Research on Computational Fluid Dynamics (CFD) Modeling in the Numerical Simulation of Spillways and its Comparison with Experimental Findings

For Kurdistan Engineering Union

by

Marif Mahmood Karim

B.Sc. in Civil Engineer

M.Sc. in Hydraulic

12/12/2023

KEU ID: (9894)

2023

TABLE OF CONTENTS

Page

CHAPTER 1

INTRODUCTION

1.1 General

The spillway is one of the most important parts of a dam to ensure the safety of the dam during the flood. flood control, dam construction is the most important issue for the entire world through navigation, hydroelectric power generation, fishing, and recreation. Technical improvement of dam design and analysis is necessary for better management of water resources because water is very important for protection. The reservoir overflows when the difference between inflow and outflow exceeds its capacity.

The spillway has been established for all dams as a safety measure against overstepping. When the water level exceeds the full supply level (FSL) it is provided to safely carry water from the reservoir.

Any spillway has five basic components: An entrance channel, a control structure, a discharge carrier, an energy dissipater, and an outlet channel.

According to the definition of United States Department of the Interior Bureau Reclamation (USBR), the spillway is provided for storage and detention dam to release over plus water or floodwater that cannot be collected in the allotted storage space, and for diversion dam to bypass flows excessively those turned into the diversion system.

1.2 Ogee spillway

Due to proper function, low cost, floodwater control capability, and high safety factor, the ogee spillway is used in the hydraulic structures.

A spillway as sketched in Figure1.1 is called ogee-crested (overflow) spillway which is the most common among several types of spillways due to its ability to discharge extra water from upstream to downstream efficiently and safely when properly designed and implemented. Three zones can be noted: the crest, the face, and the toe, each with its separate problems.

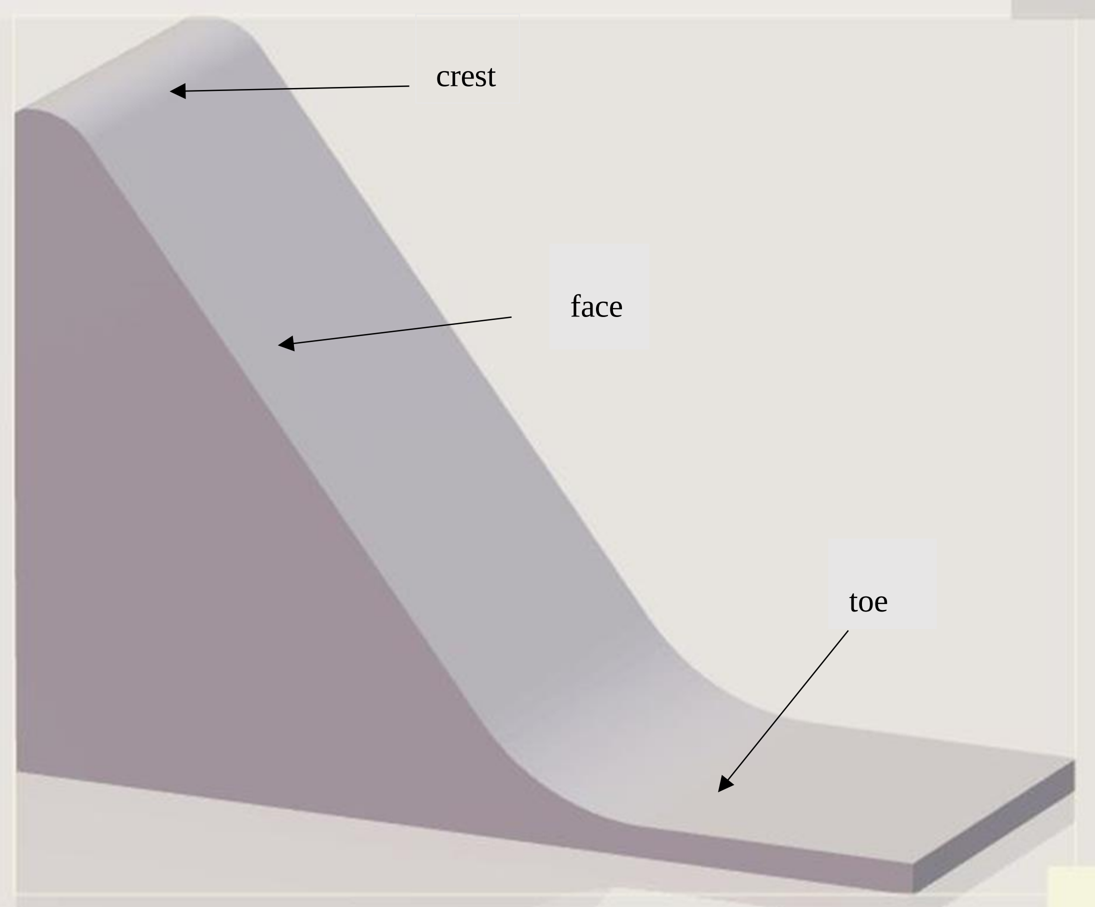

Figure 1.1 General view of a spillway

1.3 Computational fluid dynamics (CFD)

Computer programs are rapidly developed with the development of computer technology and the physical investigations now are faster and easier, also with the developing of the computational fluid dynamics (CFD) power in the hydraulic engineering, Investigation of flow on hydraulic structures is increasingly being used by numerical methods. The validated and agreement results of numerical models with physical models are led to numerical models can be used for designs.

In principle, CFD does not replace the measurements completely, but the amount of experimentation and the overall cost and time can be significantly reduced. Numerical models are usually cheaper and faster than physical models, and furthermore, it can be used for multiple-purpose at one time.

The results of CFD simulation are never 100% reliable. The input data may be accompanied by too much speculation or inaccuracy, the mathematical model of the problem at hand may be inadequate, the accuracy of the results is limited by the available computing power.

1.4 Study Objective

The object of this thesis is to perform a numerical model based on an experimental study by Kanyabujinja (2015) which initially held to find the water surface profiles, pressure, velocity, and shear stress on the surface of a spillway in to different model of spillway to led how can make a safety spillway against shear stress and cavitation that is the dangerous problem in spillway. This is done in some cases of different discharges simulated with a CFD package called Flow-3D and comparing the results with experimental research to approve the suitability of Flow-3D software.

CHAPTER 2

FLOW MECHANISM OVER SPILLWAYS

2.1 General

The spillway is a structure built to provide controlled flood and to ensure that the flow does not exceed the dam's full supply level (FSL) and convey water from the dam to the downstream, Usually, a river where a dam is built, and energy dissipation are taking place along its slope. Weirs are provided for storage and holding dams to release surplus water or floodwater that cannot be contained in the allocated storage space, and for diversion dams to bypass flows exceeding those turned into the diversion system. (USBR, 1987).

2.1.1 Spillway classification

Spillways are usually classified according to their most prominent features, either as it pertains to the control, to the discharge channel, or to some other feature. Spillways are often called "controlled" or "uncontrolled", depending on whether they are gated or ungated. Commonly referred to types are free overfall (straight drop), ogee (overflow), labyrinth, side channel, open channel (trough or chute), tunnel, drop inlet (shaft or morning glory), conduit, baffled apron drops, culvert, and siphon. (USBR, 1987), spillway types are shown in Figure 2.1.

Spillways are classified into four separate categories, each of which will serve satisfactorily for specific site conditions when designed for the anticipated function and discharge. Spillway can be categorized into different types based on various criteria: Overflow Spillway, Chute Spillway, Side Channel Spillway and Limited Service Spillway. (USACE,1992)

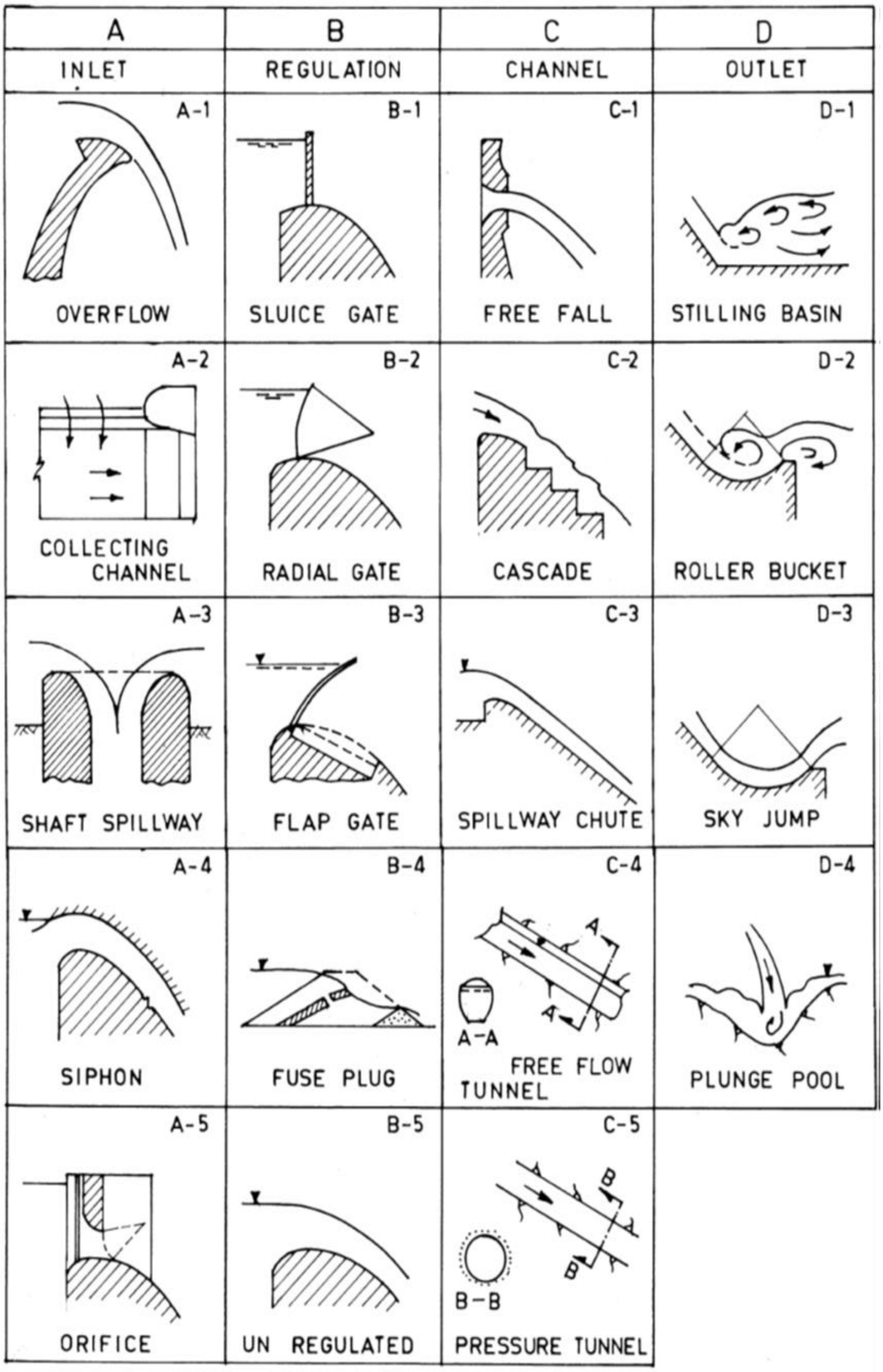

Figure 2.1 Classification of Spillway (Vischer et al,1988).

Spillways have been classified according to various criteria:

I. According to the most prominent feature

Ogee spillway, chute spillway, side channel spillway, shaft spillway, siphon spillway, a straight drop or over fall spillway, tunnel spillway/culvert spillway, labyrinth spillway and stepped spillway.

2. According to Function

Service spillway, auxiliary spillway, fuse plug or emergency spillway.

3. According to Control Structure

Gated spillway, ungated spillway, and orifice of sluice spillway (Khatsuria, 2005).

2.1.2 Functions of a spillway

The main function of the spillway is to pass the excess water from the reservoir to the downstream river, there are exactly seven functions that can be assigned to the spillway as discussed by (Takasu et al. 1988) Maintaining normal river water functions (compensation water supply), discharging water for utilization, maintaining initial water level in the flood-control operation, controlling floods, controlling additional floods, releasing surplus water (securing dam and reservoir safety), lowering water levels (depleting water levels in an emergency).

2.2 Ogee (Overflow) Spillways

The ogee or overflow spillway is the most common type of spillway. It has a control weir that is ogee or S-shaped. It is a gravitational structure that requires a sound foundation, preferably located on the main river way, although there are many spillways located on the sides in excavated channels due to foundation problems. The structure is naturally divided into three zones: the crest, the rear slope, and the toe. An ogee crest and apron may comprise an entire spillway, such as the overflow portion of a concrete gravity dam, or the ogee crest may only be the central structure for some other type of spillway. Because of its high discharge efficiency, the nappe-shaped profile is used for most spillway control crests. (Khatsuria, 2005).

the ogee spillway crest is basically a sharp-crested weir with an empty space below the lower nappe replaced with concrete. (USACE,1990).

CHAPTER 4

CASE-STUDY

3.1 General

In this chapter, a 3D numerical model of hydraulic phenomena is simulated based on an experimental study by Kanyabujinja (2015). The experimental was done in hydraulics laboratory at the Faculty of Engineering at Stellenbosch University. The numerical models were simulated by using the Flow-3D software. The hydraulic aspects were simulated such as the hydrodynamic flow characteristics, water surcharge on upstream and along the spillway downstream face. The following section includes in detail of accomplished work.

3.2 Physical model set-up and experimental procedure

From the previous study, the physical modeling tests were done in the Hydraulic Laboratory of Stellenbosch University, located in the Western Cape Province, Republic of South Africa. Two different model of ogee spillways were carried out one after another. Furthermore, the flume sides were made of transparent glasses, the length of flume is 22 m, 1.25 m high, and 0.60 m in wide. The ultimate flow rate that could be obtained in the laboratory from a constant head tank was 130 l/s. The experiments were set-up in a recirculating flume where by the outflow system was set up in a way that allowed the flow to be re-used. Figure 3.1 shows the experimental setup by Kanyabujinja (2015).

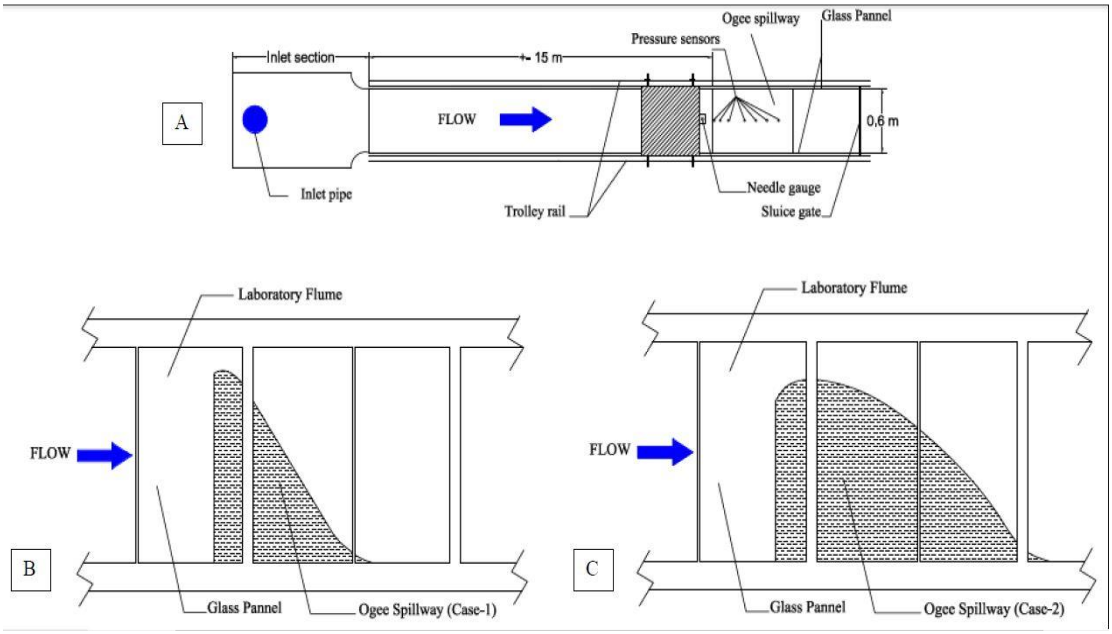

Figure 3.1 A: Plan view, B: Side view of Case-1, C: side view of Case-2 (Kanyabujinja, 2015).

