NUMERICAL MODELLING OF OGEE SPILLWAY HYDRUALICS USING CFD AND ITS VALIDATION THROUGH PHYSICAL MODEL TESTS

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR THE AWARD OF DEGREE
OF

MASTER OF TECHNOLOGY

IN

HYDRAULICS AND WATER RESOURCES ENGINEERING

Submitted By

DOLON BANERJEE

(Roll No. 2K16/HFE/09)

Under the supervision of

PROF. (DR.) BHARAT JHAMNANI

DEPARTMENT OF CIVIL ENGINEERING

Delhi Technological University

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

JULY, 2018

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CANDIDATE'S DECLARATION

I, Dolon Banerjee, Roll No. 2K16/HFE/09 of M.Tech. Hydraulics and Water Resources Engineering, hereby declare that the project Dissertation titled **"NUMERICAL MODELLING OF OGEE SPILLWAY HYDRAULICS USING CFD AND ITS VALIDATION THROUGH PHYSICAL MODEL TESTS"** which is submitted by me to the Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not been used for the award of any Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Place: Delhi, INDIA **DOLON BANERJEE**

Date:

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly Delhi College of Engineering)

Bawana Road, Delhi-110042

CERTIFICATE

I hereby certify that the Project Dissertation titled **"NUMERICAL MODELLING OF OGEE SPILLWAY HYDRAULICS USING CFD AND ITS VALIDATION THROUGH PHYSICAL MODEL TESTS"** by **Dolon Banerjee**, Roll No. **2K16/HFE/09**, Department of Civil Engineering, Delhi in partial fulfillment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the student under my supervision. To the best of my knowledge, this work has not been submitted in parts or full for any Degree or Diploma to this University or elsewhere.

Place, Delhi, India

Date:

Prof. (Dr.) BHARAT JHAMNANI

Professor, Civil Engineering

Delhi Technological University

Delhi, INDIA

ACKNOWLEDGEMENT

First and foremost, I would like to express my heartfelt appreciation to my study leader and proponent, Professor Dr. Bharat Jhamnani for his immense support and encouragement. Without his valuable guidance, advice and instructions, this work would not have been successful and timeous. Apart from inestimable academic support gained from him, he is also a professional person, hardworking and humble to everyone. He is a model to me and I owe him much respect.

I would like to thank Dr. Rakesh Kumar Arya for his advice and generosity to discuss different challenges of my research project. Besides I would also like to thank my mom who despite the distance has always been the source of strength towards the accomplishment of this thesis. Her love is too immense to express adequately in words but I owe them a very special gratitude. This is the case also with my friends who gave me enormous encouragement and support in one way or another along this journey.

I extend my deep-felt thanks to my postgraduate colleagues for their companionship during these two years. I acknowledge also each and every one who has contributed to the success of this thesis.

DOLON BANERJEE

M.Tech (HFE), DTU

Roll NO. 2K16/HFE/09

ABSTRACT

As a part of the design process for hydroelectric generating stations hydraulic engineers typically conduct some form of model testing. The desired outcome from the testing can vary considerably depending on the specific situation, but often characteristics such as velocity patterns, discharge rating curves, water surface profiles, and pressures at various locations are measured. Due to recent advances in computational power and numerical techniques, it is now possible to obtain much of this information through numerical modeling.

Modern Computational Fluid Dynamics (CFD) modeling are becoming common design and analysis tools in the engineering field. Nowadays, project designs involve the use of CFD techniques along with physical scale modeling to analyze the complex rapidly varied and turbulent flows which would not be easily analyzed by physical modeling. In particular, the consideration and/or use of CFD modeling in the Hydraulic Engineering field remains on the increase. Apart from being used for comparison with other design techniques, CFD may in future become a standalone modeling technique in hydraulic structures design.

This research aims to use CFD models to validate the simulation of the flow over a ogee dam spillway which was installed in the Hydraulic Laboratory of Stellenbosch University. To achieve this simulation of the flow which involves an interaction between water and air, the flow behavior has been mapped by the Volume of Fluid (VOF) and the realizable "k-ε" turbulence numerical models. The Volume of Fluid (VOF) and the realizable "k-ε" models simulate the free surface of two-phase flow and the flow turbulence, respectively. Firstly, it subsequently presents the geometry and dimensions of the physical models, the testing procedure and the experimental test results achieved from this modeling exercise. For CFD modeling, a commercially available Computational Fluid Dynamics (CFD) package, Ansys-Fluent, was used. To model the physical model, the use of Reynolds-averaged Navier-Stokes equations in combination with the realizable k-ε eddy-viscosity closure model was adopted. The process of CFD model development and the underlying theory of it are discussed in this

thesis. In order to determine the required mesh size, the mesh sensitivity tests were conducted on the 2 dimensional models.

Finally, the pressure readings and water levels along with the water surcharge produced by numerical models are discussed through a validation process by comparing the CFD model in 2D and 3D results with the results obtained from physical models. The outcome proved that CFD models are able to map the behavior of both flow phases since they exhibited a close correlation to those achieved in the physical models. Even though some slight differences in values were revealed, the graphical trend remains reasonably similar for all test results.

CONTENTS

List of Figures

List of Tables

List of symbols and abbreviations

Symbol	Description
C	Coefficient of discharge
CFD	Computational Fluid Dynamics
DPM	Dispersed Phase Model
F_r	Froude number
FSL	Full Supply Level
G	Gravitational force
G_b	Turbulent kinetic energy generation due to buoyancy
G_k	Turbulent kinetic energy generation due to mean velocity gradients
H	Water level measured above the crest
H_e	Head on the crest
H_o	Design head for ogee weir
H_v	Velocity Head
ICOLD	International Commission on Large Dams
K_a	Abutment contraction coefficient
L	Lateral crest width
L'	Net length of the crest
N	Pier contraction coefficient
P	Weir height
PDEs	Partial Differential Equations
PISO	Pressure-Implicit Split-Operator

Q	Discharge
R	Radius of abutment
RANS	Reynolds-average Navier-Stokes
R_e	Reynolds number
USBR	United States Bureau of Reclamation
V	Velocity
VOF	Volume of Fluid

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Over the past 30 years numerical modeling techniques have been rapidly developing as computational power has enhanced to the point where numerical solutions are now possible for many applications. This development has led to the widespread use of numerical modeling as a standard design tool in many engineering disciplines.

Despite the wide range of numerical modeling applications, the fundamental principles upon which all numerical models are based is similar for all models. Problems begin with a set of partial differential equations that describe the underlying physics of the particular situation. Some type of numerical method, such as finite element analysis or the finite volume method is then used to formulate a set of algebraic equations that represent the partial differential equations. An approximate solution to those algebraic equations is then obtained through some form of either an iterative or matrix solution. This solution is often very computationally intensive, which makes the use of modern computational power so important to the use of numerical models. In most cases, the numerical model solutions are verified or calibrated through comparisons to field observations or physical model experiments before being applied in practice. Even after extensive model verification, sound engineering judgment is required to ensure the accuracy of any model output.

An increasing need of water supply, flood control, navigation, hydroelectric power generation, fishing and recreation has made dam construction a high priority throughout the world. As water is of critical importance to protect, technological improvements in the design and analysis of dams are needed for a better management of water resources.

There are several types of spillways but the most common type is the ogee-crested spillways because of their ability to release surplus water from upstream to downstream efficiently and safely when properly designed and implemented. These types of spillways, which are also called as the overflow spillways, have larger capacities, higher hydraulic conformities, and easily adaptable to all type of dams. The ogee-shape holds over the downstream side until it reaches the determined slope on the downstream face. This slope is maintained for the remainder of the spillway as far as the base of the dam and finally, the flow enters a suitable energy dissipating basin.

All dams are equipped with spillways as a safety measure against overtopping. They are provided to safely carry water away from the reservoir, when water levels exceed the full supply level (FSL). In the recent years, there has been an increase in the frequency of large floods leading to high inflows into reservoirs. From the catalogue of Dam Failure in South Africa, it is clear that inadequate spillway design is the main cause of dam failure with a rate of 58% (Hattingh, 2012)[22]**.** Therefore, the appropriate design of spillways remains relevant so as to avoid overtopping of the non-overspill part of dams (Mays, 1999)[33]**.** Ogee spillways, which have a control weir with an S-shape in profile, have been substantially applied. They are deemed to be the most commonly used spillways due to their proper function, ability to control floodwater and high safety factor (Daneshkhah & Vosoughifar, 2012)[13].

Although much is understood about general ogee shape and its flow characteristics, a slight distortion in the standard design automatically affects its flow properties (Kim & Park, 2005)[29].

For over 100 years, physical modeling was the only analysis tool available used as the baseline to validate other methods (Johnson & Savage, 2006)[28]. To date, with recent advances in computational and numerical techniques, new design tools are evolving to assess rapidly varied flow situations (Savage *et al*, 2004)[40]. This development has led to the widespread use of numerical modeling as a standard design tool in various disciplines of engineering. This study seeks to examine the ability and the application of commercial Computational Fluid Dynamics (CFD) software namely “ANSYS-FLUENT” to model ogee spillways.

Computational Fluid Dynamics (CFD) is a branch of numerical modeling that has been developed for solving problems involving fluid flow. This technique has been gradually accepted by the Hydraulic/ Dam engineering community not only as an investigative tool in the research institutions (Kjellesvig, 1996; Savage & Johnson, 2006)[28] but also as a useful design tool (Gessler, 2005)[18]. With this complementary use of CFD techniques and physical scale modeling in Hydraulic Engineering, it has become not only important but also necessary to use physical models to validate CFD modeling (Guo*et al*, 1998)[21].

In addition, it has been of paramount benefit to use CFD modeling to identify the early problematic flow features for prototype cases. In the spillway flow field, CFD modeling is commonly used to analyse the complex hydraulic conditions such as air entrainment, flow separation, turbulence and shock waves. Savage and Johnson (2006)[28] employed Reynolds Average Navier-Stokes (RANS) equations to study the flow over an ogee spillway. Their results for predicting the pressure heads over the spillway surface and flow rates were in good agreement with the experimental results. Although a number of previous studies, such as Savage & Johnson (2006)[28], Guo*et al*. (1998)[21], Chatila&Tabbara (2004)[10], shows a good agreement between CFD and physical modeling, a need for validation of CFD modeling with particular physical models of particular cases is still arising as there is no single universal spillway design that could work for every flow scenario. In addition to this, there is still a significant lack of calibration and validation studies between CFD and physical modeling (ICOLD, 2012)[27].

1.2 OBJECIVES

The main purpose of this work is to compare the numerical simulation of the hydraulics of the ogee spillway by employing physical modeling. This is done for a few cases of different discharges simulated with the CFD package called Ansys-Fluent and compared to the laboratory test data.

The specific objectives of this research include:

1. To perform 2D CFD simulations of a selected spillway for different scenarios with the same spillway geometry and different flow conditions applied in the laboratory.
2. To perform the validation of the results by comparing it with the experimental results.
3. To carry out the analysis of the scouring phenomenon of the downstream and improve the downstream scour condition by applying corrective energy dissipating methods.

4. Also check the cavitations phenomenon and its severity for the spillway.

The main objective of this thesis is to investigate the flow parameters over a spillway using a three-dimensional numerical model. Flow characteristics such as flow rate, depths, water surface profiles, pressures on the ogee-crested spillway and downstream canal, vertical distributions of velocity are investigated both for model and prototype scale using commercial CFD code, Flow 2D. First, a validation is made comparing the numerical results with the experimental ones in the model scale.

Second, the scale effects are investigated by making additional simulations in the prototype scale and comparing these with the model scale results. Moreover, air entrainment model in Flow 2D is used to estimate the amount of air entrained as a result of the aeration device installed on the spillway. As a final step, the cavitations potential of dam spillway is investigated.

1.3 RESEARCH OUTLINE

This research project is subdivided into six chapters and structured as stated below:

Chapter one, namely general introduction, presents the background of carrying out this research, objectives and research outline. Briefly, it depicts the framework of the research project.

Chapter two consists of a literature with basic details on analytical and numerical methods carried out in the past about flow over ogee spillways. This chapter includes the hydraulics of spillways in general and ogee spillways in particular. The recent advances in the numerical modelling of spillways are also described. Along with these it includes the papers cited in following this project.

Chapter three describes the CFD modelling including modelling set-up, two and three-dimensional simulations for steady and fully hydrodynamic states.

Chapter four consists of the comparison of the results of the numerical and physical modelling and their interpretation.

In Chapter five the results are discussed and conclusions drawn. Recommendations for future research are also presented here.

CHAPTER 2

LITERATURE REVIEW

2.1 OVERVIEW

This chapter presents the literature review of the study on spillways in general and ogee spillways in particular. It details different aspects that are considered while designing an ogee spillway, the discharge computation, pressure distribution, as well as the factors defining the hydraulic crest performance of ogee spillways. It also includes a conceptual description of CFD where the design and methodology used for designing and testing an ogee spillway is examined. This section also presents the underlying theory of CFD modeling and the advantages of using CFD in flow modeling.

2.2 GENERAL CONCEPTS ON WEIRS AND SPILLWAYS

2.2.1 Weirs

A weir is referred to as an artificial barrier in a watercourse exhibiting an edge used to regulate or to measure flow rate and water depths (Ghare*et al*., 2008). They are common in different hydraulic structures such as storm water systems and stream engineering (Brown *et al*., 2012). The edge or surface over which water flows is called the crest as defined in Figure 2-1.

Based on the notch shape, Brater et al. (1996) classified weirs into four types: rectangular weirs, triangular or V-notch weirs, trapezoidal weirs and parabolic weirs.

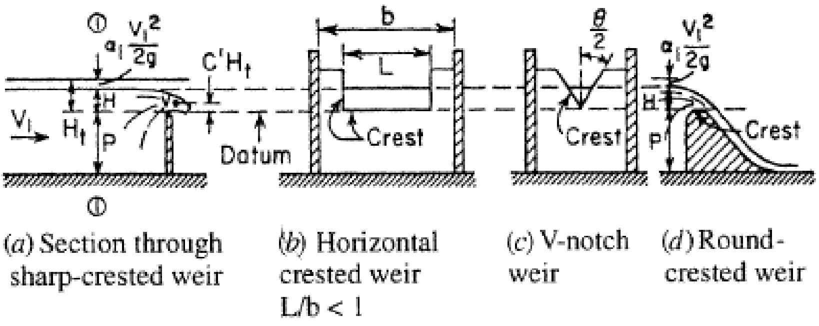

Figure 2-1: Weir definition sketch (Brater et al., 1996)

For particular situations, weir selection is made with respect to the range of discharges to be measured, the accuracy desired and the calibration needed (Brater*et al.*, 1996).

