

**ASSESSMENT ON FLOW CHARACTERISTICS
OF DAM SPILLWAY STRUCTURE USING
PHYSICAL LABORATORY MODEL AND
COMPUTATIONAL FLUID DYNAMICS**

**CHE MOHAMAD AMIRUS SHAFIQ
BIN CHE ISMAIL**

UNIVERSITI SAINS MALAYSIA

2017

**ASSESSMENT ON FLOW CHARACTERISTICS OF DAM
SPILLWAY STRUCTURE USING PHYSICAL LABORATORY
MODEL AND COMPUTATIONAL FLUID DYNAMICS**

by

CHE MOHAMAD AMIRUS SHAFIQ BIN CHE ISMAIL

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

Julai 2017

ACKNOWLEDGEMENTS

Alhamdulillah, I praise to Allah, In the name of Allah, The Compassionate, The Merciful for giving me a healthy body, strength and determination to complete my study.

The first person that I would like to thank is my supervisor Dr. Mohd Remy Rozainy Bin Mohd Arif Zainol. His dedication and support on this research for providing time and knowledge has change my several view of live. I learn a lot of thing from him for these four semesters during this work. Besides of being very friendly supervisor, he was more also being a brother that would to hear and discuss my problem.

Special thanks toward my co-supervisor from Materials and Resources Engineering School, Dr. Muhammad Khalil Bin Abdullah @ Harun and Professor Ismail Abustan for helping me to understand this research work during my supervisor were busy with his work.

I feel grateful to happy toward my parents, wife and family who are the people that most sacrifice their time, money and energy to help reach this destination. My life might would not be being this lucky without them stand by my side.

My appreciation toward the Ministry of Higher Education for their financial support through the MyBrain15(MyMaster) Scheme for my study.

Last but not least, I wish to express my special thanks to all academician, technical staff of the School of Civil Engineering and my colleagues, Firdaus, Rais Azraie, Khairi and the others for their assistance throughout the time of this work.

TABLE OF CONTENTS

	Page
ACKNOWLEDGEMENTS	ii
TABLE OF CONTENTS	iii
LIST OF TABLES	ix
LIST OF FIGURES	xi
LIST OF SYMBOLS	xvi
LIST OF ABBREVIATIONS	xviii
ABSTRAK	xix
ABSTRACT	xxi
CHAPTER ONE: INTRODUCTION	
1.1 Background of the Study	1
1.2 Mengkuang Dam's Spillway	3
1.3 Problem Statement	4
1.4 Objectives of the study	5
1.5 Scope of the Study	6
1.6 Research Contribution	6
1.7 Thesis Structure	7
CHAPTER TWO: LITERATURE REVIEW	
2.1 Introduction	9
2.2 Dam (Component of Dam)	9
2.2.1 Hazard Potential for Dam Structure	10

2.3	Spillway	12
2.3.1	Spillway Classification	12
2.3.2	Spillway Component	15
2.3.3	Selection of Spillway Size and Type	17
2.3.4	Design Consideration	18
2.4	Stilling Basin	19
2.4.1	Type of Stilling Basin	19
2.4.2	Hydraulic Characteristics	25
2.4.3	Hydraulic Jump	26
2.4.4	Flow Resistance and Scaling Effects	28
2.4.5	Review on Stilling Basin Design	30
2.5	Physical Model	30
2.5.	Review on Laboratory Model	32
2.6	Numerical Model	33
2.6.1	Computational Fluid Dynamics Model	33
2.6.2	Reviews on Computational Fluid Dynamics	34
2.6.3	Application of CFD in Model of Spillway	35
2.7	Validation of Physical Model and Numerical Simulation	37
2.8	Gap of Knowledge	37
2.9	Summary	38