3.3 Physical Model Design

In the section below, the geometric designs and hydraulic of both ogee spillways used for physical and numerical modeling are explained. The design approach was taken from the procedure documented in Small Dam Design by USBR (1987).

Figures 3.2 and 3.3 show all geometric dimensions, in millimeters, of the first and second case respectively.

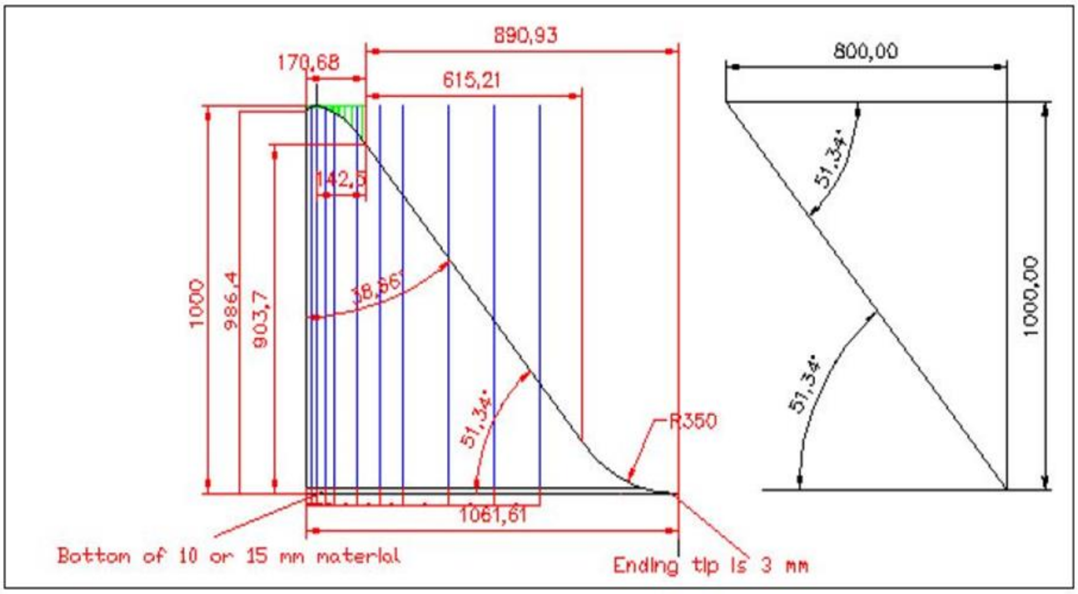

Figure 3.2 Geometric dimensions for Case-1 (Kanyabujinja, 2015).

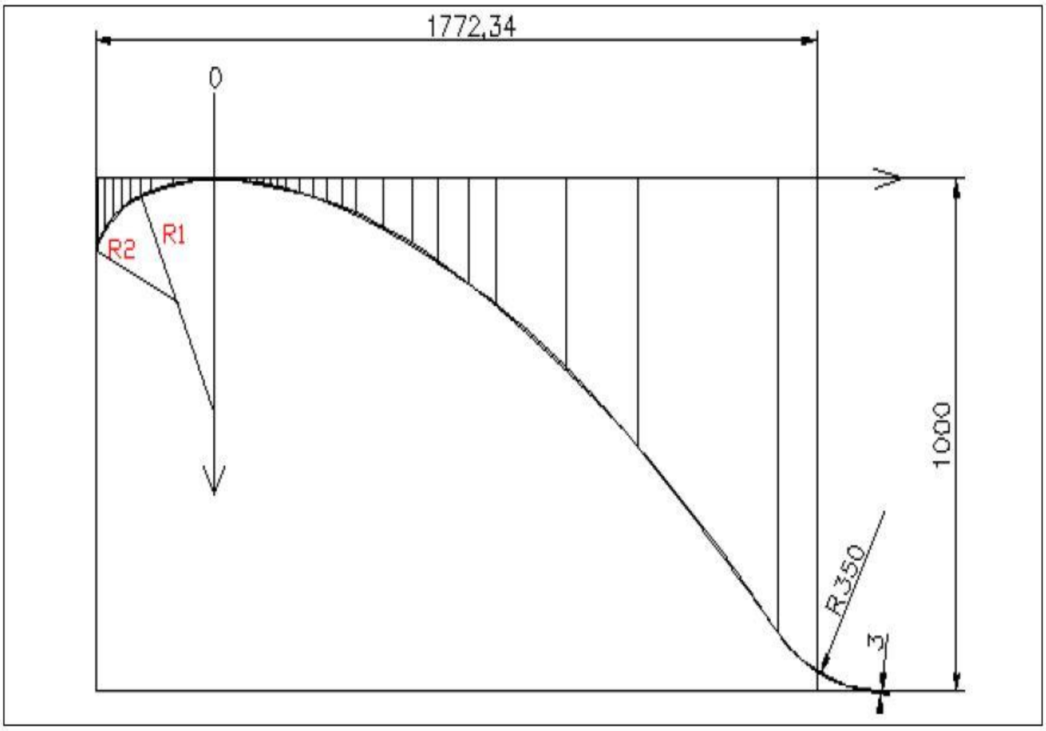

Figure 3.3 Geometric dimensions for Case-2 (Kanyabujinja, 2015).

The geometric dimensions (depth, width, and radiuses) for both cases are presented in Table 3.1.

Table 3.1 Model dimensions used by Kanyabujinja (2015)

Model Type	Spillway approach depth (m)	Crest width (m)	Radii(m)	
			R1	R2
Case-1	1.00	0.60	0.05	0.02
Case-2	1.00	0.60	0.50	0.20

3.4 Pressure sensors

For finding the pressure on the spillway surfaces seven pressure transducers (WIKA S10) were fixed under the ogee spillway chute to measure hydrodynamic pressures. WIKA S10 transducers are fabricated with a high precision to acceptable most industrial pressure measurement applications (WIKA, 2013).

This type of transducers can measure the pressure range of one meter of water (1 m H2O) with an accuracy of 0.1% of the full range. Figure 3.4 and Figure 3.5 indicated the location of the seven sensors on both physical models.

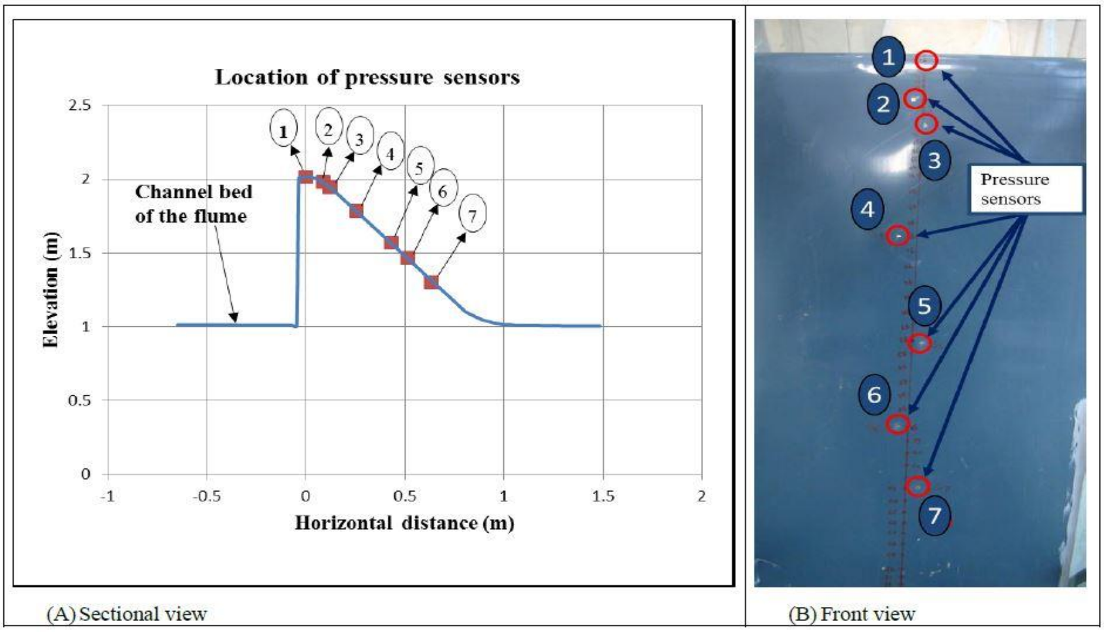

Figure 3.4 Positions of pressure transducers on Case-1(Kanyabujinja, 2015).

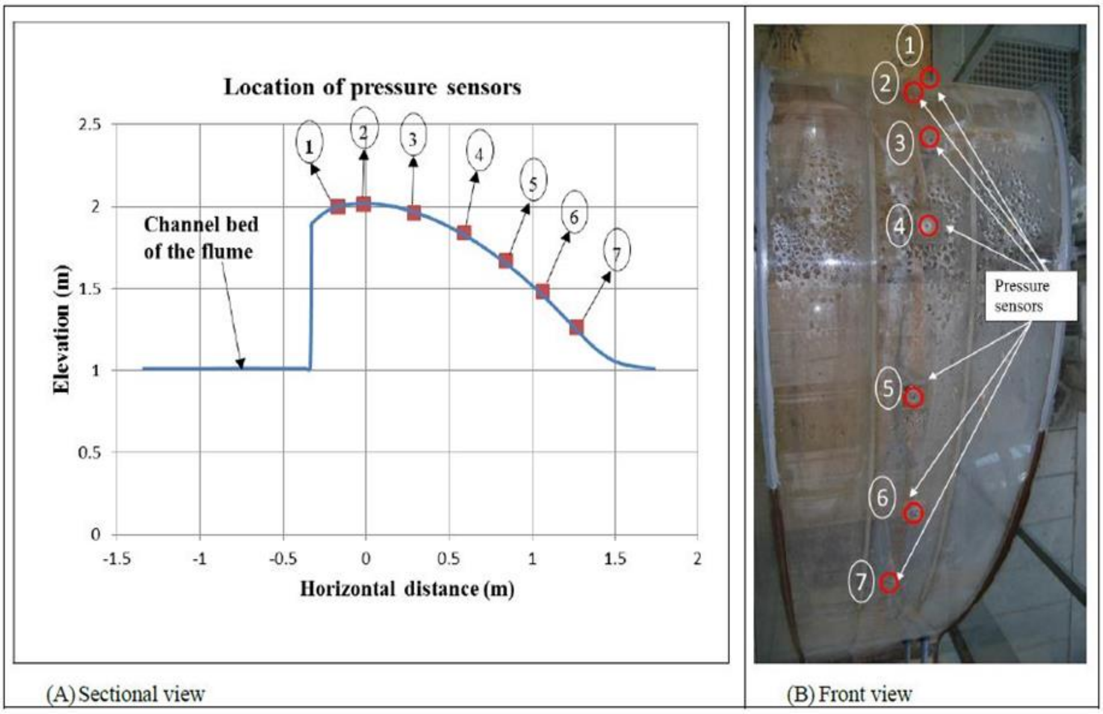

Figure 3.5 Positions of pressure transducers on Case-2 (Kanyabujinja, 2015).

3.5 The Numerical Modelling

CFD codes generally consist of three main stages. These stages are pre-processor, solver, and post-processor stages one after another.

Pre-processing creates from the input parameters of a fluid flow problem to a CFD software by a user interface. The numerical modeling begins with a computational mesh. The computational domain is formed by the number of interconnected components. In this stage, after describing the geometry and computational domain of the problem, grid generation (mesh) is done. The grid generation is a very important step because the accuracy of a numerical simulation depends on the grid quality. Through the introducing of fluid properties, appropriate boundary conditions and initial conditions are specified within the pre-processing step.

In the solver stage, discretized forms of the governing equations of fluid flow over all the computational domains are solved. In each computational cell, all the flow parameters are calculated.

A CFD simulation generates a huge amount of data, and it is not possible to post-process. Many CFD packages have been developed using visualization and post processing tools, where a large amount of data could be analyzed. There is also specialized post-processing software where 2D and 3D surface plots, contour plots, iso-surfaces, vector plots, flow lines, grid display and geometry, text data outputs and a lot of graphs combination could be drawn. Furthermore, they may also include animation tools for displaying dynamic results. The step of CFD analysis can be described as follows in Figure 3.6.

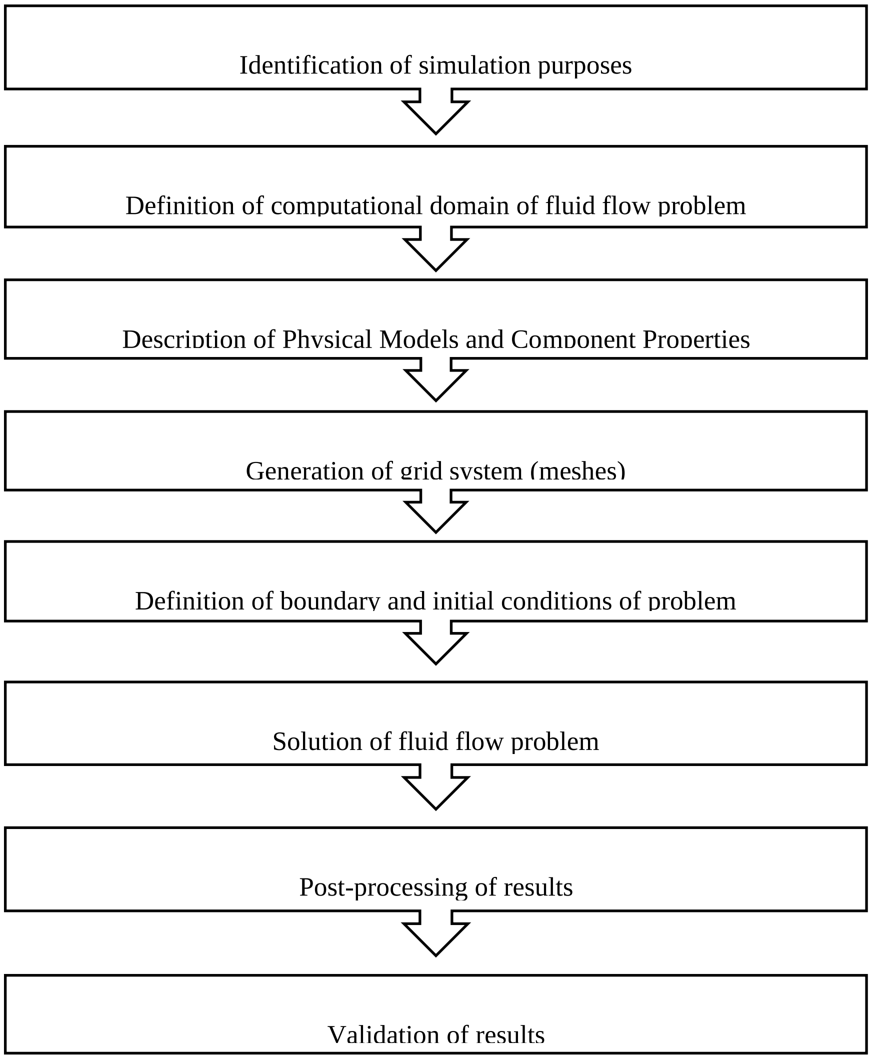

Figure 3.6 Steps in CFD analysis

In hydraulic engineer, computational modeling is a helpful tool. Packages like FLOW-3D are strongly effective and quick tools. They enable project deadlines to be met more easily and can reduce costs with alternative solutions and optimizations.

3.6 Brief introduction to Flow 3D

Flow-3D software is used to simulate numerical models that are powerful software and commercial packages developed by Flow Science Inc. (Flow Science, 2008). The software applies several excellent features to the numerical solution of the Navier-Stokes equations and continuity equations are discretized and solved in each computational cell, for open channel flow (VOF) and meshing of composite geometries using (FAVOR).

In this thesis, Flow 3D has been used in the solver step. It is a powerful tool for problems of complex fluid modeling. Flow-3D provides highly accurate simulations and using TruVOF for free surface flows. Flow 3D can solve the Navier-Stokes in three dimensions to simulate the continuity equations together with the fluid flow equations for the turbulence quantities. In order to solve the RANS equation, the software uses a finite volume method. A mesh which is a rectangular grid of cells is formed by subdividing the computational region.

In Flow-3D has many chooses for physical models that are added to or modified the basic Navier-Stokes equations. These items are describing the effects of turbulence, porous media, surface tension, air entrainment, solid deformation, heat transfer, cavitation, fluid solidification, sediment scour, moving solids, and granular flows. The software could be used in different modes such as incompressible flow compressible flow, situations or limited compressibility conditions. In addition, Flow 3D has one fluid model or two fluid models. In this study, we use one fluid uncompressed mode while modeling the free surface.

