LARGE EDDY SIMULATION OF DAMBREAK FLOW THROUGH POROUS OBSTACLE

THESIS

MASTER PROGRAM OF CIVIL ENGINEERING

FIELD: WATER RESOURCES ENGINEERING

By

DARU NURISMA PRAMUKTI

146060112111002

MINISTRY OF RESEARCH, TECHNOLOGY, AND HIGHER EDUCATION

BRAWIJAYA UNIVERSITY FACULTY OF ENGINEERING

MASTER AND DOCTORAL PROGRAM

MALANG, INDONESIA

2017

THESIS

LARGE EDDY SIMULATION OF DAMBREAK FLOW THROUGH POROUS OBSTACLE

By

DARU NURISMA PRAMUKTI

This thesis has been presented in front of committee of examiners

Date of the Examination: 12 Januari 2017

Supervisor I

Supervisor II

Prof. Chia-Ren Chu

Dr. Eng. Alwafi Pujiraharjo ST., MT.
NIP. 19700829 200012 1 001

Malang,
Brawijaya University
Head of Civil Engineering Master Program

Dr. Eng. Alwafi Pujiraharjo ST., MT.
NIP. 19700829 200012 1 001

Acknowledgement

I would never be able to finish my thesis without the guidance and support of my advisors, help from friends, and also support from my family.

Foremost, I would like to express my sincere gratitude to my advisor Prof. Chia-Ren Chu. I have been amazingly fortune to have an advisor who gave me the freedom to explore on my own and at the same time the guidance to recover when my thesis has a problem and my step faltered. Deepest gratitude also for Prof. Tso-Ren Wu, Associate Prof. Alwafi Pujiraharjo and Prof. Ming-Jyh Chern who have supported me throughout in writing thesis with their assistance and knowledge and who has willing to participate in my final defense as my reviewers.

Special thanks to Ting-Wei Lan, Yu-An Lin and all the members of Water Resources Egineering group, who as the good friends, was always willing to help and give their best suggestions. My thesis would not have been possible without their helps. Not forgetting to my love Indah Tri Pujiastuti who always supporting me, hugging me and encouraging me with her best wishes, and for all people that I cannot mentioned one by one. Thank you for giving me support.

As I complete my education at National Central University and University of Brawijaya, I would also like to deliver thank to Double Degree Program of University of Brawijaya, Indonesia that have been cooperation in this program with National Central University, Taiwan. I am one step closer to my goal and I got the great experience in education and life during my study time. Finally, I must express my very profound gratitude to my parents and my big family for providing me with unfailing support and encouragement throughout my years of study. Thank you.

LIST OF CONTENTS

COVER	i
VALIDATION PAPER	ii
ACKNOWLEDGEMENT	iii
LIST OF CONTENTS	iv
LIST OF TABLES	v
LIST OF FIGURES	vi
NOTATIONS	ix
摘要	x
ABSTRACT	xi
CHAPTER I: Introduction	1
1.1. Background	1
1.2. Reseach Objectives	3
CHAPTER II: Experimental Facility and Test Conditions	5
2.1. Experimental set-up	5
2.2. Measuring Technique and Calibration	7
CHAPTER III: Formulation and Numerical Solution	13
3.1. Governing Equations	13
3.2. Numerical Setup	14
CHAPTER IV: Experimental and Numerical Results	17
4.1. Case 1: Dambreak Flow through without obstacle	17
4.2. Case 2: Dambreak Flow through a solid obstacle	25
4.3. Case 3 and Case 4: Dambreak flow through porous obstacles	29
4.4. Drag Coefficient (C_D)	42
CHAPTER V: Conclusions	45
Bibliography	47
Appendix A	
Appendix B	

LIST OF TABLES

Table 1. Flow parameters of the flume experiments 18

Table 2. Simulation results of different computational grids for Case 1 19

Table 3. Simulation results of different Courant numbers for Case 1 21

Table 1. Simulation results of different convergence criterion for Case 1 22

Table 2. Information of parameters used for simulation 25

Table 3. Simulation results of different computational grids for Case 2 26

Table 4. Simulation results of short porous obstacle Case 3 with Caman-Kozeny method 32

