INVESTIGATION OF HYDRAULIC PERFORMANCE ON UNSYMMETRICAL SMOOTH SPILLWAY

MOHD SAFARUL IZMI BIN SAIDIN

UNIVERSITI SAINS MALAYSIA

2021

INVESTIGATION OF HYDRAULIC PERFORMANCE ON UNSYMMETRICAL SMOOTH SPILLWAY

by

MOHD SAFARUL IZMI BIN SAIDIN

Thesis submitted in fulfilment of the requirements for the degree of Master of Science

June 2021

ACKNOWLEDGEMENT

Bismillahirrahamanirrahim and Alhamdulillah, all praise and glory to Almighty Allah s.w.t for his grace and blessing throughout the journey of Master by Research, with His permission gave me strength and spirit which enabled me to complete the study successfully. Firstly, I would like to take this opportunity to thank Dr. Mohd Sharizal Abdul Aziz as my supervisor and Associate Professor Dr. Mohd Remy Rozainy Mohd Arif Zainol, my co-supervisor, for providing invaluable guidance, suggestions, endless support and comments throughout this wonderful project. Their patience and corporation towards completing my thesis are deeply appreciated. Next, I am very grateful and fortunate enough to get consistent support from the Assistant Engineer, who are En. Jamari Sadli, En. Muhammad Nabil Semail, and Puan Nurul Akma Sunai for their contributions and helps in laboratories during the period of my study. I would also like to thank my postgraduate research team members for their morals support, ideas, and suggestions which are truly appreciated. Finally, big thanks to my wife, kids and family members for their endless love and support.

TABLE OF CONTENTS

LIST OF TABLES

Page

LIST OF FIGURES

LIST OF SYMBOLS

±	Plus minus sign
C_d	Dimensionless crest coefficient (-)
D	Diameter (m)
D_ω	Cross-diffusion term
d	Water depth (m)
d_1	Incoming water depth (m)
d_2	Tailwater depth (m)
d_m	Water depth model (m)
d_p	Water depth prototype (m)
E_1	Incoming energy loss
E_2	Tailwater energy Loss
ΔE	Energy dissipation of hydraulic jump (-)
Fr	Froude number (-)
Fr_1	Incoming Froude number (-)
Fr_2	Tailwater Froude number (-)
Fr_m	Froude number of model (-)
Fr_p	Froude number of prototype (-)
G_κ	Generation of turbulent kinetic energy due to mean velocity gradients
G_b	Generation of turbulent kinetic energy due to buoyancy
g	Gravitational acceleration (m/s^2)
H_t	Total head on the crest (m)
L	Effective length of the weir (m)
Q	Water discharge flow (m^3/s or l/s)
Q_m	Discharge flow model (m^3/s or l/s)

Q_p	Discharge flow prototype (m^3/s or l/s)
R	Radius (m)
R^2	Coefficient of determination (-)
u	Water velocity (m/s)
V_m	Velocity model (m/s)
V_p	Velocity prototype (m/s)
v	Water velocity (m/s)
Y_M	Contribution of the fluctuation dilatation in turbulence (-)
ε	Epsilon (-)
κ	Kappa (-)
λ	Lambda (-)
μ_{eff}	Effective viscosity (m^2/s)
σ_ε	Prandtl number for ε (-)
σ_k	Prandtl number for κ (-)
v	Kinematic viscosity (m^2/s)
ω	Omega (-)

LIST OF ABBREVIATIONS

2D	2-Dimensional
3D	3-Dimensional
CI	Interference coefficient
CFD	Computational fluid dynamic
DES	Detached Eddy simulation
GCI	Grid convergence index
GEP	Gene expression programming
GIT	Grid Independent Test
LES	Large-eddy simulation
MAPE	Mean absolute percentage error
MARE	Mean absolute relative error
MSE	Mean squared error
PIV	Particle image velocimetry
PKW	Piano Keys Weir
PMF	Probable maximum flood
QCSS	Quarter-circular crested stepped spillway
RAM	Random access memory
RANS	Reynolds-averaged Navier-Stokes
RKE	Realizable κ-ε
RMSE	Root mean square error
RNG	Renormalization Group κ-ε
RSM	Reynold stress model
SKE	Standard κ-ε
SKW	Standard κ-ω
SPH	Smooth particle hydrodynamics
SST-KW	Shear Stress Transport κ-ω
USBR	United States Department of the Interior Bureau of Reclamation
USM	Universiti Sains Malaysia
VOF	Volume of Fluid
WSP	Water surface profile

PENYIASATAN PRESTASI HIDRAULIK KE ATAS ALUR LIMPAH LICIN YANG TIDAK SIMETRI

ABSTRAK

Alur limpah rata/licin merupakan rekabentuk klasik yang digunakan untuk melimpahkan air secara besar-besaran daripada takungan empangan air. Beberapa kajian mengenai ciri-ciri hidraulik alur limpah telah dilakukankan secara eksperimen dan numerik/berangka dalam beberapa dekad terkini untuk memberikan gambaran yang lebih baik mengenai tingkah laku aliran dan meramalkan kawasan kritikal yang boleh membahayakan struktur dan persekitaran. Satu kajian eksperimen dilakukan ke atas model berskala makmal untuk menyiasat ciri asas hidraulik alur limpah licin/rata tidak simetri dengan tiga ukuran kadar alir yang dinamakan sebagai Q1, Q2, dan Q3. Kemudian, pengesahan simulasi menggunakan ujian bebas grid (GIT), analisis ukuran sela masa dan simulasi pemerhatian masa turut dilaksanakan. Dalam simulasi, persamaan purata Reynolds Navier-Stokes (RANS), skim persamaan isipadu bendalir (VOF) dan model gelora *Standard κ-ω* (SKW) dengan keadaan sempadanan yang sesuai telah digunakan untuk mensimulasikan alur limpah rata/licin yang tidak simetri geometrinya dengan empat saiz/ukuran kadar alir iaitu Q1, Q2, Q3 dan Q4. Keputusan simulasi memberikan ramalan yang baik mengenai ciri-ciri hidraulik aliran air pada alur limpah rata/licin tidak simetri dan menunjukkan kesepakatan yang baik dengan perbandingan hasil eksperimen dari segi profil permukaan air dan halaju. Pada objektif terakhir, empat modifikasi model arka sesekat (capah -45 darjah, capah -15 darjah, tumpu 15 darjah dan tumpu 45 darjah) telah disimulasikan dengan menggunakan aliran kadar alir air Q3 dan Q4 dan prestasi lompatan hidraulik semua model dibandingkan dengan geometri normal. Model tumpu 15 darjah menunjukkan pelesapan tenaga

tertinggi untuk kesemua modifikasi model dengan menggunakan kadar aliran air Q3 dan Q4, iaitu sebanyak 5.06% dan 2.49% lebih tinggi daripada model geometri biasa. Modifikasi arka sesekat pada alur limpah yang tidak simetri akan memberi kesan kepada prestasi pelesapan tenaga di lembangan penenang dan memberi pendedahan baik yang boleh dipertimbangkan dalam rekabentuk alur limpah di masa hadapan.

