gives us a scope to study the behaviour of various parameters (such as pressure, velocity etc.) at each of these elements.

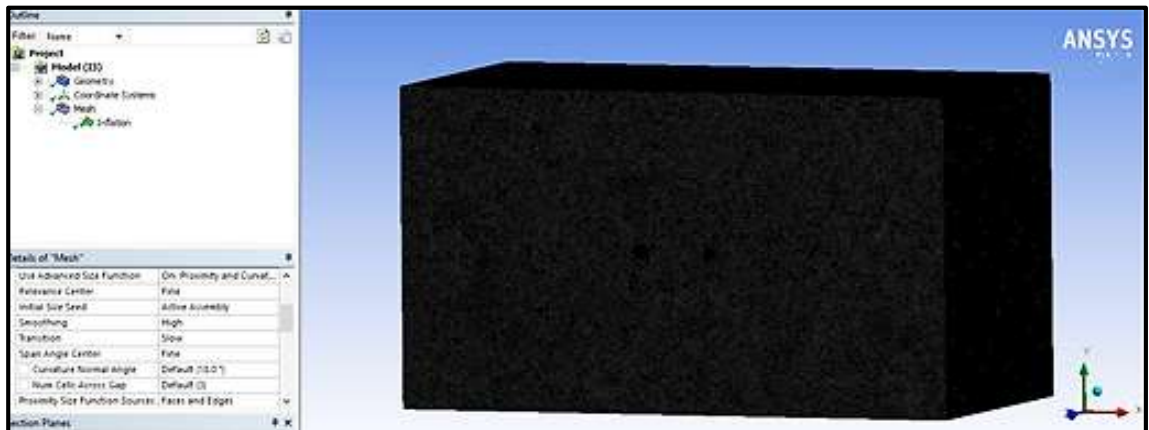


Figure 3.6 Meshing the geometry

The present procedure is to produce a computational mesh through the tunnel capacity. For this step, it necessary to utilize the ANSYS Meshing application in order to generate a mesh for the CFD analysis, Figure 3.6. The procedure to create the object mesh is:

- a) Select the ANSYS Meshing application in order to generate a mesh for the CFD analysis as in Figure 3.7.

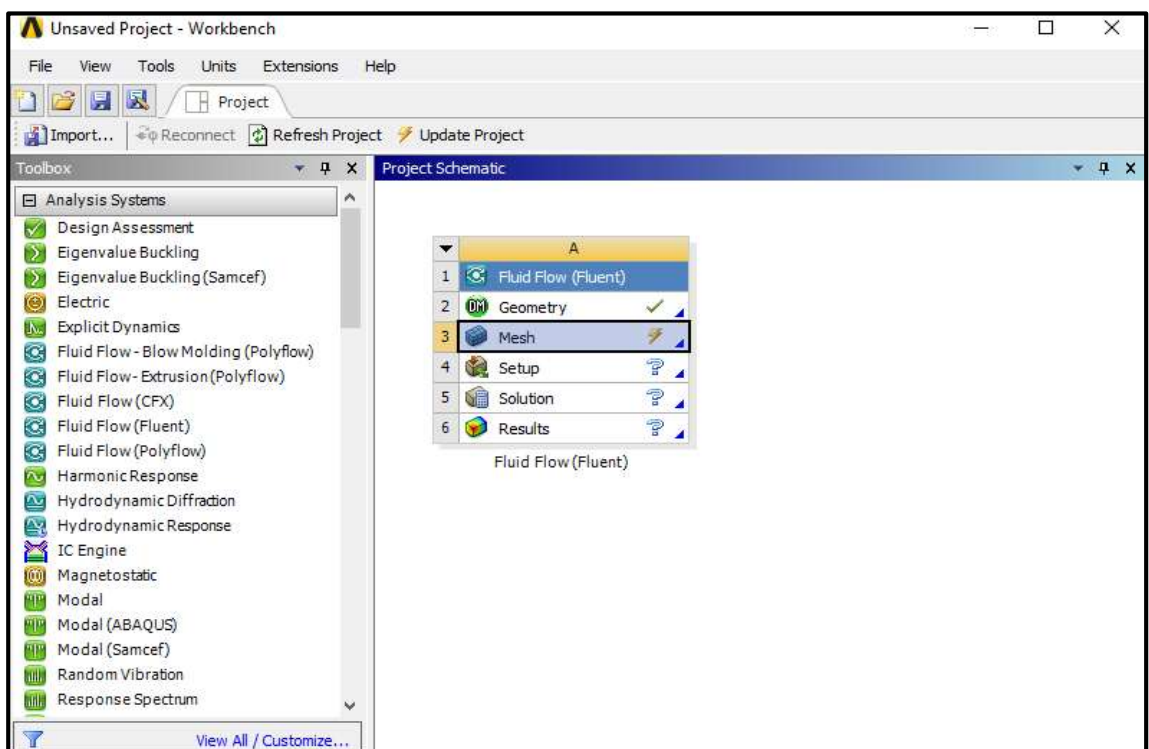


Figure 3.7 ANSYS mesh application

b) Specify the mesh quality control parameters in order to select the best size and type of mesh as in Figure 3.8. The results accuracy will depend on these parameters.

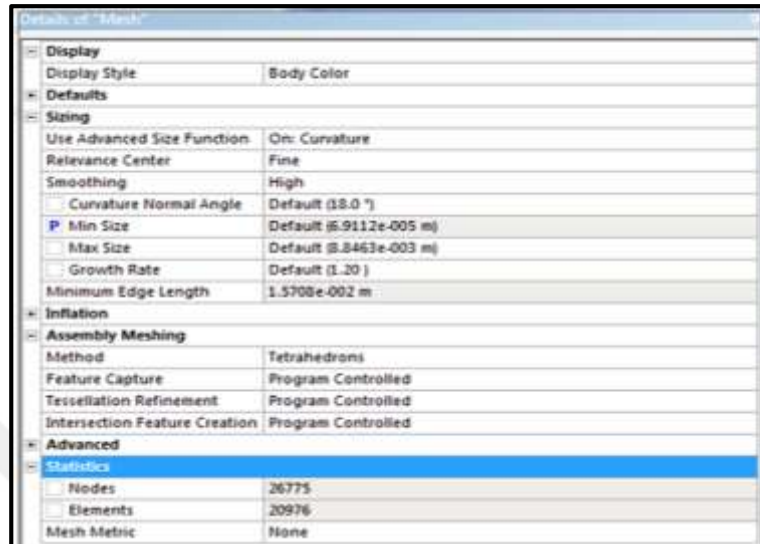


Figure 3.8 Mesh quality control parameters

c) In order to view the advance mesh statistics, the mesh metric will present the bar graph to observe the mesh element quality. The bar graph will specify the minimum and maximum element quality values as presented in Figures (Figure 3.9, Figure 3.10).

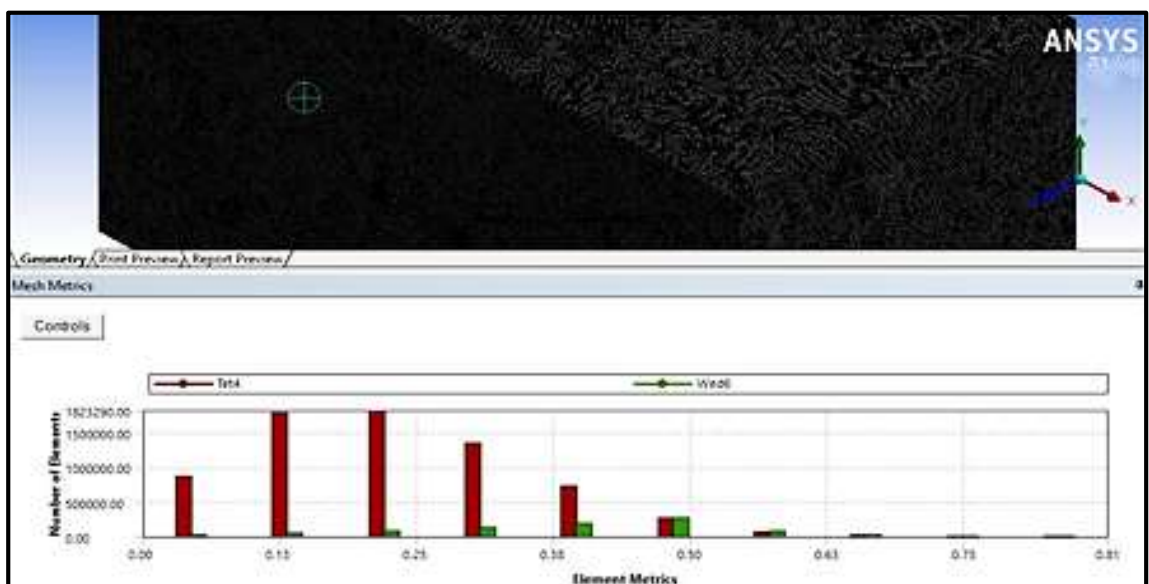


Figure 3.9 Mesh element quality values

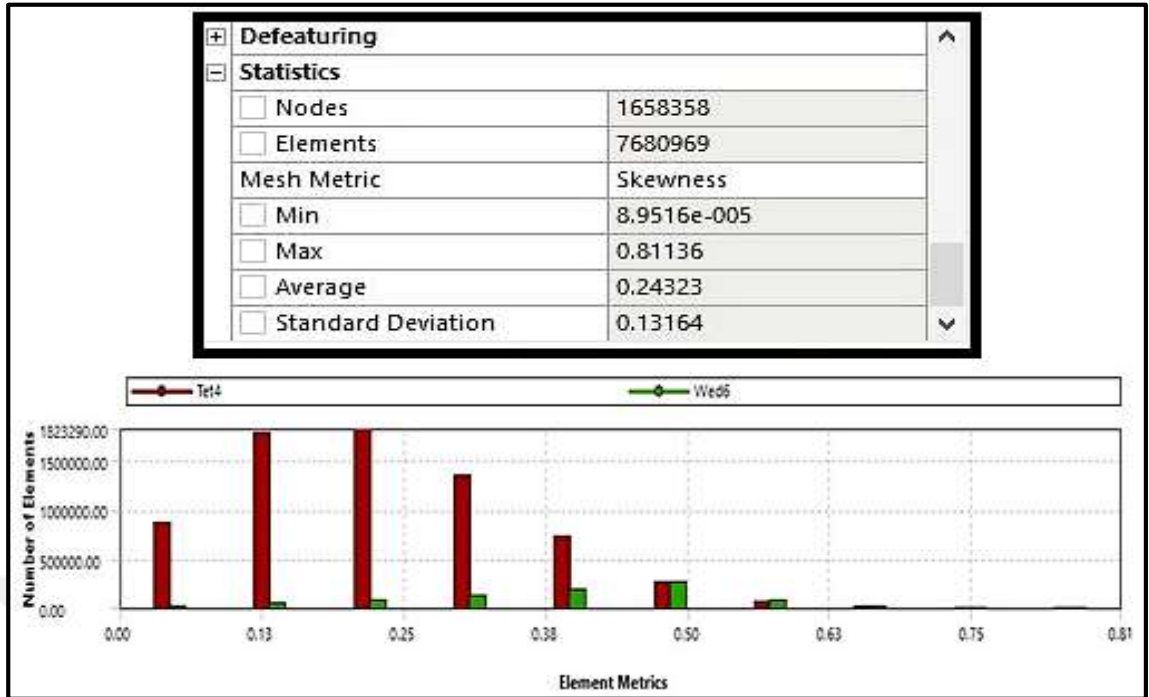


Figure 3.10 Mesh metric outcomes skewness

The best numerical mesh outcomes summarized in Table 3.1.