Table 8. Some reference to determine the value of porous drag (fd) 33

Table 9. Simulation results of short porous obstacle Case 3 with Forchheimer method 34

Table 5. Simulation results of short porous obstacle Case 4 with Forchheimer method 35

Table 6. Simulation results of maximum and average wave height for all cases 36

Table 7. Prediction errors for all four cases at location x = 1.18 m 37

LIST OF FIGURES

Figure 1. Side view and top view of the experimental setup 5
Figure 2. (a) Schematic diagram of the square cylinders in staggered arrangement, (b) front view, and (c) side view of the obstacles 6
Figure 3. Photographs of the flume experiment. (a) Water flume; (b) Solid obstacle; (c) Short porous obstacles; (d) Tall porous obstacles 8
Figure 4. Photograph of the experimental instrument. (a) Amplifier (NET-406, Nijin Co.) and wave gauge (NET-030, Nijin Co.); (b) data logger (Agilent 34970A) 8
Figure 2. Calibration curve of the wave gauge 9
Figure 3. Photographs of the dambreak flows at time t = 0.60 sec (a) Case 1 at t = 0.60 sec; (b) Case 2 at t = 0.60 sec; (c) Case 3 at t = 0.60 sec; (d) Case 4 at t = 0.60 sec 10
Figure 7. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 1 (without obstacle). The initial water depth behind the gate ho = 0.26 m and initial water depth in flume h1 = 0.02 m 10
Figure 8. Measured wave heights at location x = 1.18 m and y = 0.08 m Case 2 (solid obstacle, hf = 0.05 m) 11
Figure 9. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 3 (short porous obstacles, hf = 0.10 m) 11
Figure 10. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 4 (tall porous obstacles, hf = 0.20 m) 12
Figure 11. Comparison of averaged-measured wave heights at location x = 1.18 m and y = 0.08 m for all cases, initial water depth ho = 0.26 m and h1 = 0.02 m 12
Figure 12. Computational domain and grid arrangement of grid A and the obstacle installed in zone II. (smallest grid size: Δx = 4 mm; Δy = 4 mm; Δz = 5 mm in zone II) 17
Figure 13. Comparison of measured and simulated wave heights of different computational grids for Case 1 (without obstacle) 20

Figure 14. Simulation results of different Courant numbers for Case 1 without obstacle. The initial water depth behind the gate ho = 0.26 m; in flume h1 = 0.02 m 21

Figure 15. Simulation results of different convergence criterion for Case 1 22

Figure 16. Profiles of instantaneous velocity on the central plane at location x = 1.18 m, y = 0.08 m when t = 0.70 sec for Case 1 (without obstacle) 23

Figure 17. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 1 (without obstacle). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; and (f) t = 1.0 s 24

Figure 18. Comparison of measured and simulated wave heights of Case 2 26

Figure 19. Velocity vectors on the central plane (y = 0.08 m) for Case 2 (solid obstacle, h_f = 0.05 m). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s 27

Figure 20. Profiles of instantaneous velocity after obstacle on the central plane of the flume (x = 1.18 m, y = 0.08 m) for Case 2 (solid obstacle) 28

Figure 21. Profiles of instantaneous velocity before obstacle on the central plane of the flume (x = 0.92 m, y = 0.08 m) for Case 2 (solid obstacle) 29

Figure 22. Comparison of measured and simulated wave heights uses solid method for Case 3 at x = 1.18 m and y = 0.08 m, h_f = 0.10 m. The square cylinders are installed in their actual positions 30

Figure 23. Comparison of measured and simulated wave heights for Case 3 (short porous obstacles) at location x = 1.18 m and y = 0.08 m using the drag function of Carman-Kozeny (1937) 31

Figure 24. Comparison of measured and simulated wave heights for Case 3 (short porous obstacles) using Forchheimer model at location x = 1.18 m and y = 0.08 m 33

Figure 25. Comparison of measured and simulated wave heights for Case 4 (tall porous obstacles, h_f = 0.20 m, porosity n = 0.5) with the coefficients α = 500 and β = 1.1 35