INVESTIGATION OF HYDRAULIC PERFORMANCE ON UNSYMMETRICAL SMOOTH SPILLWAY

ABSTRACT

The smooth spillway is a classic design used to spill large volume of water from the dam reservoir. Several studies of hydraulic characteristics of the spillway have been conducted experimentally and numerically in recent decades to provide a better insight into flow behaviour and predict any critical area that could endanger the structure and environment. An experimental study was carried out on a laboratory scale model to investigate basic hydraulic characteristics of the unsymmetrical smooth spillway with three sizes of discharge namely as Q1, Q2 and Q3. Next, the simulation verification were done using grid independent test (GIT), time step size analysis, and simulation time observation. In simulation, Reynolds-averaged Navier-Stokes (RANS) equation, volume of fluid (VOF) scheme and Standard κ-ω (SKW) turbulence model with appropriate boundary conditions were used to simulate the unsymmetrical smooth spillway geometry with four sizes of discharge namely as Q1, Q2, Q3 and Q4. The simulation results provide a good prediction of the hydraulic characteristics of water flow on the unsymmetrical smooth spillway and show a good agreement in terms of water surface and velocity profiles patterns with the experimental results. In the last objective, four chute piers modification models (diverge -45 degree, diverge -15 degree, converge 15 degree and converge 45 degree) were simulated using Q3 and Q4 discharge water flows and the hydraulic jump performance of all models were compared with the normal geometry. For all modification models, the converge 15 degree model showed 5.06% and 2.49% higher energy dissipation than the normal geometry model using Q3 and Q4, respectively. The chute piers modifications on

unsymmetrical spillway will affect the energy dissipation performance at stilling basin and provide good insight which can be considered for spillway design in the future.

CHAPTER 1

INTRODUCTION

1.1 Introduction

The application of hydraulic engineering is not only applied in massive discharges but also our everyday lives. A lot of things around us are engineered to prevent accidents or damages caused by hydraulic failures. For example, the tea-pot with a spout was designed to avoid overspill during the pouring process by routing the tea properly into the cup (Noordin et al., 2013). This is just an example of how water is effectively and efficiently managed, be it in our kitchen or at the dam spillway.

Due to the massive discharge of water flow, the spillway should be adequately designed to improve the efficiency of hydraulic performance especially in terms of energy dissipation and prevent any catastrophe from occuring during operation stage (Chatila & Tabbara, 2004; Parsaie & Haghiabi, 2019b). Generally, a spillway consists of several parts, including crest weir, smooth chute or stepped chute and stilling basin at the toe of the spillway. The function of the crest is to maintain the discharge capacity, while the chute is to efficiently transfer the water flow spilling from the crest to the stilling basin at the downstream (Chanson, 2004; Parsaie & Haghiabi, 2019b).

Water flow characteristics along the spillway is considered as an open channel flow application. In general, flow characteristics along the spillway can be categorised based on dimensionless Froude number, Fr value. The Froude number can be defined as $Fr = v/(gd)^{1/2}$ where v is the approach velocity (m/s), g is the gravitational acceleration (m/s^2), and d is the water depth (m). There are 3 types of flow conditions which are subcritical flow ($Fr < 1$), critical flow ($Fr = 1$), and supercritical flow ($Fr > 1$). The subcritical flow is a gentle flow, while the supercritical condition is a rapid flow and the critical flow is the transition flow between subcritical and supercritical

(Chanson, 2004; Mohamed, 2013). All of the flow conditions were previously measured through experiment by many researchers.

Due to computer technology advancement, numerical analysis using commercial software is approached by researchers. Numerical software have improved a lot in terms of accuracy and speed of open channel flow characteristics prediction. By applying the Reynolds-averaged Navier-Stokes (RANS) model, coupled with the Volume of Fluid (VOF) scheme for multiphase flow and validated turbulence model, hydraulic characteristics of open channel flow can be predicted to produce reliable numerical results (Aydin, 2012).

1.2 Tawau Dam's Spillway

Tawau dam spillway is part of the multipurpose dam of Tawau Water Supply Scheme project (Phase III). This project is expected to complete in 2021, which is built near Taman Bukit Tawau in Jalan Gudang 4, Tawau Sabah. This multipurpose dam was built to supply 209 million litres per day of raw for domestic use. Instead of that, the Tawau Dam also will provide the clean water to the Tawau residents when the water treatment plants start to operate. The flood mitigation factor were also considered in the design where an additional structure will be construct to contain 4.6 million litres of water per day (Online, 2018).

1.3 Problem Statements

In water engineering projects such as dams, irrigation and drainage networks, spillways become one of the important parts of flood evacuation system. The hydraulic characteristics of the spillway are very significant in the water engineering design structure to enhance its performance (Parsaie & Haghiabi, 2019b).

In the past decades, hydraulic characteristics are usually approached experimentally rather than numerically. In the experimental method, a scale model of the prototype is used, and it successfully provide the reliable results for researchers. The physical models are highly cost, and takes a long time to get the results. In addition, they are prone to errors due to scale effects that increases in severity as the ratio of the prototype to model size increases.

The available design charts for chutes and energy dissipators apply only to certain types of chute configurations covering a limited range of flood levels. These limitations have been overcome by using physical models to visualise and understand various hydraulic design aspects. Although the physical models are reliable to study the chute flow behaviour, designing these models can be expensive and time-consuming. Due to cost and time constraints, the numerical simulation are used along with physical modelling (Dehdar-behbahani & Parsaie, 2016).