CHAPTER THREE: METHODOLOGY

3.1	Introduction	39
3.2	Spillway Model Geometry	39
3.3	Description of The Model	40

3.4	Model Scaling	46
3.5	Experimental set-up	49
3.6	Data Collection	51
3.6.1	Flow Discharge Measurements	51
3.6.2	Surface Water Depth Measurement using Digital Point Gauge	52
3.6.3	Velocity Measurement	53
3.6.4	Pressure Measurement using U-Tube	54
3.6.5	Visual Photograph and Video Documentation	54
3.7	Measurement Point of Surface Water Depth, Velocity and Pressure	54
3.7.1	Hydraulic Model Test	57
3.8	The Modification	59
3.8.1	Modification of Buffer Configuration of 1 st Modification	60
3.8.2	Modification on the Size and location of Buffer of the 2 nd Modification	61
3.10	Summary	63

CHAPTER FOUR: COMPUTATIONAL FLUID DYNAMICS

4.1	Introduction	64
4.2	Theory	64
4.2.1	Equation of Motion	64
4.2.2	Turbulence model	65

4.2.3	Volume of Fluid (VOF) Model	69
4.3	Description of the CFD Model	70
4.4	Simulation Procedure	71
4.5	Case Studies	72
4.5.1	3 rd Modification	73
4.6	Pre-Processing	74
4.6.1	Grids	74
4.7	Solver	77
4.7.1	Solver set-up and Solution Control	78
4.8	Post-Processing	80
4.9	Selected Planes and Location for the Studies	80
4.10	Summary	81

CHAPTER FIVE: RESULT AND DISCUSSION

5.1	Introduction	82
5.2	Observation from Physical Model	82
5.2.1	Water levels Observation	82
5.2.2	Velocity profile	84
5.2.3	Pressure	90
5.2.4	Cross-Waves	91
5.3	Modification Testing Results	93
5.4	Numerical Simulation	97
5.4.1	Grid Sensitivity Test	97
5.5	Cross-wave in Numerical Model	100

5.6	Case 4 (Original Design)	101
5.7	Case 8 (1 st Modification)	106
5.8	Case 12 (2 nd Modification)	110
5.9	Case 13 (3 rd Modification)	114
5.10	Velocity Contour and Volume of Fraction	117
	5.10.1 Upstream	118
5.11	Stilling Basin	124
	5.11.1 Case 4 (Original Design)	125
	5.11.2 Case 8 (1 st Modification)	126
	5.11.3 Case 12 (2 nd Modification)	127
	5.11.4 Case 13 (3 rd Modification)	128
5.12	Pressure Profile	128
5.13	Summaries for all Cases	129
5.14	Hydraulic Jump	130
5.15	Energy Loss	131

CHAPTER SIX: CONCLUSION

6.1	Conclusions	132
6.2	Recommendation for Future Studies	134

REFERENCES	136
-------------------	-----

APPENDICES

LIST OF PUBLICATIONS

LIST OF TABLES

	Page
Table 2.1 Hazard Potential Classification	11
Table 2.2 Summaries of Previous Study for Hydraulic Flow Using CFD	36
Table 2.3 Accuracy Based on Relative Error	37
Table 3.1 Flow Discharge for Hydraulic Spillway Spillway Model and Prototype	47
Table 3.2 Ultrasonic Flow Meter Specification	52
Table 3.3 Detail of Measurement Points	55
Table 3.4 The Study Cases of The Hydraulic Spillway Model	59
Table 3.5 Summary of Stilling Basin Configuration	47
Table 4.1 Numerical Modelling Case Study	73
Table 4.2 Grid Sensitivity Study	74
Table 4.3 Boundary Condition of The Model	77
Table 4.4 Properties of Water	78
Table 4.5 Under Relaxation Factors Used in the Simulation	79
Table 4.6 Discretization Used in the Simulation	79
Table 5.1 Distance Measured for the Cross-Wave	93
Table 5.2 The Length, L and Depth, D_2 of Hydraulic Jumps in The Stilling Basin For The Original Design, 1st Modification and 2nd Modification	95
Table 5.3 Result of the Hydraulic Model Velocity Reduction Between end of Chute and after Stilling Basin	96
Table 5.4 Energy Dissipation	96
Table 5.5 Grid Sensitivity Study Details for Case 4	100

Table 5.6	Velocity Reduction	130
Table 5.7	Hydraulic Jump	130
Table 5.7	Energy Loss	131