Figure 26. Comparison of simulated wave heights at location x = 1.18 m and y = 0.08 m for all cases 36

Figure 27. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 3 (short porous obstacles) uses solid method, h_o = 0.26 m; h_1 = 0.02 m; h_f = 0.10 m. (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s 38

Figure 48. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 4 (tall porous obstacles, h_f = 0.10 m). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s 39
Figure 259. Velocity vectors on the central plane of the flume for Case 4 (tall porous obstacles). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s 40
Figure 30. Comparison of instantaneous velocity behind the obstacles on the central plane of the flume at location x = 1.18 m at the same time t = 0.7 sec 41
Figure 31. Comparison of instantaneous velocity in front of the obstacles on the central plane of the flume at location x = 0.92 m at the same time t = 0.7 sec 41
Figure 32. Drag coefficients of dambreak flow through obstacles at different times 43

NOTATIONS

A	frontal area of the obstacle (m^2)
$C_d = F_d/0.5\rho U^2 A$	drag coefficient (dimensionless)
F_x	dynamic force on the sidewall (N)
H	height of water flume (m)
h_o	initial water depth (m)
h_1	initial water flume (m)
h_f	height of the baffles (m)
L	length of the water flume (m)
P_d	dynamic pressure (Pa)
P_s	static pressure (Pa)
t	simulation time (sec)
W	width of the water flume (m)
η	wave height (m)
ρ	density of water (kg/m^3)

摘要

水利工程師對潰壩洪水的預測非常感興趣，這類極端事件中水位和速度的變化是防洪控制的重要參數。從前的研究大多數使用水槽實驗和二維淺水方程來研究洪水波在平滑通道或通過不可滲透障礙物。本研究使用三維數值模型和水槽實驗來研究潰壩波與孔隙結構之交互作用。通過大渦模擬（LES）模型模擬周圍之自由水面及流場之流動情形，並通過流體體積　(VOF）方法來計算水面的變化。多孔障礙物對水流的影響由多孔阻力模型模擬。並通過波高計和數位照相機測量水表面高度等實驗數據的比較來證明模式本身的可信度與準確度。待驗證成功後，本模式即可被用於研究孔隙結構物對潰壩流的影響。數值結果表明，多孔隙阻力模型可以用於模擬多孔障礙物的阻力。

關鍵詞：潰壩流，多孔障礙物，大渦模擬，水槽實驗。

Abstract

The prediction of dambreak floods is of great interest to hydraulic engineers. Variations of water level and velocity in these extreme events are important parameters for the design of hydraulic systems and for flood control. Most of the previous studies used flume experiments and two-dimensional shallow water equations to investigate the flood waves over smooth channels or through impermeable obstacles. This study uses a three-dimensional numerical model and flume experiments to investigate dambreak flows through porous obstacles on the channel bed. The flow motion was simulated by a Large Eddy Simulation (LES) model, and the variation of water surface was solved by the Volume of Fluid (VOF) method. The effect of porous obstacles to the water flows was simulated by the a porous drag model. The variation of water surface in the water flume was measured by a wave gauge and a digital camera. The simulated water surface was validated by the measured surface elevation. Furthermore, the verified numerical model was then used to study the influences of the porosity of obstacles to the dambreak flows. Numerical results reveal that the resistance of the porous obstacle can be simulated by a porous drag model.

Keywords: Dambreak flow, Porous obstacles, Large Eddy Simulation, Water flume experiment.

CHAPTER I: Introduction

1.1. Background

Dambreak flood wave is one of the most complex type of rapidly varied flows. During the dam failure, large amount of stored water in the reservoir is suddenly released onto channel. The prediction of the free surface form, water depths and flood wave location is essential for reducing losses of lives and flood damages. The orientation of such obstacles with respect to the main flow direction also influences the flow behavior and complexity of the inherently turbulent flow. In addition, dambreak flow exerts significant impact force on the structures in the channel, due to its rapid propagation speed.