Alternatively, the numerical simulation are used for many years to estimate the hydraulic performance of smooth spillway because the computational cost is lower than physical modelling (Dae & Park, 2005). The water surface and velocity profiles are considered as crucial in the open-channel flow study. Both profiles, will provide hydraulic characteristics of open channel flow for energy dissipation calculation. This kind of characteristics can be predicted by combining the VOF and suitable turbulence model in the computational fluid dynamics (CFD) simulation. However, studies on unsymmetrical smooth spillway using elliptical weirs combined with chute pier are scarce.

1.4 Objectives

This study is conducted to investigate the hydraulic characteristics of the unsymmetrical smooth spillway and comprises the following objectives:

i. To investigate the water surface and velocity profiles of the physical model of unsymmetrical smooth spillway.

ii. To analyse the effects of water discharge on hydraulic performance using CFD and validate the results with experimental results.

iii. To investigate the influence of chute pier angle on the hydraulic jump performance of smooth spillway.

1.5 Scope and Limitation of Study

This study involves 3-dimensional (3D) simulation and experimental results were used for the verification and validation of the simulation. For the first objective, the experiments were conducted on a scale model of unsymmetrical smooth spillway with 3 types of discharge flow (Q) based on 100, 1,000 and 10,000 years return period for Q1, Q2 and Q3 respectively. Due to the space limitation, only three Q models were done experimentally in the laboratory while the probable maximum flood (PMF) for Q4 model was conducted in simulation.

For second objective, simulation using ANSYS were done to estimate and capture hydraulic characteristics of unsymmetrical smooth spillway such as velocity and water surface profiles. For the first step of simulation analyses, both hydraulic characteristics were validated using 5 turbulence models which were Standard κ-ε (SKE), Renormalization Group κ-ε (RNG), Realizable κ-ε (RKE), Standard κ-ω (SKW), and Shear Stress Transport κ-ω. For the verification, the grid independent test

(GIT), time step size and simulation time were adopted to get suitable and stable simulation setups for this type of geometry.

Next, the simulations were conducted using 4 types of discharge (Q1, Q2, Q3, and Q4). The selected turbulence model, SKW with appropriate mesh and boundary setup was run through the Fluent module in ANSYS by applying RANS equation combined with VOF.

Finally for third objective, the performance of energy dissipation in the stilling basin region was evaluated for the smooth spillway via multiple chute piers modifications (diverge -45 degree, diverge -15 degree, converge 15 degree, and converge 45 degree) using Q3 and Q4 discharge types. The same simulation setup from second objective was used in this study.

The numerical simulation has already become a new trend in the hydraulic investigation of smooth spillway. However, the hydraulic characteristics of unsymmetrical spillway are still hard to come by. Meanwhile, the common weirs been used for the smooth spillway are either the ogee or labyrinth weir type. The water flow on the elliptical weir combined with chute piers on the smooth spillway can still be further investigated. The effects of multiple geometry combination on the hydraulic performance of spillway is highlighted in this study. Therefore, a detail investigation of the effects of unsymmetrical smooth spillway with elliptical weir and chute pier on energy dissipation performance via simulation was performed.

Overall, the current research is focused on the investigation of unsymmetrical smooth spillway hydraulic characteristics where water spills over the elliptical weir and flows down through the chute pier towards the stilling basin at the downstream. The overall hydraulic characteristics of the spillway and energy dissipation performance of hydraulic jump were analysed using Fluent ANSYS software.

1.6 Thesis Outline

The outline of this thesis begins with Chapter 1 on the introduction of the investigation of hydraulic characteristics of unsymmetrical smooth spillway by simulation and validation with experimental results.

The literature review on types of spillway including smooth and stepped spillways are explained in detail in Chapter 2. The categories, factors, and other theoretical backgrounds of hydraulic characteristics of the spillway are also elaborated. In addition, the compilation studies on spillway conducted experimentally and numerically are presented in this chapter, focusing the review point of views of research on the study of both methods to get insight into hydraulic characteristics of the smooth spillway.

Chapter 3 on Methodology gives details on how the study was conducted to achieve all listed objectives, geometry models, instrumentation and measurement methods. This chapter also focuses on the turbulence model type as well as the verfication methods used in this study.

The findings of simulations and experimentals of unsymmetrical smooth spillway analysis are presented and discussed in Chapter 4. The results and discussions of all drawn objectives are laid out.

Chapter 5 presents a conclusion to answer the objectives of the study, contributions and recommendations for a brief future outlook.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Most of past studies were conducted numerically on symmetrical spillway by using ogee crest weir or labyrinth weir with a smooth chute or stepped chute in the middle and with the stilling basin at the toe. In order to investigate free surface flows and their interactions with hydraulic structures, modelling by CFD is therefore attractive in terms of cost and time; more importantly, the detailed flow description is obtained for the entire flow field with sufficient accuracy (Assy, 2001; Chatila & Tabbara, 2004). The water surface profile and velocity can be predicted faster and more economical compared to experimental method even though it's more convincing due to the real-time approach (Dehdar-behbahani & Parsaie, 2016; Kirkgöz et al., 2009; Savage & Johnson, 2001). Both parameters were considered as crucial information in the open-channel flow study because we are dealing with the free surface while in the close-system, the hydraulic behaviour tends to follow the shape of the geometry (W. Li et al., 1989).

A spillway is a hydraulic structure which is critical for the dams to release surplus or flood water to prevent damages to dam structures due to reservoir storage exceeding the designed allowable capacity (Chatila & Tabbara, 2004). Two main aspects that should be considered during design stage is the hydraulic and structure of spillway. The capacity of both aspects should be analysed properly to prevent any failure during operations (USBR, 1987).

For hydraulic, the open channel water will flow down over the spillway, accelerated by gravity along the chute, and flow further through the stilling basin

(Chanson, 2004; Tabbara et al., 2005). There are 3 types of spillway classified based on their functions which are service spillway, emergency spillway, and auxiliary spillway (Novak et al., 2014; USBR, 1987). Instead of that classification, generally, there are 2 types of common spillway used worldwide based on their geometries, which are smooth spillway and stepped spillway. However, certain researchers only focus on the water flow behaviour or characteristics on sub-component of spillway such as stilling basin, labyrinth weirs and chute piers. All of these spillways are studied through experimental and/or numerical approaches for the past decades.