Table 3.1 Best numerical mesh results values

Parameter	Value
Elements	7680969
Nodes	1658358

Step 3: CFD Simulation Arrangement

The best mesh results for selected geometry will transfer to the CFD analysis utilizing the ANSYS Fluent. The procedure of this work is as in follows:

1. Start ANSYS Fluent

The Fluent Launcher permits start the ANSYS Fluent depended on the already processed geometry as shown in Figure 3.11.

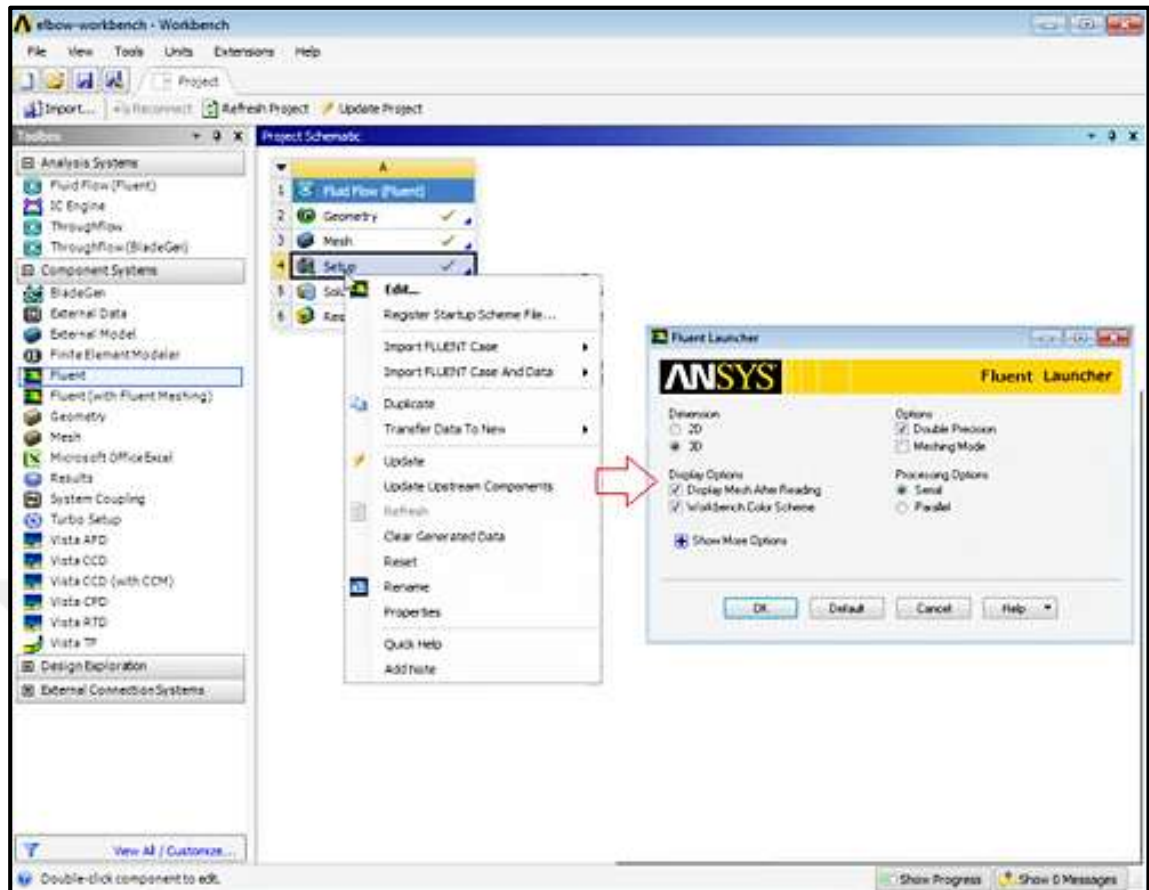


Figure 3.11 Fluent launcher

To solve the study scenarios, the general procedure will apply based on the meshed geometry conditions as in the sequence below:

- a) Change the units for object dimensions. This step involves:
 - i) Select length in the Quantities list.
 - ii) Select the millimeters in the Units list.
- b) Check the mesh results.

Specifies the water flow direction. In Figure 3.12 shown the steps.

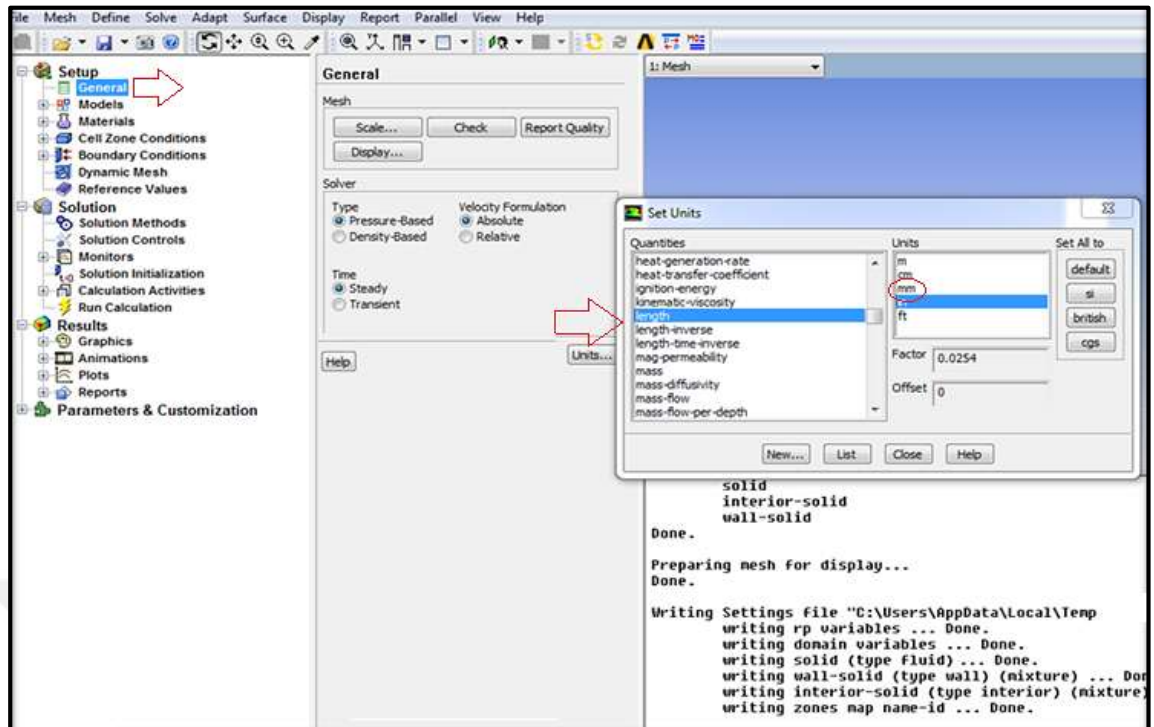


Figure 3.12 General CFD settings

Step 4: Setup the Model

In this step, the researcher will choose the model based on the previous studies which presented in chapter two. The model will cover the aerofoil requirements which can give all the needed information based on the working parameters in the selected type of spillway. The independent procedure specified as in the follow Spillway sequence:

- a) Select the model type from Models Code. In our case study the model will be the SST $k-\omega$ turbulence model as presented in chapter two.
- b) Enable the $k-\epsilon$ turbulence model; use the best k-epsilon Model list as in Figure 3.13.
- c) Create the materials using the Create/Edit Materials dialog box.

Step 5: Set up the boundary conditions

FLUENT software solves the 3D Reynolds-Navier-Stokes equations for time-averaged mass speed, pressure, and density.