3.6.1 Geometry

The simple objects can be created by Flow-3D, but the complex objects, or for easy it can use the Auto-Cad, Solid work software or any software has (stl) format. The geometry of the spillway for each model was drawn in Auto-Cad software in 2D and extrude to 3D then, exported as a stereolithographic (stl) format. The stl file was directly imported into the Flow-3D, but the coordinates of the original point should be known in Auto-Cad or other software. See Figure 3.7.

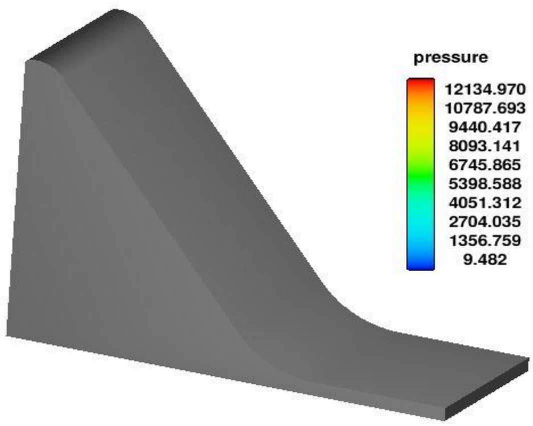

Figure 3.7 Spillway geometry in Flow-3D

3.6.2 Meshing

A mesh is a subdivision of the flow domain into relatively small regions in a CFD numerical model, the smallest part of the subdivided called cells, in which numerical values such as velocity, pressure and shear stress are computed. Cell sizes in each block mesh can affect both the simulation time and the accuracy of the results so it is very important to minimize the number of cells while including enough resolution and adequate flow detail. In this study, the computational domain was divided into the cells and the mesh sizes in x,y, and z directions are 0.02 m.

3.6.3 Data Sharing

FLOW-3D can generate output in the form of numerical data, images, and animations for sharing in presentations and embedding in reports. Figure 3.8 and 3.9 show the 3D and 2D computational domain Case-1 that is 1.6 m long, 0.6 m width and 1.3 m height, and the mesh sizes in x,y and z directions are 0.02 m. And Figure 3.10 and 3.11 show the 3D and 2D illustration of the 3D computational domain (Case-2) that is 2.6 m long, 0.6 m width and 1.3 m high including,

For Case-1: There are total number of cells (active and passive) = 177954 mesh cells in the simulation, and total number of active cells =132940 active cells include: real cells (used for solving flow equations) = 116274, open real cells =116274, fully blocked real cells = 0, external boundary cells =16666, inter-block boundary cells = 0 (Flow-3D report).

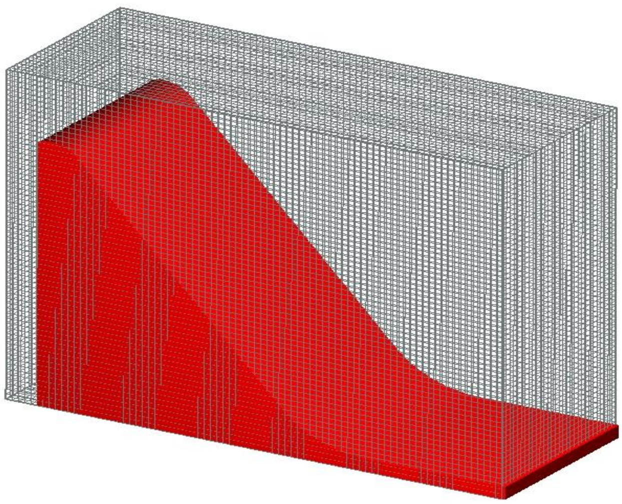

Figure 3.8 Representation of the geometry meshing of 3D model Case-1

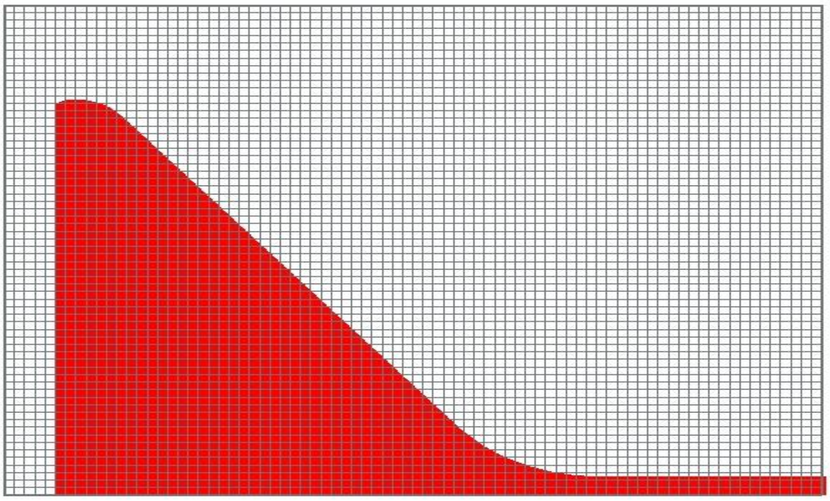

Figure 3.9 Representation of the geometry meshing of 2D model Case-1

For Case-2: total number of cells (active and passive) = 285154, total number of active cells = 178690, active cells include, real cells (used for solving flow equations) = 157740, open real cells = 157740, fully blocked real cells = 0, external boundary cells = 20950, inter-block boundary cells = 0 (Flow-3D report).

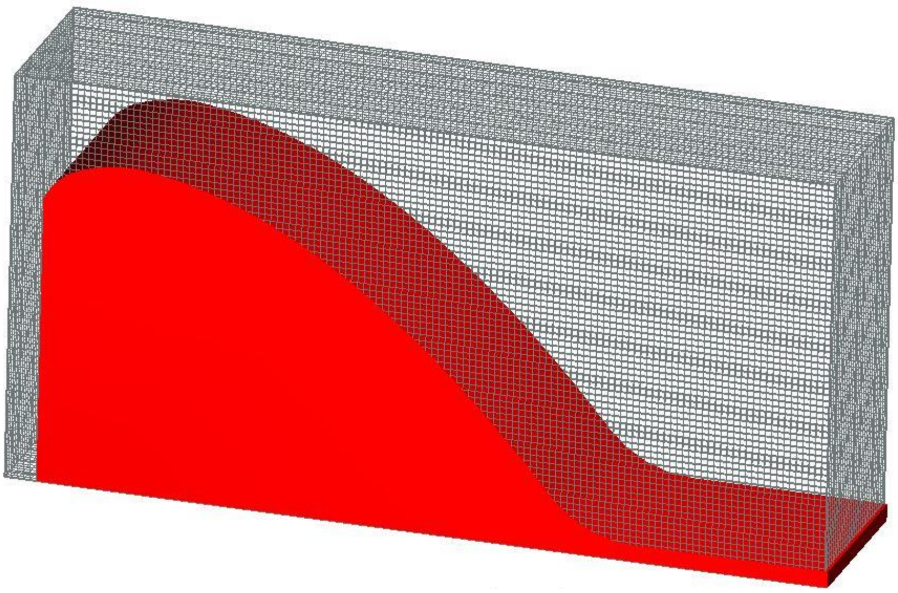

Figure 3.10 Representation of the geometry meshing of 3D model Case-2

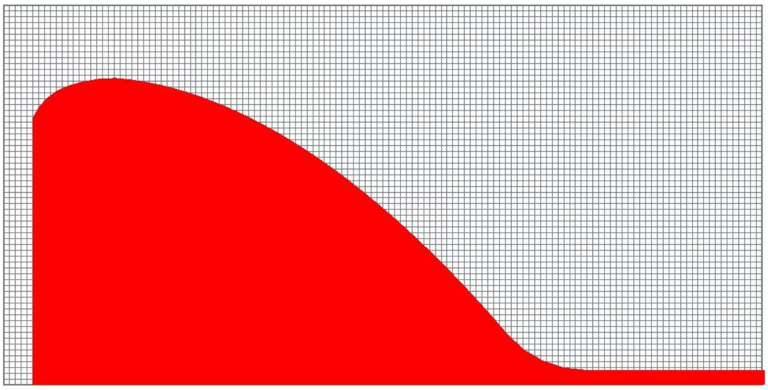

Figure 3.11 Representation of the geometry meshing of 2D model Case-2

3.6.4 Specifying Boundary Conditions and initial conditions

When solving continuous equations and Navier-Stokes equation, appropriate initial conditions and boundary conditions must be applied.

Flow-3D has many types of boundary condition; each type uses for the specific condition of models. The boundary conditions in Flow-3D are symmetry, continuative, specific pressure, grid overlay, wave, wall, periodic, specific velocity, outflow, and volume flow rate. As shown in Figure 3.12.

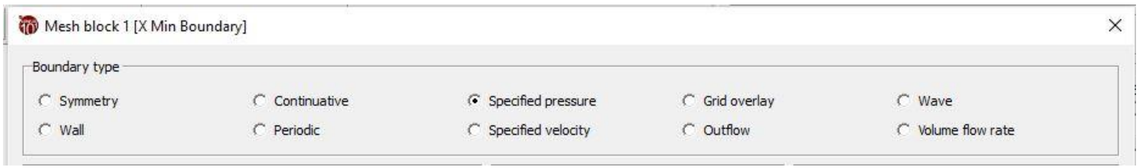

Figure 3.12 Type of boundary conditions in Flow-3D.

Figure 3.13 and 3.14 indicates the boundary conditions for the simulated spillway model in Case-1and Case-2 respectively, as shown the upstream boundary (X min) is stagnation pressure condition (Hydrostatic pressure with zero velocity), the downstream (X max) is outflow boundary while the bottom (Z min) is computed as wall boundary and the top (Z max) is symmetry boundary, for both (Y min) and (Y max) were labeled as symmetry. While in ANSYS for the same models and discharges the boundary conditions are used as follows:

For water inlet (x min), a (velocity-inlet) boundary condition was used, and in (X max) the outlet was selected, while for (Y min, Y max, and Z min) the wall boundary was choosing, and the Z max labeled as air-inlet, as showed in Figure 3.15.

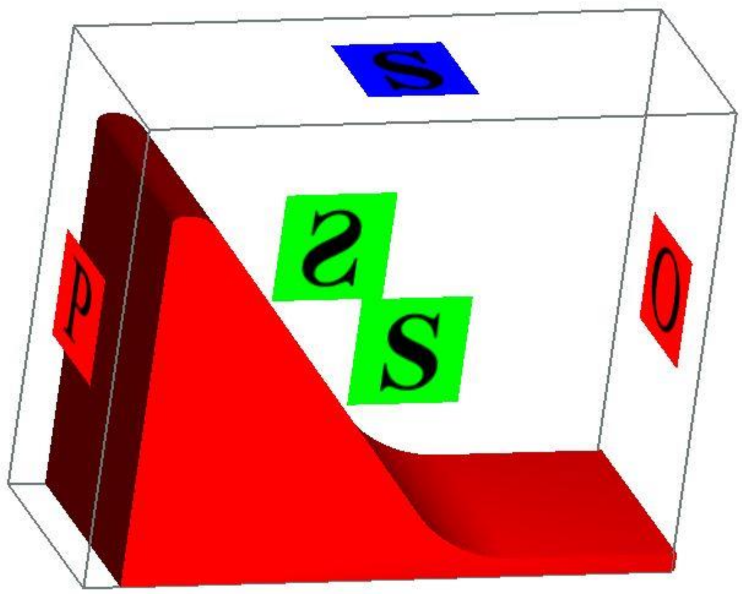

Figure 3.13 Boundary conditions for Case-1

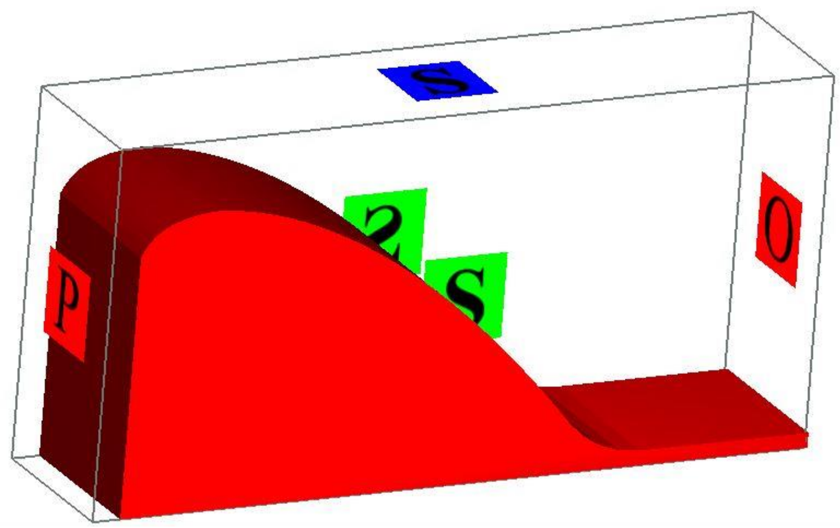

Figure 3.14 Boundary conditions for Case-2

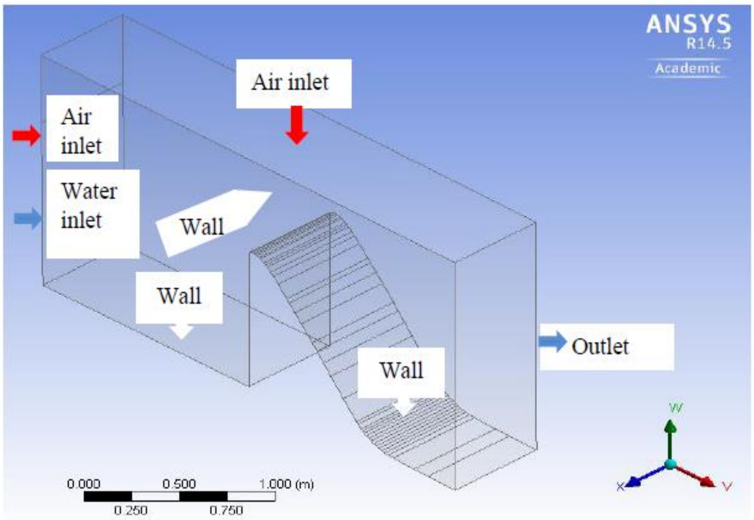

Figure 3.15 Geometry and boundary conditions in Ansys (Kanyabujinja, 2015)

For the initial condition, a fluid area is defined within the reservoir and it is located at the crest of spillway as shown in Figure 3.16. Due to the stagnation pressure conditions, the flow velocity is set to zero.

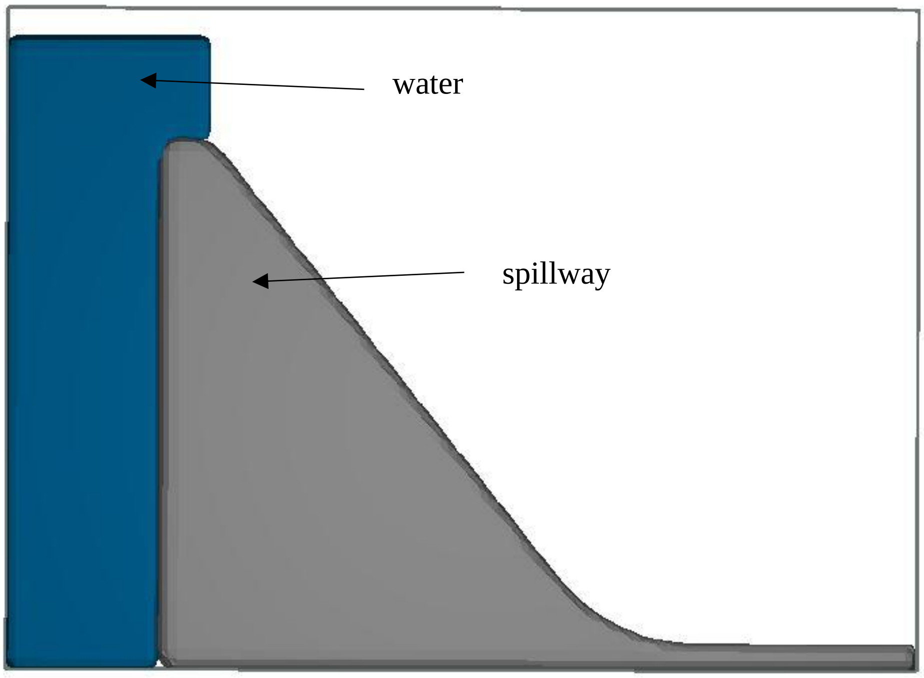

Figure 3.16 Initial condition in Case-1

CHAPTER 5

RESULT AND DISCUSSION

4.1 Introduction

In this study simulated results by the Flow-3D software of two different spillway models with different discharges are compared with laboratory results and also another numerical software Ansys to assess the precision of the two numerical models in modeling the physical observations. The aim of this investigation is to evaluate the numerical simulations based on different models and discharges. In this part, results have presented the comparison of the two numerical and experimental results of pressure distribution along the spillway with different discharge. And also, pressure development with increasing discharge along the lower nappe of the spillway is presented the calculated pressures on the spillway surface from the experimental study and those computed from computational fluid dynamics (CFD) models (Flow-3D and Ansys) are being compared. After that, the effects of different discharges on the pressure distribution over the surface of spillway with two different shapes of spillway are presented. Surface profiles for each discharge are also illustrated for each model, besides the variations of velocity magnitude with different discharges and the shear stress distribution with increasing discharge. The relationship between velocity, pressure and shear stress are presented.