2.2.2 Spillways

Spillways are control appurtenances which are constructed at the dams and/ or impounding reservoirs to provide the controlled release of flows exceeding the dam's full supply level (FSL) to the downstream side. Excess water is conveyed downstream while an appropriate structure dissipates the high kinetic energy of the flow that may lead to serious scour of the channel bed. If the scour is not properly controlled at the spillway toe, it may extend backward and endanger the entire spillway as well as the dam. In some cases, the energy dissipaters are included along its slope for reducing the amount of space required to efficiently discharge the flow without jeopardising the dam (USBR, 1973).

Of all dam safety measures, the spillway capacity is of paramount importance for different kinds of dams, especially earth fill and rock fill dams, which may probably fail once, overtopped.

Concrete spillways must be built with adequate structural protection from frost damage and erosion/abrasion from water and water-borne debris. Ruskin Dam spillway in Canada, as a prototype case, was resurfaced to repair concrete damages (Lihe, *et al.* , 2011). Such remedies must be avoided as they are very expensive. Each dam, thus, should be equipped with an adequate device to prevent the overtopping.

Another tragic disaster occurred on 22, February, 1994 at Merriespruit Dam where 500,000 m3 of mud flows moved for 2 km and killed seventeen people (Strydom&Willams, 1999). Therefore, all dams should be constructed with high safety device to prevent such risks. A part from the functions discussed here above, Takasu& Yamaguchi (1988) provided seven more functions of a safe spillway:

1. Maintaining normal river water functions
2. Discharging water for utilisation
3. Maintaining initial water level in the flood-control operation
4. Controlling floods
5. Controlling additional floods
6. Releasing surplus water

2.2.3 Spillway classification

Spillways can be classified based on various factors: Function (service spillway and auxiliary spillways); regulatory or control structure (Gated spillway, ungated spillways and orifice of sluice spillway) with the latter being the most pertinent feature (Khatsuria, 2005). According to Şentrürk (1994), while selecting a spillway type, the following factors should be considered:

- Type of the dam to be constructed,
- Dam storage and outlet capacity,
- Topography and geological conditions of the spillway site,
- Hydrological and hydraulic factors, and
- Cost and risks involved.

Factors involved in the selection of each spillway type are briefly discussed in the Table 2-1.

Table 2-1: Factors governing the spillway type selection

Spillway type	Selection factors
Ogee Spillway	This type is suitable for a variety of situations because of its high efficiency to control flows (Savage *et al.*, 2001), such as in most control crests of other spillways and in high dams.
Chute Spillway	Provided to dams where the site permits a limited channel with sufficient depth of excavation for the foundation.
Side channel Spillway	Chosen as it is suitable for dams where the sides are steep and have a considerable height above the dam such as in canyons.
Shaft/Tunnel Spillway	Used for dams that are located in narrow canyons and in dam sites where the space downstream of the dam is not enough for other spillway types.
Siphon Spillway	Selected when the dam must operate automatically without mechanical tools, with a small discharge to be conveyed.
Free over-fall Spillway	This type is specifically appropriate for arch dams.
Labyrinth Spillway	Adopted as an economical measure for passing large floods as they provide an increased unit discharge when compared to conventional weirs for a given head (Darvas, 1971). They are used, therefore, in dams where an extra crest length is needed.
Stepped Spillway	Suitable for dissipating the energy of the overflow and to reduce the size of the energy dissipater needed at downstream of the spillway.

2.2.4 Ogee or overflow spillways

Several types of spillways currently exist, as discussed in the preceding section, but only the uncontrolled ogee-shaped spillways will be reviewed in this thesis, as others are outside of the scope of this thesis.

An ogee spillway, as shown in Figure 2-2, exhibits a control weir that is in form of ogee or S-shaped profile. According to IS-6934, the crest of an ogee spillway is basically a sharp-crested weir with an empty space below the lower nappe replaced with concrete. In fact, the shapes are based on a simple parabola designed conditionally to fit the trajectory of the lower nappe. The profile below the upper curve is prolonged tangentially, along the slope, to support the sheet on the face of the overflow and flow up to the apron of a stilling basin or into the spillway discharge channel.

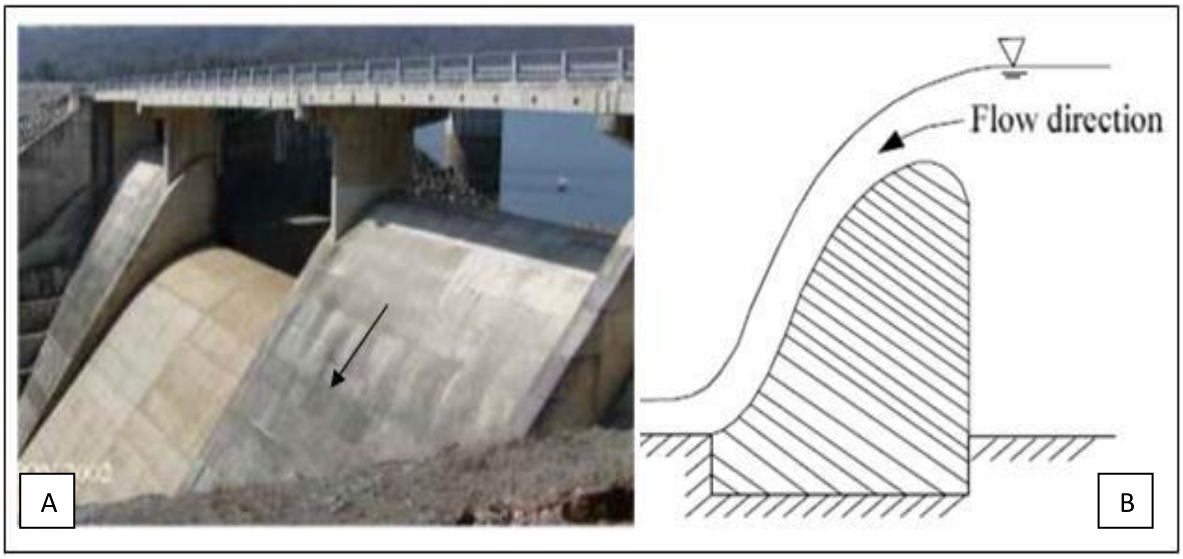

Figure 2-2: Ogee spillway type. A: Ogee spillway front view and B: Sectional view. Source: (Chanson, 2002)

2.2.4.1 Specific functions of an ogee spillway

An Ogee spillway is the most commonly used type especially in high dams. Its ability to pass flows efficiently and safely, when properly designed and with relatively good flow measuring capabilities, has enabled engineers to use it in a wide variety of situations (Savage *et al.* 2001).

Based on the regulatory or control structure, three different designs for spillway control are distinguished and can be classified as: uncontrolled devices which do not use a hydraulic gate in its operation, movable crest devices and regulating devices. Uncontrolled crests are generally applied on small spillways and weirs when the release of water is only required in case the reservoir head exceeds the design level. One of the advantages of this design is that the constant supervision of an operator, maintenance and repair costs are not needed.

Movable crest and regulating devices are often employed when there is a sufficiently long uncontrolled crest or when the spillway crest is located under the normal operating level of the reservoir.

2.2.4.2 Discharge computations

2.2.4.2.1 General

The most commonly used equation for computing the discharge of ogee spillways was developed from early experiments by James B. Francis (Horton, 1907). This discharge relationship, which is of the same form as other weirs, is often referred to as the "Weir Equation", as shown in Equation (2-1):

$$Q = CLH_e^{1.5} \qquad (2\text{-}1)$$

Where

Q= Discharge (m^3/s)

L= Lateral crest length or width (m)

C= Discharge coefficient ($m^{0.5}/s$)

He= The total energy head upstream from the crest which is defined as follows:

$$H=h+H_v$$

Also,

$$H_v = \frac{v^2}{2g} \qquad (2\text{-}2)$$

The following terms from equation (2-2) mean:

h = Measured water level above the weir crest (m)

H_v = Velocity head (m/s)

It is also important to note that, the total head on the crest, H_e, does not count for approach channel friction losses or other losses.

2.2.4.2.2 Determination of discharge coefficients

An ogee spillway is characterized by a relatively high value of discharge coefficient because of its shape. However, this coefficient is not constant. It is influenced by a number of factors including the depth of the approach, relation of the actual crest shape to the ideal nappe shape, the upstream face slope downstream apron interface and downstream submergence.

2.2.4.3 Pressure distribution

The design head is generally chosen to give the maximum hydraulic efficiency, in keeping with the operational requirements, structural stability and economy. At the design head "H_o", ogee crests operate with the design discharge coefficient "C_o", exhibiting the atmospheric pressures.

For heads "H_e" less than the design head, the coefficients of discharge "C" are less than the design coefficient of discharge "C_o" and positive pressures develop on the crest. For heads greater than the design head, the coefficient of discharge "C" become greater than the design coefficient of discharge with negative pressure on the crest, thereby, increasing the discharge capacity.

Table 2-2: Pressures on an ogee crest for design and non-design flow conditions (Chanson, 2004)

Upstream head (1)	Pressure on crest (2)	Discharge coefficient (3)
$H_1 = H_{des}$	Quasi-atmospheric	$C = C_{des}$
$H_1 > H_{des}$	Less than atmospheric	$C > C_{des}$
$H_1 < H_{des}$	Larger than atmospheric	$C < C_{des}$
$H_1 \ll H_{des}$	Larger than atmospheric	$C \approx 1.70\ m^{0.5}/s$

Design engineers should not allow crest pressures to become too negative, as accentuated crest negative pressures cause cavitation damage, spillway destabilization and possible failure of the entire structure. Chanson (2004) shows, in Table 2-2, that crest pressures decrease linearly with the increase of the upstream head on the crest thereby increasing the discharge coefficients.

2.2.4.4 Aeration effects

Naturally, air flows are commonly encountered at the water flow surface (Chanson, 2004). The falling nappe is considered aerated and it is subjected to atmospheric pressures. The aeration observed has led to the suggestion that the point at which aeration commences coincides with the point at which the boundary layer depth meets the free surface (Henderson, 1996; Keller, 1972; Cain & Wood, 1981). Figure 2-3 depicts boundary layer development in the flow over the ogee spillway.

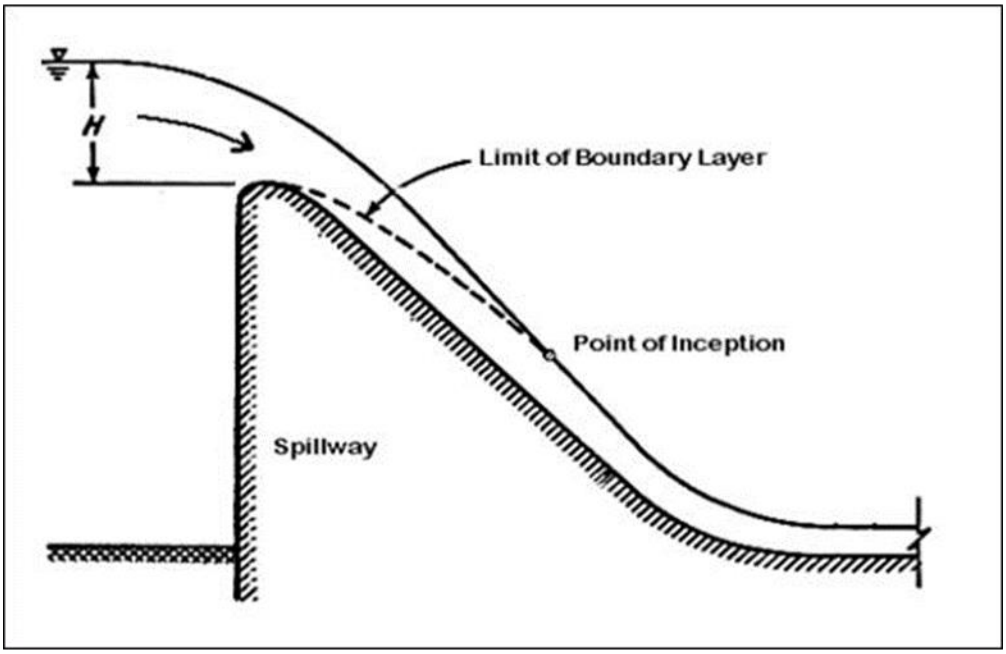

Figure 2-3: Boundary layer developed on ogee spillway flow (Bhajantri et al., 2006)

The insufficient aeration of the nappe will cause the reduction in pressure thereby leading to the abrupt change of the nappe shape for which the spillway crest is designed. These pressures can theoretically be as low as the vapour pressure of water, which causes structural damage due to destructive cavitation and vibration, and are therefore to be avoided from an operational and structural point of view.

Şentrürk (1994) discusses three problems solved for an aerated spillway flow:

- Increase in flow depth due to air entrainment,
- Aeration of the lower nappe avoids pulsating flow,
- The downstream channel is free of cavitation if the lower nappe is aerated.

2.3 STATUS OF RESEARCH WORK IN THIS AREA

With the development of the computational power in the hydraulic engineering, numerical methods have been increasingly used in investigating the flow over spillways. After numerical models were validated with physical model tests and started to be used as a design tool. Most of the literature on Flow-3D modeling discusses how the program uses a finite difference solution scheme and the Volume of Fluid (VOF) method, developed by Hirt and Nichols (1981), which allows the model to include only the water portion of the flow. Use of this method results in significant reductions in simulation times as the motion in the surrounding air is neglected and this type of programming allows a sharp interface between water and air to be created without the use of very fine meshes required by other CFD programs. Flow-3D also uses a Fractional Area/Volume Obstacle Representation (FAVOR) method (Hirt and Sicilian, 1985)[24] to define obstacles. This method allows Flow-3D to use fully structured computational grids that are much easier to generate than the deformed grids used by most other CFD programs.

Olsen and Kjellesvig (1998) modeled numerical water flow over a spillway in two and three dimensions for various geometries in order to estimate the spillway capacity. RANS equations were solved with k - ε turbulence model on a structured non-orthogonal grid. The results were compared with experimental study where there was a good agreement on discharge coefficient. Moreover, the pressure distribution on spillway was also acceptably closer to the physical model results.