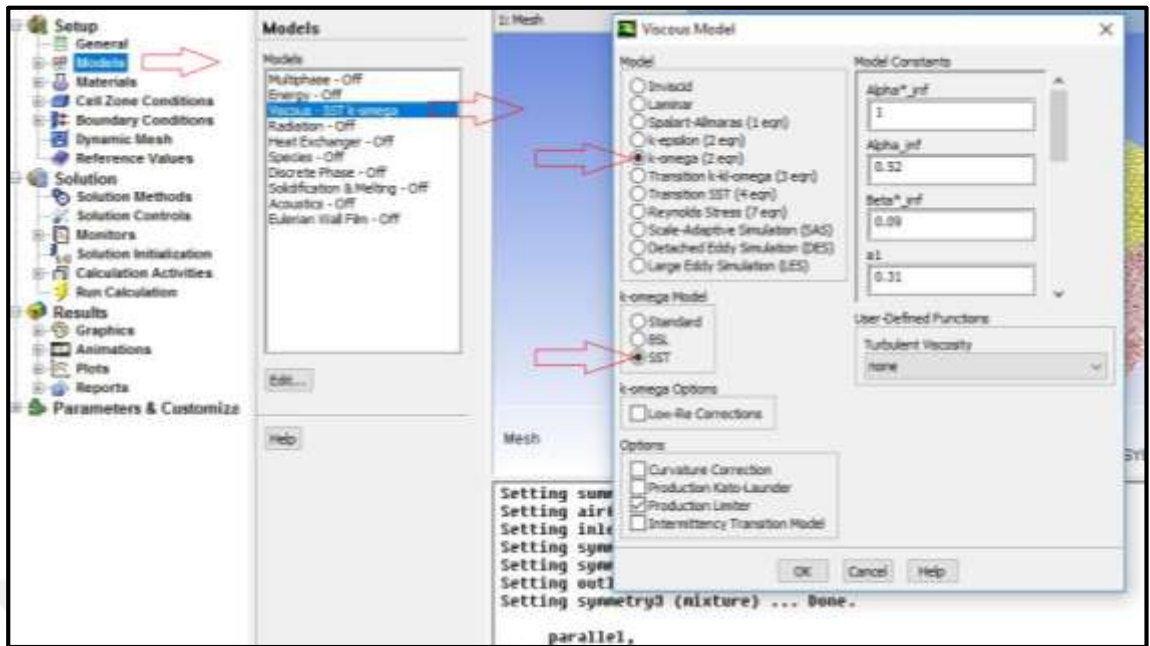


Figure 3.13 Model selection

The software is an integrated and complex Navier-Stokes fluid flow prediction system, able of varied and compound multidimensional fluid flow complications. It uses a flexible, multi-block grid system, a graphical interface and several sophisticated modelling tools, making it suitable water flow simulation. For that the main input representation for the present study is the water velocity as shown in Figure 3.14.

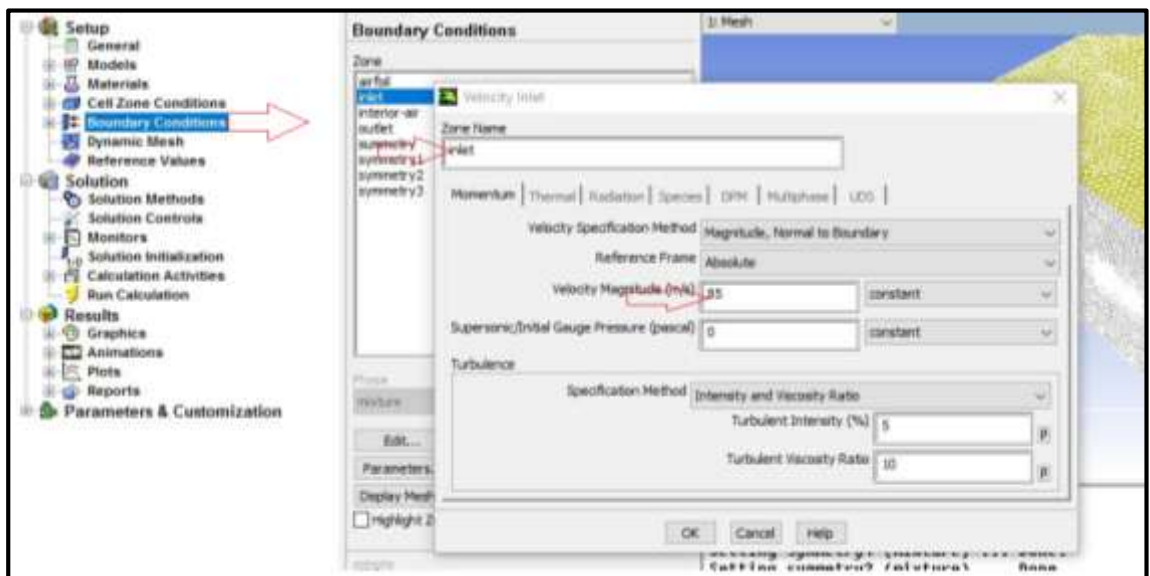


Figure 3.14 Set up the boundary conditions

The input representation will control the boundary conditions of the system simulation. The counties change in simulation parameters will be in this step.

Step 6: Run the solution

Run the solver is an important phase in ANSYS system. It is a powerful tool to describe the water over stepped spillway behaviour. The essential points in this step are the date of working variables, and the number of iteration. In Figure 3.15 present the details.

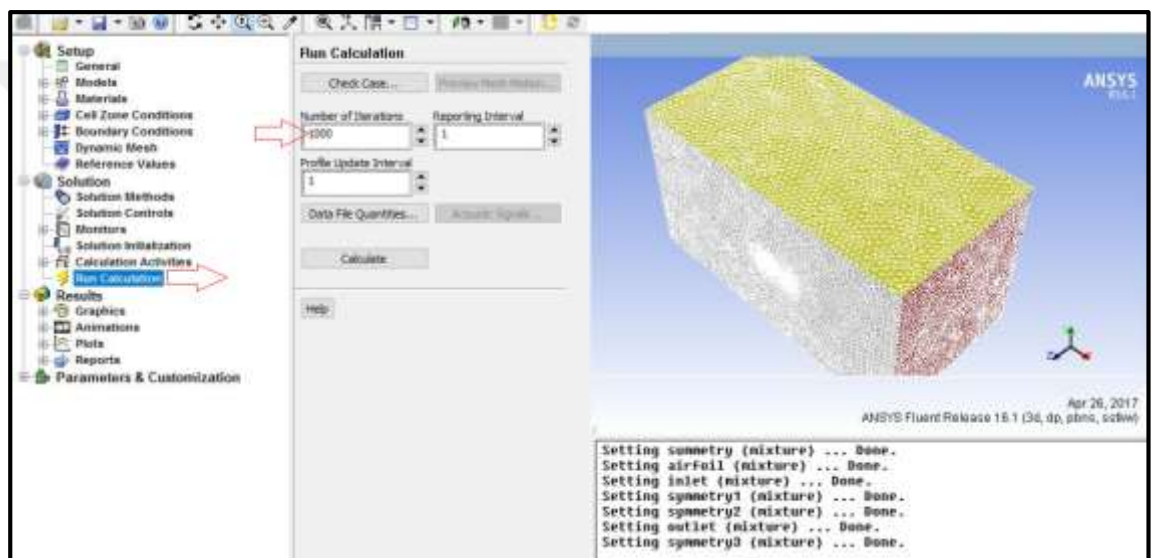


Figure 3.15 Run process

Fluid Flow Analyser, FLUENT offers solutions of incompressible, steady-state/transient, single-phase laminar/turbulent fluid flow solutions in compound geometries. Therefore, the system simulation can observe brief details about the possible changes in the scenarios which can present a clear water over stepped spillway differences.

3.3 Summary

This chapter presents the methodology of the present work. It involves the dataset web location, and the general procedure of the CFD ANSYS processes. Also, the sequence

of operation highlights and tests the mesh requirements. It provides the best mesh results and specifies the number of nodes and elements. This step is crucial to give the best simulation results. Based on that the researcher specifies all the essential parameters used to investigate the water over stepped spillway behaviour. At the end, the researcher validates the simulation work by comparing the with the previous results to ensure the methodology process.



4. RESULTS AND DISCUSSION

The hydraulic characteristic changes in the spillway effects have been investigated by various spillway design analysis. It typically requires understanding the spillway specific shape design to find the water behavior moving on the spillway surface which produces the traditional responses. The present work produces a set of simulation tests using various spillway and velocities. Numerical simulations are performed to study the influences of the step edges and cavities on the water flow features. The parameters affecting the mixture flows include step edge and cavity shape. The simulation applied by ANSYS CFD Fluent software which is validating by the baseline. This step is essential to consider the correctness of the CFD technique. Validation testing is a way to reduce and estimate modeling errors and to confirm that the CFD model being resolved is a respectable demonstration of actuality. For examining the effect of the difference of any given variable in the simulation the CFD simulation outcomes achieve the purpose of this study and offer brief discussion depended on spillway geometries. The first section presents the results of ANSYS and plots its own numerical outcomes, while the second section presents an analysis of the comparative study among the two existing types of spillway and looks at the variances.

4.1 Stepped Spillway Results

Numerical modelling is an extensively applied computer-aided technique the Navier-Stokes equation, which is based on the preservation of collective power, mass, and momentum. The stepped spillway prototypes are modeled using CFD software. The Navier-Stokes equation is estimated by Fluent software using the Finite Element Method (FEM). In CFD, the FEM methodology is the most extensively used method for preserving the dynamic equations. Instead of solving equations numerically, volume control technology converts the governing equation into algebraic equations. The volume control integration equations are solved in an embedded form, and the numerical simulation uses seven distinct types of spillway models with varied step sizes and configurations, as indicated in Table 4.1. For each example, the slope of all models is kept constant, and there are two types of slopes investigated in this study. The

discharge characterizes the stepped spillway model's entrance of boundary condition. Pressure, velocity, and friction volume are used to determine the spillway's opening boundary condition.

Table 4.1 Design parameters for numerical simulation of several tiered spillways

Model	Step Shape	Slop
M1	Triangle step	Type 1
M2	Triangle step	Type 2
M3	Rectangle step	Type 1
M4	Rectangle step	Type 2

4.2 ANSYS Evaluation of Triangle Steps (Case 1)

The Stepped Spillway is the most crucial component in the dams. For that, this study aims to investigate the aerodynamic characteristic in dam Stepped Spillway. The flow characteristic can be recognized by observing the water flow behavior in various types of Stepped Spillways and water velocity. Various shape of the dam Stepped Spillways produces various flow behavior. This creates a difference in velocity based on the physical law recognized as Bernoulli's Principle. While this velocity is trying to balance, the water flow is trying to move in the direction of the lower level. In order to characterize the aerodynamic forces acting on the dam Stepped Spillway. Current study observes the aerodynamic behavior around the darn Stepped Spillway by using CFD ANSYS. The part of CFD in design changes field of relies on the phase of study. Computational fluid dynamics in the shape of a two-dimensional structured mesh flow analyzer by using the Navier-Stokes equations averaging Euler and Reynolds. The numerical model is prepared and run with CAD software that creates boundary conditions for the analysis. Outcomes of analysis act a very significant part in optimizing design components so that there is a smallest velocity on the Stepped Spillway surface and swirl of water flow.