4.2 Comparison between observed and CFD results

4.2.1 Water-surface profile and flow surcharge

The water surface profiles in two models of spillway are studied along the nappe of the spillway from upstream to downstream as shown in Figure 5.1 to Figure 5.4. Water surcharge for different discharges in both physical and Ansys are similar to each other also the results from the Flow-3D are very close to the experimental and Ansys results.

It can be seen in Figure 4.1, Figure 4.2 for Case-1 and Figure 4.3, Figure 4.4 for Case-2 that, the surface profile and surcharge of water on spillway have a good agreement. It should be noted that the water surface profile on the spillway refers to the depth of flow perpendicular to the face of the spillway. The two maximum discharge profiles of 130 l/s and 117l/s are tested in Case-1 and Case-2 as shown in Figure 4.1 and Figure 4.3 respectively. The surface profiles for other flow rates are presented in Appendices.

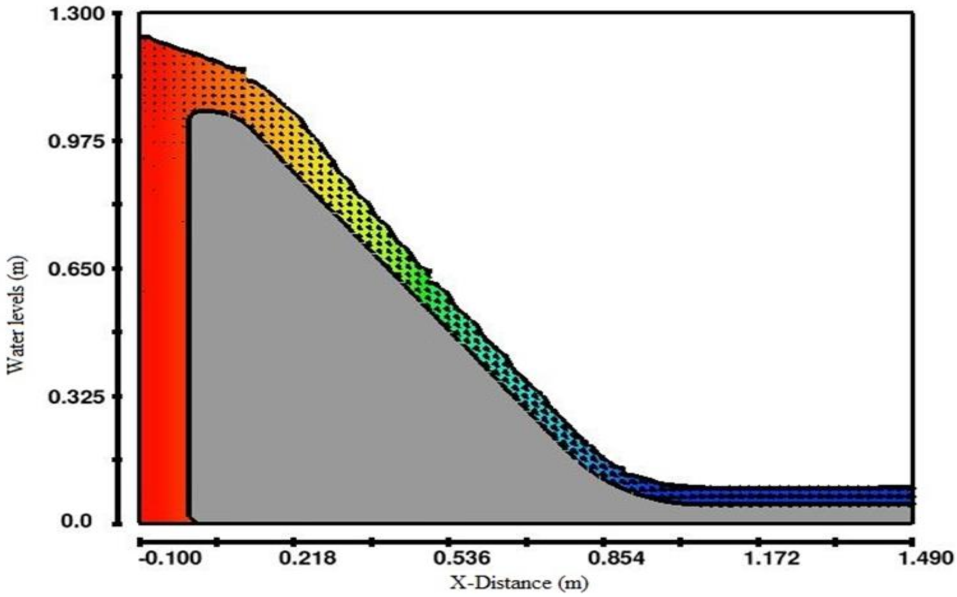

Figure 4.1 Surface profile of Case-1 simulated by Flow-3D

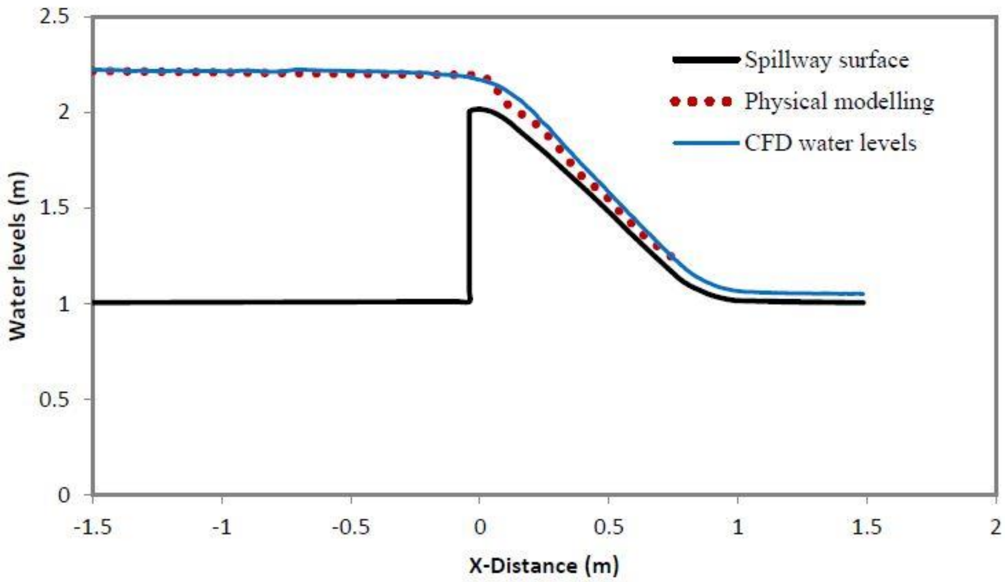

Figure 4.2 Comparison of surface profiles for observed and CFD models for Q = 130 l/s (Case-1) (Kanyabujinja, 2015).

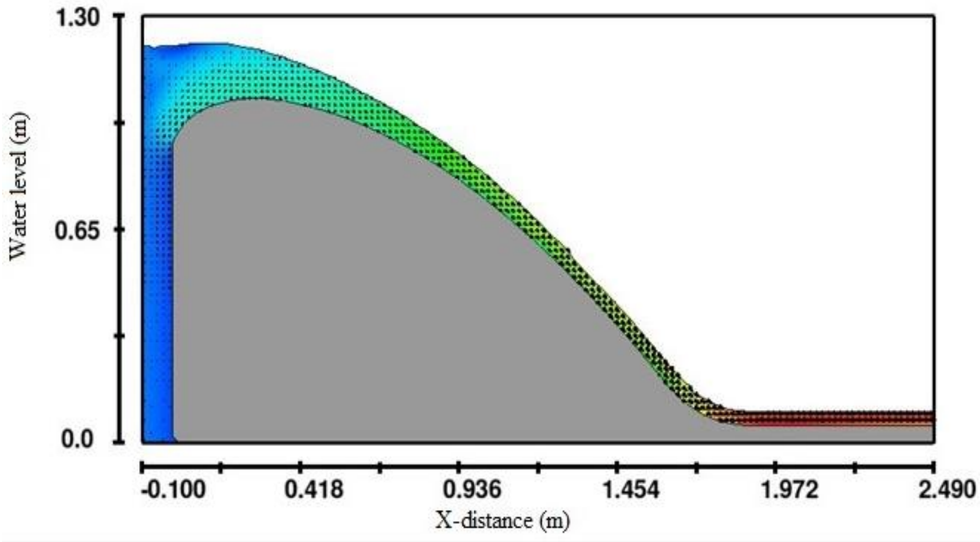

Figure 4.3 Surface profile of Case-2 simulated by Flow-3D

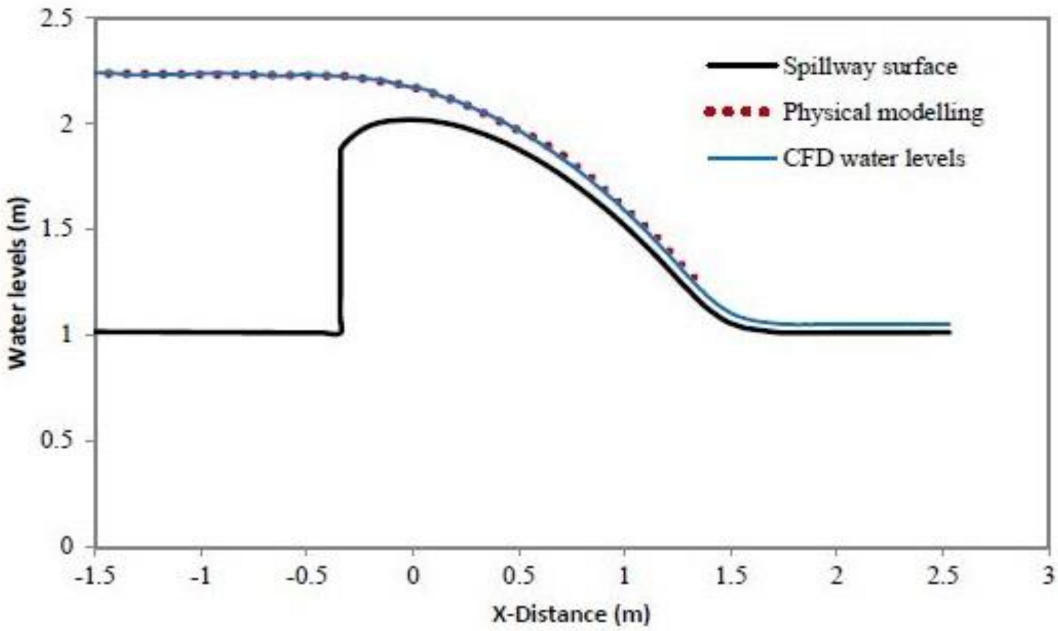

Figure 4.4 Comparison of surface profiles for observed and CFD models for Q = 130 l/s (Case-2) (Kanyabujinja, 2015).

The results of the flow surcharge simulation are presented in Table 4.1 and Figure 4.5 for Case-1 the surcharges obtained from observed and CFD modeling for Flow-3D and Ansys with comparison each other with observed modeling, the simulation of CFD models are very close to observed and have a good similarity between the observed model and CFD surcharge measurements. The maximum difference between observed and Ansys results which were found corresponding to 14 mm, but the maximum difference between observed and Flow-3D was 8 mm. It can be said that the Flow-3D results are closer to observed than the Ansys software.

Table 4.1 Physical and CFD surcharge values for variable discharges Case-1

		1	2	3	(4)=(1)-(2)	(5)=(3)-(1)
No.	Discharge l/s	Physical water surcharge (mm)	Flow-3D water surcharge (mm)	Ansys water surcharge (mm)	Observed-Flow-3D (mm)	Observed - Ansys (mm)
0	0	0	0	0	0	0
1	23	70	64	78	6	8
2	35	86	81.6	97	4.4	11
3	41	98	90	106	8	8
4	56	120	113.3	125	6.7	5
5	71	137	130.2	149	6.8	12
6	89	156	150	169	6	13
7	108	175	170	187	5	12
8	130	197	190	211	7	14

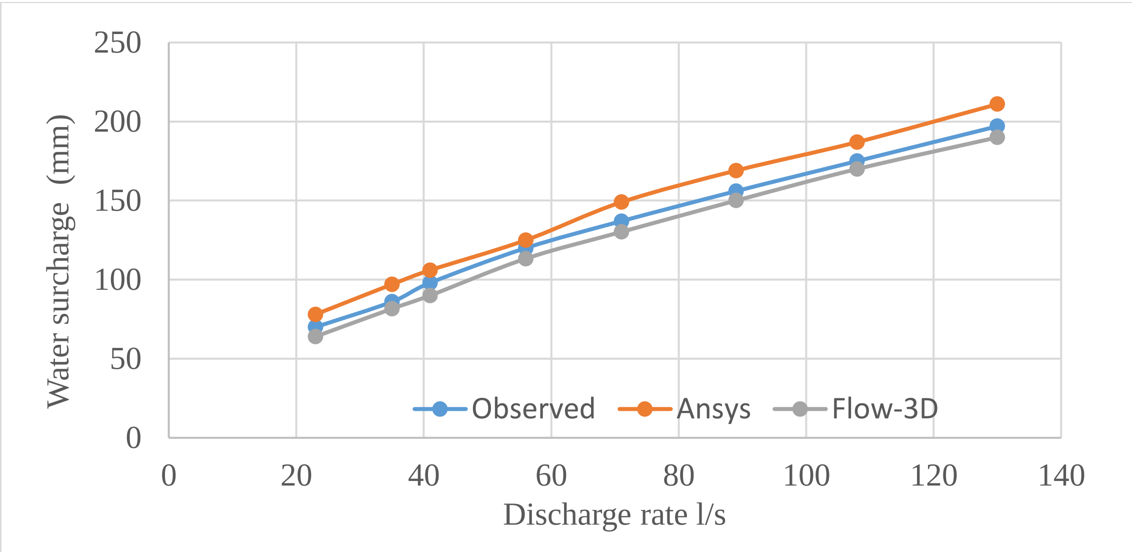

Figure 4.5 Observed surcharge comparison with Flow-3D and Ansys (Case-1)

In Table 4.2 and Figure 4.6, it can be seen that at initial points Flow-3D predictions are very close to observed data. Ansys predictions are closer to the observed data compared to Flow-3D. Overall predictions of Flow-3D are close to both observed and Ansys predicted data.

Table 4.2 Physical and CFD surcharge values for variable discharges (Case-2)

No.	Discharge l/s	1 Pysical water surcharge (mm)	2 Flow-3D water surcharge (mm)	3 Ansys water surcharge (mm)	(4)=(1)-(2) Observed-Flow-3D (mm)	(5)=(3)-(1) Observed - Ansys (mm)
0	0	0	0	0	0	0
1	25	79	77	82	2	3
2	37	104	105	112	-1	8
3	45	118	108	125	10	7
4	51	128	115	137	13	9
5	77	166	141	171	25	5
6	85	179	147	187	32	8
7	95	192	170	203	22	11
8	117	218	210	223	8	5

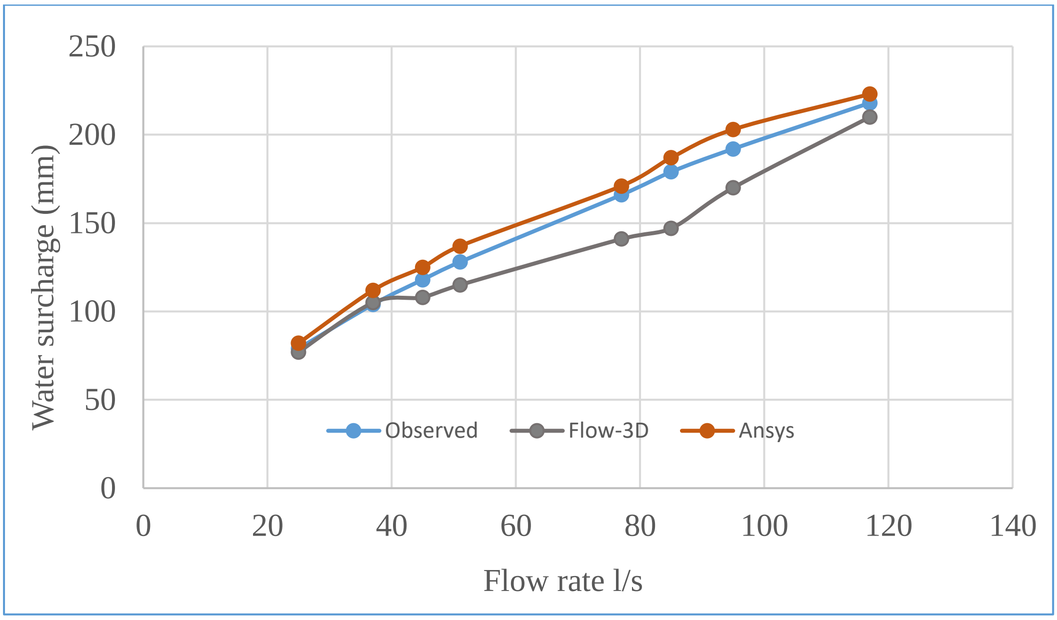

Figure 4.6 Observed surcharge compared with Flow-3D and Ansys simulation (Case-2)

4.2.2 Pressure distribution for Case-1

Table 4.3 presented the comparison of the results for each experimental and CFD (Flow-3D, Ansys) data for different points in variable discharges, and variation of their pressure along the spillway nappe. The results are displayed that, Flow-3D has good agreement observed and numerical simulations, and in the most trail, the Flow-3D results are between the observed and Ansys results.