Yakun, et al. (1998) presented a study about numerical modeling of spillway flow with a free drop and initially unknown discharge. Flows over different spillway profiles were studied. The discharge, profiles of the free drop and the pressure distributions on the walls were computed from the numerical model. The numerical results were in good agreement with the measured ones.

Unami, et al. (1999) solved a 2-D numerical spillway model to verify the applicability of the model to practical design. An unstructured triangular mesh system was used. Both finite element and finite volume methods were used for resolving of 2-D free surface flow equations. In addition to the flow equations, air entrainment model was also added to the system. The study proved that the model was valid as a primary analysis tool for hydraulic design of spillways.

Savage and Johnson (2001) completed their study using Flow 3D in order to compare the flow parameters over a standard ogee-crested spillway between physical model test results, existing spillway literature design guidelines created by USACE and USBR and numerical simulation results. The numerical simulations were solved with Reynolds Averaged Navier-Stokes equations using finite volume method. This study showed that numerical tools were sufficiently advanced to calculate discharge and pressure on the spillway. Although physical model studies were still considered the basis from which other methods were compared, the numerical simulations had an improved accuracy over the design nomographs for obtaining the discharge capacities and pressures.

Teklemariam, et al. (2002) tried to apply Flow 3D in hydraulic engineering applications. As a result of case studies, CFD analysis provided a considerable design support for advanced hydraulic engineering projects.

Ho, Boyes, Donohoo, and Cooper (2003) made comparisons of crest pressures and discharges over a standard ogee spillway from 2d and 3d simulations in Flow-3D to USACE data and empirical discharge equations. Their study found that simulated 2d and 3d crest pressures followed the general trend of data published by USACE, however, in both cases the CFD results predicted slightly larger negative pressures. It was also found that the 2d simulation over-predicted the flow-rates by about 10 to 20 percent depending on the headwater elevation. The 3d CFD simulation results were much better, within 5 percent of the empirical calculations for the three headwater levels considered. This document also goes on to discuss the successful application of Flow-3D software for analysis of spillway hydraulics on three real structures in Australia.

Kim and Park (2005) analyzed the flow structure over ogee spillway in consideration of scale and roughness effects. The commercially available CFD package, Flow 3D was used in this study. RANS equations were solved and RNG model was used as turbulence closure. It was obtained that numerical errors due to the roughness effects were insignificant and if the length scale ratio is less than 100 or 200, the scale effects of the model were in an acceptable error range.

Ho, Cooper, Riddette, and Donohoo (2006) also prepared a document reviewing the application of Flow-3D to eight spillway upgrade projects in Australia. Their report discusses that the general result for numerical model flow-rates for head levels equal to or greater than the design level yields a 5 percent overestimation when compared to physical models. They also state that although some pressure fluctuations may result from limitations in grid resolution, the general trend of CFD pressures is in reasonable agreement physical model data. They conclude that CFD is a viable technology for use in the design and rehabilitation of spillways. The report also states that, “Further benchmark tests against established data or design guides (USACE-WES, 1952) will provide.

Johnson and Savage (2006) showed the influence of the tailwater on the spillway. In 2001, Savage and Johnson investigated the flow parameters and compared the results of physical test studies, existing spillway literature design guidelines created by USACE and USBR and numerical model studies. This study was used in order to show that the pressure along the spillway under the submergence effect could be accurately predicted using numerical models. The comparison indicated that numerical modeling can estimate the flow rate and pressure distribution on the spillway accurately. Moreover, it was stated that numerical models can provide more details about velocity and pressure distribution than the physical models.

Bhajantri, et al. (2006) described the formulation and development of a two dimensional free surface flow numerical model for flow over a spillway. The objective of study was to investigate the hydraulic characteristics of the flow over the spillway. Pressures, velocities and other non-dimensional hydraulic parameters such as the Froude number and the cavitation index were analyzed. Simulations were completed using inviscid weakly compressible flow equations. Numerical model results showed reasonable agreement with the results measured from the physical model tests.

Jacobsen and Olsen (2010) investigated the capacity of a complex spillway and calculated the stage-discharge curve with a three dimensional numerical model solving RANS equations using finite difference method with the standard $k - \varepsilon$ turbulence model. A fixed orthogonal grid was used in the computations. The results were compared with a physical model study. The deviation between the computed and measured values of the rating curve was under 2% for most of the discharges whereas at some points it increased to a maximum value of 10% where the flow was most complex. As a result of the study, it is stated that the use of numerical modeling to compute spillway capacity caused considerable savings in both cost and time for hydraulic engineering design.

Zhenwei, et al. (2012) studied flow over a spillway using numerical model that utilizes VOF method with multidimensional two phase flow. The water flow in the whole spillway was simulated defining unstructured hexahedral grid using a $k - \varepsilon$ turbulence model. Numerical modeling results showed good agreement with experimental results in flow parameters such as free surface elevation, pressure and flow velocity. It was stated that the flow characteristics obtained from numerical analysis can provide detailed data for the design.

Daneshkhah and Vosoughifar (2012) worked about the impacts of different turbulence models on flow parameters for ogee spillways using Fluent software. The results calculated by the numerical models were compared with the experimental results. RNG $k - \varepsilon$ turbulence model showed more accurate results over ogee spillway.

Azmoudeh and Kamanbedast (2013) calculated the flow parameters using Flow 3D in order to determine appropriate location of the aeration system of the chute for preventing the cavitation risk. In this study, Gotvand dam's spillway and chute was worked as a case study. Cavitation numbers were computed by the numerical model and measured from the physical model at different locations of the aerator systems. A good agreement between the numerical and physical modes was obtained. It was found that total average difference between the results was 0.03% which was in acceptable range for a CFD analysis.

Singh, et al. (2013) presented a case study of iterative simulations of the flow over the spillway of Ratle Hydro Electric Project, for various modifications which were intended to obtain a hydraulic design, compatible with the topography on the downstream side. Physical model tests with a scale of 1:55 were also completed in the scope of this study. RANS equations were solved and RNG turbulence model was used as the closure model.

As a result of this study, although the use of a coarse mesh caused errors in the discharge values, the trajectory of water was simulated quite accurately by Flow 3D.

Additional literature discussing successful applications of Flow-3D and other CFD software is available. This summary provides a general overview of the abilities and future promise of CFD software in the hydro-electric industry. It has been found that further studies investigating the applicability of CFD modeling to additional spillway geometries and configurations is necessary. Supplementary correlations between physical and numerical model results would enhance the confidence in the software and possibly make it a viable alternative to physical modeling in some applications. It should also be noted that in some situations, the most optimal design approach may be to use a combination of both CFD and physical modeling as discussed in above readings. This research project will attempt to provide additional confidence in CFD through comparisons between results from Flow-3D.

2.4 COMPUTATIONAL FLUID DYNAMICS MODELING

2.4.1 Definition

Computational Fluid Dynamics (CFD) is a computer-based tool that is used to represent and analyze systems that involve fluid flow, heat transfer and chemical reactions, by using numerical methods that are based on partial differential equations describing these systems (Versteeg&Malalasekera, 2007).

2.4.2 Background and development of CFD modeling

Over 100 years, hydraulic engineering practice has been relying on physical modelling for the design of most hydraulic structures. Because of the rigid and expensive nature of scale models, more alternative methods have been developed for the sake of accuracy and time optimization. Modern Computational Fluid Dynamics (CFD) was born in the 1950’s when digital computers were introduced (Chung, 2002). There has been a rush in development and application to all aspects of fluid dynamics from the late 1960s where the aerospace industry profited from its use, especially for the design and manufacturing processes (Parviz & John, 1997).

In hydraulic engineering, computational modelling of spillway flows is increasingly being used in the industry. However, a validation from a physical model is still required to ensure that the physical processes are accurate. Several approaches have been developed, including modelling in one, two or three dimensions which use various equations and discretization techniques. One-dimensional models are generally applied to verify the river stage and the water surface profile upstream and along the length of the spillway (Song & Zhou 1999). On the other hand, a one-dimensional model has presented some limitations as it is not adapted to variable-geometric structures such as junctions, steep slope, curved surfaces or to any other change of size or shape of the channel (Franz & Melching, 1997).

2.4.3 Governing equations

The fundamental principles for all numerical models remain similar where the problems are stated, physically, by a set of partial differential equations. In the same way, CFD techniques are governed by a number of equations which must be solved in each control volume. Depending on the properties of fluid flow to simulate, the physical phenomena are represented by mathematical statements that are referred to as governing equations of fluid flow and heat transfer (Versteeg & Malalasekera, 2007). These equations include mass conservation or continuity, momentum and energy equations also known as the Navier-Stokes equations. Based on the theory documented by Wendet (2009), mass conservation and momentum equations can be described as follows:

2.4.3.1 Mass conservation (continuity) equation

The mass conservation equation or continuity equation states that the mass of a closed system of substances will remain constant, regardless of the processes acting inside the system. Fluid flow has a fixed mass even if its shape and volume may change as it moves. The illustration is made in Equation (2-3) and Figure 2-4:

$$dm = \rho dv \qquad (2\text{-}3)$$

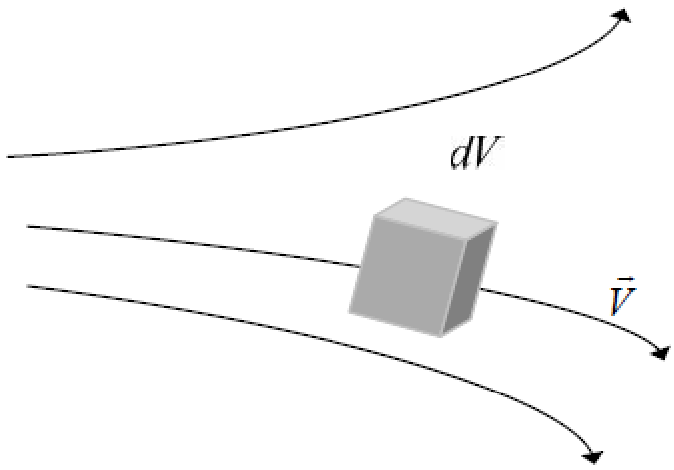

Figure 2-4: Definition sketch for moving fluid (Kositgittiwong, 2012)

During the time interval Dt, the principle of conservation of mass states that the rate of change of mass of a fluid element is zero as the mass flow entering is equivalent to the mass flow leaving as shown in equation (2-4).

$$\frac{D(dm)}{Dt} = 0 \qquad (2\text{-}4)$$

Further we may write Equation (2-6) as follows:

$$\frac{D\rho}{Dt} + \rho\left[\frac{1}{dV}\frac{D(dV)}{Dt}\right] = 0 \qquad (2\text{-}6)$$

If the product rule is applied for each of the spatial differentiations and the definition of substantial derivative is used, Equation (2-6) is written in a more compact notation and becomes Equation (2-7).

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\,\vec{V}) = 0 \qquad (2\text{-}7)$$

Where,

dm= Change of mass in the system (kg)

ρ = Density (kg/m^3)

V = Velocity (m/s)

Dt = Time interval (s)

∇= Partial derivative of a quantity with respect to all directions

2.4.3.2 Momentum conservation equation

The momentum equation is a statement of Newton's Second Law and relates the sum of all the forces acting on a particle of fluid (Chadwick, *et. al.*, 2004; Versteeg & Malalasekera, 2007). The moving fluid element model is sketched with more details in Figure 2-5.

The general equation for conservation of momentum can be written as follows:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\left(\frac{\partial P}{\partial x}\right) + \nabla.(\mu\nabla\mu) + S_m \qquad (2\text{-}8)$$

where

P= Static pressure (Pa)

μ = Kinetic viscosity (m^2/s)

S_m= Source term (Constant)

ρ= Density (kg/m^3)

v= Overall velocity vector (m/s)

The left hand side of Equation (2-8) contains terms as defined for the mass conservation equation and its right hand side contains the pressure source term and the diffusion source term respectively. When the VOF-method is used to treat multiphase-phase models, the mass conservation equation becomes slightly modified and the momentum conservation equation will remain the same because of the dependence on of the variables ρ and μ on the volume fraction.

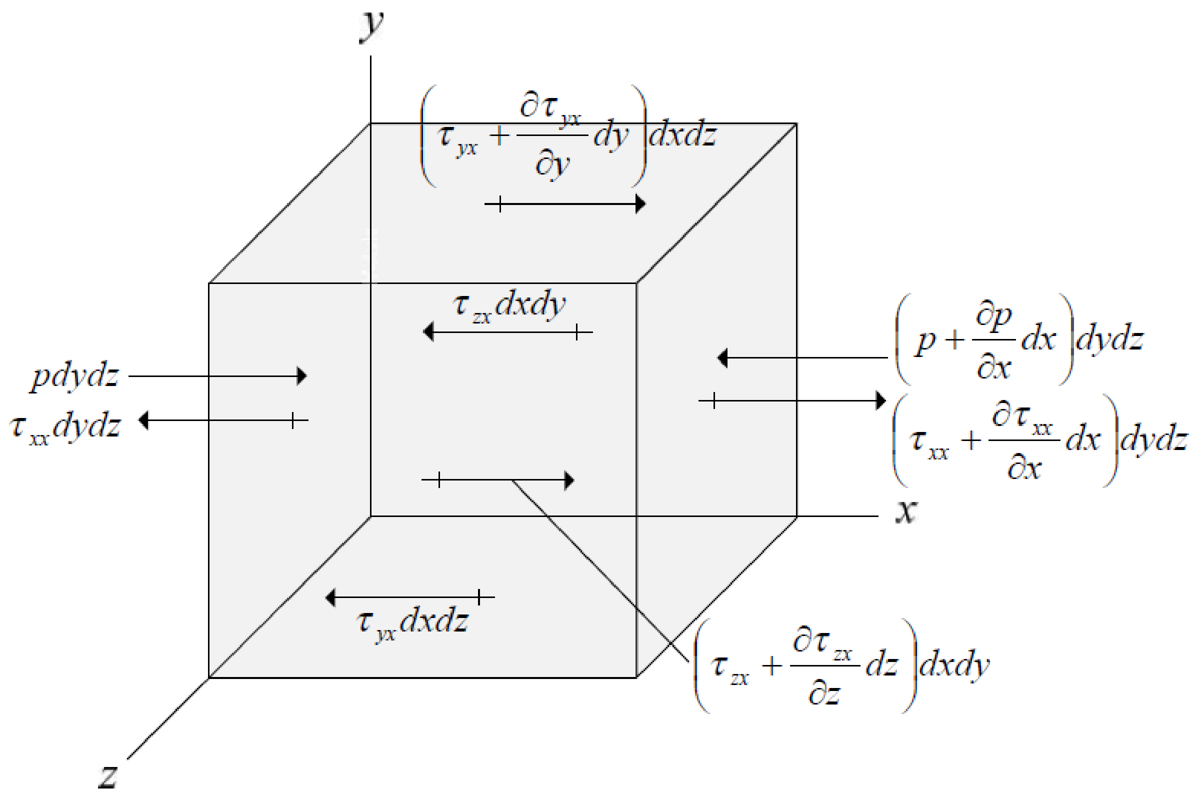

Figure 2-5: Moving particle element model for the x component (Kositgittiwong, 2012)

2.4.4 Multiphase modelling

Multiphase modelling is a technique which simulates flow in the simultaneous presence of different phases. All the three phases (gas, liquid and solid) are identifiable with a distinct particular inertial response to an interaction with the flow and the potential field. There are four main categories of multiphase flows; gas-liquid, gas-solid, liquid-solid and three-phase flows (Murrone & Villedieu, 2011). Such complex systems have been made possible with the availability of two broad approaches for the numerical calculation, namely Euler-Euler (Volume of Fluid model or VOF model) and Euler-Lagrange (Discrete Phase Model) approaches.