LIST OF FIGURES

	Page
Figure 1.1	Mengkuang pumped storage scheme (Source: Salleh et. al., 2011) 4
Figure 1.2	Reservoir surface impounded area upon the completion of Mengkuang Dam Expansion (Source: Salleh et al., 2011) 4
Figure 2.1	Component of a dam 10
Figure 2.2	Typical side channel and chute spillway (Source USBR, 2008) 14
Figure 2.3	Several types of labyrinth spillway (Source: USSD, 2011) 15
Figure 2.4	Step configurations for energy dissipation (Source: Gonzalez and Chanson, 2007) 18
Figure 2.5	Stilling basin Type II (source USBR, 2008) 20
Figure 2.6	Stilling basin Type III (source USBR, 2008) 21
Figure 2.7	Stilling basin Type IV (source USBR, 2008) 21
Figure 2.8	All shapes of the buffer blocks (Source: USBR, 2008) 24
Figure 2.9	Hydraulic jump (Source: USBR, 2008) 28
Figure 3.1	Detail Plan View of the Mengkuang Spillway section 40
Figure 3.2	The Mengkuang Dam's Spillway Physical Model 41
Figure 3.3	The detail scaled Mengkuang Dam`s spillway hydraulic model. (a) Plan view (b) Side view 42
Figure 3.4	Water circulation delivery system for the hydraulic spillway model 43
Figure 3.5	Flow chart of the overall study 45
Figure 3.6	General Process of Physical Model 48
Figure 3.7	Type of Jump in The Stilling Basin 50
Figure 3.8	Steps involves in physical modelling 51
Figure 3.9	A digital point gauge for measuring water depth 53
Figure 3.10	Location of measurement points 56

Figure 3.11	(a) Buffer block (front-view) (b) Buffer block (side-view)	57
Figure 3.12	(a) Stilling basin drawing (b) Stilling basin on the	58
Figure 3.13	Side-view of the original configuration	58
Figure 3.14	Plan-view drawing of 1 st modification on the model (mm)	60
Figure 3.15	Side-view drawing of the of 1st Modification on the model (mm)	61
Figure 3.16	Front and side view of the suggested buffer block of 2nd Modification	61
Figure 3.17	Plan-view drawing of buffer blocks for 2nd Modification on the model	62
Figure 3.18	Side-view of 2nd Modification on the model	62
Figure 3.19	Suggested position of buffer blocks of 2nd Modification on the model	63
Figure 4.1	Example of Distribution of F values	69
Figure 4.2	Problem solving steps in the CFD analysis	72
Figure 4.3	3 rd Modification of The Stilling Basin	74
Figure 4.4	The meshed geometry model of the spillway; (a) coarse (b) medium and (c) fine	75
Figure 4.5	(a) Tetrahedral mesh (b) Polyhedral mesh	76
Figure 4.6	The boundary conditions set up	77
Figure 4.7	(a) Selected plane for this study (b) Section A-A of the selected plane	81
Figure 5.1	Comparison of water surface profile for Case 1, 2, 3 and 4	83
Figure 5.2	Velocity contour for Case 1	85
Figure 5.3	Velocity contour for Case 2	86
Figure 5.4	Velocity contour for Case 3	87
Figure 5.5	Velocity contour for Case 4	88
Figure 5.6	Comparison of velocity profile for Case 1, 2, 3 and 4	89
Figure 5.7	Pressure profile for Case 1, 2, 3 and 4	91