Hydraulic characteristics can be strongly affected by the geometrical design of spillway, Froude number, and Reynolds number. The considerations of these criteria in the design are crucial to provide optimal hydraulic performance (Chatila & Tabbara, 2004). To measure the hydraulic performance of spillway in the scope of research, experimental and/or numerical approach are applied. The experimental approach can provide the reliable data if properly implemented. Instead of that, due to the development of high-performance computing, many researchers are also considering the numerical approach in their studies because it can provide good prediction of water flow on the spillway. Both methods and their results that produced by researchers must be thoroughly reviewed to prevent any redundant studies. The aim of this chapter is to discuss the main components of spillway and related paper reviews on the broad aspect of hydraulic behaviour results and relevant methods that been used by previous researchers regarding spillway components.

2.2 Components of Spillway and Studies

In this section, the spillway will be divided according to its sub-components from the upstream to the toe of the spillway. In the upstream region of the spillway, crest is constructed to contain the flood water and to release it slowly down through the chute spillway. After the weir, the chute spillway which has a downward discharge angle channel play a role to route the water down into the stilling basin (Kirkgöz et al., 2009). In the stilling basin, the energy dissipation process will occur whereby the installed multiple-blocks will promote a hydraulic jump that will dissipate the massive water flow energy to prevent any damages.

2.2.1 Crest

In general, the crest will make the water discharge more smoothly by spilling over the structure without significant water pressure. In terms of design, Reese & Maynord (1987) have developed a design procedure for spillway crests through experiments with various conditions by considering length of crest, minimum pressure and maximum pressure on structure, hydraulic and structure stability and few other critical criteria for spillway design where the procedure can be applied in various types of crest design.

Based on the conducted reviews, the crest operation can also be divided into 2 types of operation, either controlled crest (equipped with water gate) or uncontrolled crest (without gate) (S. Hong et al., 2015). The controlled and uncontrolled crests were determined based on the design requirement, space and cost constraint. Due to these kind of designs, Assy (2001) used finite difference method to predict water surface profiles on controlled and uncontrolled flows over two types of crest (simple and circular crests) using Neumann condition on body contour. It was found that the predicted results agreed well with the experimental results. The method is also easy to analyse for a variety of crest geometries.

2.2.1(a) Ogee Crest

This type of crest can be considered a common crest design and always be used in construction of dam. During operation of spillway, the water level will increase gradually behind it and reach the peak of the ogee crest weir before flowing down the tangent slope and reaching the toe of the stilling basin (USBR, 1987). The design of this crest as shown in Figure 2.1, is well established and has been used for decades in the world.

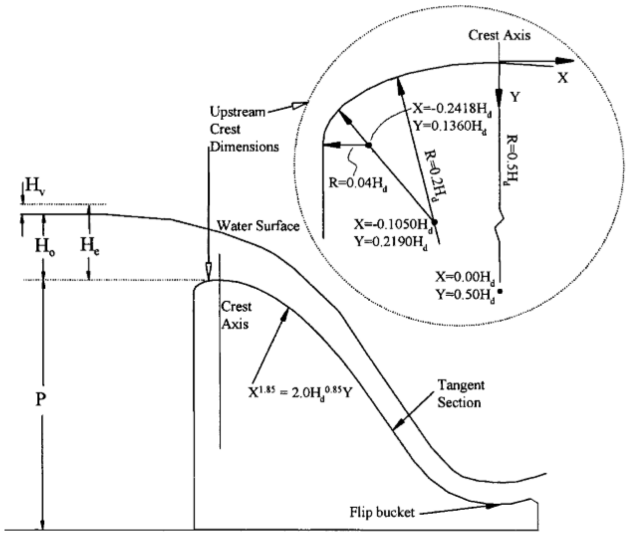


Figure 2.1 Standard Ogee Crest Smooth Spillway (Savage & Johnson, 2001)

W. Li, Xie, & Chen in 1989 conducted a 2-dimensional (2D) numerical approach to determine the discharge coefficient of ogee crest spillway adopting the boundary fitted coordinate system to map the complex domain. It was found that the discharge coefficient was reduced by a scan method. However, the finite element showed good agreement with experimental results. Yakun et al. (1998) also conducted the numerical modelling using iterative method to predict the discharge coefficient, water surface profile, and pressure distribution on 2D ogee crest type spillway.

The flow parameters on the spillway were also studied using 3D numerical method focusing on discharge and pressure by Savage & Johnson (2001). The spillway geometry hydraulic characteristics were simulated in CFD program adopting governing equation solved by Reynolds-averaged Navier Stokes to model the physical model setup. Both parameters were previously scarcely attempted by researchers numerically. Chatila & Tabbara (2004) used the CFD software (ADINA) to predict 2D water surface profiles on the ogee crest smooth spillway. The numerical prediction using κ-ε turbulence model went well with the measured results.

Dae & Park (2005) investigated numerically on a scaled model and the surface roughness effects on the hydraulic characteristics of water flow on the smooth spillway using Flow-3D software. Based on the results, surface roughness slightly affect the flowrate, crest pressure and maximum velocity of the spillways. The surface roughness study was extended by Dargahi (2006) using 2 types of turbulence model (κ-ε and RNG) to simulate the ogee crest spillway geometry and compared it with experiments. The water surface profiles, mean velocity profiles and wall shear stress results were analysed in this study. In the same year, Johnson & Savage (2006) extended their numerical comparison study in 2001 by including tailwater effects on the pressure distribution on ogee crest spillway model. The findings showed that the numerical model accurately predicted the rate of flow over the spillway and the pressure distribution on the spillway.

The ogee spillway study continued in 2008 with experimental works done by Tullis & Neilson (2008) which study the degree of submergence, S and head-discharge relationships on the ogee crest smooth spillway. Tullis (2011) also extended the study on the relationship between submerge discharge coefficient and the upstream and downstream of ogee crest spillway.

In the numerical analysis, mesh analyses were considered an essential validation process. By using the mean squared error (MSE) analysis, Kirkgöz et al. (2009)

provided a good comparison of SKE and SKW turbulence models compared to experimental data on the flow over chute spillway. Salmasi (2018) predicted the discharge coefficient by using new established data instead of relying on USBR design charts for ogee type spillways. After adopting gene expression programming (GEP) and multiple regression technique, the results showed that GEP was superior to the regression analysis in predicting the discharge coefficient.