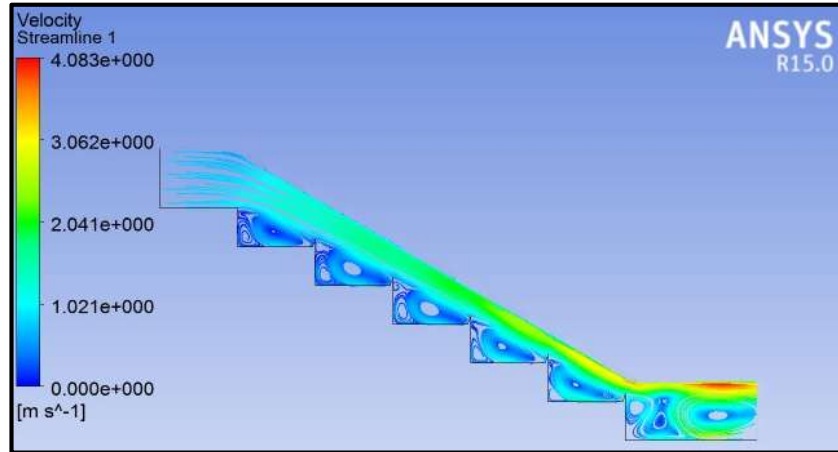
4.2.1 Triangle steps

Flow Pattern Vector plots and the stream line plots as shown in Figure 4.1 for the triangle steps case clearly represent the flow patterns over dam Stepped Spillway. The ANSYS software simulated the chosen cases by defining them according to boundary conditions. There are five basic properties employed in the CFD-ANSIS simulation process which are given in Table 4.2. Numerous situations are produced by means of the behavior of the velocity and speed in different dam Stepped Spillway subjected to water motion.

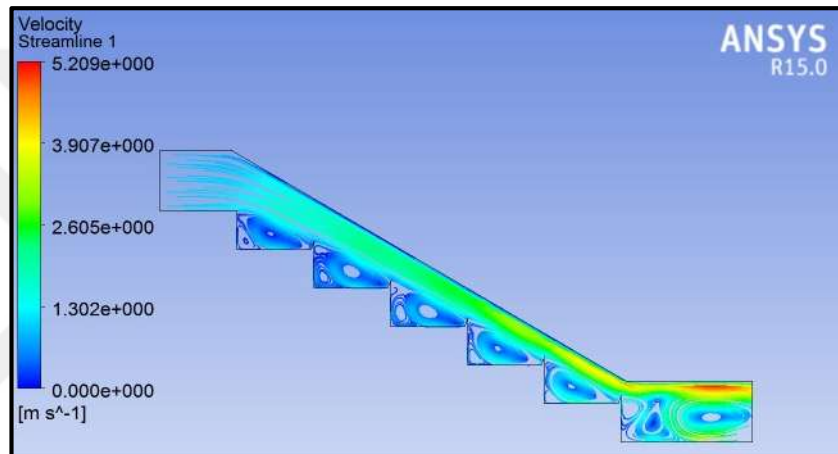
Table 4.2 Operating parameters

	Parameter	Input Value
1	Water flow speed	0.2, 0.8, 1.2 m/s
2	Operating temperature	288.16 K
3	Density	1016 Kg/m ³
4	Model	Realizable k-ε Model
5	Fluid	Water

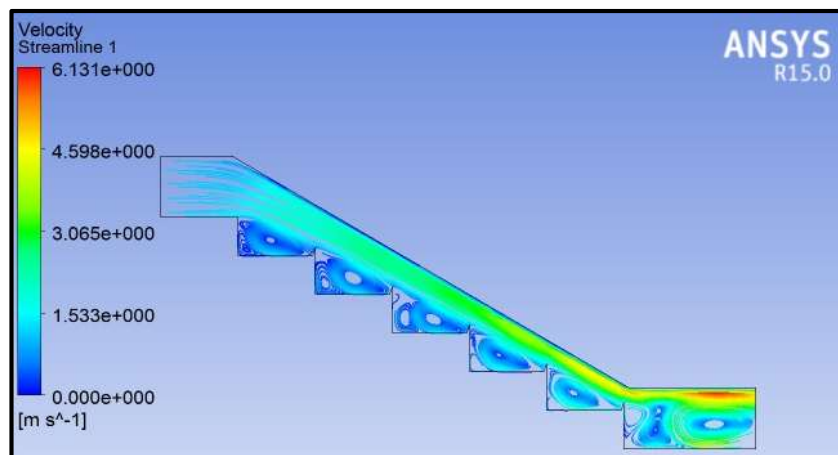
The flow characteristic can be recognized by observing the water flow behavior in various kinds of Stepped Spillways. The influence of velocity variation on the dam Stepped Spillway indicates the variances. The flow speed water of is crucial because it control the effect of water movement which changes the surface velocity on Stepped Spillways.



Velocity Flow Pattern of 0.2 mps



Velocity Flow Pattern of 0.8 mps

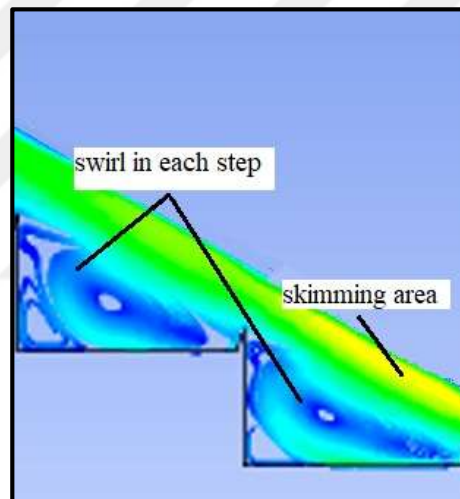


Velocity Flow Pattern of 1.2 mps

Figure 4.1 Velocity flow pattern of the triangle spillway simulation results

It is observed that the maximum velocity affects in the down of the dam Stepped Spillway as shown in the red color of the velocity profile. This observation approved

that there is a high down force in this point, the affected force component will have applied perpendicularly on the longitudinal axis of the dam Stepped Spillway section. But with the increase of water velocity, the vortex appears as shown in Figure 4.2 in the velocity profile. The vortex will increase the drag force which cause several aerodynamic problems. Due to boundary layer separation at the suction side of the dam Stepped Spillway stationed below the critical edge vortex which broke down that resulted in loss of velocity. It is seen that the water characteristics have been changed by the changes of water velocity; therefore, the speed forms detect a rise in the water velocity at the side of surfaces of the dam Stepped Spillway and reduction in the upper surface as revealed in the red area. In current case, the force will increase because there is an inverse relationship between the pressure and the water speed.



Velocity Flow Pattern of 0.2 mps

Figure 4.2 Velocity profile of the triangle spillway simulation results

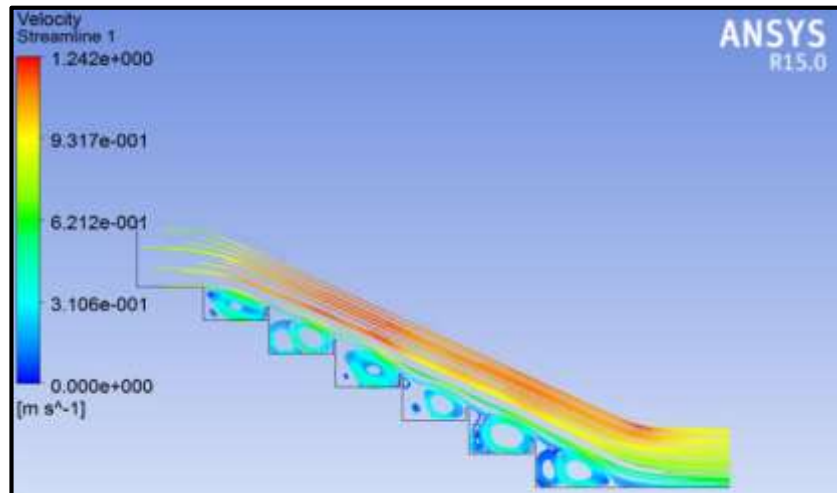
The velocity contours observe a rising in force surface of the dam Stepped Spillway and step pole surface reduction. This result indicates that the resistance counter to the flow course will create a negative force which means that the force will increase because there is a positive relationship between the force and the water velocity in the dam Stepped Spillway surfaces.

4.3 ANSYS Evaluation of Rectangle Dam (Case 2)

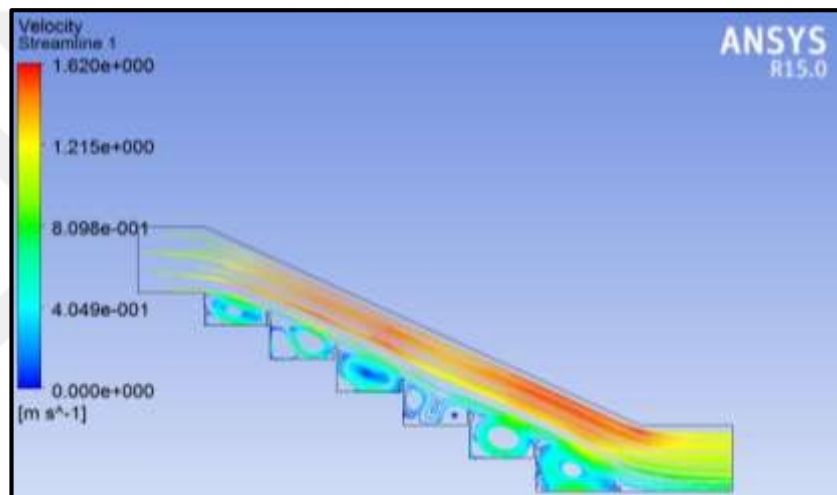
The present case observes the aerodynamic behavior around rectangular dam Stepped Spillway by using CFD ANSYS. The numerical model is also prepared and run with CAD software which creates a boundary conditions for the analysis. This analysis outcome acts a very significant part in optimizing design components so there must be a minimum velocity on the Stepped Spillway surface and swirl of water flow.