Figure 5.8	(a) Two locations of crossed wave and (b) closed-view for 1 st crossed wave	91
Figure 5.9	Simple schematic drawing of the cross wave	92
Figure 5.10	Water surface profile along the centerline for Case 4, 8 and 12	93
Figure 5.11	Comparison of velocity between physical model, coarse, medium and fine grid for Case 4	98
Figure 5.12	Velocity magnitude for physical and numerical of the coarse grid for Case 4	99
Figure 5.13	Velocity magnitude for physical and numerical of the medium grid for Case 4	99
Figure 5.14	Velocity magnitude for physical and numerical of the fine for Case 4	100
Figure 5.15	Two Position of cross-waves (numerical model result)	101
Figure 5.16	Water surface profile along centreline of the spillway for Case 4	102
Figure 5.17	Water surface level for physical model and numerical analysis for Case 4	102
Figure 5.18	Velocity magnitude profile along centreline of the spillway for Case 4	103
Figure 5.19	Velocity magnitude for physical model and numerical analysis for Case 4	104
Figure 5.20	Comparison of the physical and numerical results of pressure for Case 4	105
Figure 5.21	Instantaneous pressure for physical model and numerical simulation for Case 4	105
Figure 5.22	Numerical pressure contour for Case 4	106
Figure 5.23	Water surface profile along centreline of the spillway for Case 8	107
Figure 5.24	Water surface level for physical and numerical model for Case 8	108
Figure 5.25	Velocity magnitude profile along centreline of the spillway for Case 8	109
Figure 5.26	Velocity magnitude for physical and numerical model for Case 8	109
Figure 5.27	Numerical pressure contour for Case 8	110

Figure 5.28	Water surface profile along centreline of the spillway for Case 12	111
Figure 5.29	Water surface profile of physical and numerical model for Case 12	111
Figure 5.30	Velocity magnitude profile along centreline of the spillway for Case 12	112
Figure 5.31	Velocity magnitude for physical and numerical model for Case 12	113
Figure 5.32	Pressure contour for Case 12	114
Figure 5.33	Water surface level along centreline of the spillway for Case 13	115
Figure 5.34	Velocity along centreline of the spillway for Case 13	115
Figure 5.35	Pressure profile along centreline of the spillway for Case 13	116
Figure 5.36	Pressure contour for Case 13	117
Figure 5.37	Velocity contour for Case 4	118
Figure 5.38	Velocity vector along for Case 4 (side-view)	119
Figure 5.39	Contours of Volume Fraction of Case 4 (side-view)	119
Figure 5.40	Velocity vector on the upstream (Plan-view)	120
Figure 5.41	Velocity vector on the upstream (side view)	120
Figure 5.42	Contours of Volume Fraction on the upstream	121
Figure 5.43	Velocity contour along the model for Case 8	122
Figure 5.44	Velocity contour along the model for Case 12	123
Figure 5.45	Velocity contour along the model for Case 13	124
Figure 5.46	Velocity vector along the centreline on stilling basin (side view)	125
Figure 5.47	Contours Contours of Volume Fraction along the centreline on stilling basin	125
Figure 5.48	Velocity vector on stilling basin for Case 4	126
Figure 5.49	Velocity vector on the stilling basin for Case 8	127
Figure 5.50	Velocity vector on the stilling basin for Case 12	127

Figure 5.51	Velocity vector on the stilling basin for Case 13	128
Figure 5.52	Pressure Profile along centreline of the spillway for Case 4, 8, 12 and 13	129

LIST OF SYMBOLS

ρ	Density
μ	Dynamic viscosity
p	Static pressure
t	Time
τ	Shear stress
\bar{u}_i	Mean velocity
u'_i	Fluctuating velocity
ν	Kinematics viscosity
Fr	Froude number
Fr_m	Froude model parameter
Fr_p	Froude prototype parameter
g	Gravitational constant
k	Turbulence kinetic energy
ε	Turbulence dissipation rate
TI	Turbulence intensity
U	Inlet velocity
v	Velocity magnitude
L	Characteristic length

Pr Prandtl number of energy

F Fraction number

LIST OF ABBREVIATIONS

3-D	Three Dimensional
CFD	Computational Fluid Dynamics
RNG	Renormalization-group
PISO	Pressure-Implicit with Splitting of Operators
PMF	Probable Maximum Flood
SDF	Spillway Design Flood
VOF	Volume of Fractions
URF	Under Relaxation Factors

**PENILAIAN TERHADAP CIRI-CIRI ALIRAN STRUKTUR ALUR LIMPAAH
EMPANGAN DENGAN MENGGUNAKAN MODEL FIZIKAL MAKMAL
DAN PENGKOMPUTERAN DINAMIK BENDALIR**