2.4.4.1 Comparison of DPM and VOF

Comparing DPM to VOF modelling, DPM modelling application proves to be more challenging than VOF. VOF is more useful and computationally affordable. This inconvenience of DPM is due to the formulation of position vectors as the solids respond to shear stresses. In VOF formulations, physical laws do not possess the position vectors and the velocity appears as the major variable thereby revealing all fluid flow patterns necessary. This is due to the fact that the turbulent fluids which are subjected to shear stress deform continuously when the stress is applied (Panton, 1984). As a common drawback, both models cannot be used with density-based solvers; only the pressure based solver is allowed.

2.4.5 Aeration effects in multiphase modelling

Chatila & Tabbara (2004) conducted a study with a CFD software package called ADINA-F to compare water surface profiles over an ogee spillway geometrically similar to the physical model. The spillway profile was free of piers and water free surface was measured at the centre line to avoid an influence from the wall boundaries. The results achieved show that, even if qualitative solutions are reasonably consistent with general flow patterns, an inconsistency was found. In all three simulated discharges, ADINA-F, predicted water surface levels much lower than the experimental water levels along the spillway length. These inconsistencies were found to be caused by air entrainment effects which were not accounted for the CFD model used. They used different CFD software in their research, called Flow-3D for modelling the Keeyask spillway. The results produced by Flow-3D included the effects of air entrainment because of the use of the volume of fluid method (VOF).

2.4.6 Advantages of using CFD

CFD has grown from a mathematical field to become a crosscutting tool in nearly every branch of fluid dynamics. In the engineering field, CFD is used to analyze new designs before they are implemented. A number of clear advantages can be summarized as follows:

- CFD can give an insight into flow patterns, weight losses, mass and heat transfer, flow separation, etc. Thence, all of these parameters may help implementers with a much better and thorough understanding of what is happening on the field.

- CFD reveals complex features that could not be achieved by physical modelling such as high temperature.

- CFD can be used to test dangerous experiments with cheaper means and without risks such as accident scenarios or safety studies

2.5 SUMMARY

The literature review shows that the design process of an ogee spillway is reasonably well understood. These theories provide the design parameters such as the design head (Ho) and the discharge coefficients (Co) for which the ogee profile can be designed and the discharges computed. However, this is not enough to ensure a better performance and stability of a spillway; it is necessary to determine and analyze all hydrodynamic pressures generated from water flows (Johnson & Savage, 2006).

These hydrodynamic pressures are complex and they are determined at both accelerating and decelerating regions to predict the pressure distribution on a spillway. In addition, aeration of the nappe is the key factor which must be considered in spillway design since it has a great impact on the spillway performance Şentrürk (1994).

The advances in numerical methods as well as the development of computing power are attempting to improve the quantification of hydrodynamic pressures and aeration in the spillway flow nappe. These factors are very important for improving the spillway performance and to limit negative pressures which could damage the concrete surface of a spillway.

Amongst numerical methods, Computational Fluid Dynamics (CFD) is found to be increasingly used in the spillway flow field. A clear advantage of these methods lies in their ability to simulate the effects of turbulence and multiphase flow since most of the flows in nature is turbulent and multiphase.

However, the CFD method is not used as a standalone method in the spillway design, the physical modeling is always required for a validation. This comparative study is used so as to allow engineers to verify the degree of accuracy of CFD results.

CHAPTER 3

THE NUMERICAL MODELING

3.1 INTRODUCTION

This chapter presents, firstly, the numerical procedure that was used to develop a CFD model and the logic behind each stage. Secondly, the formulations of physical phenomena are identified and discussed in this chapter. These formulations were based on turbulence modelling as the flow over an ogee spillway, like most of the other flows, is turbulent in nature. Finally, the results obtained from three-dimensional models for pressures are presented with a comparison between the results produced by steady state tests.

3.2 SET-UP PHYSICS

The physical modeling tests for this study were carried out in the Hydraulic Laboratory of Stellenbosch University, located in the Western Cape Province, Republic of South Africa. The physical modeling consisted of one ogee spillway installed, in a glass flume of 1.25 m high, 0.60 m wide and 22 m in length. The maximum flow rate that could be obtained in the laboratory from a constant head tank was 130 l/s. The conceptual laboratory set-up is shown in Figure 3-1.

In this research, a physical model was tested. The primary spillway model is 1:30 scale of a large sized-dam. This scenario was performed with the overall objective of getting more details at the crest of an ogee spillway. However, the experimental results were not scaled up to the prototype as the objective of this study was to compare the results produced by CFD models with physical model results.

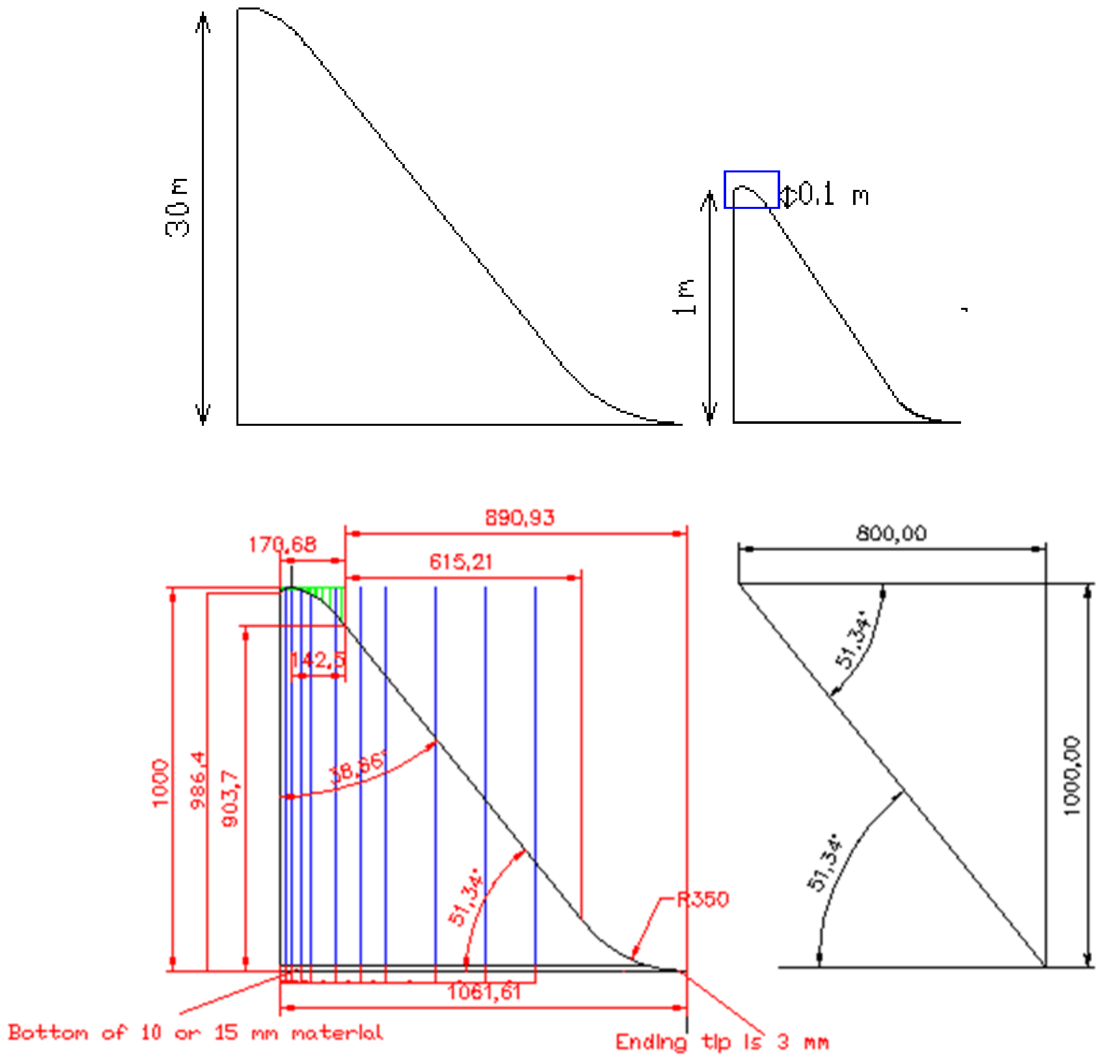

Figure 3-1: Physical model geometry of 1:30 scale model (Placide Nshuti Kanyabujinja et al)[38]

The geometric dimensions for the case is summarized in Table 3-1. It is noted that since the flume of 0.6 m wide was used in the laboratory, the prototype width modeled by the 1:30 model is 30*0.6=18 m.

Table 3-1: Model dimensions(Placide Nshuti Kanyabujinja et al)[38]

Model Type	Spillway Approach Depth	Crest Depth	Radii	
1:30 scaled	1.00	0.60	0.05(R_1)	0.02(R_2)

Table 3-2 presents the design heads and discharge coefficients for the spillway model.

Table 3-2: Design Surcharge, coefficient of discharge and design discharge for the model(Placide Nshuti Kanyabujinja et al)[38]

Model Type	Design Surcharge(H_o) in m	Coefficient of Discharge(C_o)	Design Discharge(m^3/s)
1:30 model	0.10	2.17	0.042

3.3 NUMERICAL PROCEDURE

The numerical procedure is divided into three main dependent stages, namely: pre-processing, solving and post-processing. Each stage must be completed as listed, before starting the other. In the subsequent sections, these three stages are thoroughly explained.

3.3.1 Pre-processing

The first stage of numerical modelling was to transform a real hydraulic structure into a computable model. This involved the geometry building and boundary definition for the numerical domain. Over 50% of the time spent on CFD modelling was devoted to defining the solution domain and grid sizing. To achieve this, Ansys Fluent software was used. Ansys Fluent is a CFD package created by Ansys Inc., which is based on finite volume method. It uses both the Volume of Fluid (VOF) and Fractional Area-Volume Obstacle Representation (FAVOR) methods to simulate the free surface and the location of obstacles respectively (Fluent, 2009)[17].

3.3.1.1 Geometry

Using the Ansys design modeller, CFD model geometry was built with the same dimensions as the case of available physical model. The prototype dimensions which this physical model represent were not used to build their geometry for application in the numerical modelling. In the geometry builder, both surface and body were frozen to allow a fluid flow to pass through the modelling domain.

Figure 3-2 and Figure 3-3 are two and three-dimensional geometries developed for physical model.

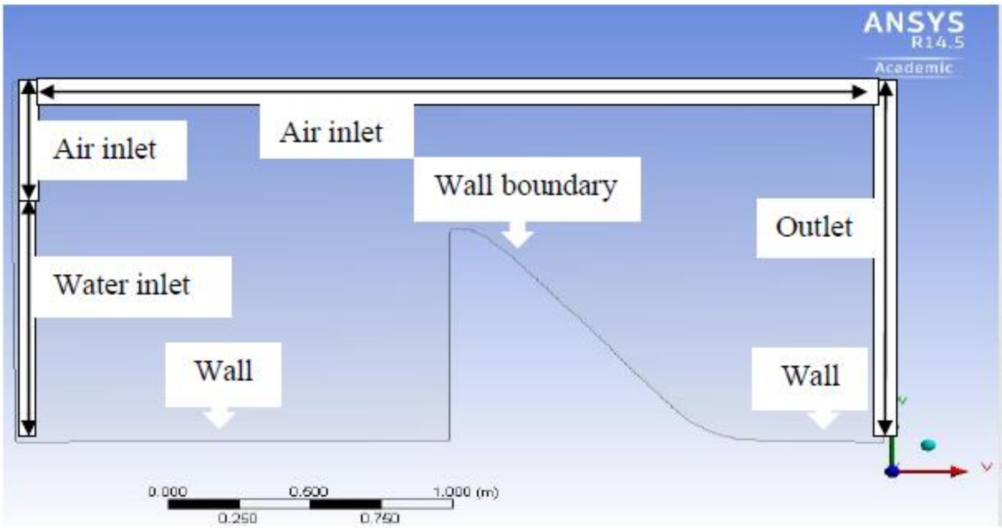

Figure 3-2: Model geometry constructed in 2-D with boundary labels

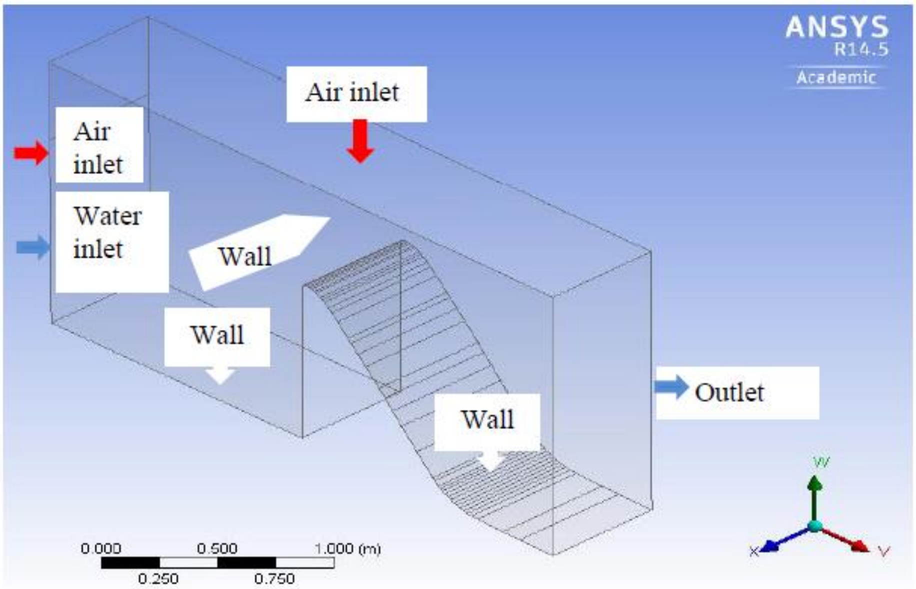

Figure 3-3: Model geometry constructed in 3-D with boundary labels

3.3.1.2 Meshing

The meshing process is a very important stage in CFD modelling which requires much attention. In order to analyze the fluid flow, the domain had to be split into smaller cells within which the governing equations would be solved. Basically, the accuracy of a CFD solution depends upon the number of cells in the mesh. The finer the meshing, the better the solution accuracy. A number of smaller and non-overlapping computational cells were performed in order to obtain the most suitable mesh for better solution accuracy. The areas of high solution interest have been meshed up with the finer mesh while the coarser mesh was implemented in areas with less solution interest. The main features of interest in the numerical domain are:

- The spillway crest,
- The downstream part where the turbulent phenomenon occurs, and
- The region of interface between the two phases (water and air).