The study of dambreak problems began a century ago with Ritter, in 1892 who studied the dambreak flow of inviscid fluids in a horizontal channel using the shallow water approach. In recent years, dambreak flow studies have been mainly directed at flume experiments and numerical simulations because of difficulties in getting field data. Laboratory tests concerning dambreak flows were carried out in straight and curved channels (Miller and Chaudhry, 1989; Bell et al., 1992). Aureli et al. (2000) experimentally studied the stage hydrographs induced by dambreak flow over two trapezoidal bumps and compared with numerical results. Dambreak flow experiments performed by Stansby et al. (1998) used more than one method to record the water levels.

There are several previous researchers used flume experiments to investigate dambreak flows over a triangular bottom sill using digital imaging measurements (Soares-Frazao, 2007; and Aureli et al., 2008). Hunt (1987) simulated the dambreak floods on a dry, sloping channel using the method of characteristics. A normalized solution has been calculated and plotted for a dam-break flood on a sloping channel that is initially dry.

The solution neglects channel bed resistance and assumes that the dam was removed instantaneously. Comparisons of the inviscid solution with experimental measurements for two different channel roughnesses domenstrated that both flow depths and wetting-front arrival times increased as the channel roughness increased. Bellos et al. (1992) experimentally investigated two-dimensional flood waves resulting from instantaneous break of a dam, and measured the movement of waves on dry and wet beds using wave meters.

There are many numerical studies concerning dambreak flows in rectangular channel. Tseng and Chu (2000), and Tseng et al. (2001) used two-dimensional depth-averaged numerical models to simulate dambreak flows in open channels. Soares-Frazao and Zech (2007) investigated experimentally and numerically dambreak flows against an isolated obstacle. The aimed is to provide data about the influence of obstacles on dam-break waves. The experimental set-up consists of a channel with a rectangular-shaped obstacle, representing a building, placed immediately downstream from the dam.

Soares-Frazao and Zech (2008) also investigated the transient dambreak wave in an idealized city in order to understand the effects of flow depth and velocity in urban areas. They also assessed the ability of the finite-volume numerical model, commonly used in inundation modelling, to reproduce the fast transient flow interacting with multiple obstacles.

Cagatay and Kocaman (2010) used Reynolds-Averaged Navier-Stokes (RANS) and k-ε turbulence model involving the shallow-water equation to study the free surface profiles during the initial dambreak stages. By comparing the predicted and measured surface profiles, they found that the prediction using the RANS model is better than that of k-ε model. Kocaman and Ozmen-Cagatay (2010) studied the effect of lateral contraction on dambreak flows. Several numerical studies based on shallow water equation have been compared with experimental data of the dambreak flows over the channel with bottom obstacles (Aureli et al., 2008; Kocaman and Ozmen-Cagatay, 2011).

Hu et al. (2012) used a laboratory flume to investigate wave dissipation by vegetation in combined current-wave flow. They defined an empirical relation between drag coefficient and Reynolds number, which can be useful for numerical modeling. The characteristics of drag coefficient variation and in canopy velocity dynamics are incorporated into an analytical model to clarify the effect of vegetation-induced dissipation on wave-current flows. Wu and Hsiao (2013) employed a three-dimensional Large Eddy Simulation (LES) model with a macroscopic model for porous media to investigate solitary waves interacting with permeable breakwaters.

Stoesser (2014) summarized recent hydraulic and hydro-environmental researches using Large Eddy Simulation (LES) model. It is used increasingly to address research and engineering problems of modern hydraulics. LES directly computes the large-scale fluid motion and employs a model to account for the effects that small-scale turbulence imposes on large eddies. It has demonstrates a wide range of applicability and highlights some important research contributions that have been made in studies of large-scale turbulence-dominated flows in which LES delivered accurate predictions.

1.2. Research Objectives

In view of the previous studies, simulation of the dambreak flows through porous obstacle by three-dimensional LES model is not yet available. The porous obstacles could influence the dynamics of the dambreak flow. The purpose of the present study is to investigate experimentally and numerically the dambreak flow in the presence of porous obstacles in the channel, and to investigate the applicability of a porous media model for dambreak flows. This study focused on the formation of turbulent flows around obstacles. The turbulent flows were simulated numerically by VOF-based LES model.