Table 4.3 Physical and CFD pressure values for variable discharges Case-1.

Q (l/s)	Sensors	1	2	3	4	5	6	7
	Physical and numerical sensor pressure values (m)							
23	observed	0.017	0.005	0.009	0	-0.037	-0.049	-0.025
	Ansys	0.022	0.003	0.014	0.01	0.008	-0.003	0.01
	Flow 3D	0.021	-0.007	0.011	0.005	-0.018	-0.011	-0.003
35	observed	0.032	0.004	0.011	-0.002	-0.035	-0.055	-0.022
	Ansys	0.02	0.003	0.018	0.015	0.01	-0.005	0.014
	Flow 3D	0.012	-0.01	0.028	0.01	-0.025	-0.047	-0.0003
41	observed	0.028	0.003	0.006	0.001	-0.037	-0.054	-0.029
	Ansys	0.017	0.001	0.019	0.017	0.012	-0.005	0.016
	Flow 3D	0.006	-0.014	0.026	0.026	-0.0004	-0.025	0.019
56	observed	0.012	-0.003	0.009	0.005	-0.03	-0.055	-0.018
	Ansys	0.007	-0.004	0.021	0.023	0.015	-0.007	0.021
	Flow 3D	0.006	-0.023	0.003	0.03	0.007	-0.053	0.044
71	observed	-0.005	-0.008	0.008	0.005	-0.029	-0.052	-0.015
	Ansys	-0.024	-0.018	0.019	0.035	0.022	-0.007	0.032
	Flow 3D	-0.009	-0.033	0.02	0.021	0.007	-0.034	0.045
89	observed	-0.029	-0.026	0.006	0.014	-0.022	-0.049	-0.01
	Ansys	-0.024	-0.018	0.019	0.035	0.022	-0.007	0.032
	Flow 3D	-0.023	-0.043	0.015	0.034	0.017	-0.003	0.041
108	observed	-0.056	-0.034	0.002	0.019	-0.019	-0.056	-0.012
	Ansys	-0.047	-0.028	0.017	0.041	0.027	-0.006	0.039
	Flow 3D	-0.037	-0.052	0.009	0.043	0.056	0	0.038
130	observed	-0.094	-0.05	-0.006	0.02	-0.017	-0.04	-0.004
	Ansys	-0.072	-0.039	0.013	0.047	0.031	-0.003	0.045
	Flow 3D	-0.057	-0.08	-0.00002	0.049	0.036	0	0.043

From Figure 4.7 to Figure 4.14 it can be seen that the magnitudes of pressure which computed by Flow-3D are in the acceptable range if compared with Physical data. However, the Flow-3D results are not completely similar to the observed and Ansys data but it gives the same conclusion for changing the pressure in different points and the variation of pressure due to the changing of discharges.

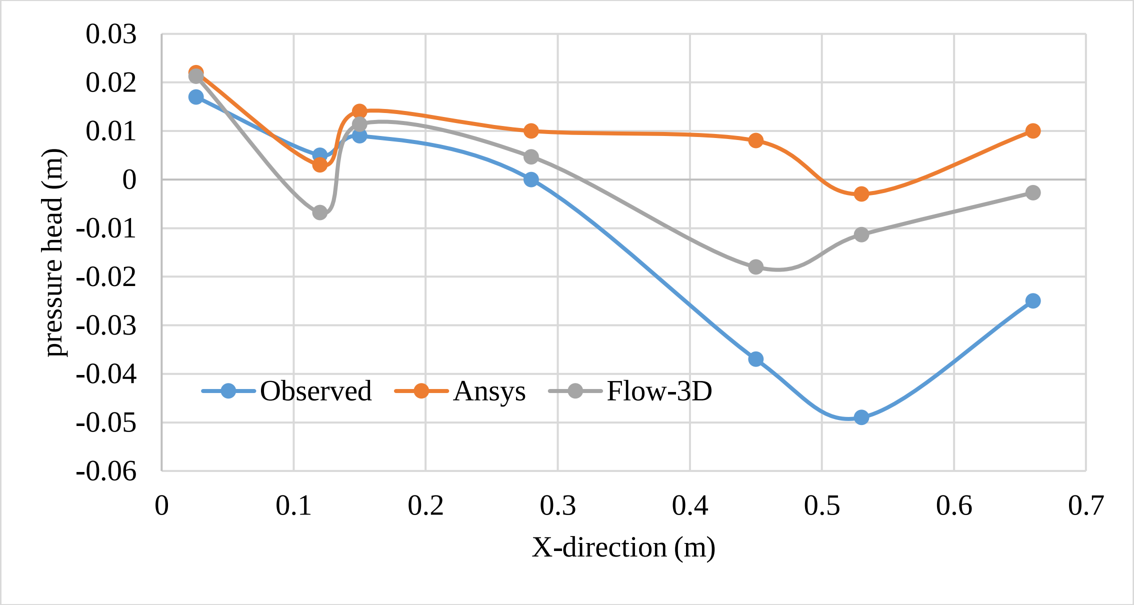

Figure 4.7 Comparison between observed and CFD results for Q = 23 l/s

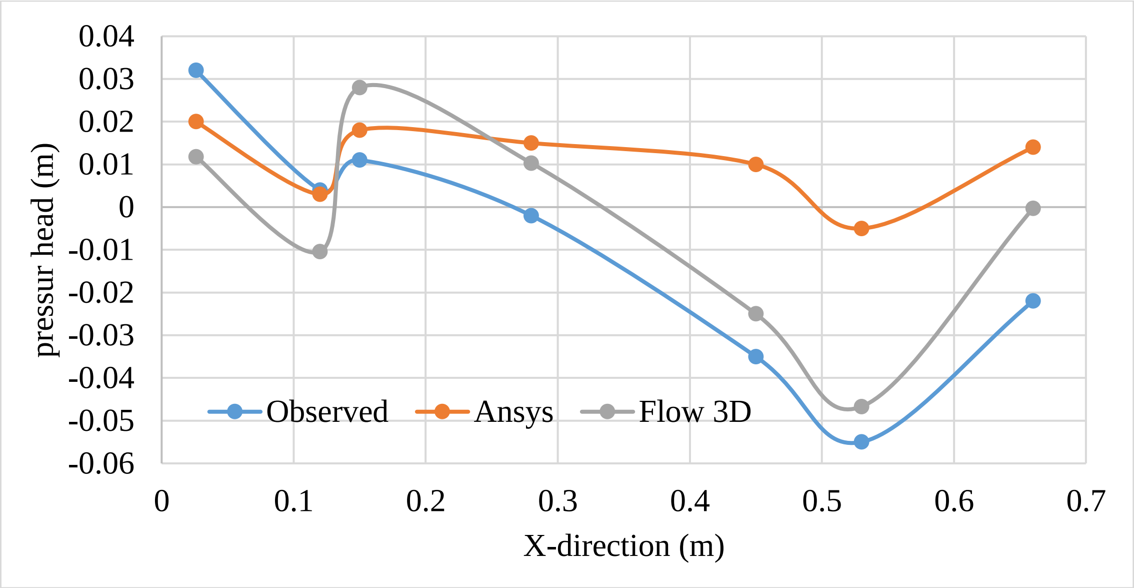

Figure 4.8 Comparison between observed and CFD results for Q = 35 l/s

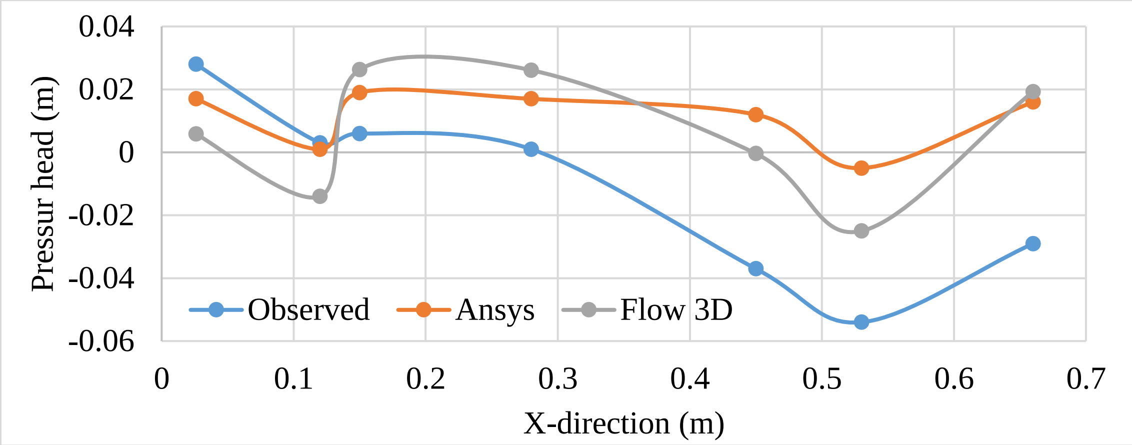

Figure 4.9 Comparison between observed and CFD results for Q = 41 l/s

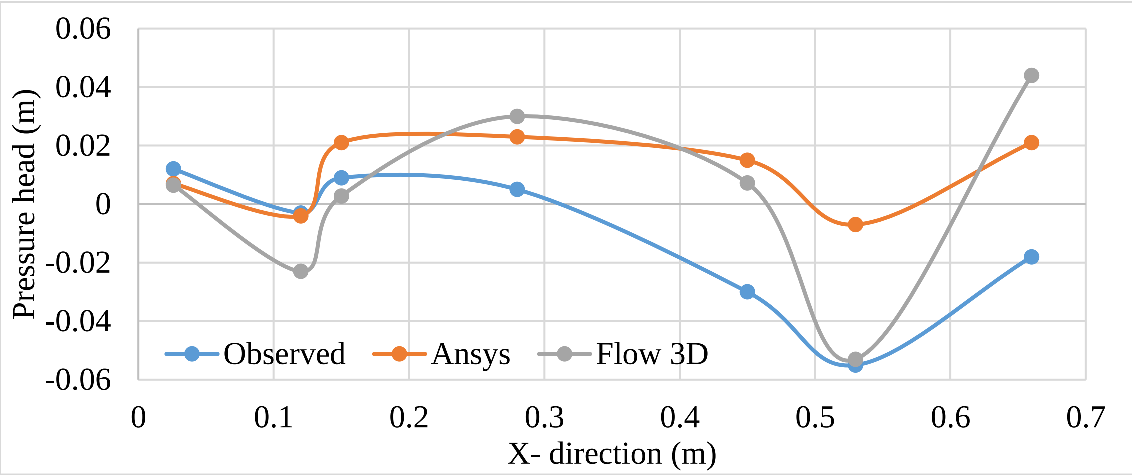

Figure 4.10 Comparison between observed and CFD results for Q = 56 l/s

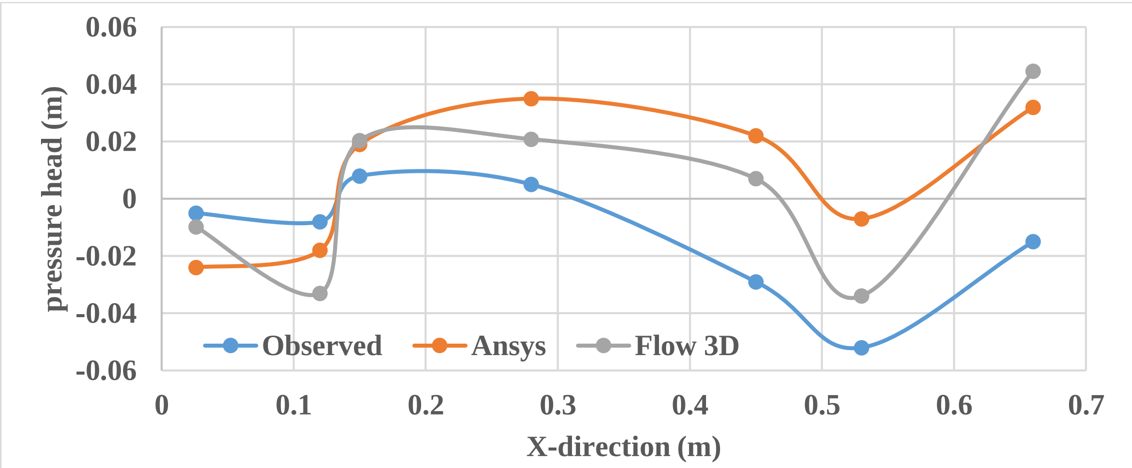

Figure 4.11 Comparison between observed and CFD results for Q = 71 l/s

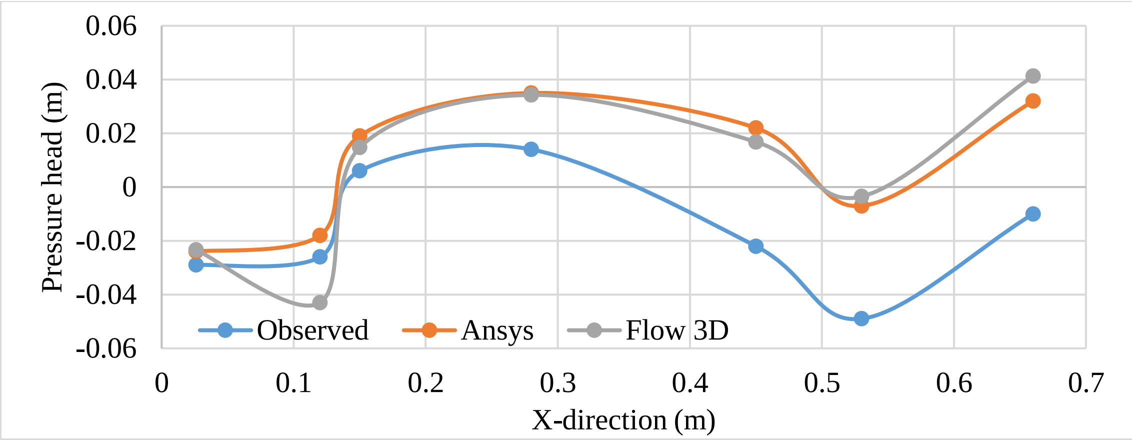

Figure 4.12 Comparison between observed and CFD results for Q = 89 l/s

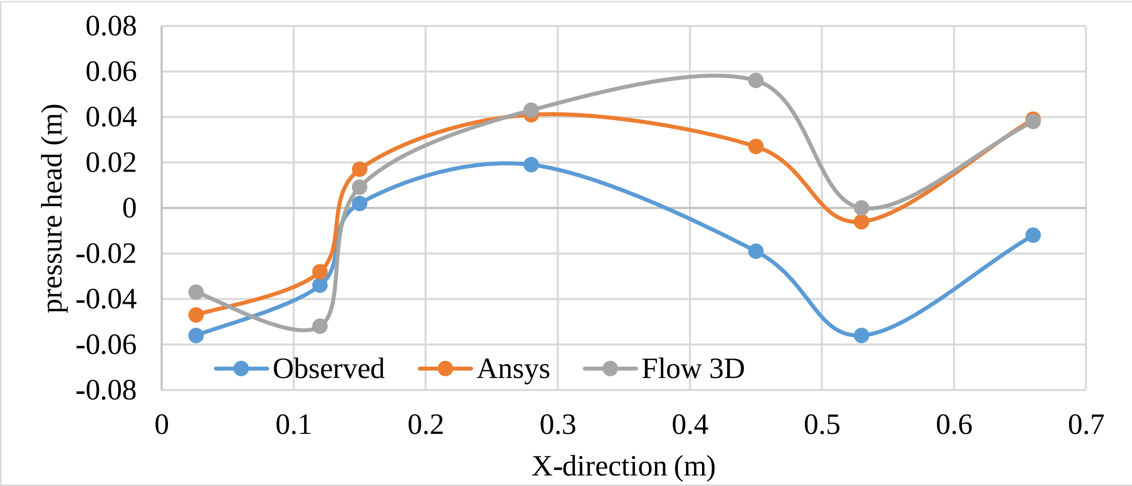

Figure 4.13 Comparison between observed and CFD results for Q = 108 l/s

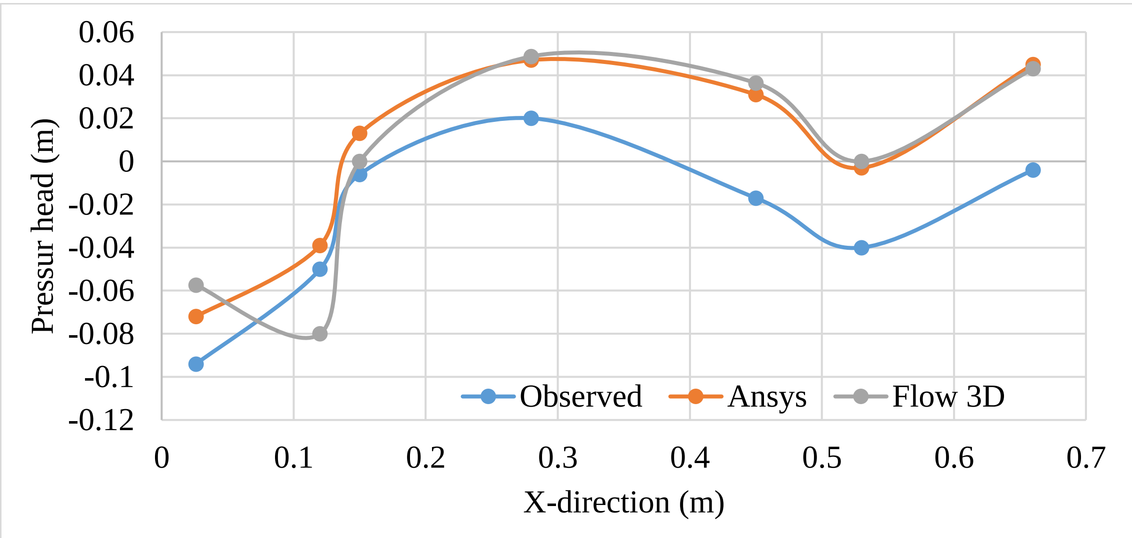

Figure 4.14 Comparison between observed and CFD results for Q = 130 l/s

The experimental and numerical data in Table 4.4 and Figures 4.7 to Figure 4.14 show the pressure distribution on the spillway lower nappe.