In 2019 trend, in order to predict the discharge coefficient of ogee spillway, Roushangar et al. (2019) applied artificial intelligence. Ryan et al. (2019) applied 2D numerical method to simulate flooding situation using smooth particle hydrodynamics (SPH) at the upstream area of spillway focusing on water level characteristics of various flow rates.

2.2.1(b) Labyrinth Weir

Labyrinth weir is another type of spillway crest. Alternatively, the labyrinth weir is efficient to facilitate large discharges per unit width (Cassidy et al., 1985). The water capacity that can be facilitated by the weir is larger than to the ogee crest type. The advantage of this crest is due to the increment of the crest length thus increasing the dam safety by increasing discharge capability to pass larger PMF, and improve discharge efficiency (Anderson & Tullis, 2013; Crookston & Tullis, 2013a). The capacity of labyrinth weir can be designed making use of the linear discharge equation of weir:

$$Q = \frac{2}{3} C_d L \sqrt{2g} H_t^{1.5} \qquad (1)$$

Where C_d is a dimensionless crest coefficient; g is aacceleration of gravity; L is an effective length of the weir; and H_t is a total head on the crest. The total head is the depth of water above the crest plus local velocity head ($v^2/2g$) of the approach flow.

Another type of labyrinth weir is a Piano Keys Weir (PKW). Both designs of weirs are shown in Figure 2.2. Figure 2.2 shows a geometric design variability of different types of PKW alone with optional features and optional ramps (inlet) of rectangular labyrinth wier geometries. Type-A PKW has a flat crest, while optional features along PKW are a half-round crest with a parapet that makes it different than the other labyrinth weir.

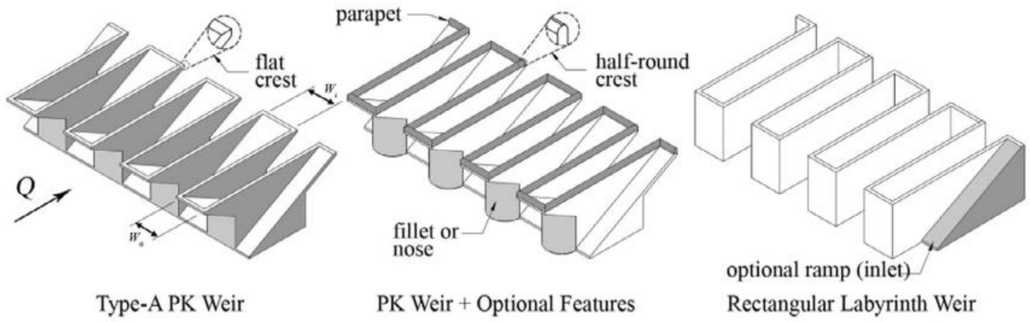


Figure 2.2 Labyrinth Weir and Piano Keys Weir (PKW) (Crookston et al., 2018)

The water flow behaviour of this type of crest was studied by Cassidy et al. (1985). Discharge characteristics of this labyrinth with triangular plan indicate that the efficiency decreases when the water lever increases. To gain more depth about this, Anderson & Tullis (2013) conducted the experiment involving two types of labyrinth weir which were PKW and multiple configuration of trapezoidal labyrinth weir. Comparing both types of weir, the trapezoidal have higher discharge eficiency compared to PKW. Crookston & Tullis (2013a) provide extended version of discharge flow characteristics on labyrinth spillway inclusive of multiple geometry shape effects on the hydraulic performance and nappe flow behaviour.

Erpicum et al. (2016) conducted an experiment to study the connection between surface tension and the scale effects using four size of scale models of PKW labyrinth. Instead of the scale effects, Dabling & Tullis (2018) studied incoming flow angle, β efficiency approach on the labyrinth weir. Based on their results, the discharge eficiency dropped as much as 11% due to the 45 degree approach of flow angle.

Instead of normal labyrinth, labyrinth side weirs were also considered in this review. Aydin (2012) and Aydin & Emiroglu (2013) conducted a numerical comparison study on labyrinth side weir focusing on six turbulence models (Spalart Allmaras Model, SKW, RNG, SKE, RKE and Reynolds stress model (RSM) performance. The grid convergence index (GCI) and discharge capacity were analysed and the result had good agreement with the experiment. While for the numerical analysis of PKW, Crookston et al. (2018) applied LES and RNG turbulence models to predict the discharge capacity of PKW models. The root mean square error (RMSE) and Mean absolute percentage error (MAPE) were used for the validation process. Overall, the prediction results showed good agreement with the experimental and emperical equations in terms of discharge performances.

2.2.1(c) Other Crest Geometries

Instead of ogee crest and labyrinth weir, many researchers studied the flow behaviour on the basic shape of the crest such as flat crest, circular crest broad crest, and many others. Montes (1994) used a numerical approach to predict the water surface and bottom pressure profiles of water flow along the rectangular channel with mild slope and steep slope with two types of angle (45 degree and 60 degree). Olsen & Kjellesvig (1998) used κ-ε turbulence models to predict the discharge coefficient, water surface, velocity profiles, and bottom pressure for the 2D and 3D spillway cases which agreed well with the experiment results. In 1999, Song & Zhou (1999) used large-eddy simulation (LES) method to analyse water surface profile over tunnel spillway. Andersson et al. (2013) used the SKE and SSG turbulence model to simulate a unique spilling reservoir geometry. The prediction results were than compared with the experimental results. Mesh convergence analysis had been done prior to the results analysis. The velocity and water surface profiles were compared and shown acceptable agreement with the physical model.

For the circular crest, Assy (2001) used finite difference method to predict water surface profiles on controlled (with gate) and uncontrolled (without gate) flows over two types of crest (simple crest and circular crest). The prediction results agreed well with the experimental results. Instead of numerical approach, Bagheri & Heidarpour (2010) conducted a theory experiment comparison of discharge coefficient and crest velocity based on vortex theory on circular crested weirs as shown in Figure 2.3. The achieved results were within 5% tolerance of the experiments. Rahimzadeh et al. (2012) also focused on the circular type of crest by conducting 3D numerical studies using 4 types of turbulence models (RSM, RNG, RKE, and SST-KW) performance where the RSM turbulence model outperformed the rest. Moreover, Akoz et al. (2014) provided comprehensive numerical study validation and verification on hydraulic performances of flow over the semicylindrical weir. The numerical analysis were validated and verified carefully using y^+ analysis, multiple size of mesh, six turbulence models (SKE, RNG, RKE, Modified KW, SST-KW, and RSM) and GCI.