4.3.1 Rectangle spillway

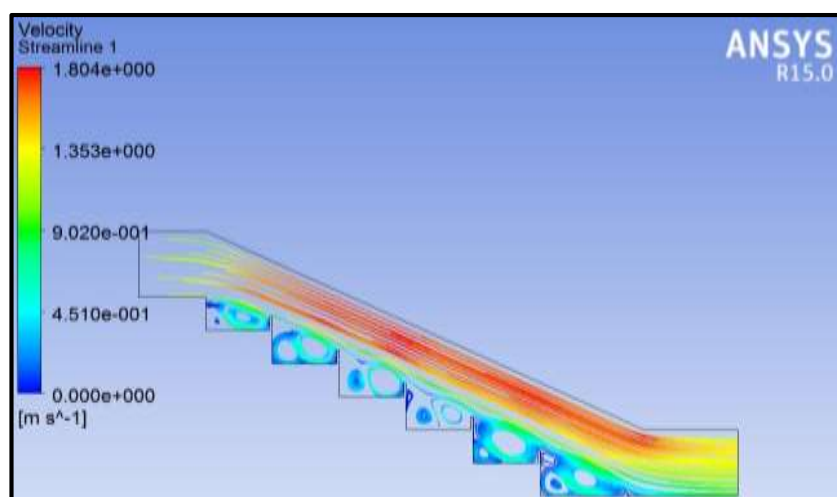
Flow Pattern Vector plots and the stream line plots as shown in Figure 4.3 for the single column case clearly represent the flow patterns around the rectangular dam Stepped Spillway. The ANSYS software simulated the chosen cases with meaning of boundary conditions. There are five basic properties employed in the CFD-ANSYS simulation process are the same in the first case flow characteristic can be recognized by observing the water flow behavior in this type of Stepped Spillways. The influence of velocity variation on the dam Stepped Spillway indicates the variances. The water flow speed is crucial because it control the effect of water movement which changes the surface velocity on Stepped Spillways.



Velocity Flow Pattern of 0.2 mps



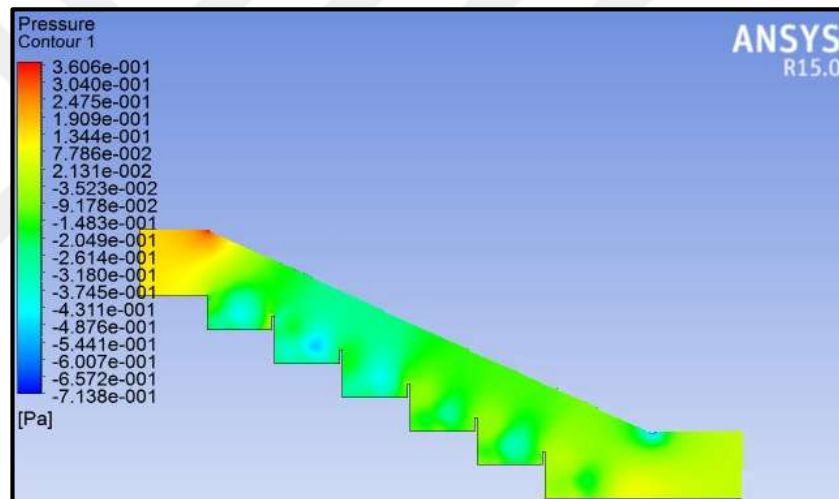
Velocity Flow Pattern of 0.8 mps



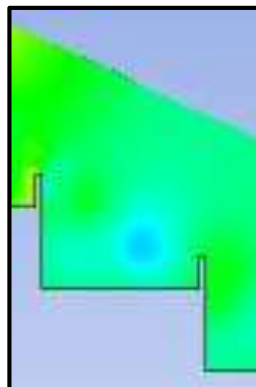
Velocity Flow Pattern of 1.2 mps

Figure 4.3 Velocity flow pattern of the rectangle spillway Dam simulation results

It is observed that the maximum velocity affects in the downstream of the dam Stepped Spillway as shown in the red color of the velocity profile. This observation approved that there is a high down force in this point, the affected force component will have applied perpendicularly on the longitudinal axis of the dam Stepped Spillway section. But with the increase of water velocity, the vortex appears which will increase the drag force and that causes several aerodynamic problems. Due to boundary layer separation at the suction of the dam Stepped Spillway. It is seen that the water characteristics have been changed by the changes of water velocity; therefore, the velocity contours observe an increase in the water speed of the dam Stepped Spillway. In this case, the force will increase because there is an inverse relationship between the force and the water speed as shown in pressure plots.



Pressure Pattern of 0.2 mps



Pressure Pattern on Step

Figure 4.4 Pressure profile of rectangular spillway Dam simulation results

In Figure 4.4, compression shapes in the horizontal surfaces of the three stages are correspondingly, visible for the three flow rates. At both extreme and normal flow rates, the compression on the horizontal surface of the steps firstly reductions and then rises, and again at the edge of the step, decreases. At the lower bound of the flow rate, the upward trend observed in the previous cases occurred less so that the trend of decreasing compression distribution was roughly noticed.

The minimum pressure obtained from the simulation is a significant component of continuous compression, and the lowest average compression is wanted to evaluate the potential creation of cavitation. The pressure contours observe a rise in the head surface of the dam Stepped Spillway and reduction in the downstream surface.

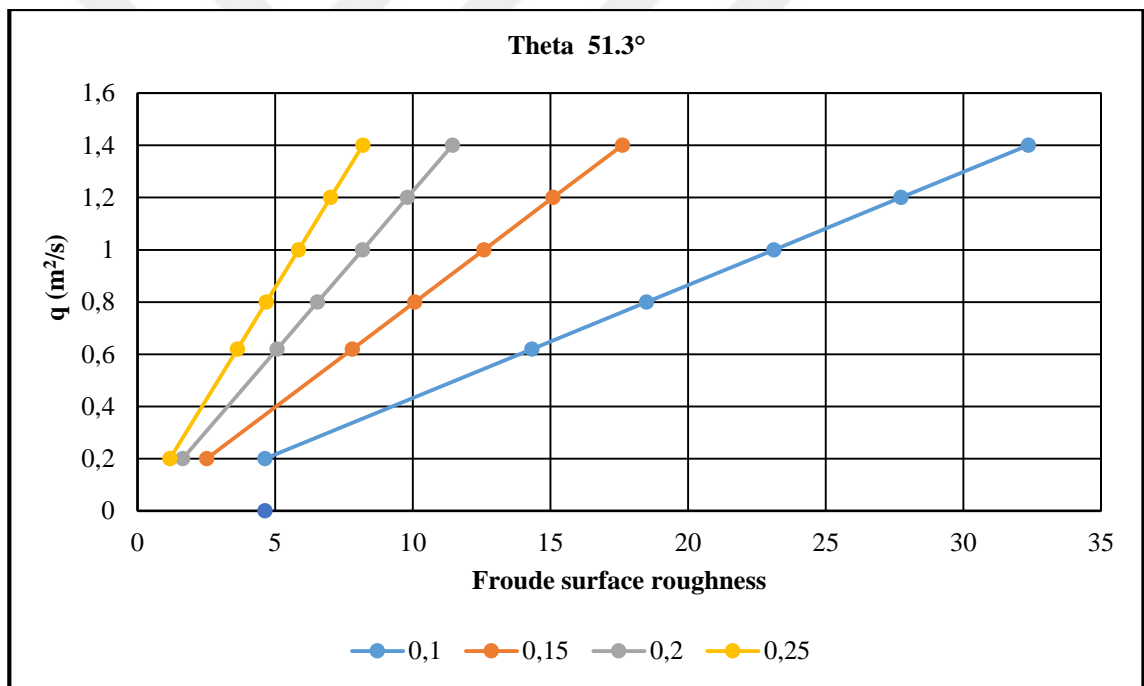


Figure 4.5 The relationship between Froude surface roughness and discharge with slope 51.3°

This result indicates that the resistance counter to the flow course will create a negative force which means that the force will increase because there is a positive relationship between the force and the water velocity in the dam Stepped Spillway surfaces. Based on the specific water discharge, Table 4.2 shows the step height and Froude surface

roughness (F). The initial length climbs in the same number of steps as the discharge rises. The Froude surface roughness, identified by the meaning of surface roughness, increases as the number of steps increases ($k_s = h \cos \Theta$), which is based on unit discharge (q) and step height.

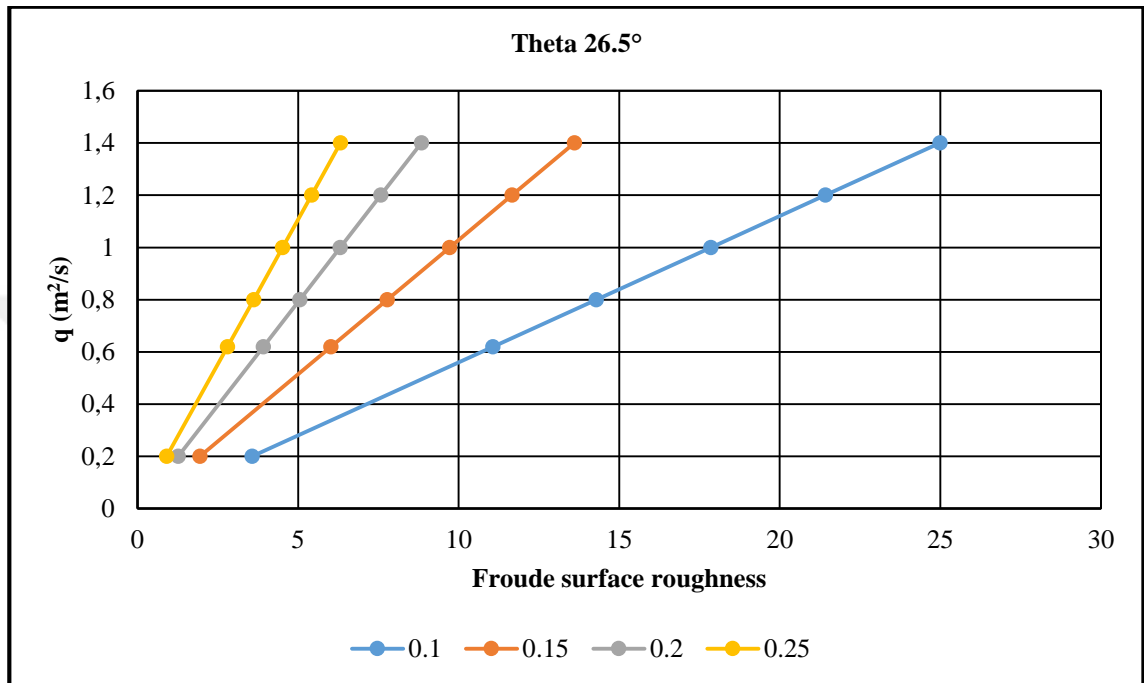


Figure 4.6 The relationship between Froude surface roughness and discharge with slope 26.5°

The variation of the Froude surface roughness by taking in consideration the discharge at various step heights is displayed in Figure 4.5 and Figure 4.6. The direction of the figure displays that it is at the highest step, the Froude surface roughness is the minimum below all circumstances of various discharges. Under the same discharge, the Froude surface roughness increases as the step height is reduced.