In Table 4.4 and Figure 4.15, it is clear that with the increasing discharge the pressure distribution is decreased and it is also clear that the negative pressure can be seen in two regions; the first region is located at the ogee curve and the second region at the end of the sloping line after ogee curve. Depending on the Flow-3D data in Table 4.4 it can be denoted that in the first region the negative readings reduce with increasing the discharges and vice versa.

Table 4.4 Pressure distribution along the spillway for variable discharge by Flow-3D Case-1

X-direction	sensor 1	sensor 2	Sensor 3	sensor 4	sensor 5	sensor 6	sensor 7
	0.026	0.12	0.15	0.28	0.45	0.53	0.66
Discharge l/s	Pressure distribution						
23	0.021	-0.007	0.011	0.005	-0.018	-0.011	-0.003
35	0.012	-0.0104	0.028	0.0103	-0.025	-0.047	-0.0003
41	0.0058	-0.014	0.0263	0.0261	-0.0004	-0.025	0.019
56	0.006	-0.023	0.003	0.03	0.007	-0.053	0.044
71	-0.01	-0.033	0.0203	0.0208	0.007	-0.034	0.044
89	-0.023	-0.043	0.015	0.034	0.017	-0.0035	0.0413
108	-0.037	-0.052	0.009	0.043	0.056	0	0.038
130	-0.057	-0.08	-0.00002	0.049	0.036	0	0.043

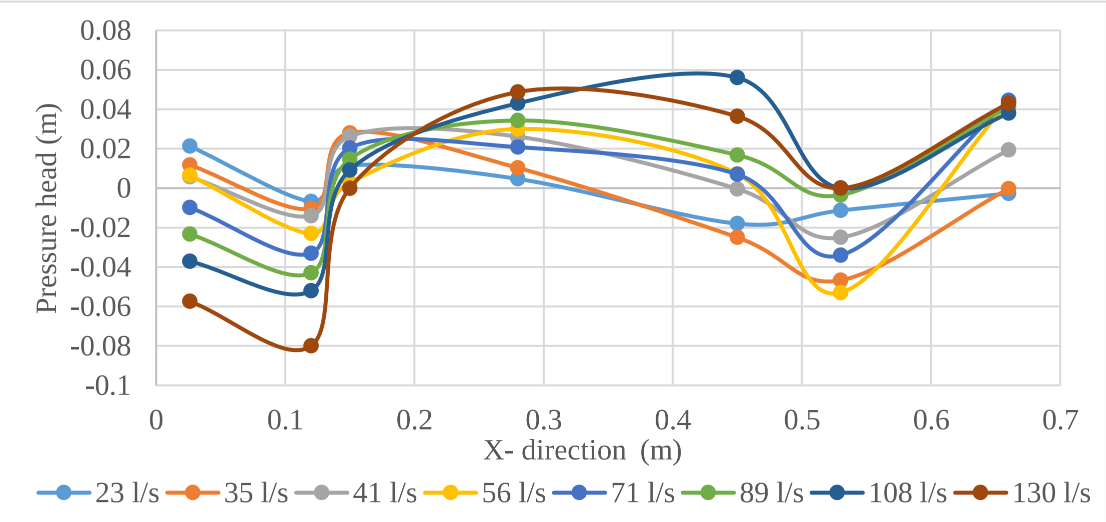

Figure 4.15 pressure distribution along the spillway for variable discharges by Flow-3D Case-1

Figure 4.16 and Figure 4.17 present the pressure results obtained from experimental modeling and Flow-3D modeling respectively for Case-1, for all seven points, and for all discharges tested. The sensors situated at the crest of ogee spillway (Sensors 1 and 2) show the decrease of pressure from positive to negative value. This pressure reduction is proportional to the increase of discharge. The greatest negative pressure showed in the results is -0.094 m documented from sensor1in physical and -0.08 from sensor 2 in Flow-3D for a discharge of 130 l/s.

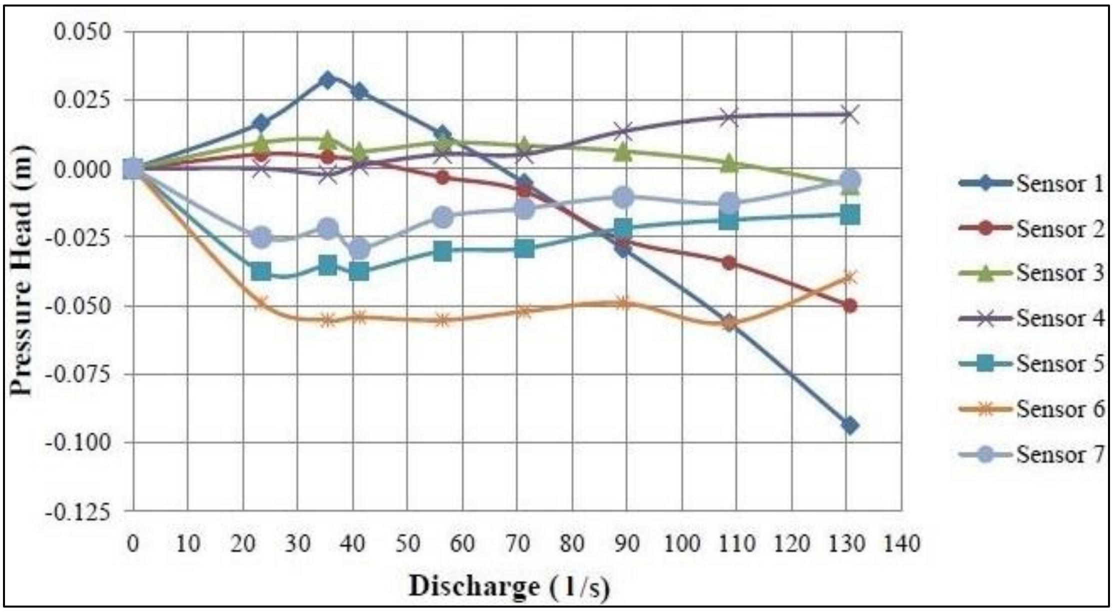

Figure 4.16 The pressure readings obtained during physical testing of Case-1

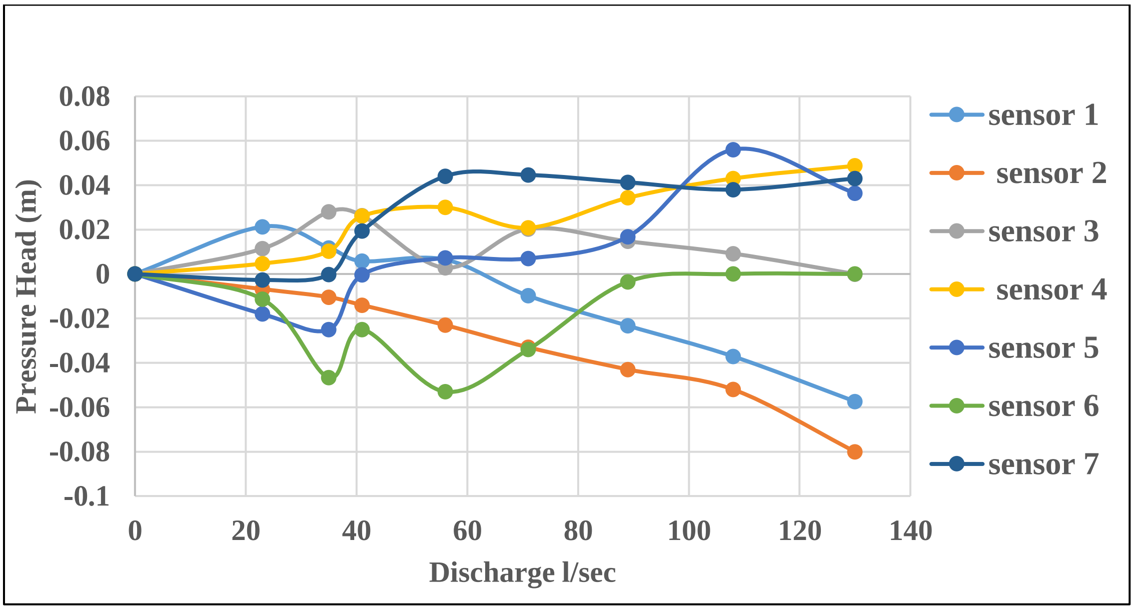

Figure 4.17 The pressure readings obtained by Flow-3D of Case-1

4.2.3 Pressure distribution for Case-2

Table 4.5 and Figure 5.18 to Figure 5.25 present the comparison of the results for each experimental and CFD (Flow-3D, Ansys) at seven points for variable discharges and variation of their pressure along the spillway nappe, due to increased discharge from 25 l/s to 117 l/s. The data are displayed that Flow-3D results are very similar to Ansys results and generally have a good agreement with observed results but in the negative pressure regions, Flow-3D results are closer to experimental if comparing with Ansys.

Table 4.5 Physical and CFD pressure values for variable discharges Case-2.

Q l/s	Sensors	1	2	3	4	5	6	7
	Physical and numerical sensor pressure values (m)							
25	Observed	0.09	0.062	0.025	0.005	-0.017	0.037	0.068
	Ansys	0.086	0.047	0.016	0.008	0.003	0.005	0.019
	Flow 3D	0.093	0.052	0.0246	0.0096	0	0.012	0.0151
37	Observed	0.106	0.083	0.03	0.003	-0.024	0.037	0.07
	Ansys	0.106	0.064	0.025	0.012	0.003	0.007	0.029
	Flow 3D	0.103	0.0609	0.046	0.0092	-0.0095	0.0207	0.032
44	Observed	0.119	0.108	0.052	0.022	-0.016	0.096	0.095
	Ansys	0.115	0.071	0.03	0.014	0.003	0.008	0.033
	Flow 3D	0.12	0.081	0.0305	0.0183	-0.0073	-0.0039	0.0452
51	Observed	0.125	0.114	0.055	0.023	-0.022	0.096	0.097
	Ansys	0.132	0.08	0.04	0.018	0.004	0.01	0.037
	Flow 3D	0.13	0.0864	0.0411	0.0259	0	0.021	0.0366
78	Observed	0.148	0.134	0.064	0.032	-0.032	0.099	0.099
	Ansys	0.143	0.094	0.044	0.022	0.005	0.012	0.051
	Flow 3D	0.135	0.0936	0.0627	0.0306	-0.0064	0.0214	0.0231
85	Observed	0.155	0.141	0.073	0.035	0.003	0.1	0.101
	Ansys	0.149	0.099	0.048	0.023	0.005	0.012	0.056
	Flow 3D	0.1387	0.0963	0.0644	0.0303	0	0.0247	0.0246
95	Observed	0.163	0.147	0.077	0.038	0.004	0.103	0.102
	Ansys	0.154	0.104	0.051	0.025	0.005	0.013	0.059
	Flow 3D	0.1277	0.1051	0.0549	0.027	0	0.0243	0.0177
117	Observed	0.176	0.16	0.084	0.043	0.006	0.105	0.106
	Ansys	0.167	0.115	0.058	0.029	0.006	0.015	0.069
	Flow 3D	0.0996	0.0961	0.0705	0.04	-0.003	0.023	0.0247

In Table 5.5 and Figure 4.18 to Figure 4.25 it is clear that the pressure values which computed by Flow-3D are in the style. However, the Flow-3D results are not completely similar to the observed, but it gives the same conclusion, and it has a good similarity with Ansys and observed.