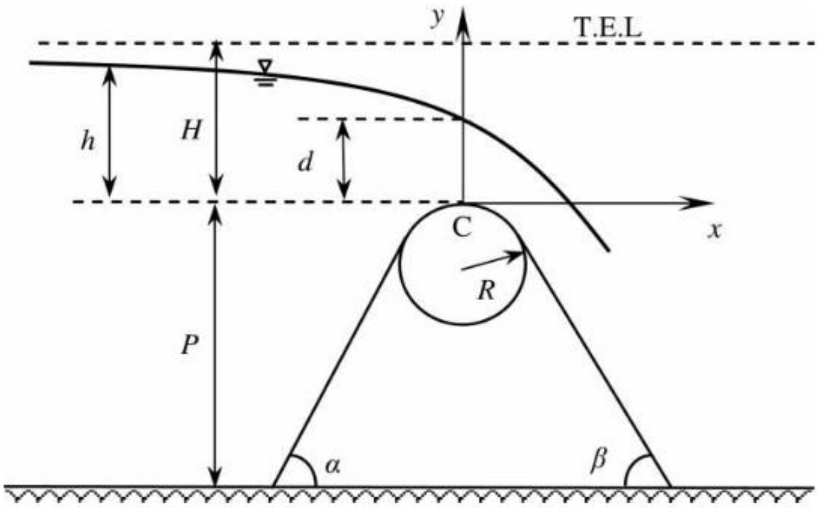


Figure 2.3 Example of Circular-Crested Weir (Bagheri & Heidarpour, 2010)

Hager & Schwalt (1994) studied high tailwater submergence flows over long-crested weirs where discharge coefficients remain constant. The water surface profile, pressure, and velocity were measured. The undular flow were also observed in this experiment. Ohtsu et al. (2001) studied undular-jump formations based on the Froude number, turbulent boundary layer, and Reynold number. Hargreaves et al. (2007) replicated the broad crested weir studied by Hager & Schwalt (1994) and simulate the model using 3 turbulence models (RNG, SKE and RSM). The 3D model results provided predictions closer to the experimental values compared with the 2D model (Hargreaves et al., 2007). In the broad weir study, J. et al. (2010) applied the 3D numerical model on the flow around spur dikes using SKW turbulence models. The water flow characteristics on the riverbed such as effects on spur dike angle on the water flow, velocity, and the shear stress at the river bed were analysed and compared with the experimental results. Shamloo & Pirzadeh (2015) extended the broad crested weir study by using multiple configuration of rib and proceeded with a numerical comparison study on various depths and Reynolds number. In 2019, Daneshfaraz et al. (2019) studied the flow behaviour over the broad crest with multiple hole configuration by using the numerical comparison method with 3 different types of turbulence models (SKE, RNG, and LES).

Futhermore, Mohamed (2013) conducted a numerical-experiment comparison using SKE turbulence models on prismatic chutes. The results showed good agreement with water depth comparison while pressure and velocity distribution produced small results deviation. Marsooli & Wu (2014) also conducted a study on multiple types of weir. The trapezoidal, triangular, and rectangular weir shape were simulated in 2D and 3D. The validation and verification were observed using the RMSE method and the transient numerical analysis were presented carefully by them.

Besides of the crest, a few researchers also aimed their studies on water flow profiles only. For example, Gadge et al. (2016) investigated numerically and experimentally the nappe flow profiles (bottom and upper) of flow through orifice with multiple varied heights of orifice and design heads. The SKE, RNG, RKE, and SKW turbulence models were used in this study.

2.2.2 Chute

In this review, there are two main types of chute spillway geometry that were studied which are the smooth and stepped chutes. Both types of geometry are used in a lot of research to study multiple fluid characteristic and behaviours when water flows down the spillway. For the uncontrolled crest, the length of the chute will be longer to improve energy dissipation of water compared to controlled crest which have the gate to regulate the water discharge (S. Hong et al., 2015).

2.2.2(a) Smooth Chute

Castro-Orgaz, (2009) developed an analytical equation to predict free surface profile at the chute region which agreed well with experiments. Castro-Orgaz (2010) and Castro-Orgaz & Hager (2010) extended their studies focusing on turbulent velocity profiles at the inception point and chute flow development prediction.

Valero & Bung (2016) conducted an extensive study on the air-water interaction in the non-aerated region of the smooth spillway flow whereby the inception point and air layer were analysed experimentally. A general inception location formulation were derived in the prediction of air entrainment.

Besides the air entrainment phenomenon, the cavitation phenomenon on the spillway has also been a great concern among researchers. Aerators and ramps are used in chute spillways to prevent or reduce the cavitation level on the spillway structure. The aerator performance at the chute region of Bergeforsen dam spillway were studied by Teng & Yang (2016). The prediction results using the RKE turbulence model agreed well with the physical model which the deviation was within acceptable limits. Kumcu (2017) conducted a numerical comparison study on the smooth spillway with the ram and the average deviation is 3.2% compared with physical model results. Later studies conducted by Aydin (2018), R. Bai et al. (2018), Lian et al. (2017), and Yang et al. (2019) also focused on air entrainment characteristics using bottom inlet aerator on the spillway which functioned to prevent cavitation damages in the negative pressure region.

Bhate et al. (2018) conducted a hydraulic experiment by evaluating methods to mitigate cavitation on a controlled flow spillway (orifice spillway). Three types of cavitation mitigation methods were studied which are stopping the occurrence, applying cavitation resistance material, and inducing the aeration. However, the aeration method is still considered as the most practical and economical option to mitigate the cavitation damage of spillway structure. Bung & Valero (2018) and Valero & Bung (2018) purposed new method to predict free surface interaction between water and air in high velocity flow condition.

Gadge et al. (2018) purposed a new equation based on the verification and validation of numerical comparison studies as a basic guideline for designing upper surface profile of orifice spillway. The proposed guideline is based on the discharge coefficient, bottom and roof profile of spillway criteria (Gadge et al., 2019).