Along the spillway crest, the surface needed to be adequately resolved with finer grid resolution in order to monitor the pressure accurately. The regions where the interaction between water and air was supposed to happen were meshed with a refined mesh to allow a clear interface.

In two-dimensional meshing, there are many options that can be applied such as triangular and quadrilateral meshing. In this study the meshing, parameters were defined as follows:

- A structured grid, consisting of triangular mesh cells was used because of its superiority in producing more accurate results.
- At the spillway crest and in the turbulent area downstream, a refinement was applied.
- In order to allow a stable flow and accurate solution, a relevance center and a medium smoothing were also included in the setup parameter.

Figure 3-4 depicts the schematic representation of two-dimensional and 3-5 shows the three-dimensional model meshing for this case.

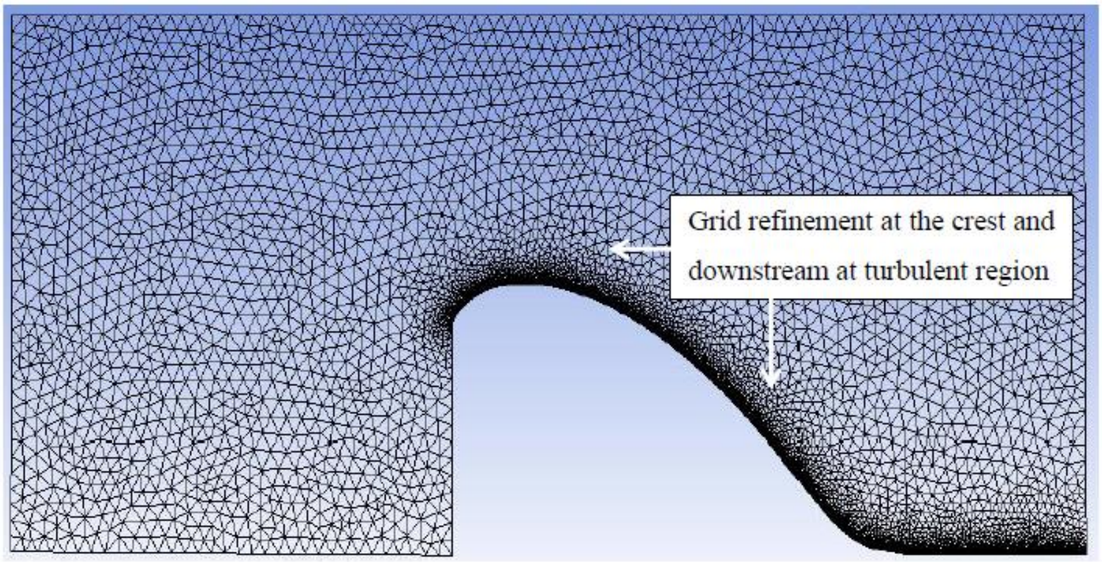

Figure 3-4: Representation of the geometry meshing of 2D model

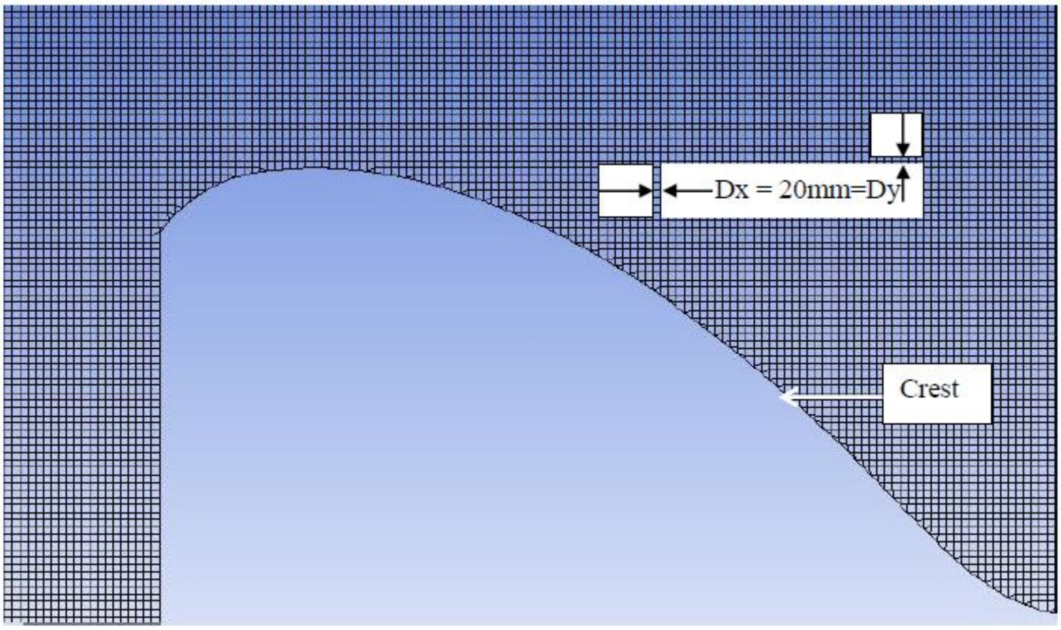

Figure 3-5: Representation of the geometry meshing of 3D model

3.3.2 Solver

3.3.2.1 Model Set-up

After creating the geometry of the model and meshing it up in the Ansys-Fluent package, the mesh was imported into fluent solver. Various parameters within the solver had to be set before the simulations could be started.

a. Turbulence Model Selection

Since most of the fluid flows are turbulent in reality, Computational Fluid Dynamics Modelling (CFD) use turbulent model to simulate fluid flows. Among the linear turbulence models, the widely used, two-equation model is based on the turbulent kinetic energy equation k and the turbulent eddy dissipation ε or the turbulent frequency ω. In this study, the realisable $k{-}\varepsilon$ model was selected as it contains a new formulation for the turbulent viscosity as well as new transport equation for the kinetic energy dissipation rate (Shigh et al., 1995)[39].

b. Multiphase model selection

The volume of fluid model (VOF) was chosen to simulate the multiphase flow. Two Eulerian phases: air and water, were defined as primary and secondary phase respectively. Another alternative for setting Eulerian phases was tried, that was, to define water as primary phase and air as secondary phase but this resulted into flow instability.

c. Boundary conditions

The precision of boundary conditions on the solution domain plays a capital role for the accuracy of the results. The boundary conditions to be specified consist of flow inlet and outlet boundaries, which have to be defined with the flow properties such as turbulence parameters, velocity and pressure.

Walls and internal faces which have a direct interaction with the flow have been defined as well. All of these boundary conditions are discussed below:

i. **Inlet boundary**: The inlet section is at the upstream of the spillway and consists of the inlet of water at the bottom and the inlet of air at the top. For water inlet, a “velocity-inlet” boundary condition was selected as it was the best option that produced a stable flow in the solution domain. For discharge tested in physical modelling, the input velocity was calculated and set uniformly at the inlet. The air boundaries were defined as an inlet pressure with the atmospheric pressure conditions.

ii. **Outlet boundary**: The outlet of the domain at the downstream part was specified with an outlet pressure so that water and air can flow out freely. To ensure atmospheric conditions, the air phase was allowed to flow back into the model.

Table 3.3 Below summarises other parameters that were defined for the running of the simulations.

Model/ Solver parameter	Type	Value
$\boldsymbol{k-\varepsilon}$ Model (2eqn)	C2- Epsilon	1.9
	TKE Prandtl number	1
	TDR Prandtl number	1.2
Multiphase Model	Volume Fraction Cutoff	1e-06
Phase interaction	Continnum surface stress constant	0.0728
Cell Zone conditions	Operating pressure (Pa)	101325
	Gravity enabled (m/s^2)	9.81
Boundary conditions	Turbulence intensity (%)	5
	Roughness constant	0.5
Phase densities	Density of water (kg/m3)	998.2
	Density of Air (kg/m^3)	1.225
Reference values	Enthalpy (j/kg)	0
	Temperature (k)	288.16
	Ratio of specific heats	1.4
Scheme (Piso)	Skewness-Neighbor coupling	1
	Neighbor Correction	1
Under relaxation factors	Pressure	0.3
	Density	1
	Body Forces	1

iii. Walls: Wall boundary conditions (walls and channel bed) were specified to simplify the operational conditions with no-slip conditions and to be in stationary conditions at all times.

d. Initialisation

For each test, it is necessary to initialise the model. This is done by setting initially all pressures to the atmospheric conditions. For these conditions, the solution domain was entirely filled with air. The flow computations were set to start from the inlet boundaries to allow stable conditions to be far from the spillway crest. For each model the tests were run under steady state conditions until it attained a convergence.

After a successful convergence, the time length of five minutes was used to match the time of experimental tests for unsteady tests. The simulation results obtained are documented in post-processing section.

3.3.2.2 Tests Conducted

Numerical simulations were accomplished in several scenarios; each model had to be tested in 2D for a flow rate of 130 l/s. The 3D geometry was the full dimensions of the physical model while 2D modelled the centreline. These scenarios were adopted to evaluate their accuracy in simulating the hydrodynamic pressures over ogee spillway. All results achieved in this work will be discussed in the subsequent sections.

i. Steady states

Upon a completion of a steady state simulation, the model convergence had to be monitored. In this study, convergence was judged by simultaneous examination of residual levels and nearness to zero of the mass balance between the flux entering and of that leaving the model. The difference in fluxes should not exceed 1% for a successful convergence.

ii. Fully hydrodynamic tests

To get a fully hydrodynamic test running, only small changes had to be made in the numerical model. These are to determine and set the time step for fully hydrodynamic testing and to define simulation length. Fully hydrodynamic tests were performed with the time step equal to 0.001.

iii. Fully hydrodynamic test duration

Running the fully hydrodynamic tests in a CFD model, the target was to perform tests for the same five minutes as it was done for experimental tests. However, the preliminary simulation results proved that such time length was not necessary as the pressure results did not exhibit a fluctuating trend.

All pressure readings appeared to be in the same range; there were neither upward nor downward trends. Before starting the fully hydrodynamic states tests, the model had to become stable. Therefore, the pressure recorded during this test duration did not show any fluctuation in terms of trend. This resulted in a fixed duration of twenty seconds (20 s) for all other flow rates.

3.3.3 Post-Processing

3.3.3.1 Flow development

With the large extent of data recorded during simulations, different ways were used to visualise flow features. The density and path line contours were employed in the visualisation of flow pattern in the domain. Figure 3-6 included below; indicate the plotted densities of the multiphase flow in this case.

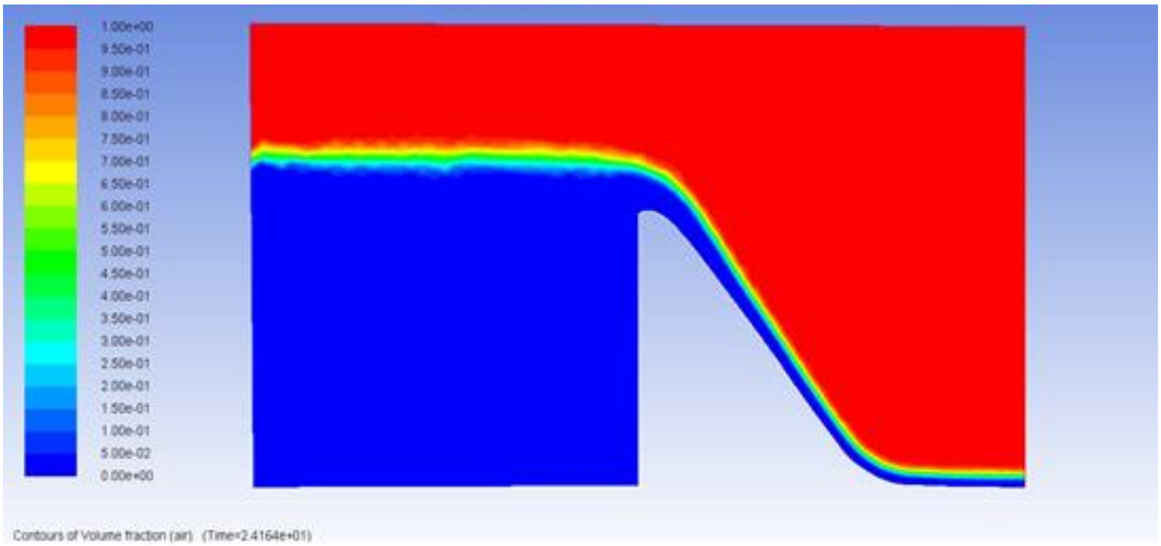

Figure 3-6: Density contours for a flow rate of 130 l/s simulated by CFD model

Water (dark blue color) has a density of 998.2 kg/m^3, air (red color) comprises a density of 1.225 kg/m^3 and the yellowish color in between represents the interface. By plotting the path line contours, the flow trajectories and the interaction between water and air phases have been visually tracked as depicted in Figure 3-7.