This page is intentionally left blank

CHAPTER II: Experimental Facility and Test Conditions

2.1. Experimental set-up

The experiment was carried out in a rectangular flume of 2.18 m length, with the test section 0.16 m in width, and 0.4 m in height. Figure 1 shows the side view and top view of the water flume. A sluice gate was located at the right-hand side of the flume. The thickness of the sluice gate was 0.02 m.

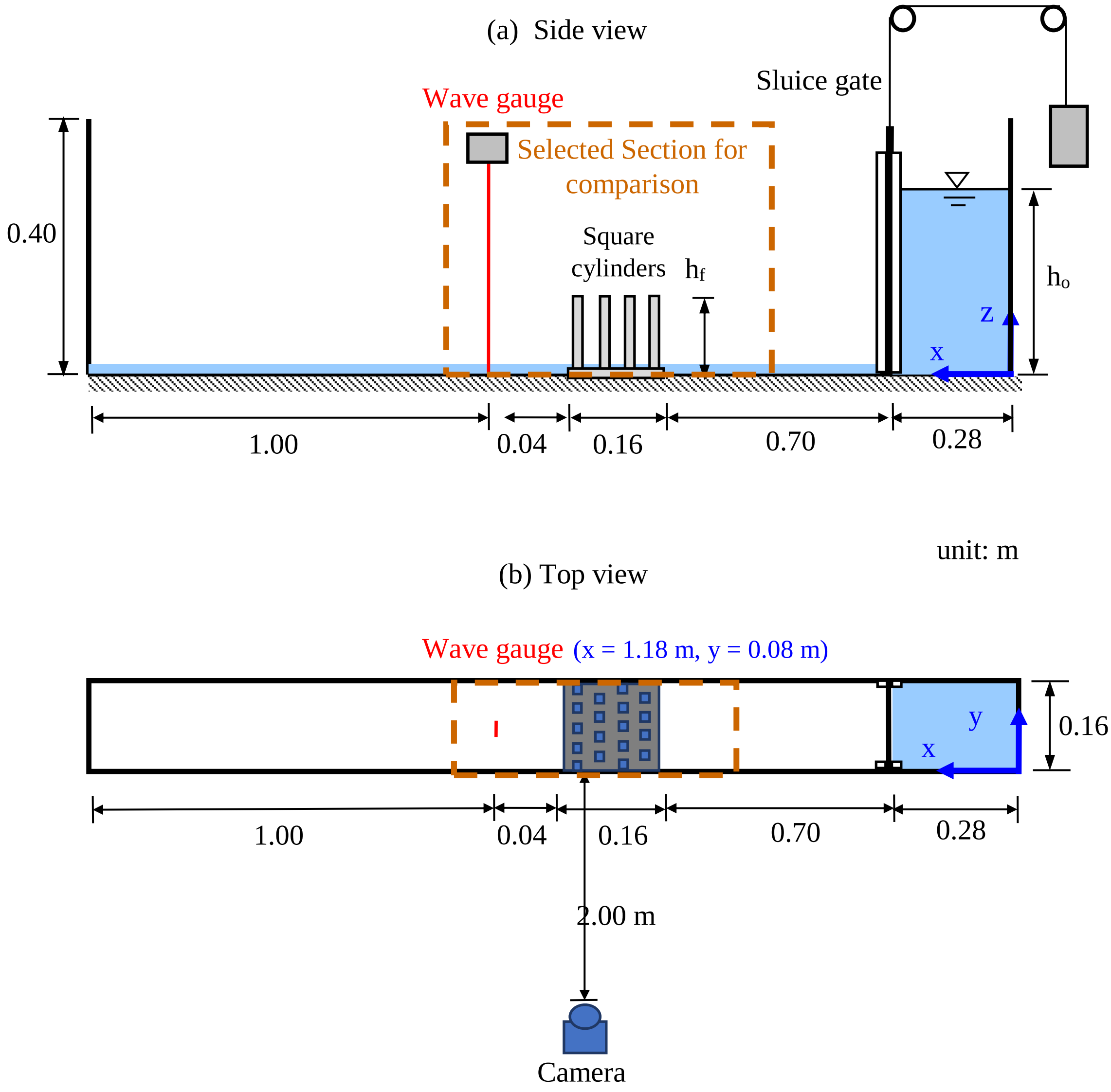
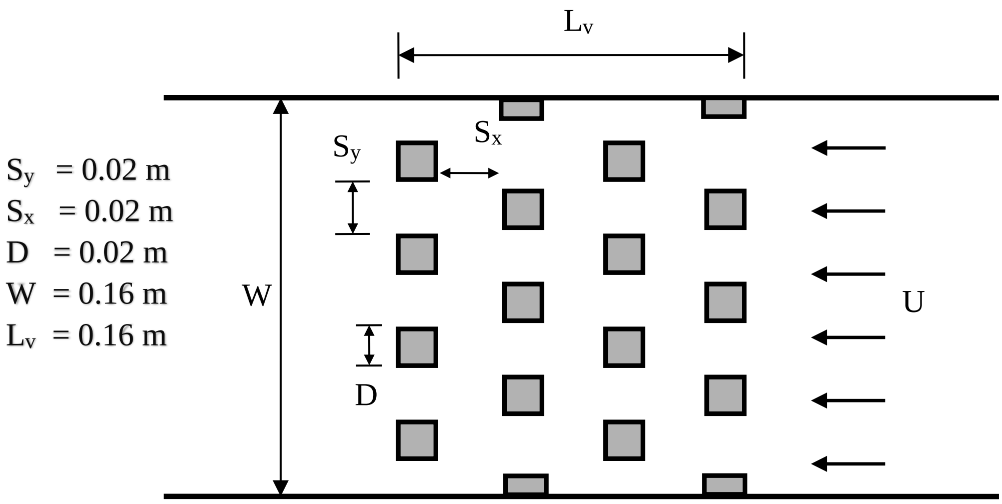