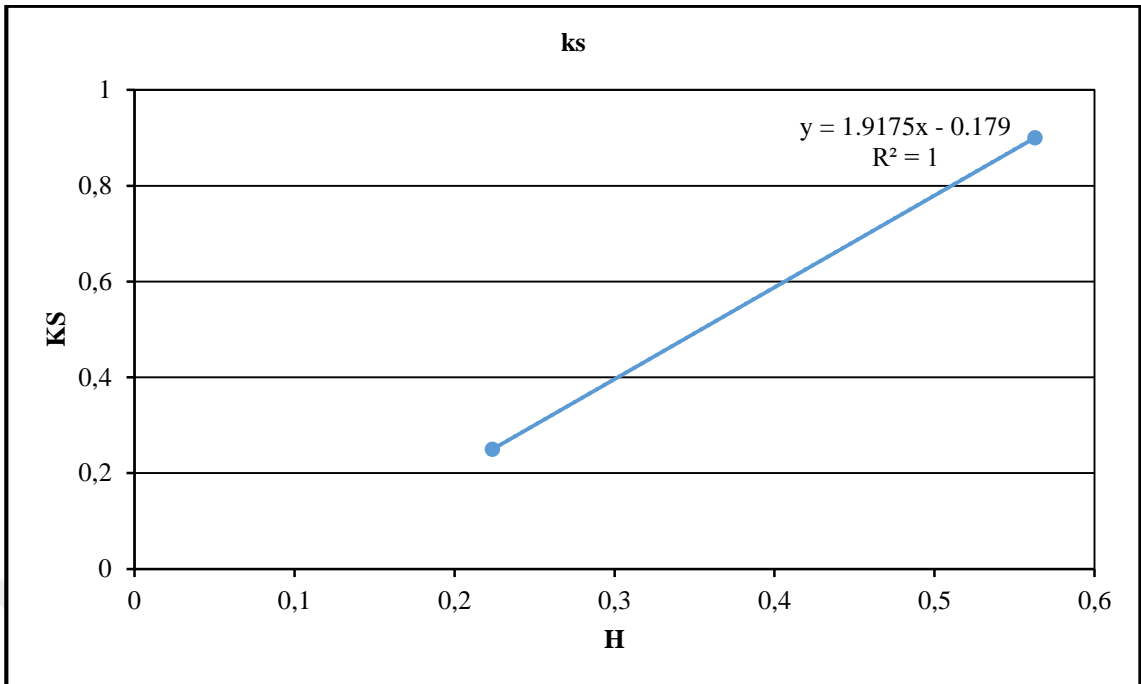


Figure 4.7 The relationship between spillway step and ks

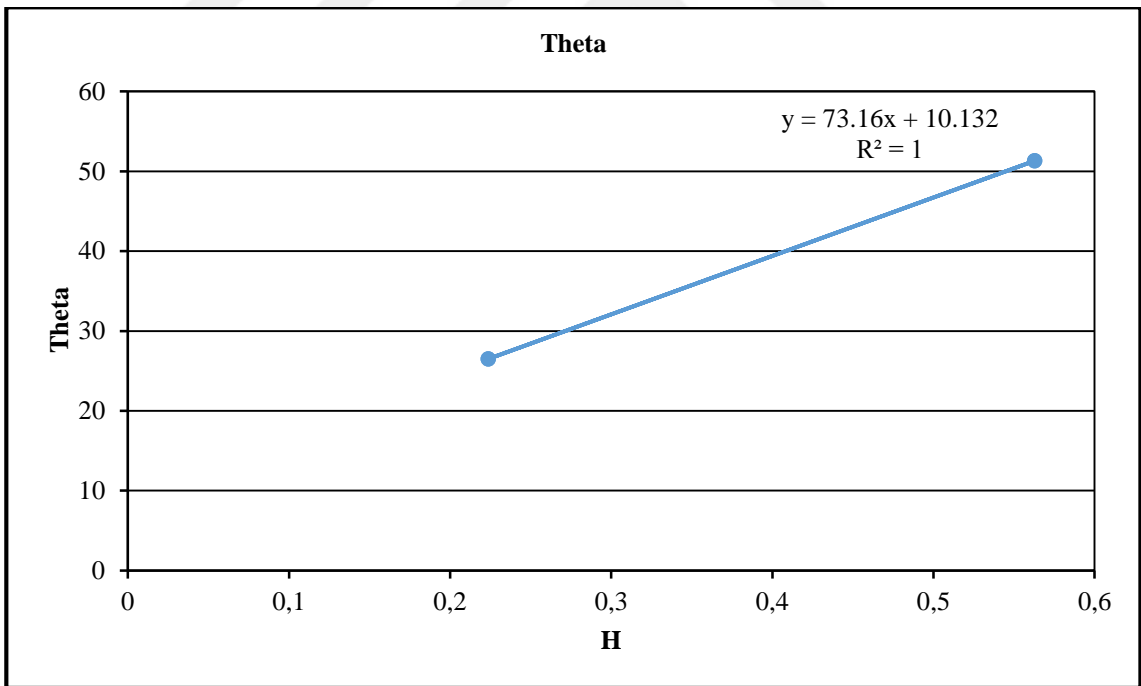


Figure 4.8 The relationship between spillway step and angle

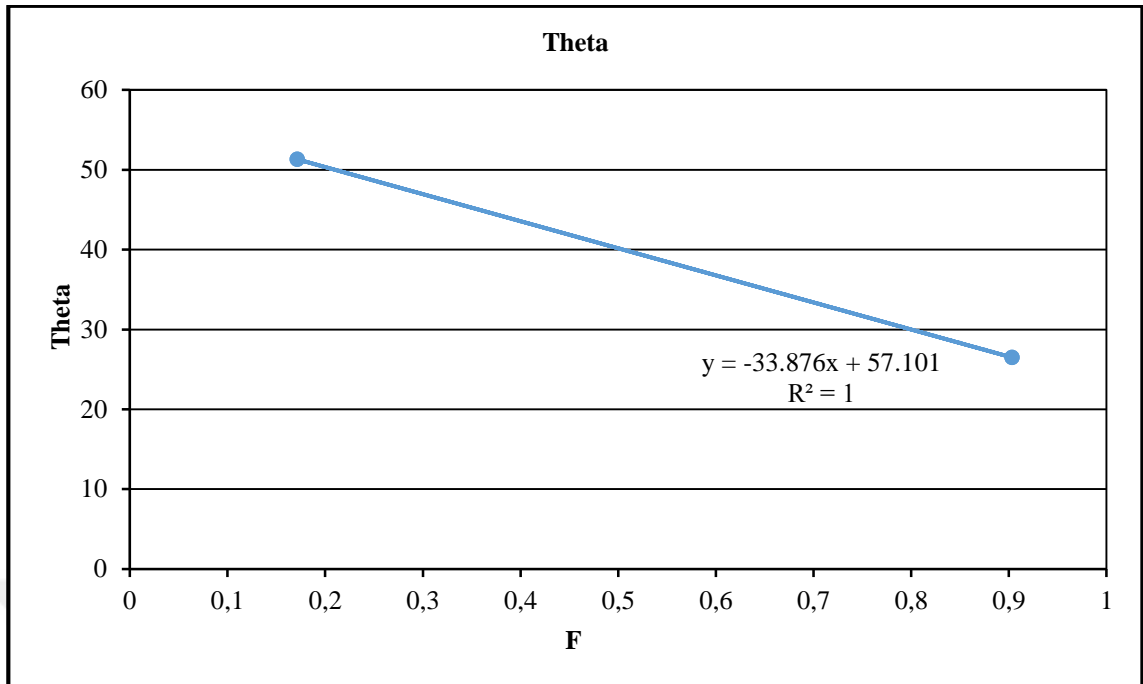


Figure 4.9 The relationship between spillway step and angle

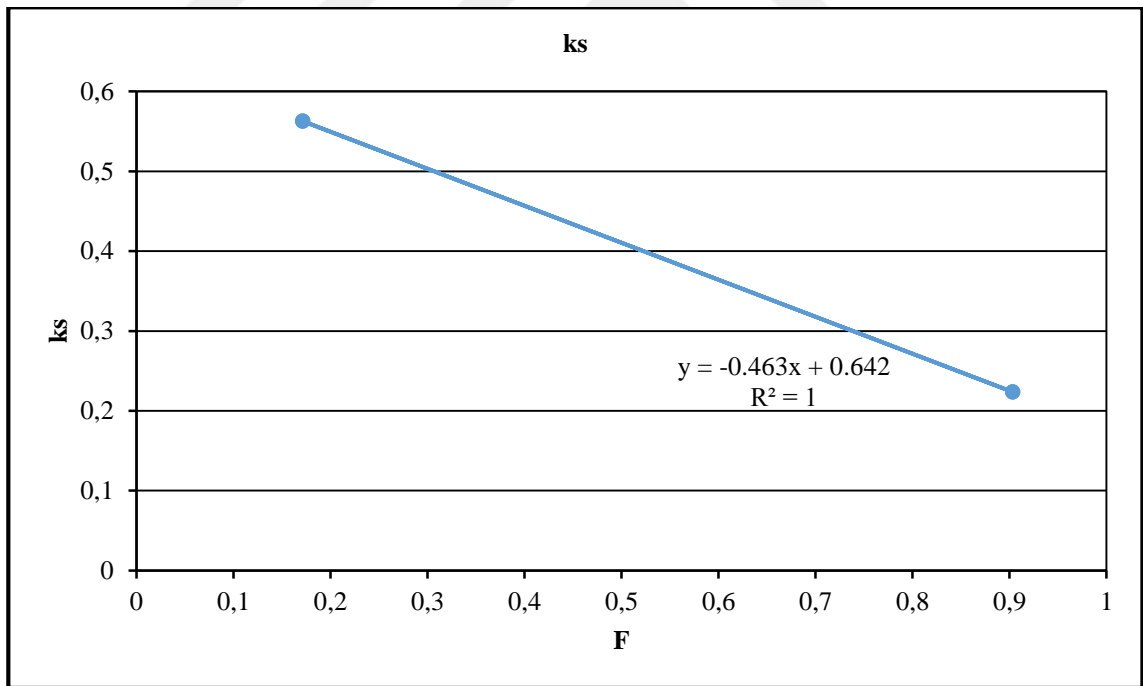


Figure 4.10 The relationship between spillway step and ks

The Figures (Figure 4.7, Figure 4.8, Figure 4.9, Figure 4.10) and Tables (Table 4.3, Table 4.4) shows the relationship between surface roughness ks, theta and Froude (F) at different step heights. As the surface roughness of Froude rises, the starting length