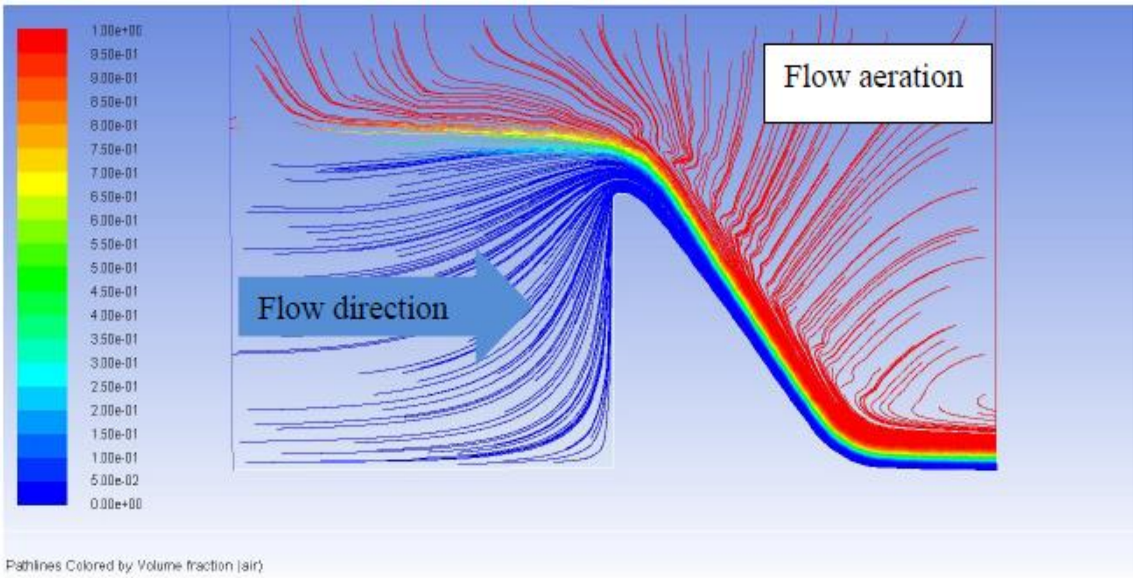

Figure 3-7: Simulated path lines of particles for 130 l/s colored by volume fraction of air

In Figure 3-7, the blue path lines show that water moves from the inlet to the outlet following the same path direction, which confirms that the principle of mass conservation has been achieved. In addition, the aeration conditions over the weir are also significant. In all discharges simulated the aeration conditions were observed in the flow.

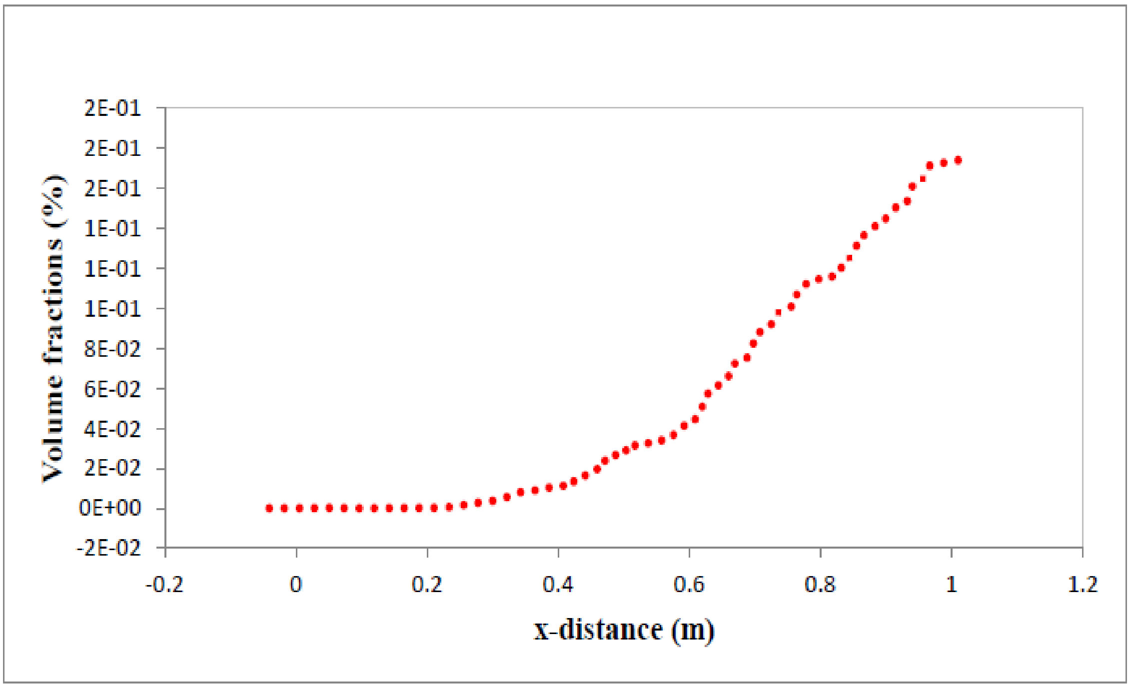

Figure 3-8: Aeration of Flow over ogee spillway model for Q=130 l/s

Figure 3-8 present the rate of volume fraction of air in the flow of 130 l/s . They provide a better representation of the self-aeration of the flow as discussed before.

They also confirm the reason attributed to the pressure fluctuations in section 3.7.1.1 and 3.7.1.2. The pressure sensors located upstream exhibit a steady trend in pressure reading while those located downstream show fluctuations in the pressure readings.

3.3.3.2 Water levels (Surcharge)

Since the general purpose of the multiphase model include capturing flow behaviour and phase interaction, the interface between two phases played a capital role in surface flow determination. The flow levels were measured along the spillway crest by plotting the middle of phase interface, that is, at 0.5 volume fraction. The water surface was plotted at the centreline to eliminate any influence from the boundary walls.

The measurements of water free surfaces acquired form steady and fully hydrodynamic state tests produced the same water free surfaces. However, to make a fully satisfied decision on the accuracy of numerical water surcharge, the results obtained from CFD models were compared to results from physical modelling as a validation process.

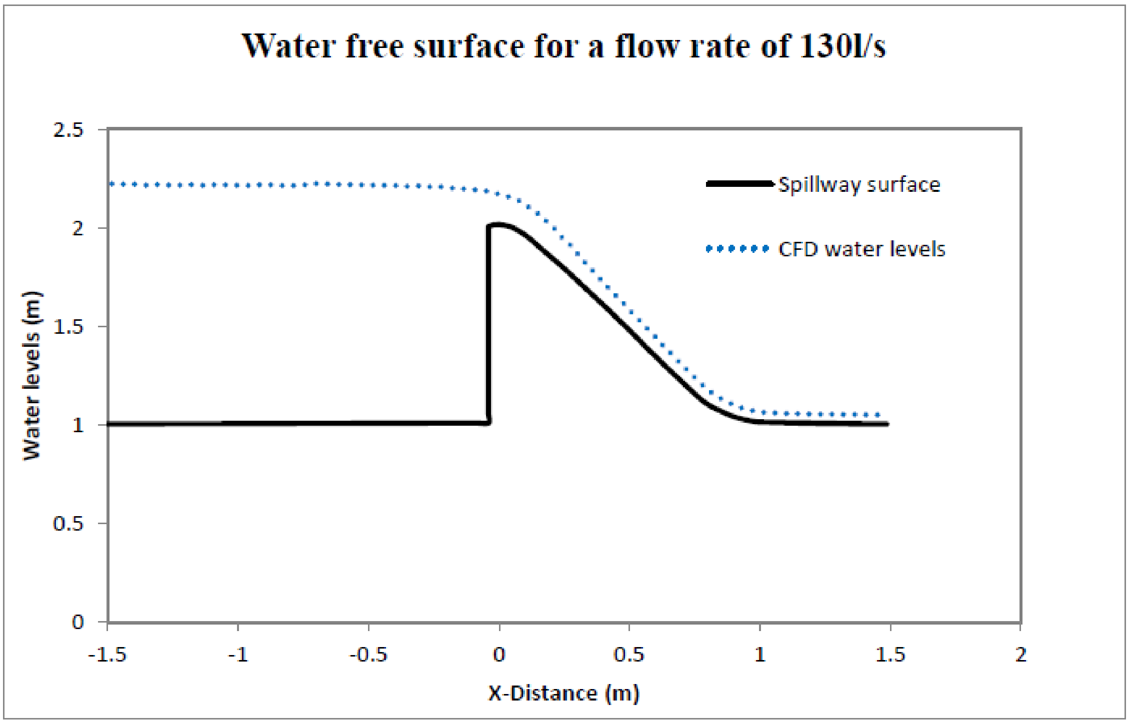

Figure 3-9: Water free surface over spillway model for 130 l/s

Figure 3-9 show that at the spillway crest, water levels are slightly higher than those found in the downstream area. This shows that two actions simultaneously happen in the flow over the crest, that is, the formation and gradual thickening of the turbulent boundary layer along the spillway profile, and a continuous increase in the velocity and decrease in the depth of the main flow. As displayed in the graphical representation of velocity contours in Figure 3-10, in the upstream part, a low approach velocity corresponds to higher water depths whereas in downstream part, the supercritical conditions lead to high velocity magnitude. Using Equation 3-1 the Froude number (Fr) was determined to grasp the flow characteristics.

As demonstrated by the results, the Froude number remains subcritical at the upstream part and develops into supercritical state gradually after the crest. After entering the spillway, the flow drops at a critical depth thereby developing into supercritical depths along the chute. The CFD model indicated the location of the critical flow section (Froude number = 1) near the spillway crest. As seen from the figures (Figure 3-10) depicting the velocity contours, it is quite evident that the velocities become higher at the downstream portion of the spillway. Special energy dissipaters should be provided at this portion of spillway so as to protect the structure from scour due to high velocities.

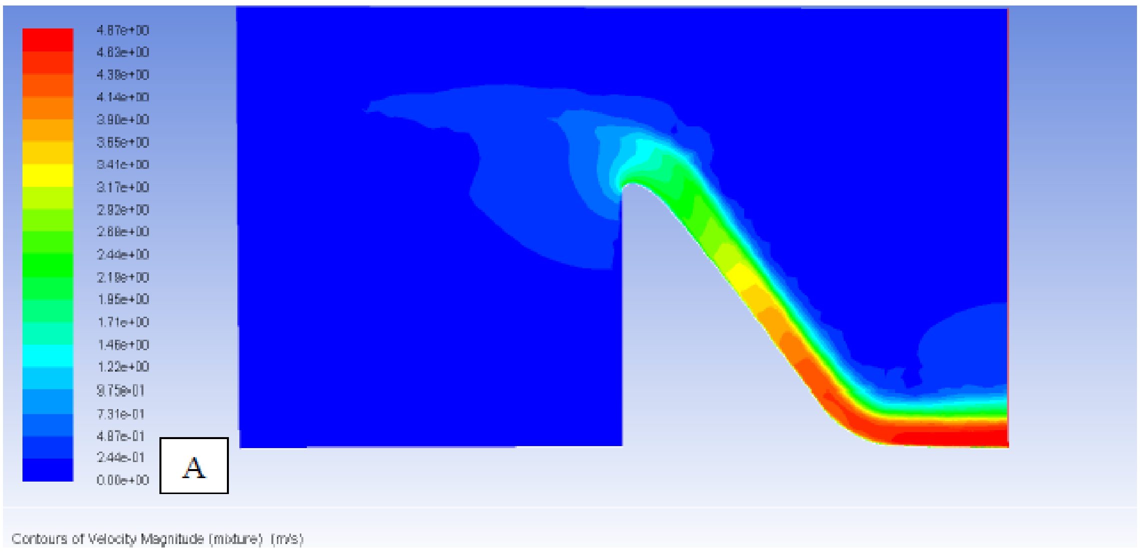

Figure 3-10: Velocity distribution along the spillway model for 130 l/s. A: Density contours

As can be seen in Figure 3-10, air (dark red colour) is entrained in the water at the crest. The velocity vectors (dark blue colour) show that water has a tendency to pull away from the crest thereby reattaching to the spillway face downstream.

3.3.3.3 Pressure results

In this section, the steady and fully hydrodynamic pressures extracted from the models for flow rate is presented. For this spillway case, a comparison was made in form of graphical representation of the results obtained for each discharge tested. The steady state pressure results at each of the 7 recording locations were determined through the average pressures recorded for 2500 iterations, while the average fully hydrodynamic pressure results were determined by averaging of the simulated pressures recorded over the run time of twenty seconds. Table 3-4 and Table 3-5 present the results produced by the 2D-numerical model for steady and unsteady states respectively.

Table 3-4: CFD simulated steady state pressure results for 2D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.118	-0.034	0.009	0.041	0.017	-0.031	0.036

Table 3-5: CFD simulated fully hydrodynamic pressure results for 2D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.120	-0.034	0.009	0.040	0.017	-0.029	0.036

Table 3-6: CFD simulated steady state pressure results for 3D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.072	-0.038	0.012	0.046	0.030	-0.004	0.044

Table 3-7: CFD simulated fully hydrodynamic pressure results for 3D model

Flow Rate(l/s)	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
	CFD modelling: Average Sensor pressure (m)						
130	-0.073	-0.035	0.012	0.047	0.031	-0.003	0.044

Table 3-4 and Table 3-5 present the average pressure readings of the fully hydrodynamic and steady state which indicate a close similarity in the results for 2D models. Also Table 3-6 and Table 3-7 present the average pressure readings of the fully hydrodynamic and steady state which indicate a close similarity in the results for 3D model. Most of the results achieved in both states are equal or close to each other. This model offered an advantage of testing flow rates greater than the design discharge.

In this context, the performance of a spillway for the head that is less, equal and/or greater than the design head has been assessed. For the flow rates less than the design discharge (130 l/s), the hydrodynamic conditions exhibit a positive pressure reading, while for the discharges other than 130 l/s the pressures become negative with the increase of discharge.