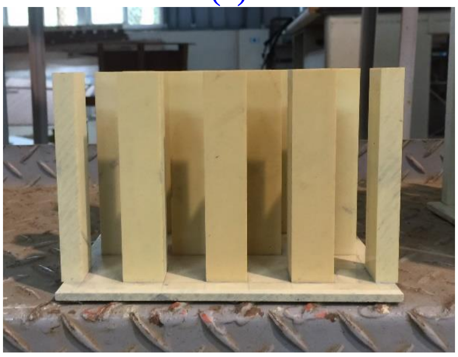
Figure 1. Side view and top view of the experimental setup.

The gate was suddenly lifted up by a heavy object, and the gate motion was recorded by a high speed camera (Exilim, Casio Inc.). The resolution of the photograph was 512 x 384 pixels with 100 frames per second (fps) video capture rate. The video images were analyzed using a Matlab code that can track the water surface at each frame.

(a)



(b)



(c)

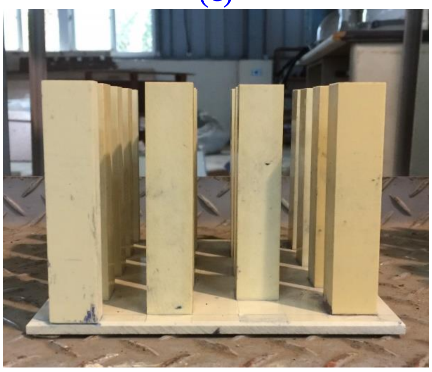


Figure 2. (a) Schematic diagram of the square cylinders in staggered arrangement, (b) front view, and (c) side view of the obstacles.

