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

CHAPTER FOUR: COMPUTATIONAL FLUID DYNAMICS

CHAPTER FIVE: RESULT AND DISCUSSION

LIST OF TABLES

LIST OF FIGURES

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
υ	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 LIMPAH 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 pelesapan 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 m^3/s, 0.017 m^3/s, 0.021 m^3/s dan 0.067 m^3/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 Fluent^{TM} digunakan sebagai penyelesai untuk kajian ini. Selepas mendapat keputusan dari CFD, nilai-nilai itu dibandingkan dengan data yang diperoleh 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 pelesapan tenaga, susunan asal menghasilkan nilai tertinggi secara purata. Dengan pengesahan keputusan untuk kedua-dua kaedah, persetujuan yan baik telah dicapai. Untuk hubungan plotan berselerak antara dat ujikaji dan simulasi, nilai pekali regresi (R^2), berada dalam julat 0.97 ke 0.99 telah diperoleh. Keputusan akhir telah menunjukkan halaju air dihujung alur limpah boleh dikurangkan lebih banyak dengan mempunyai blok penampan 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 limpah.

ASSESMENT ON FLOW CHARTERISTICS 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 m^3/s, 0.017 m^3/s, 0.021 m^3/s and 0.067 m^3/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 Fluent™ 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