ABSTRAK

Alur limpah ialah suatu struktur yang melepaskan air lebihan dari empangan ke kawasan hilir. Tanpa alur limpah, air yang melepasi akan menyebabkan empangan runtuh ketika musim hujan lebat. Ketika melepaskan aliran air yang besar, profil permukaan air yang tinggi dan halaju yang tinggi akan berlaku. Situasi ini cenderung untuk aliran melepasi dan keluar dari alur limpah. Halaju yang tinggi menyebabkan tekanan yang rendah atau negatif berlaku di lantai alur limpah dan hakisan di kawasan hilir. Tekanan negatif menyebabkan peronggan dan mungkin menyebabkan kerosakan yang teruk di kawasan hilir. Matlamat kajian ini dijalankan untuk mengkaji ciri-ciri aliran hidraulik alur limpah seperti kedalaman air, halaju, tekanan dan pelepasan tenaga di penenang lembangan. Kajian ini menggunakan model fizikal dan Pengkomputeran Dinamik Bendalir (CFD). Satu model fizikal alur limpah berskala 1:20 telah dibina dandipasang di Makmal Hidraulik, Universiti Sains Malaysia. Kajian ini telah dilakukan terhadap empat kadar alir yang berbeza ($0.012 \text{ m}^3/\text{s}$, $0.017 \text{ m}^3/\text{s}$, $0.021 \text{ m}^3/\text{s}$ dan $0.067 \text{ m}^3/\text{s}$) dengan sebanyak 62 titik pengukuran untuk setiap kes. Data diambil dan direkodkan di setiap titik yang dipilih. Untuk memastikan air yang dilepaskan selamat daripada menyebabkan kerosakan yang teruk, kajian ini kemudiannya difokuskan ke kawasan penenang lembangan. Untuk CFD, ANSYS FluentTM digunakan sebagai penyelesaian untuk kajian ini. Selepas mendapat keputusan dari CFD, nilai-nilai itu dibandingkan dengan data yang diperolehi dari model fizikal untuk pengesahan. Berdasarkan kepada reka bentuk asal, panjang dan kedalaman lompatan hidraulik melebihi dinding tepi model untuk lepasan air. Penurunan halaju

untuk semua empat jenis susunan penenang lembangan berada dalam lingkungan 54% ke 64%. Panjang dan kedalaman lompatan hidraulik turun seperti yang dicadangkan pada perubahan kedua. Untuk pelepasan tenaga, susunan asal menghasilkan nilai tertinggi secara purata. Dengan pengesahan keputusan untuk kedua-dua kaedah, persetujuan yang baik telah dicapai. Untuk hubungan plotan berselerak antara data ujikaji dan simulasi, nilai pekali regresi (R^2), berada dalam julat 0.97 ke 0.99 telah diperolehi. Keputusan akhir telah menunjukkan halaju air diujung alur limbah boleh dikurangkan lebih banyak dengan mempunyai blok penampungan yang bersaiz lebih besar sementara mengekalkan panjang penenang lembangan yang sedia ada. Kesimpulan yang boleh dibuat daripada kajian ini ialah, CFD dapat digunakan untuk simulasi atau menggantikan model fizikal kepada aliran alur limbah.

ASSESSMENT ON FLOW CHARACTERISTICS OF DAM SPILLWAY STRUCTURE USING PHYSICAL MODEL AND COMPUTATIONAL FLUID DYNAMICS

ABSTRACT

Spillway is a structure that release surplus water from a dam into downstream area. Without spillway, water will be overtopping the dam and collapse during heavy raining season. During releasing large flow discharge, higher water surface profile and high velocity flow occur. This situation have the tendency for the flow to overtopping and breaching out the spillway. High velocity cause low or negative pressure on the spillway slab, scouring and erosion at downstream area. Negative pressure leads to cavitation and may cause critical damage on spillway structure. This study aims to investigate the flow characteristics along spillway such as water depth, velocity, pressure and energy dissipation on the stilling basin. The study were attempt on the physical experiment and Computational Fluid Dynamic (CFD). A 1:20 scaled spillway physical model was constructed and assembled in Hydraulic Laboratory of Universiti Sains Malaysia. This study has been conducted on four different flow rate ($0.012 \text{ m}^3/\text{s}$, $0.017 \text{ m}^3/\text{s}$, $0.021 \text{ m}^3/\text{s}$ and $0.067 \text{ m}^3/\text{s}$) with total of 62 measurement points for each case. The data were measured and recorded on the selected points. To ensure the release flow is safe from causing severe damage on the downstream area, the study then was focused on stilling basin. For CFD, ANSYS FluentTM used as the solver for this study. After obtaining the required results from CFD, the values were compared to the physical model data for validation. Based on the original design, hydraulic jump length and depth were overtopping the side bank wall of the model for all flow discharge. Three modifications on the stilling basin were suggested to reduce the depth and length of hydraulic jump. The velocity reduced from these four tested stilling basin