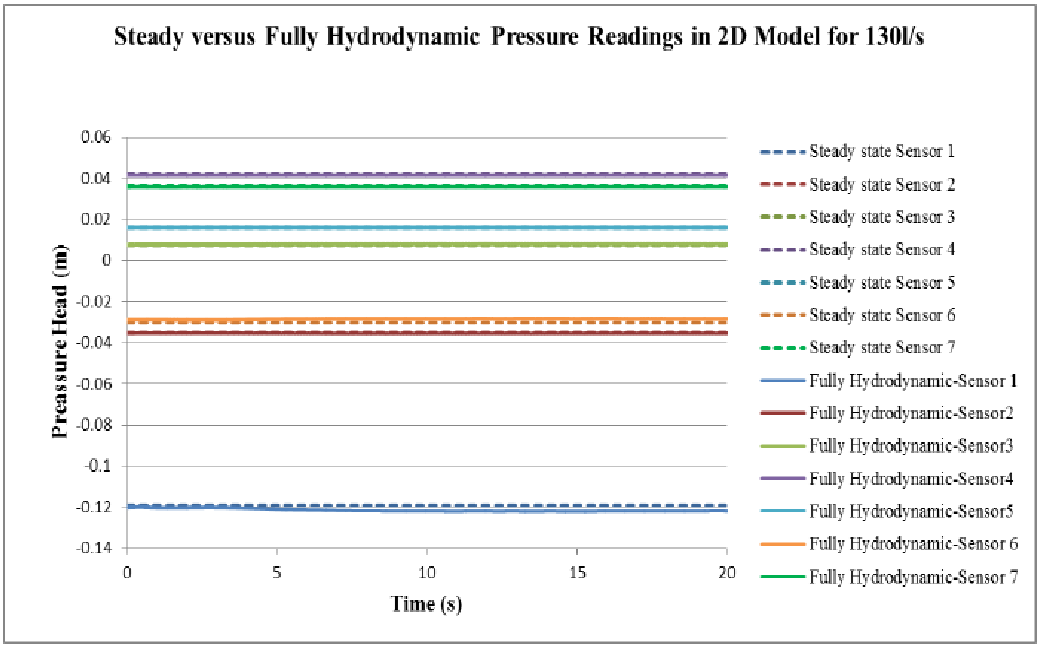

Figure 3-11: Comparison between 2D-simulated steady and fully hydrodynamic state models for a discharge of 130 l/s

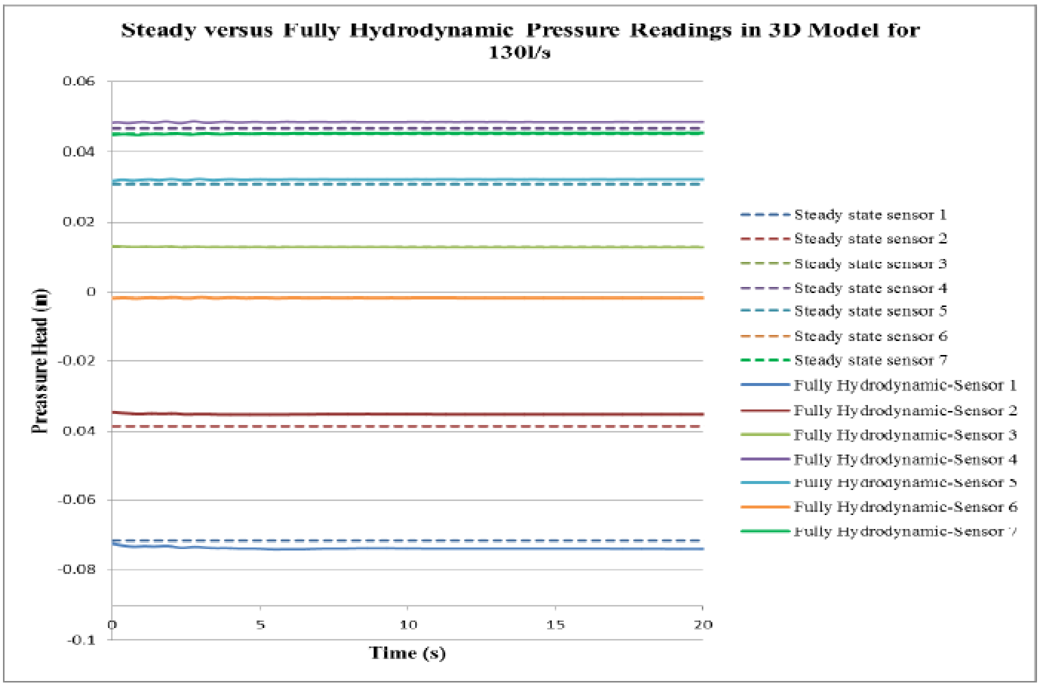

Figure 3-12: Comparison between 3D-simulated steady and fully hydrodynamic state models for a discharge of 130 l/s

3.4 Discussion of numerical model results

After analysing CFD results obtained from spillway case, the following remarks can be drawn:

- The steady state simulations remain a near constant from the beginning until the end whilst the fully hydrodynamic simulations produced, at the beginning, unstable pressure readings which oscillated slightly for 5 to 7 seconds.
- From the comparison between two and three dimension modelling, no significant variation in pressure reading was encountered. The pressures obtained from 2D numerical models are in close agreement at all the recording locations on the spillway face. However, the discrepancy between these models and the degree to which they are accurate will be determined in validation process.

CHAPTER 4

RESULTS AND DISCUSSION

The investigations conducted in the physical and numerical modelling were centred on the surcharge and hydrodynamic effects of flow as discussed in the preceding chapters. This chapter focuses on the validation of CFD modelling through a comparison of results obtained for equivalent flow rates from the physical modelling discussed in the preceding chapter. However, there are a number of factors, known and unknown, which might have affected the comparison between numerical and physical modelling results.

4.1 COMPARISON BETWEEN PHYSICAL AND CFD MODEL RESULTS

The following section is devoted to the validation of the CFD results achieved in this study by comparing the CFD model results with the physical model results.

4.1.1 Water surcharge

The results of surcharge presented in this section are divided into two main parts: (1) Flow surcharge upstream of the spillway and (2) the water surface along the entire spillway model.

4.1.1.1 Upstream water surcharge

The flow surcharge upstream of the crest (at 5H) was an important key consideration during the CFD modelling as it showed the quality of the CFD set-up and confirmed that the flow was behaving similar to the physical modelling. Table 4-1 compare the physical and CFD modelling water surcharge that were obtained from this case.

Table 4-1: CFD and physical modelling water surcharge for ogee spillway

SN°	Discharge (Q/l)	(1) Physical model water surcharge (mm)	(2) CFD water surcharge (mm)	(3) = (2) - (1) Difference (mm)
1	0	0	0	0
2	23	70	78	8
3	35	86	97	11
4	41	98	106	8
5	56	120	125	5
6	71	137	149	12
7	89	156	169	13
8	108	175	187	12
9	130	197	211	14

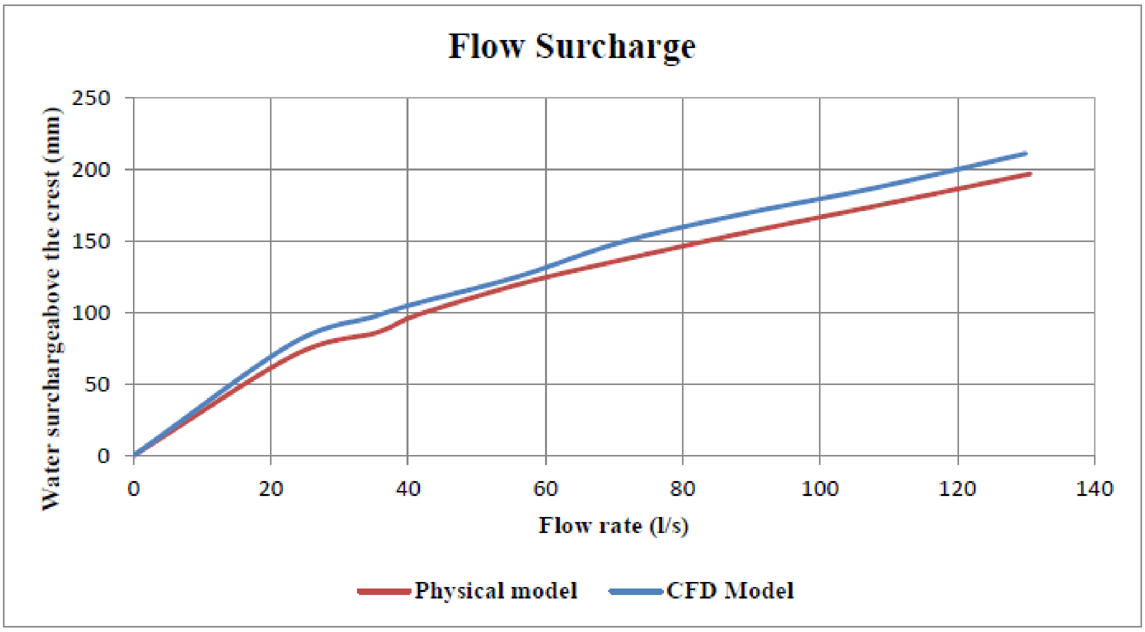

Figure 4-1: Physical modelling and CFD surcharge measured at 5H upstream

The flow surcharge results, shown in Table 4-1and Figure 4-1, shows the surcharge measurements obtained through CFD modelling and physical modelling. As can be observed from the difference, CFD model simulations exhibit a great similarity between the physical model and CFD surcharge measurements. The maximum difference which was found corresponds to 14 mm.

4.1.1.2 Water surface profiles

In the spillway model the water surface profile was investigated along the spillway chute from upstream to downstream of the spillway. The study of the flow surcharge at the flow rate indicated that the water surface profiles obtained from CFD modelling are similar to those of the physical model. It should be noted that the water surface profiles obtained along the spillway refer to the flow depth measured perpendicular to the spillway face. The profile of 130 l/s is presented in Figure 4-2. The water free surfaces, as shown in Figure 4-2, include the surcharge readings obtained through CFD modelling and physical modelling.

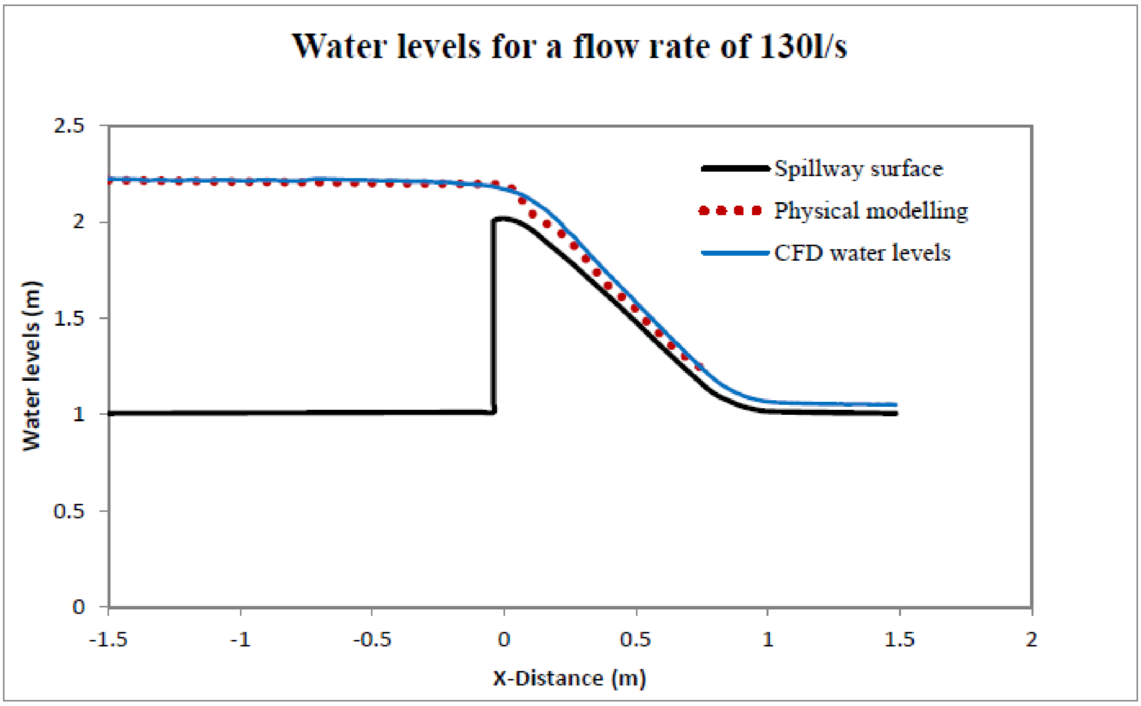

Figure 4-2: The comparison of CFD and experimental free surfaces for 130 l/s

From Figure 4-2, it can be seen that the water free surfaces simulated by CFD models are in good agreement with those measured in physical modelling. It should be noted that the CFD models can simulate the surcharge of flow for ogee spillways consistently.

4.1.2 Pressure results

The average pressure results obtained from CFD and physical modelling are compared in this section to determine the accuracy of CFD models. As discussed in before**,** the comparison of steady and fully hydrodynamic pressure readings indicated a close similarity in all states. However, both steady and fully hydrodynamic pressures were compared to the physical modelling results in this section. Figure 4-3 compares CFD and physical model average pressure for a discharge of 130 l/s.

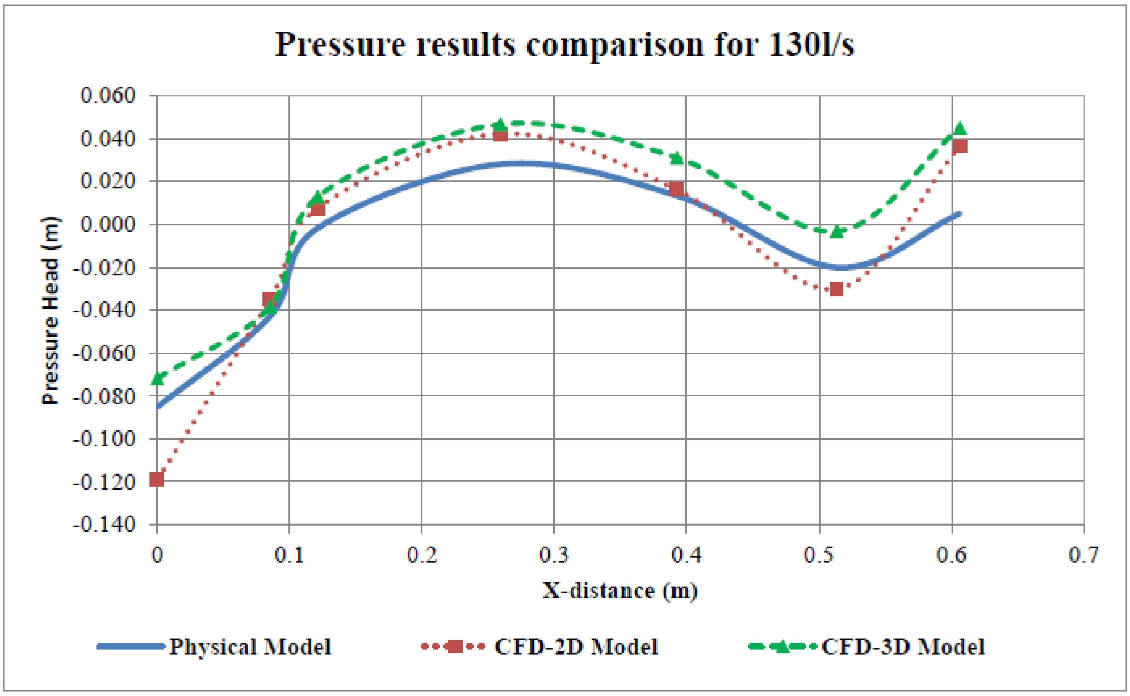

Figure 4-3: Comparison of CFD model and avg Physical model results for 130 l/s

As can be seen in Figure 4-3, the pressure results presented for 2D and 3D models produced the reasonable agreement. The graphical comparison indicates a great agreement between CFD and physical model results. By observing all the results, it can be seen that a better correlation was achieved for this higher flow rate.