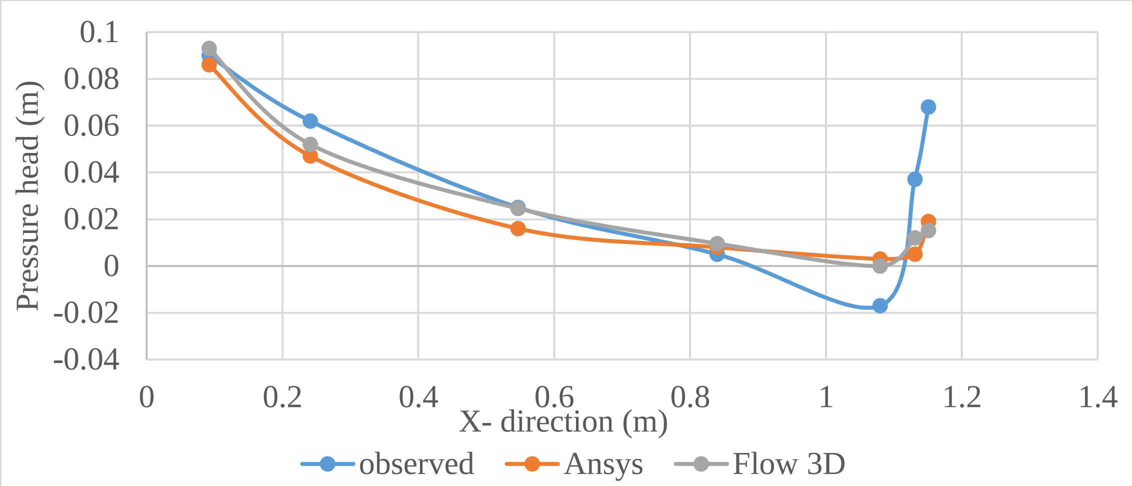

Figure 4.18 Comparison between observed and CFD results for Q = 25 l/s

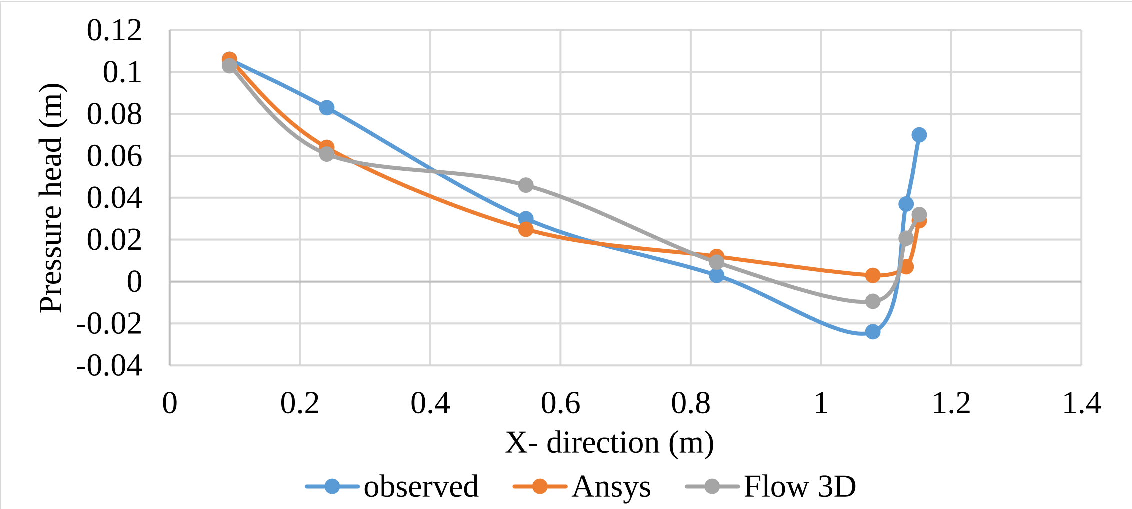

Figure 4.19 Comparison between observed and CFD results for Q = 37 l/s

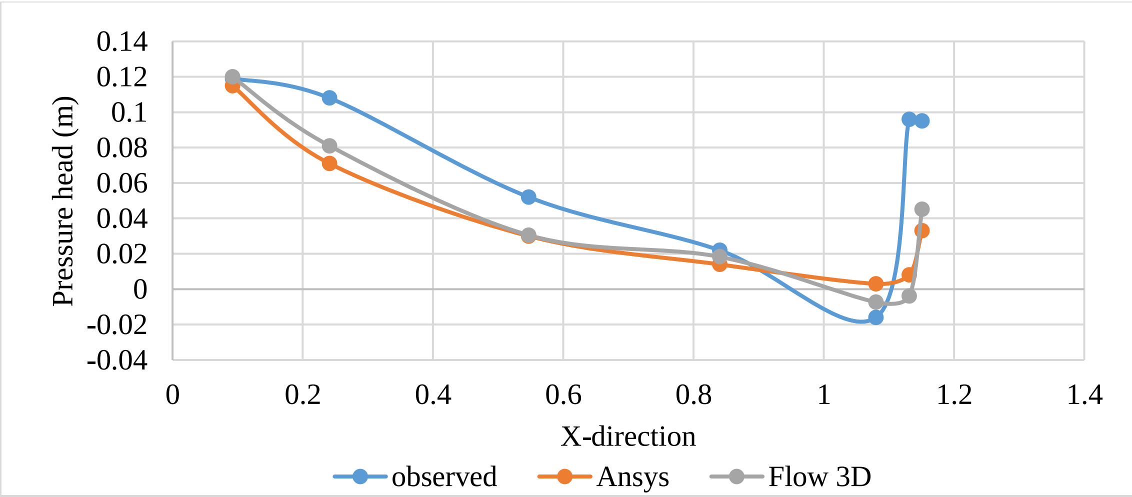

Figure 4.20 Comparison between observed and CFD results for Q = 44 l/s

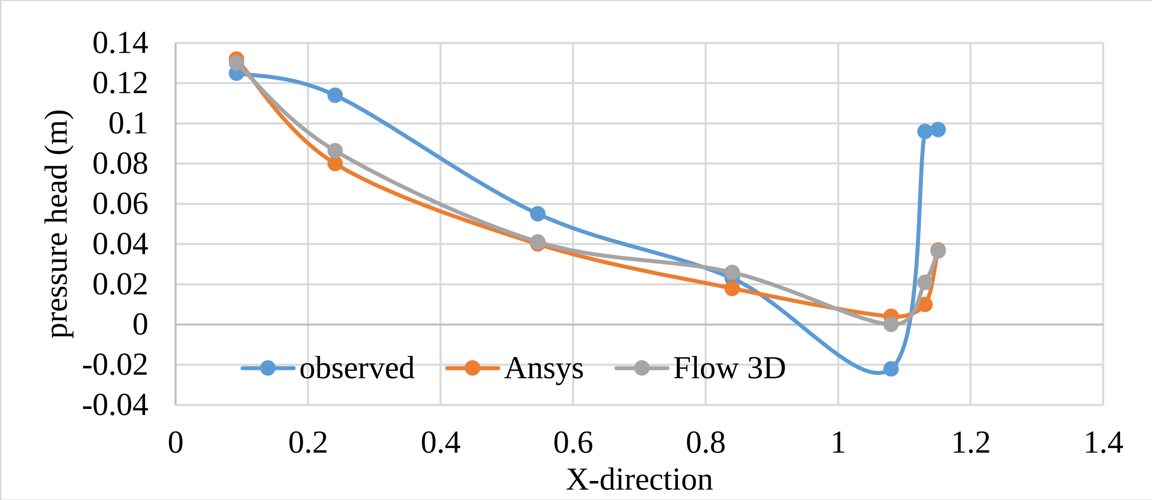

Figure 4.21 Comparison between observed and CFD results for Q = 51 l/s

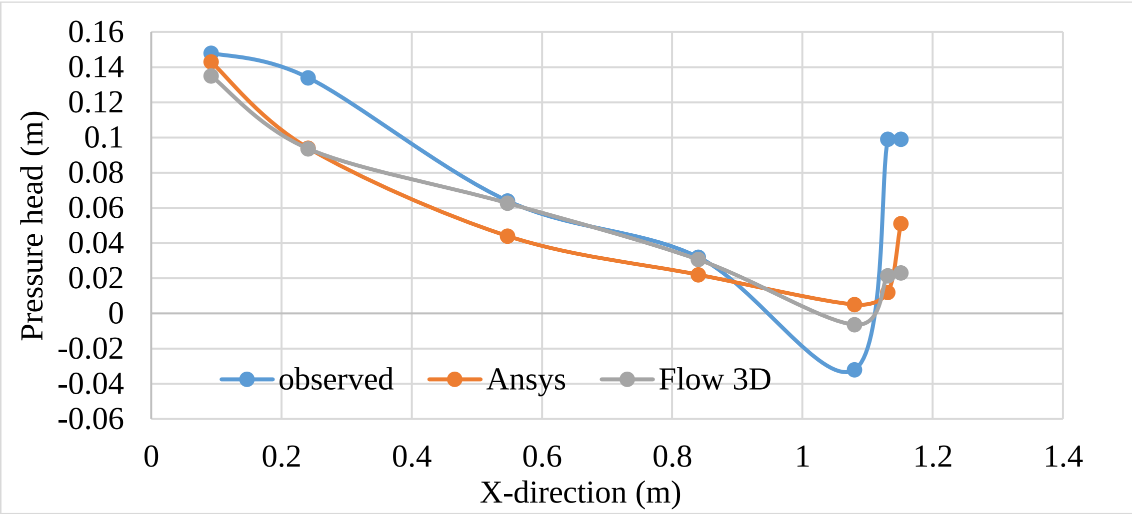

Figure 4.22 Comparison between observed and CFD results for Q = 78 l/s

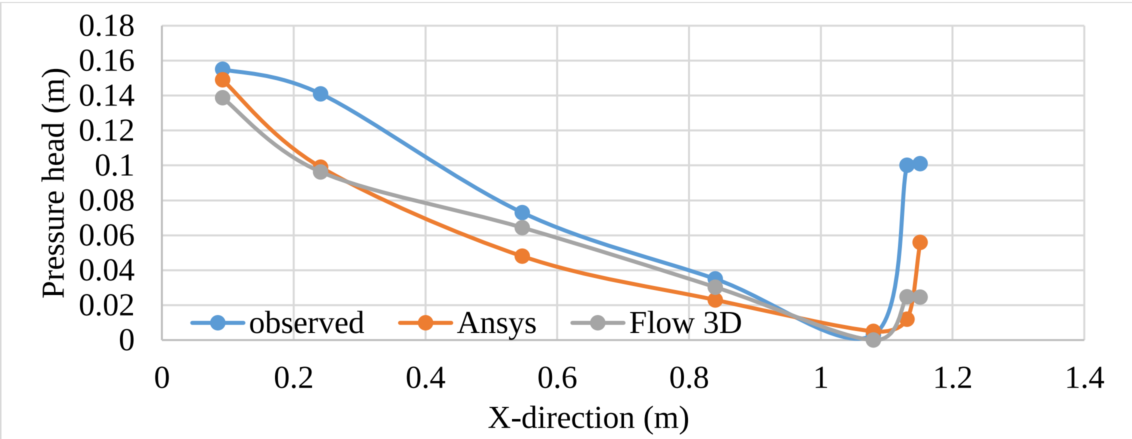

Figure 4.23 Comparison between observed and CFD results for Q = 85 l/s

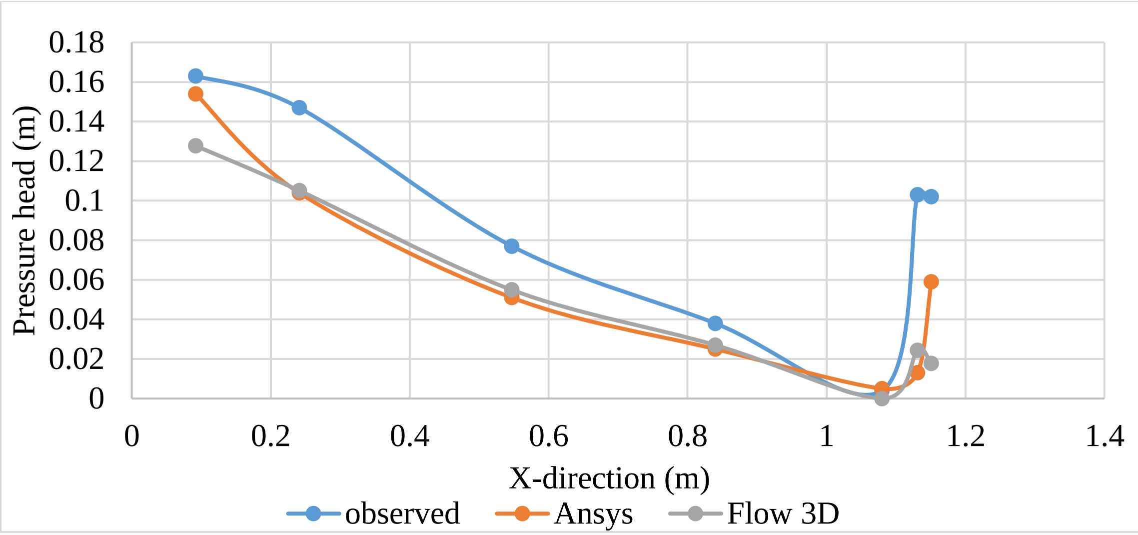

Figure 4.24 Comparison between observed and CFD results for Q = 95 l/s

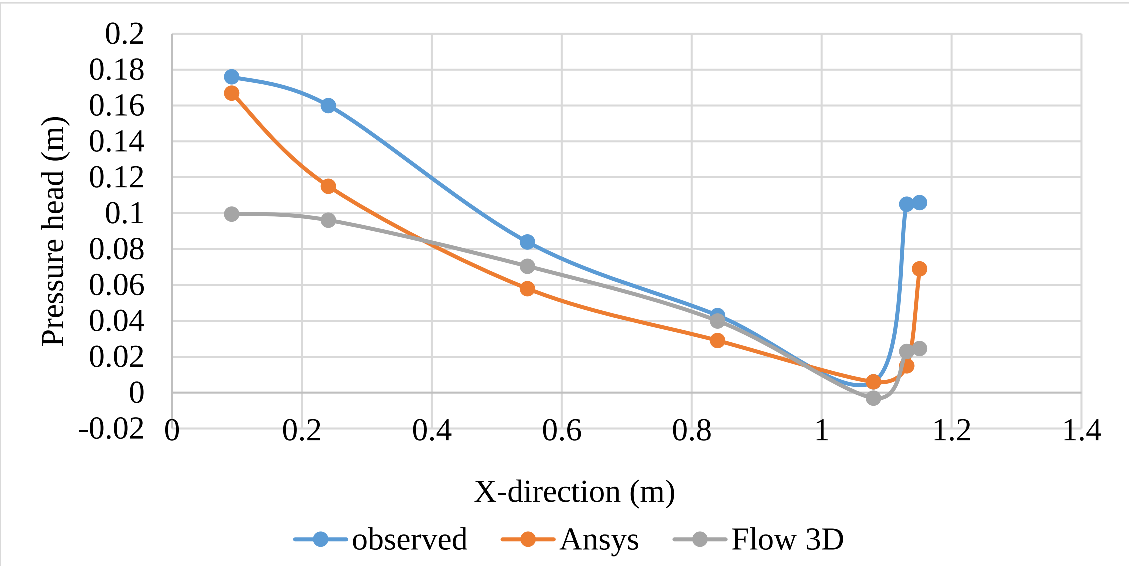

Figure 4.25 Comparison between observed and CFD results for Q = 117 l/s

For experimental and numerical data shown in Table 4.6 and Figures 4.26 for distribution pressure along the spillway lower nappe, this can be improved by application of Computational Fluid Dynamics (CFD) models and can be observed that the pressure which obtained in physical, Ansys and Flow-3D modellings are very close together, however at the end of the spillway the observed model has some differences with CFD models but generally all models in the same direction, also the Ansys results are very close to the Flow-3D results because both of them are CFD software and used same empirical formula. In Case-2 it can be denoted that the pressure changes proportionally with increasing the discharge on the points located on the spillway, except to last points as shown in Table 4.6 and Figure 4.26, but it is not true for Case-1 due to change in the spillway profile.

Table 4.6 Pressure distribution along the spillway for variable discharge by Flow-3D Case-2

	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
X-distance	0.092	0.241	0.547	0.84	1.08	1.131	1.151
Discharge l/s	Pressure distribution over the spillway (m)						
25	0.093	0.052	0.025	0.009	0	0.012	0.015
37	0.103	0.061	0.057	0.009	-0.009	0.021	0.032
44	0.12	0.081	0.03	0.018	-0.007	0.059	0.045
51	0.13	0.086	0.041	0.026	0	0.021	0.036
78	0.135	0.093	0.063	0.03	-0.006	0.0214	0.023
85	0.139	0.096	0.064	0.03	0	0.025	0.024
95	0.128	0.105	0.055	0.027	0	0.024	0.017
117	0.099	0.096	0.07	0.04	-0.003	0.023	0.011

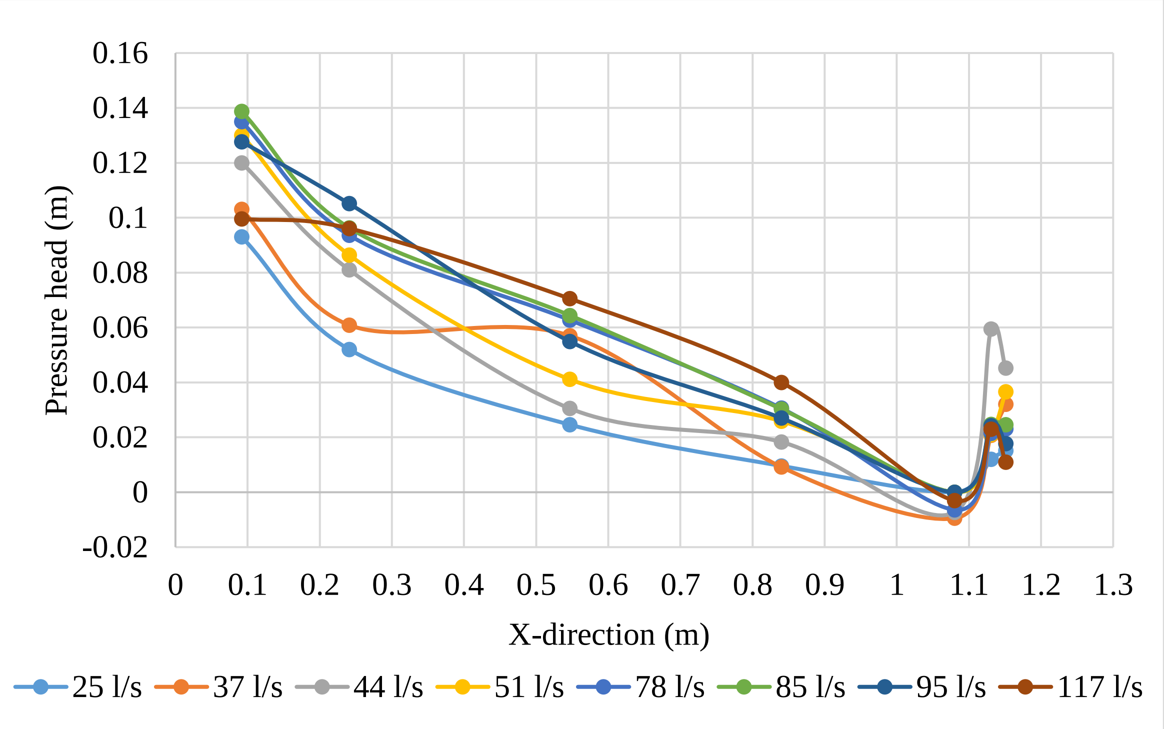

Figure 4.26 Simulated pressure distribution along the spillway for variable discharges by Flow-3D Case-2

Figure 4.27 and Figure 4.28 show the physical and Flow-3D results respectively, and they are the good representation of the pressure increase on the ogee spillway for the variable discharges. From physical and Flow-3D simulations summary graph shown the pressures on the spillway increase with the increase of discharge, with some fluctuations.