ranging from 54% to 64%. The length and depth of the hydraulic jump reduce as suggested on the 2nd modification. As for energy dissipation, original configuration yield the most value averagely compare to the others. By validating the results for both method, a good agreement was achieved for the water surface profile, velocity and pressure. The scatter plot relationship between physical experiment and numerical analysis having the regression coefficient (R^2) ranging from 0.97 to 0.99. End results shows that the velocity, length and depth of hydraulic jump at the end of spillway can be reduce more by having a bigger dimension with additional row of buffer blocks while having the same length of stilling basin. Thus, it can be concluded that CFD can be used to simulate the flow characteristics in spillway as an alternative to physical experiment of spillway.

CHAPTER ONE

INTRODUCTION

1.1 Background of the Study

Dam is one of the mega structure in the hydraulic field of study. Construction of a dam usually will involve several acres of land, which are selected based in several necessary data in order to be able properly function and withstand desired water collection on its storage volume. The purposes of dam construction are to help nearby community to benefit from the irrigation, flood control, water supply, hydropower, navigation and recreation (Mahato and Ogunlana, 2011; Zarfl et al., 2015; and Zhao et. al., 2013).

However, in order to have a dam that operate safely with minimum risk and damage incident, the surplus water of the structure needs to be released safely in intake rivers (Saunders et. al., 2014). A spillway is needed to convey the surplus water out from the storage area.

A spillway needs to be designed well and compatible to the dam which will involve a lot of parameters. Examples of the parameters are the flow discharge, head level, and type of foundation. In order to have a good spillway structure, the study of hydraulic characteristics of the flow needs to be performed by various methods and safety feature before finalizing the design.

The flow phenomena that appear in the spillway have been theoretically, experimentally and numerically studied its performance (Simões et al., 2012 and Ho et al. 2006). It involves the study whether in the past spillway design or the unknown hydraulic characteristics behaviours such as the flow regimes and skimming flow, geology and safety before the construction is carried out (Lesleighter et al., 2008,

Lesleighter, et al. 2016). Most these studies were done in the past by using the hydraulic physical model.

The hydraulic physical model was constructed in order to study the real-life performance of the structure. Usually, the model has been scaled down to desired size in due to limitation of facilities and cost. By observing and recording the data of the measurement such as water depth, velocity and pressure, the designer will be able to solve the problem that arise. Several problems that might arise are such as, high velocity and low pressure of flow. However, due to scaling effect and some other factors that cannot be identified during physical experiment, such as cavitation, the actual behaviour of flow in spillway cannot be fully identified.

Furthermore, the initial cost of the physical model is high and time consuming. Qualitative and empirical terms are the other factors that affecting the efficiency of the spillway known, but for physical model, there is no exact method for predicting them. Therefore, numerical method is used to resolve this issue. By numerical simulation method such as Computational Fluid Dynamics (CFD), it is possible to resolve the hydraulic flow problems and obtain some valuable information.

CFD is a method that plays a very effective role in analysis and predicting the performance of the spillway with less cost and effort (Bhajantri et al, 2006). The gathered information can lead to a new discovery and helps the designer to improve their design with less time and cost. In this study, the application of the comprehensive CFD model will be used in the design of spillway and discussed in details.

To avoid flow from spillway causing some serious casualty to the intake river, an energy dissipation device needs to be well designed and provide for the structure. The device will function by interrupting or the flow and therefore reducing its kinetic energy. By reducing the kinetic energy, the high velocity of the flow will drop sharply