To compare CFD with physical modelling results, the differences between the steady state pressures and the physical model average pressures were calculated. These differences were calculated in order to determine the accuracy of the CFD model results. For positive figures, it means that the CFD model overestimates the physical model pressures and vice versa.

Table 4-2 and Table 4-3 present the differences of 2D and 3D steady state with physical model average pressures.

Table 4-2: Differences between 2D-Steady State simulated gauge pressures and physical model average gauge pressure readings

Flow rate (l/s)	Difference between 2D steady State Over mean Physical Model Pressures (m)						
Sensor	**1**	**2**	**3**	**4**	**5**	**6**	**7**
23	0.004	0.003	0.007	0.003	0.013	0.008	0.020
35	-0.015	0.002	0.007	0.008	0.010	0.004	0.021
41	-0.016	0.001	0.008	0.009	0.011	0.004	0.022
56	-0.003	0.004	0.007	0.013	0.008	0.005	0.023
71	0.001	0.002	0.006	0.016	0.006	-0.004	0.025
89	0.011	0.004	0.006	0.019	0.006	-0.006	0.027
108	-0.003	0.007	0.010	0.017	0.005	-0.005	0.029
130	-0.034	0.008	0.009	0.014	0.003	-0.010	0.031

Table 4-3: Differences between 3D-Steady State simulated gauge pressures and physical model average gauge pressure readings

Flow rate (l/s)	Difference between 3D steady State Over mean Physical Model Pressures (m)						
Sensor	**1**	**2**	**3**	**4**	**5**	**6**	**7**
23	0.000	-0.003	0.004	0.006	0.027	0.032	0.028
35	-0.016	-0.002	0.005	0.012	0.026	0.032	0.030
41	-0.014	-0.002	0.007	0.013	0.026	0.033	0.031
56	-0.009	-0.003	0.006	0.015	0.024	0.035	0.031
71	-0.023	-0.012	0.004	0.023	0.026	0.027	0.040
89	0.000	-0.002	0.008	0.017	0.022	0.024	0.036
108	0.003	-0.001	0.011	0.019	0.020	0.022	0.039
130	0.013	0.004	0.015	0.019	0.018	0.017	0.040

Table 4-2 and Table 4-3 present the difference of gauge pressures between CFD model and the physical model results. The differences calculated for all sensors proved a good correlation between the observed and simulated pressures. The results presented above compared the average pressures. To confirm their accuracy, however, 2D and 3D-CFD pressure readings have been compared with those obtained from physical modelling.

4.2 Final remarks

Based on the validation process of ogee spillway hydraulics conducted in this chapter, the following remarks can be highlighted:

- The physical modelling results were used as benchmarks for the water surcharge and free surfaces, for this flow rate. The values produced by a CFD model predicted free surface results that reflect the general flow characteristics over ogee spillways.
- The pressure results obtained from physical and CFD modelling show similar pressure distribution patterns on the spillway face and the pressures are in reasonable agreement. However, CFD models did not simulate the pressure fluctuations as observed in the physical model pressures.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Introduction

Based on the results obtained from both physical and CFD models, conclusions are presented and recommendations for further research are proposed in this chapter.

5.2 Conclusions

Mathematical modelling plays a capital role in the design and analysis of hydraulic structures. However, this often necessitates physical models, considered as the most established form of hydraulic modelling, to ensure reasonable accuracy of the results. This study was devoted to carrying out a CFD validation of ogee spillway hydraulics. To achieve this objective, an ogee spillway model was adopted in the study and the most established method (physical modelling) was chosen as comparison baseline.
The findings in this thesis can be summarised as follows:

- The pressure readings obtained from this case, a negative pressure reading was encountered for the discharges greater than the design discharges with a clear flow separation from the crest. In addition, an accentuated sub-atmospheric pressure appeared, especially for heads which are greater than $1.33H_o$.

- A reasonable agreement was achieved between physical modelling and CFD water surcharge, with a maximum difference of 13 mm.

- For CFD modelling, the model domain was developed with the same dimensions as the physical model with regard to minimising errors as much as possible. Triangle mesh in 2D was selected and cut cell meshing was used for 3D modelling where the minimum grid size in 2D and 3D models was 4mm and 20mm respectively.

- The Volume of Fluid (VOF) and the Realisable "$k\!-\!\varepsilon$" models chosen for this study to model the multiphase flow and turbulence, respectively, simulated successfully the flow over an ogee spillway. The investigations proved that CFD models were able to develop a flow surcharge and simulate the pressures similar to the physical model test results. However, they were unable to accurately determine the pressure fluctuations similar to those obtained from physical modelling.

Concerning the comparison between physical and CFD modelling, it is clear that physical modelling still proves to be the more established of the two methods. Although CFD tools still have limitations (including grid resolution, run times and numerical instabilities to name a few), there are many instances where they may offer an increased accuracy over the designs and provide an insight into the required application. CFD models can provide more detail about velocity and turbulence than a physical model can and may be more economical in some cases.

5.3 Recommendations

The current study has provided an insight into the CFD modelling of ogee spillways. Based on these concluding remarks, the following recommendations are made:

- This study attempted to validate the uncontrolled ogee spillway hydraulics with Ansys-Fluent software. The same validation is recommended for a controlled ogee spillway to assess the use of this software in the design and testing processes of this particular type.

- Other types of turbulence models, apart from Realisable $k\!-\!\varepsilon$ and Volume of Fluid model (VOF) within Ansys-Fluent should be assessed to determine their capabilities for turbulence and multiphase modelling.

- Ansys-Fluent takes a very long time to produce results. Normally, a five minutes test length in physical modelling would take more than one month to provide results for a single discharge scenario. Therefore, methods should be developed to reduce simulation periods.

References

[1] AIAA, 1998. Guide for the Verification of Computational Fluid Dynamics Simulations. *s.1: AIAAG-077-1988*.

[2] Berger, C. R., & Winant, E.H. (1991). One dimensional Finite Element Model for Spillway. *Hydraulic Engineering, Proceedings 1991.* Nashville: National Conference , ASCE.

[3] Bhajantri, M. E., & Deolalikar, P. (2006). Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model. *Sadhana*, 743-754.

[4] Bradley, J. N. (1945.). *Studies of flow characteristics, discharge and pressures relative to submerged dams.* Hydraulic Laboratory Rep No. 182, Denver.

[5] Brater, F. E., King, W. H., Lindell, J., & Wei, Y. C. (1996). *Handbook of Hydraulics.* New York: MC Graw - Hill.

[6] Cain, P., & Wood, I. (1981). Measurements of Self-aerated Flow on a Spill way. *JI. Hyd. Div., ASCE, 107, HY11*, 1425-1444.

[7] Causon, D., Mingham, C., & Ingram, D. (1999). Advances in Calculation methods for super-critical flow in spillway channels. *Journal of Hydraulic Engineering 125 (10),* 1039-1050.

[8] Chadwick, A., Morfett, J., & Borthwick, M. (2004). *Hydraulics in Civil and Environmental Engineering.* New York: Spon Press.

[9] Chanson, H. (2002). *The Hydraulics of stepped chutes and spillways.* Lisse, Netherlands: Balkema.

[10] Chatila, J., & Tabbara, M. (2004). Finite Analytic Method in Flows and Heat Transfer. *Computers and Structures 82*, 1805-1812.

[11] Chow, V. T. (1959). *Open-Channel Hydraulics.* New York: McGraw Hill Book Company INC.

[12] Cloete, S., Olsen, J., & Skjetne, P. (2009). CFD modelling of plume and free surface behavior resulting from a sub sea gas release. *Applied Ocean Research 31(3),* 220-225.

[13] Daneshkhah, A., & Vosoughifar. (2012). Solution of Flow Field Equations to investigate the Best Turbulent Model of Flow over a Standard Ogee Spillway in Finite Volume Method. *The First International Conference on Dams & Hydropower.*

[14] Darvas, L. (1971). Performance and design of labyrinth weirs. *Journal of Hydraulic Engineering ASCE,* Issue 97(8), pp. 1246-125.

[15] Falvey, H. (2003). *Hydraulic Design of Labyrinth Weirs.* Virginia: ASCE Press.

[16] Ferzieger, J., & Peric, M. (1996). *Computational Methods for Fluid Dynamic.* Heidelberg: Springer-Verlag.

[17] Fluent. (2009). *Ansys Fluent 6.3 User's guide.* Ansys Inc.

[18] Gessler, D. (2005). CFD Modelling of Spillway performance, EWRI 2005: Impacts of global climate change. *In Proceedings of the World Water and Environmental Resources Congress.* Anchorage, Alaska: American Society of Civil Engineers.

[19] Ghare, A., Wadhai, P., Mistry, N., & Porey, P. (2008). Hydraulic and Environmental Aspects of Long Crested Weirs. *Global Journal of Environmental Research 2 (3)*, 122-125.

[20] Ghosh, S. (2014). *Flood Control and drainage Engineering, Fourth Edition.* London, UK: CRC Press/Balkema.

[21] Guot., Wen, X., & Wu, C. F. (1998). Numerical Modelling of Spillway Flow with Free Drop and Initially Unknown Discharge. *J. Hydr. Res. 36(5).* , 785-801.

[22] Hattingh, L. N. (2012). Lessons learned from Dam Safety Incidents in South Africa. Kyoto, Japan: ICOLD.

[23] Henderson, F. (1996). *Open Channel Flow.* Prentice-Hall Inc.

[24] Hirt, C. W., & Nicholas, B. (1981). Volume of Fluid Method (VOF) for the Dynamics of Free Boundaries. *J.Comp. Physics, Vol. 39*, 201-205.

[25] Houston, K. (1983). *Hydraulic Model Study of Hyrum Dam Auxilliary Spillway.* Denver: Bureau of Reclamation Division of Research Hydraulics Branch.

[26] ICOLD. (2001). Tailings Dams- Risk of dangerous occurences, lessons learnt from practical experiences. *ICOLD Comittee on Tailings Dam and Waste Lagoons* (p. Bulletin 121). Paris: United Nations Environmental Programme (UNEP), Division of Technology, Industry and Economics (DTIE) and International Commission on Large Dams (ICOLD).

[27] ICOLD. (2012). Complementary Use of Physical and Numerical Modelling Techniques in Spillway Design Refinement. *ICOLD, 24th Congress* (pp. 55-76). Kyoto: ICOLD.

[28] Johnson, M., & Savage, M. B. (2006). Physical And Numerical Comparison of Flow Over Ogee Spillway in the Presence of Tailwater. *Journal of Hydraulic Engineering, Vol. 132, No.12*.

[29] Kim, G. D., & Park, H. J. (2005). Analysis of Flow Structure overOgee - Spillway in consideration of Scale and Roughness Effects by using CFD Model. KSCE Journal of Civil Engineering, PP 161-169.

[30] Lihe, J., Mazin, E., Natalya, S., Dudley, R., & Sukumar, A. (2011). Ruskin Dam spillway shocrete assessed. *Concrete Internationwithpermission of American Concrete Institute*, 7.

[31] Loftin, M. (1999). *Water Resources Engineering",.* The McGraw-Hill Companies, Inc.

[32] Maynord, T. S. (1985). *General Spillway investigation.* Washington D.C: Department of The Army - Waterways Experiment STation-Corps of Engineers.

[33] Mays, W. L. (1999). *Hydraulic Design Handbook.* Tempe, Arizona: McGraw. Hill Companies.

[34] Murrone, A., & Villedieu, P. (2011). Numerical Modeling of Dispersed Two-Phase Flows. *Aerospace Lab*.

[35] Nikseresht, A., Alishahi, M., & Emdad, H. (2008). Complete Flow Field Computation around an ACV (Air Cushion Vehicle) Using 3-D VOF with Lagrangian Propagation in Computational Domain. *Computer and structures Vol. 86,* , No7-8.

[36] Oberkampf, l., Deland, M., Rutherford, M. B., Diegert, V. K., & Alvin, F. K. (2001). Error and Uncertainty in modelling and Simulation. *Reliability Engineering & System Safety*, 333-357.

[37] Panton, R. (1984). Incompressible flow. *John Wiley and Sons*, 675.

[38] Placide Nshuti Kanyabujinja (2014) *). Physical Experimentaion of Flow Over Ogee Spillway in the Presence of Tailwater. Journal of Hydraulic Engineering, Vol. 132, No.12..*

[39] Singh, C., & Zhou, F. (1999). Simulation of Free Surface Flow over Spillway. *Journal of hydraulic Engineering 125(9)*, 959-967.

[40] Savage, B., Frizell, K., & Crowder, J. (2004). *Brains versus Brawn: The Changing World of Hydraulic Model Studies*. [Online] Available at: http://www.usbr.gov/pmts/hydraulics_lab/pubs/PAP/PAP-0933.pdf

[41] Veersteg, H., & Malalasekera, W. (1995). *An Introduction to Computational Fluid Dynamics.* Harlow: Longman Group Ltd.

[42] Versteeg, H., & Malalasekera, W. (2007). *An introduction to Computational Fluid Dynamics: The finite volume method.* England: Prentice Hall.

[43] Wendet, F. J. (2009). *Computational Fluid Dynamics: An Introduction, 3rd Edition.* Eagle River, WI, USA: Springer.
WIKA. (2013). [Online] *Available at :http://www.wika.co.za/publish/download_datasheets_PE_en_co.aspx?ActiveID=14267*