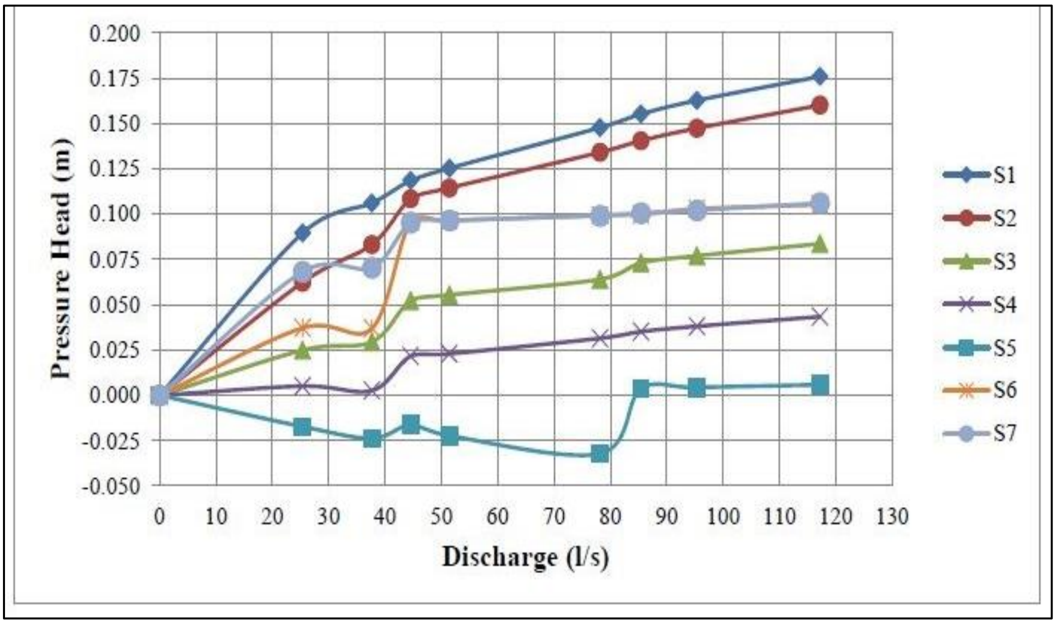

Figure 4.27 pressure readings obtained during testing of Case-2

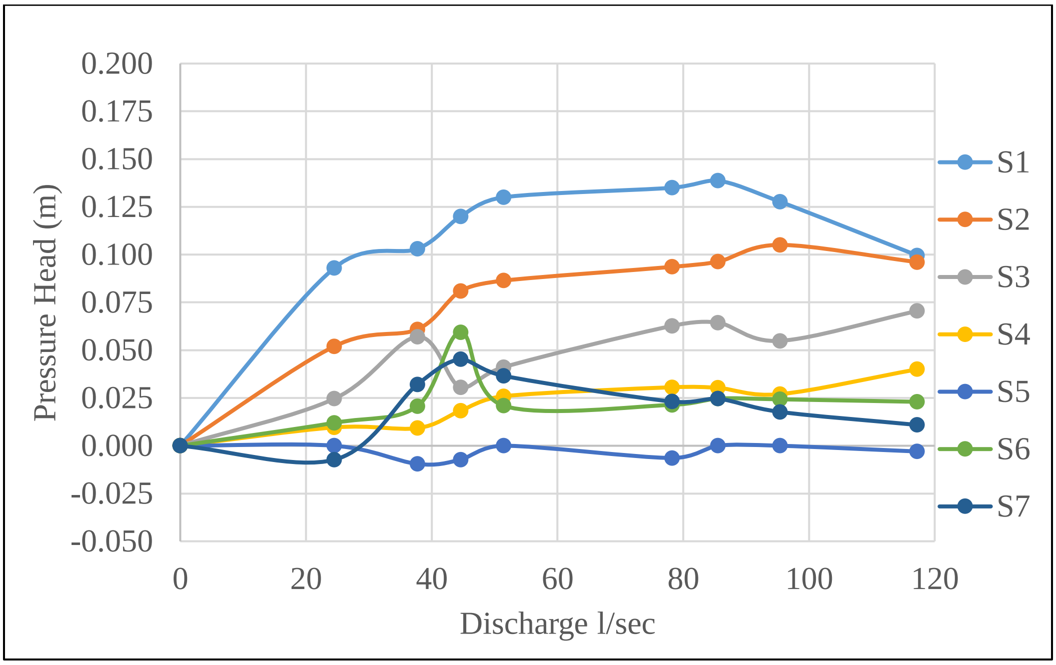

Figure 4.28 Pressure readings obtained by Flow-3D for Case-2

4.2.4 Velocity

There is a strong relationship between pressure and velocity. Therefore, it is very important to showing velocity with pressure, for the better illustration The relationship between velocity and pressure for incompressible flow (constant fluid density) is given by equation below (Bernoulli's Law).

$$P + \frac{1}{2}\rho V^2 = constant$$

Where V is flow velocity, ρ is density, and P is pressure. The two expressions are also commonly called static pressure (P) and dynamic pressure ($\frac{1}{2}\rho * V^2$). These two quantities must always add up to the same value. This means that an increase in velocity causes a decrease in static pressure, as shown in Figure 4.29.

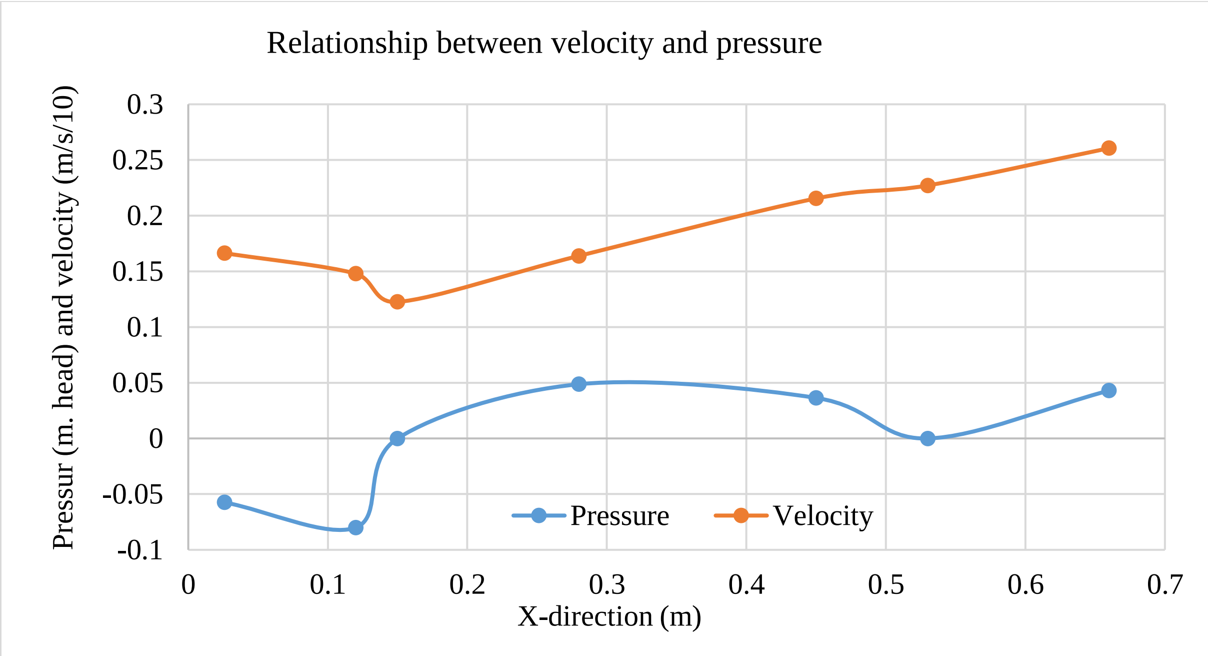

Figure 4.29 Relationship between velocity and pressure for flow rate of 130 l/s Case-1 by Flow-3D

4.2.5 Velocity in Case-1

In Table 4.7 it can be seen that velocity was increased with increasing discharge, and for the same spillway profile in variable discharges has similar velocity profiles, also the velocity increase along the spillway, from upstream to downstream gradually.

Table 4.7 Velocity values along the spillway for variable discharges (m)

Q l/s	Sensor 1	Sensor 2	Sensor 3	Sensor 4	Sensor 5	Sensor 6	Sensor 7
0	0	0	0	0	0	0	0
23	0.666	0.966	0.958	1.511	1.944	2.071	2.286
35	0.756	0.991	0.942	1.534	1.899	1.972	2.250
41	0.800	1.007	0.953	1.548	1.932	1.985	2.291
56	1.043	1.203	1.029	1.587	2.086	2.148	2.456
71	1.395	1.452	1.087	1.615	2.129	2.258	2.588
89	1.404	1.552	1.094	1.646	2.150	2.256	2.579
108	1.523	1.494	1.171	1.653	2.157	2.174	2.542
130	1.664	1.479	1.226	1.639	2.155	2.271	2.606

4.2.6 Velocity in Case-2

Same as Case-1 in Table 4.8 it can be seen that velocity was increased with increasing discharge, and for the same spillway profile in variable discharges has similar velocity profiles, and along the spillway, the velocity increased due to gravity.

Table 4.8 Velocity values along the spillway for variable discharges.

Sensors	1	2	3	4	5	6	7
X-distance (m)	0.092	0.241	0.547	0.840	1.080	1.131	1.151
Discharge l/s	X-velocity distribution (m/s)						
25	0.332	0.704	0.729	1.059	1.566	2.296	2.649
37	0.634	0.870	1.132	2.003	2.436	2.752	2.907
45	0.631	0.896	1.590	2.134	2.564	2.850	2.942
51	0.730	0.941	1.590	2.138	2.543	2.830	2.966
77	1.062	1.178	1.746	2.236	2.562	2.790	2.989
85	1.146	1.246	1.782	2.249	2.595	2.724	2.918
95	1.260	1.360	1.779	2.276	2.598	2.819	3.005
117	1.532	1.577	1.949	2.355	2.652	2.835	3.028

4.3 Shear stress distribution over the spillway face

When a fluid is moving the shear stresses are occurred, during the motion, the particles of the fluid move relative to one another. When this occurs, adjacent particles have different velocities, due to changing velocity from the wall of channel or pipe to the center the shear stress occurred. If the velocity of fluid remains the same at every point, then there is no shear stress produced and the relative velocity of particles is equal to zero. At the wall of pipe or channel, the velocity of the water will be zero and the velocity will increase toward the center of the pipe or surface of the channel. Therefore there is a relationship between shear stress and velocity of the fluid and with increasing velocity, the shear stress will increase.

There is a strong relationship between velocity, and shear stress, therefore in hydraulic investigations, it is very important to study, shear stress with velocity, as shown the relationship between velocity and shear stress in Figure 4.30.

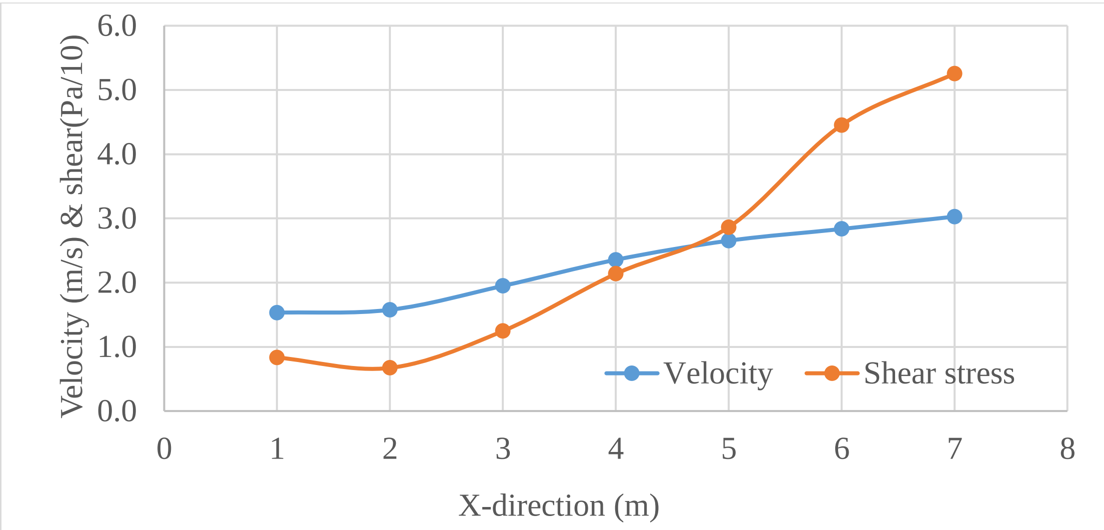

Figure 4.30 Relationship between velocity and shear stress in X-direction for Q=117 l/s, Case-2 by Flow -3D

CHAPTER 6

CONCLUSION

The numerical models developed with developing of computer technology and now it has a big role in the designs. Numerical models can now provide cost-effective methods instead of historical design methods and can provide additional information that may not be obvious in the physical model tests, and numerical models are based on equations that describe the basic physics of a particular situation. The model must be verified against physical model experiment.

Validation is usually provided by comparing the results of the numerical model with the results of a physical model.

The flow characteristics of turbulent flow over an ogee spillway were numerically simulated using the Flow-3D tool. VOF technique and the $k-\varepsilon$ turbulent model were used as numerical model. The experimental results of Kanyabujinja (2015) were used to verify the numerical model.

The most important findings of this study can be summarized as follows:

- In CFD modeling, pressure, shear stress, velocity and water surcharge were recorded for a period of five minutes for two cases.
- Flow-3D can successfully model the water surface profile of spillway for various discharge compared to physical tests.
- In Case-1, negative pressure values were observed in the ogee curve and at the end of the inclined line after the ogee curve, also provide that with increasing discharge the pressure was decreased and vice versa. According to the numerical results, the negative pressure zones could be eliminated with changing the inclined line to curve. The predictions of Flow-3D were observed to be in a good with the experimental ones, and closer to the experimental results compared to the results of ANSYS tool.

- In Case-2 it can be denoted that the pressure decreases with decreasing the discharge but it is not right for Case-1 due to change in the spillway profile. It means that can solve the cavitation phenomena by changing the spillway profile.
- In the numerical results could find a strong relationship between pressure and velocity. Therefore, it is very important to showing velocity with pressure, for the better illustration The strong relationship between velocity and pressure for one fluid incompressible flow (constant fluid density) is given by Bernoulli's Law, (P+1/2 ρv^2=constant) The two expressions are also commonly called static pressure (P) and dynamic pressure ($\frac{1}{2}\rho * v^2$). These two quantities must always add up to the same value. This means that an increase in velocity causes a decrease in static pressure. And this equation validated the Flow-3D results as shown in Figure5.29.
- When a fluid is moving, the particles are move relative to one another, due to this movement shear stress is occurring. The CFD results showed that there is a strong relationship between velocity and shear stress, it is such that the shear stress increases as the velocity increases as shown in Figure 5.46.
- The orthogonal mesh was used for 3D modeling where the grid size of meshes is equal to 20mm in X, Y, and Z direction.
- In CFD modeling, the model domain was developed in the same dimension as the physical model in order to minimize errors as much as possible.
- A reasonable agreement was achieved in the pressure distribution and magnitude of the pressure between physical and CFD models for both cases, and overall flow 3D results are closer to physical when comparing to the Ansys results.
- In most trails, the Flow-3D results in negative or low pressures regions are closer to physical results than other regions.

Comparing Flow-3D results with Ansys results:

- The water surcharge results showed that for both cases, the physical results are between Flow-3D and ANSYS results, both numerical results are very close to the physical result.
- Due to the results in both cases, it can be said that the Flow-3D data are between ANSYS and physical results, also in the most trail, the Flow-3D data are closer to ANSYS data if comparing to physical result, because in both software the same

empirical equation was used and the most boundary conditions are the same. but Flow-3D results have better validation to physical if comparing with ANSYS.

- In this numerical study for pressure and water surcharge calculation Flow-3D tool more accurate than ANSYS tool