increases. In regard to the (8) Equation, the surface roughness of Froude (F) increases with decreasing step height and increasing discharge. Figure 6 further illustrates that the Froude surface roughness values increase as the step height decreases.

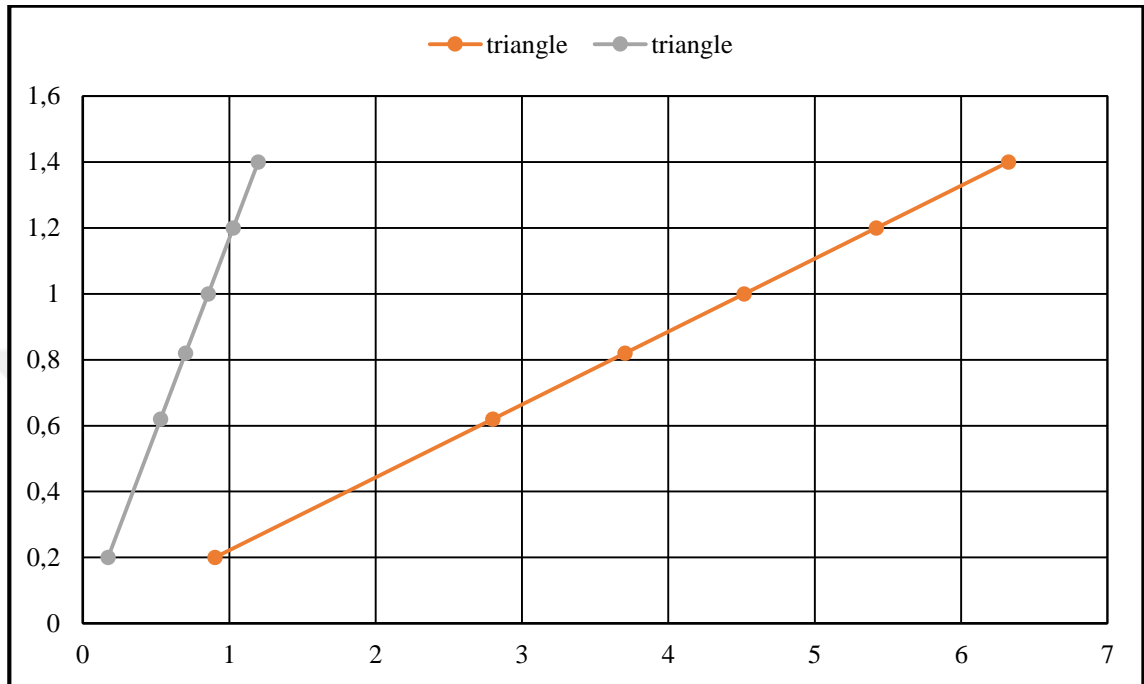


Figure 4.11 Comparison between triangle and rectangle steps

Table 4.3 Numerical results of triangle step spillway

Case	Type	Steps No	h	l	Theta	ks	q (m ² /s)	F
1	Triangle	6	0.25	0.5	26.5	0.223734	0.2	0.903306
1	Triangle	6	0.25	0.5	26.5	0.223734	0.62	2.800249
1	Triangle	6	0.25	0.5	26.5	0.223734	0.82	3.703555
1	Triangle	6	0.25	0.5	26.5	0.223734	1	4.516531
1	Triangle	6	0.25	0.5	26.5	0.223734	1.2	5.419837
1	Triangle	6	0.25	0.5	26.5	0.223734	1.4	6.323143

Table 4.4 Numerical results of rectangle step spillway

Case	Type	Steps No	h	l	Theta	ks	q (m ² /s)	F
2	Triangle	12	0.9	0.9	51.3	0.562718	0.2	0.171235
2	Triangle	12	0.9	0.9	51.3	0.562718	0.62	0.530828
2	Triangle	12	0.9	0.9	51.3	0.562718	0.82	0.702062
2	Triangle	12	0.9	0.9	51.3	0.562718	1	0.856173
2	Triangle	12	0.9	0.9	51.3	0.562718	1.2	1.027408
2	Triangle	12	0.9	0.9	51.3	0.562718	1.4	1.198643

Simulated water behavior for various stepped spillway models with different slopes ranging from steep to steep slopes. The scraping flow system can be generated by increasing the flow rate of a stepped spillway. The stepped spillway's scrape flow system is dependent on the discharge as well as the height and length of the steps. The discharge value must be greater than the crucial value for all discharges to pick the scrape flow system. L_i rises with critical depth in all regression channels of stepped spillways as critical depth rises. The product of the critical depth of the step height and the relative depth of flow is the relative depth of flow. The relative flow depth indicates that when the minimum channel slope values increase, the relative flow depth increases, as does the length of the unventilated flow area.

4.4 Slop Effect on Inspection Point

Stepped spillways acts a significant part throughout the construction of spillway. The stepped spillway Slope is a significant consideration for detecting the beginning point position and unventilated flow region length. In this case, the starting point position determines its location upwards as the downstream slope rises, the value of the unventilated flow area decreases while the downstream slope is increased for all drains. In Figures (Figure 4.12, Figure 4.13, Figure 4.14) demonstrate water volume fraction profiles for different states.