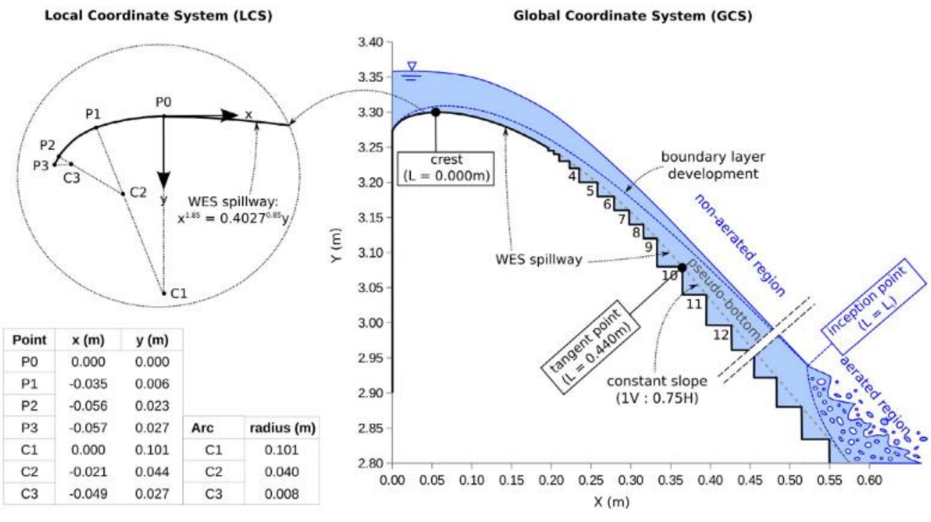
For the smooth chute, piers are also used on the chute. Gadhe, Patil, & Bhosekar (2018) studied on pier design to support the controlled gates at the upstream of the spillway crest. Instead of cavitation, piers also produce standing waves or rooster tail at the end of the piers when the water flow is in the supercritical flow condition. The nose pier performance at the upstream of the crest were studied and the flow characteristics and energy dissipation performed better on the improved design of spillway. Besides the nose piers which are installed at the upstream, chute piers also used at the downstream of the crest, and the study was conducted by Luna-Bahena et al. (2018). The findings indicate that the chute piers can stimulate the development of air entrainment and further reduce cavitation damages (Luna-Bahena et al., 2018).

In the same year, H. S. Hong et al. (2018) studied the transitional flow consisting of uncontrolled, controlled, submerged-uncontrolled and submerged controlled flow. Pedersen et al. (2018) focused on submergence flow behaviour on smooth spillway and numerical errors were thoroughly analysed. Teng & Yang (2018) used numerical simulation to investigate abnormality of models and prototype results of flow characteristics on chute spillways with flip-bucket aerators.

Based on the above reviews, all the studies focused on the smooth chute whereby the dissipation of water flow energy is quite low compared to the stepped energy. Thus, Daneshfaraz et al. (2020) studied the blocks effects on the flow at the smooth chute where the numerical studied were conducted and showed significant agreement with experimental results where the relative errors was between 0.38% to 6.89%. The block bed configuration in the experiment showed 15.4% higher energy dissipation than the smooth chute setup.

2.2.2(b) Stepped Chute

The stepped spillway is defined based on stepped chute design instead of the smooth chute. In standard stepped spillway, the stepped chute is added to the WES smooth spillway type as showed in Figure 2.4. This type of spillway replaced the smooth chute with the stepped ladder which functioned to provide macro-roughness to flow coming down from the crest. The macro-roughness of the steps effectively increases the energy dissipation by reducing the velocity of fluid even when the water depth is increases (Tongkratoke et al., 2009).



Point	x (m)	y (m)
P0	0.000	0.000
P1	-0.035	0.006
P2	-0.056	0.023
P3	-0.057	0.027
C1	0.000	0.101
C2	-0.021	0.044
C3	-0.049	0.027

Arc	radius (m)
C1	0.101
C2	0.040
C3	0.008

Figure 2.4 Geometry of Stepped Spillway (Bayon et al., 2018)

Compared to the smooth spillway flow, the hydraulic characteristics of the stepped spillway involves 3 phases of water flow along the stepped chute as shown in Figure 2.5 which are the non-aerated region, inception point, and aerated region or ‘white waters’ (Bayon et al., 2018; Chanson, 2004; Zhang & Chanson, 2017).

According to Andrade Simões et al. (2010), stepped spillways provide higher flow aeration and 500% more friction factor than smooth spillways.

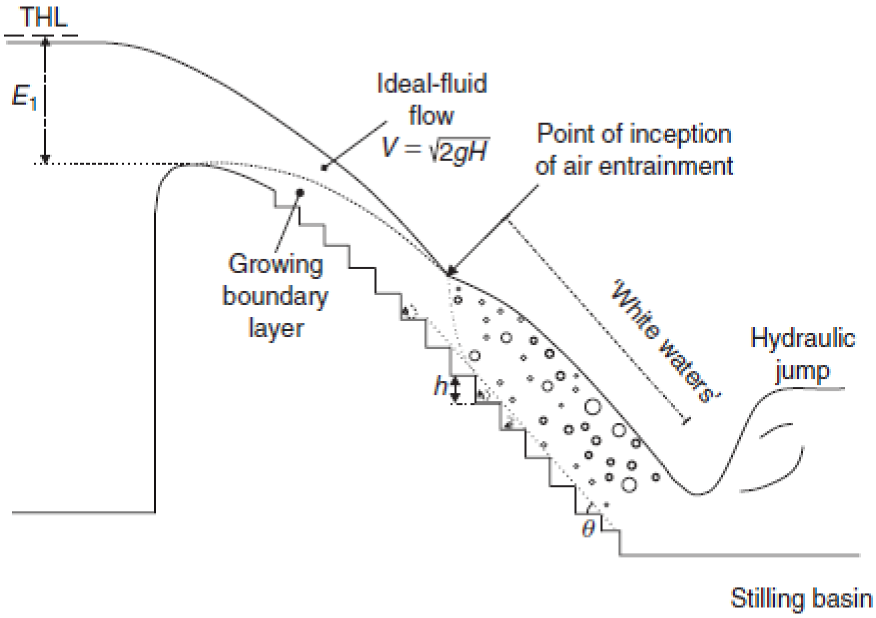


Figure 2.5 Water Flow on The Stepped Spillway (Chanson, 2004)

The flow along the stepped region can also be divided into two types namely nappe flow or skimming flow. The nappe flow is like a "water fall" type of flow which occurs during low discharge and may be combined with the large steps but when the water discharge increases on certain levels, skimming flow will develop and the water flow will skim over the steps towards the downstream (Boes & Hager, 2003a; Chakib & Mohammed, 2015).