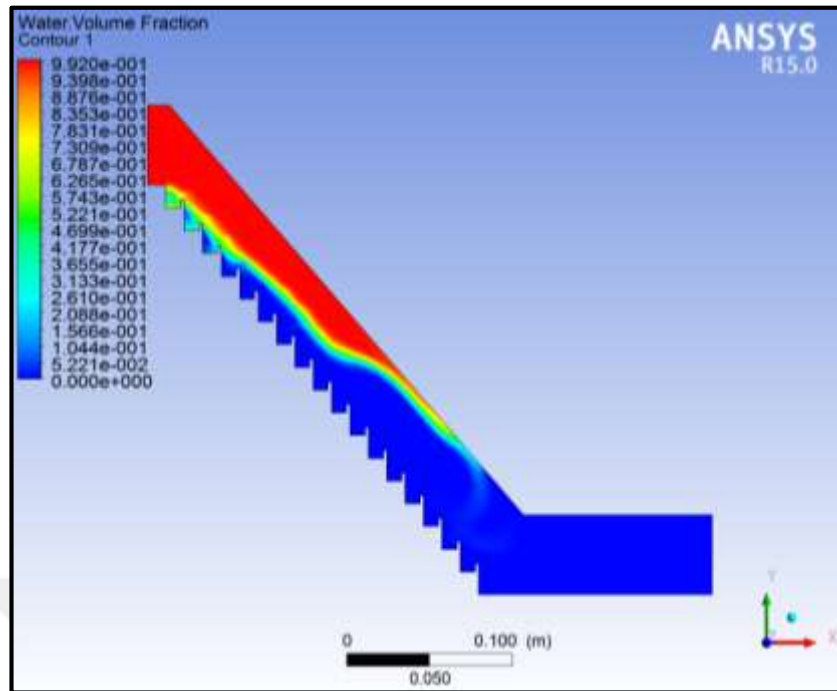


Figure 4.12 Volume of fraction for case 1

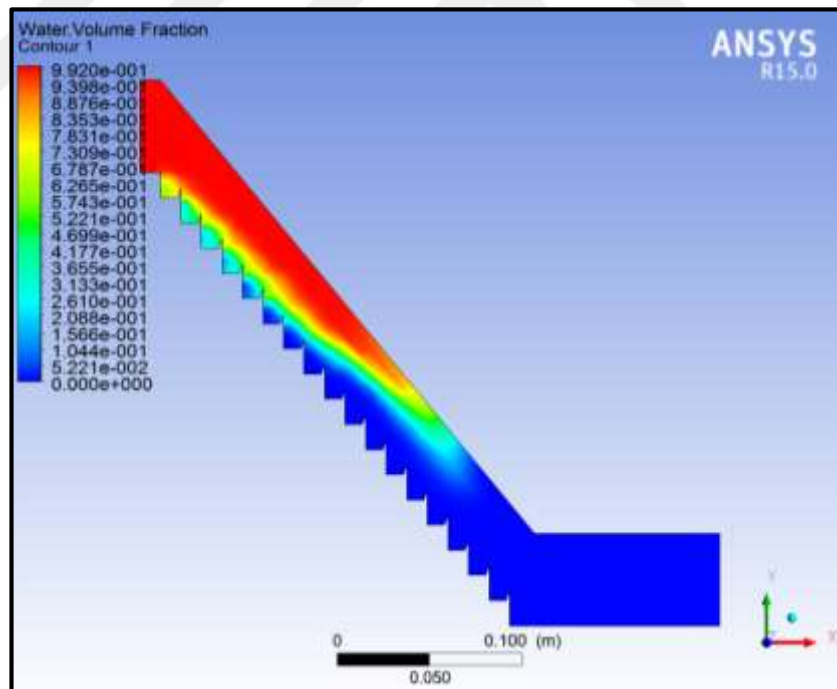


Figure 4.13 Volume of fraction for case 1, vel. 2

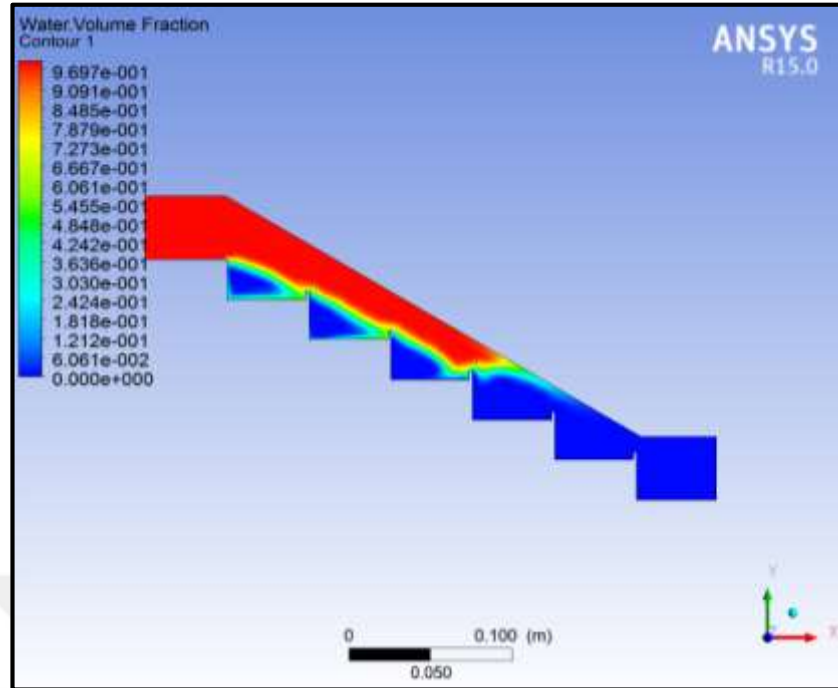


Figure 4.14 Volume of fraction for case 2

It is clear from the figure that with the rise of the water flow, the distance of the starting point L_i rises, with the difference of the channel slope. On the other hand, under the same discharge value, L_i drops as the pf channel slope increases. This suggests that, based on the constant discharge value, the high channel slope has a smaller non-ventilated flow area.

5. CONCLUSION AND RECOMMENDATION

In the present research, to explore the flow behavior and investigate the inception point position, four different stepped spillway models with variable slopes and steps are simulated utilizing velocity profile, pressure profile, and VOF using fluent ANSYS models. A skimming flow system is created by increasing the flow rate on a tiered spillway slope. The discharge, step height, and step length all play a role in the stepped spillway's skimming flow mechanism. The study's findings are used to determine the skimming flow system in all discharges; the discharge value must be greater than the critical value. Position of air entrainment (beginning point) travels down when increases the discharge rate. This result observes the importance of the spillway steps shape to generate the pressure force based on height of each step. Also, this work observed that it is feasible to improve the hydraulic behaviors of a pool configurations in stepped spillway. The pools in this study present an overcome of the flow stability and energy dissipation issues.

5.1 Future Work and Recommendation

The researcher recommends a future works as in below:

1. I recommended to develop practical shapes. This can be achieved more accurate information to the spillway conditions.
2. It's important to find a new spillway design to reduce the water energy.

REFERENCES

- Aal, G. M. A., Sobeah, M., Helal, E. and El-Fooly, M. 2018. Improving energy dissipation on stepped spillways using breakers. *Ain Shams Engineering Journal*, 9(4): 1887-1896.
- Ali, A. S. and Yousif, O. S. Q. 2019. Characterizations of flow over stepped spillways with steps having transverse slopes. In *IOP Conference Series: Earth and Environmental Science*, 1: 12-19.
- Amador, A., Van der Graaf, G., Sánchez-Juny, M., Dolz, J., Sánchez-Tembleque, F., Puertas, J. and Girona, C. J. 2004. Characterization of the flow field in a stepped spillway by PIV. In *Proc. 12th Symp. Applications Laser to Fluid Mechanics*, 3: 12-15.
- Ashoor, A. and Riazi, A. 2019. Stepped spillways and energy dissipation: a non-uniform step length approach. *Applied Sciences*, 9(23): 5036-5071.
- Barani, G. A., Rahnama, M. B. and Sohrabipour, N. 2005. Investigation of Flow Energy Dissipation over Various Stepped Spillways. *American Journal of Applied Sciences*, 2: 1101-1105.
- Bautista, E., Strelkoff, T. S. and Clemmens, A. J. 2003. General characteristics of solutions to the open-channel flow, feedforward control problem. *Journal of Irrigation and Drainage Engineering*, 129(2): 129-137.
- Chatila, J. G. and Jurdi, B. R. 2004. Stepped spillway as an energy dissipater. *Canadian Water Resources Journal/Revue Canadienne des Ressources Hydriques*, 29(3): 147-158.
- Felder, S. and Chanson, H. 2013. Aeration, flow instabilities, and residual energy on pooled stepped spillways of embankment dams. *Journal of Irrigation and Drainage Engineering*, 139(10): 880-887.
- Ghaderi, A. and Abbasi, S. 2021. Experimental and Numerical Study of the Effects of Geometric Appendage Elements on Energy Dissipation over Stepped Spillway. *Water*, 13(7): 950-957.
- Guenther, P., Felder, S. and Chanson, H. 2013. Flow aeration, cavity processes and energy dissipation on flat and pooled stepped spillways for embankments. *Environmental Fluid Mechanics*, 13(5): 503-525.

- Jahad, U. A., Al-Ameri, R., Chua, L. and Das, S. 2017. Geometry Effects on Energy Dissipation Over Stepped Spillway and HCED Iraqi Scholars Conference in Australasia 2017. ISCA, 375 page, Australia.
- Jamel, A. A. J. 2018. Numerical Simulation for Estimating Energy Dissipation over Different Types of Stepped Spillways and Evaluate the Performance by Artificial Neural Network. Tikrit Journal of Engineering Sciences, 25(2): 18-26.
- Johansson, S. 2007. Detection of internal erosion in embankment dams using temperature, resistivity and SP measurements. In Internal Erosion of Dams and their Foundations, CRC Press, pp. 141-158, France.
- John LaBoon, P. E. 2014. Appurtenant Structures for Dams (Spillways and Outlet Works) Design Standard, General Spillway Design Considerations, Bureau of Reclamation, pp. 206-211, USA.
- Karlsson, T. and Tallberg, J. 2011. Concrete Face Rock-fill dam compared to Roller Compacted Concrete dam. MSc. Thesis, Chalmers University of Technology, 59 page, Sweden.
- Li, S. 2021. Numerical simulations of flow discharge and behaviours in spillways. Doctoral Dissertation, KTH Royal Institute of Technology, 45 page, Sweden.
- Meireles, D. C. 2015. Numerical modeling of skimming flow over stepped spillways: application on small embankment dams, 10: 1-13.
- Morovati, K., Eghbalzadeh, A. and Soori, S. 2016. Numerical Study of Energy Dissipation of Pooled Stepped Spillways. Civil Engineering Journal, 2(5): 208-220.
- Niu, Z., Li, K., Bai, W., Sun, Y., Gong, Q. And Han, Y. 2021. Design of Inductive Sensor System for Wear Particles in Oil. Journal of Mechanical Engineering, 57(12): 126-135.
- Parsaie, A. and Haghiabi, A. H. 2019. Evaluation of energy dissipation on stepped spillway using evolutionary computing. Applied Water Science, 9(6): 1-7.
- Raza, A., Wan, W. and Mehmood, K. 2021. Stepped Spillway Slope Effect on Air Entrainment and Inception Point Location. Water, 13(10): 1419-1428.
- Saleh, S. M. and Husain, S. M. 2020. Computational Study to Predict the Free-Surface Flow over Non-uniform Stepped Spillway Using ANSYS-CFX. Polytechnic Journal, 10(1): 43-50.

- Salmasi, F. and Samadi, A. 2018. Experimental and numerical simulation of flow over stepped spillways. *Applied Water Science*, 8(8): 1-11.
- Saqib, N. u., Akbar, M., Pan, H., Ou, G., Mohsin, M., Ali, A. and Amin, A. 2022. Numerical Analysis of Pressure Profiles and Energy Dissipation across Stepped Spillways Having Curved Risers. *Applied Sciences*, 12(1): 448.
- Simões, A. L. A., Schulz, H. E., Lobosco, R. J. and de Melo Porto, R. 2012. Stepped spillways: theoretical, experimental and numerical studies. *IntechOpen*, 2: 6-26.
- Uday, A. M., Alturfi, H. M. J., Al-moadhen and H. S., Mohammed. 2020. Evaluation of Computational Fluid Dynamic Model in Investigating the Hydraulic Performance of Stepped Spillway, 15: 752-761.
- Van Alwon, J. A. C. O. B., Borman, D. U. N. C. A. N., Sleight, A. N. D. R. E. W. and Kapur, N. 2017. Experimental and numerical modelling of aerated flows over stepped spillways. In *Proceedings of IAHR 2017. International Association for Hydro-Environment Engineering and Research (IAHR)*, 8: 2-13.
- Varagilal, I.A. 2018. Stability analysis of gravity dams for the maximum design earthquake. Doctoral Dissertation, The NOVA University of Lisbon, 98 page, Portugal.
- Zeng, Y., Qi, X., Shu, L., Yao, Z., Zhou, L. and Wang, F. 2022. Fluid–Structure Interaction Effects of a Partially Immersed, Cantilevered Hydrofoil. *Journal of Fluids Engineering*, 144(2): 5-11.

CURRICULUM VITAE

Personal Information

Name and Surname : Amjed ALAWADI

Education

MSc	Çankırı Karatekin University Graduate School of Natural and Applied Sciences Department of Civil Engineering	2019-Present
Undergraduate	Kufa University Faculty of Engineering Department of Chemical Engineering	2008-2012