Sorensen (1985) conducted an investigation on 3 types of modification stepped spillway. The study focused on flow transition downstream of the crest, energy dissipation on the stepped spillway, toe velocities, and training wall heights along the stepped region based on experimental results. Instead of going through the experimental approach, Rajaratnam (1990) developed a prediction method of energy loss on the skimming flow region of stepped spillway. Christodoulou (1993) indicated that critical depth plays a significant role to increase energy dissipation on the stepped spillway.

Instead of nappe flow, Chamani & Rajaratnam (1999) conducted an experiment focusing on skimming flow characteristics on a large stepped spillway including velocity profiles, air concentrations and energy dissipations. They also concluded that the water depth on the stepped spillway can be measured at which the air concentration is equal to 90%.

Instead of water depth, Chanson & Toombes (2003) also focused on air concentration and interaction between air and water at the transition and skimming flow regions of stepped spillway. The interaction between air-water and turbulent level were observed in their work. Boes & Hager (2003a) carried out the experiment focusing on multiple hydraulic criteria affecting the stepped spillway design while Boes & Hager (2003b) focusing more on the scale effects, air concnetration, inception point, and skimming flow characteristics. In order to improve the understanding of skimming flow, Ohtsu et al. (2004) conducted an experiment on a few designs of stepped chute size with two types of skimming flow (parallel flow and partial parallel flow). Bung (2011) concentrated on the water region of the stepped spillway after the inception point, where the aeration starts to develop. Zare & Doering (2012) expanded energy dissipation studies by using multiple configurations of baffles and sills. In previous experiments, a lot of intrusive instruments were used to measure multiple hydraulic characteristics. Amador et al. (2006) used the nonintrusive technique, the particle image velocimetry (PIV) to investigate the turbulence characteristics of the stepped spillway during high velocities concerning cavitation in the nonaerated flow region. Frizell et al. (2013) used the PIV method to capture shear strain formations which can lead to cavitation formation on the step edges. Based on their observation, the cavitation risk can be reduced using a steep slopes of step and this were observed by reducing ambient pressures (Frizell et al., 2013).

Zhang & Chanson (2017) studied the relationship between air bubble diffusivity and eddy viscosity based on the Reynolds number increment at the downstream of the inception point region of stepped spillway flow instead of cavitation. While Felder & Chanson (2016) focused on energy dissipation performance and Darcy-Weisbach friction factors on the embankment of stepped spillway. The reinvestigation of aeration on the stepped spillway focusing on air-water interface and gas transfer mechanism in the aeration region of stepped spillway flow was done by Bung & Valero (2018). The geometry and discharge of spillway are the main factors of hydraulic characteristic of aeration flow. Ljubičić et al. (2018) combined the stepped spillway with an upward stilling basin and provided detail insight of hydraulic jump characteristics such as roller length, sequent depth, and energy dissipation.

Multiple configurations of the step spillway provided different performance of hydraulic characteristics as performed by Zhang & Chanson (2018a). They also investigated the performance and practicability of optical flow methods on hydraulic characteristics measurement on the stepped spillway (Zhang & Chanson, 2018b). In 2019, Kramer & Chanson (2019) enhanced the image-based velocimetry technique in the turbulent flow (highly-aerated) region of the stepped spillway. In the same publishing year, Parsaie & Haghiabi (2019a) focused on inception point of circular crested stepped spillway. To gain more depth, Parsaie & Haghiabi (2019b) conducted an experiment on the hydraulic characteristics of the quarter-circular crested stepped spillway (QCSS) in terms of inception length, critical depth, discharge coefficient, and energy dissipation. Likewise Parsaie & Haghiabi (2019b), Rajaei et al. (2019) studied the geometry effect using gabions on the stepped spillway which allowed water through the impervious layer and increase the energy dissipation by 16.9% on the stepped spillway.

A numerical comparison analysis on the stepped spillway was done by Chen et al. (2002). The results indicated that maximum pressures occur near the horizontal step edges which are caused by the impact of water flow. The numerical simulation (κ-ε turbulence model) quite agreed with the experimental data using unstructured grid to overcome the complex geometry (Chen et al., 2002). They also indicated that the mean minimum pressure can be used to asses the cavitation phenomena on the stepped spillway which have skimming flow on the steps and eddy flow in the corner (Chen et al., 2002). The eddy flow shown in Figure 2.6 occurs due to high velocity flow through the pseudo bottom thus increasing the pressure near the step edges as observed by Boes & Hager (2003a). Unlike Chen et al. (2002), who used SKE turbulence models, Tongkratoke et al. (2009) used the linear, non-linear, large-eddy simulation (LES) and modified non-linear turbulence models on the stepped spillway case. All the models were compared with experimental results based on air concentration and velocity distribution.

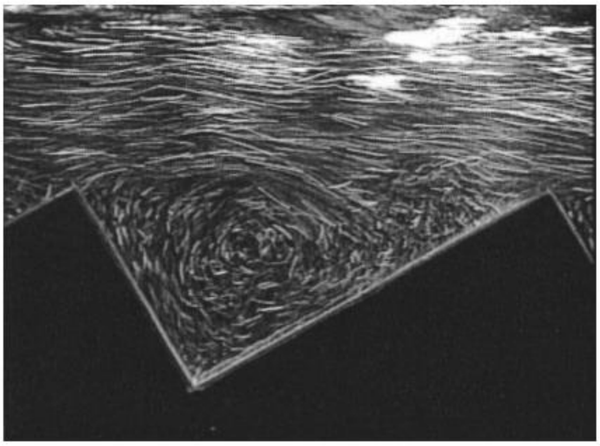


Figure 2.6 Eddy Flow at the Stepped Spillway (Boes & Hager, 2003a)

The non-linear model is considered to be more precise prediction rather than linear turbulence model κ-ε and can reduce the computation time of simulation. To reduce the numerical error and gain the best turbulence models, various turbulence