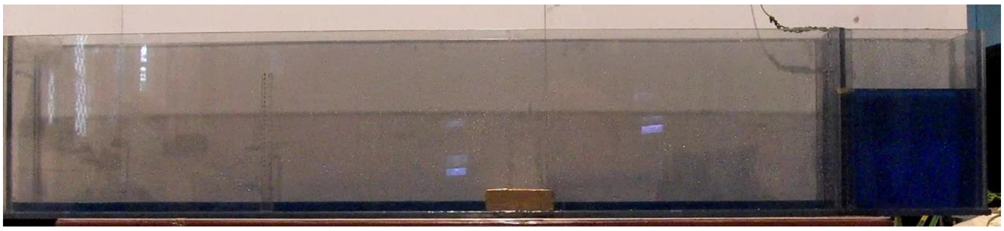
The resolution of the displacement was 0.32 mm in the horizontal direction, and 0.48 mm in the vertical direction. The experiments were conducted five times to ensure the repeatability of the experimental results. The initial water depth behind the sluice gate was 0.26 m, and the initial water depth in the flume was 0.02 m. The obstacle was a matrix of square cylinders (size 0.02 m) with the total length 0.16 m and total width 0.16 m. The cylinders are in two different heights, 0.10 m and 0.20 m. The square cylinders were installed in a staggered arrangement at downstream distance x = 0.78 m (Figure 2).

2.2. Measuring Technique and Calibration

Figure 3 are the photographs of the initial conditions of the water flume with the solid and porous obstacles before the flow began. The wave height was measured by a capacitance-type wave gauge (NET-030, Nijin Co.) installed at location x = 1.18 m. The resolution of the wave gauge was 0.03 mm.

(a)

(b)



(c)

(d)

Figure 3. Photographs of the flume experiment. (a) Water flume; (b) Solid obstacle; (c) Short porous obstacles; (d) Tall porous obstacles.

The variation of the water level at a fixed location, *x* = 1.18 m; y = 0.08 m, from the gate was measured by a wave gauge (Figure 4). The measured signal was processed by an amplifier (NET-406, Nijin Co.) and transmitted to a data logger (34970A, Agilent Inc.). First, the wave gauge has to be calibrated by several known, stationary water depths to determine the relationship between the water level and output voltage.

(a) (b)

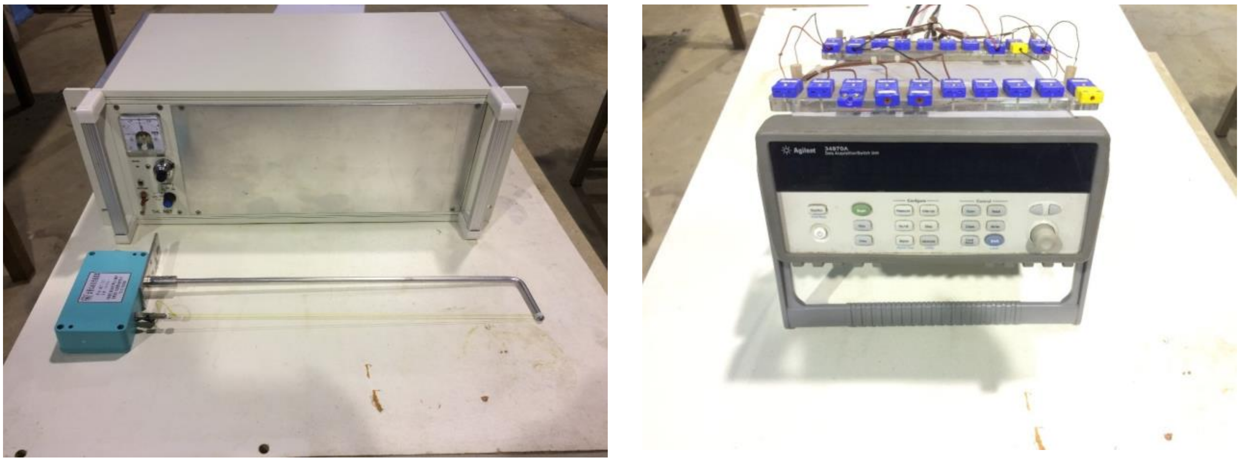


Figure 4. Photograph of the experimental instrument.

(a) Amplifier (NET-406, Nijin Co.) and wave gauge (NET-030, Nijin Co.); (b) data logger (Agilent 34970A).

The data were fitted to a straight line by linear regression (Figure 5). This calibration curve was used to determine the water depth in the flume. Figure 6 shows the photographs of dambreak flow in the flume at the same time t = 0.60 sec for all cases. There are four different cases of the laboratory experiment. Case 1 studies the dam break flow without obstacles in the flume. The results of measurements on this case can be seen in Figure 7.

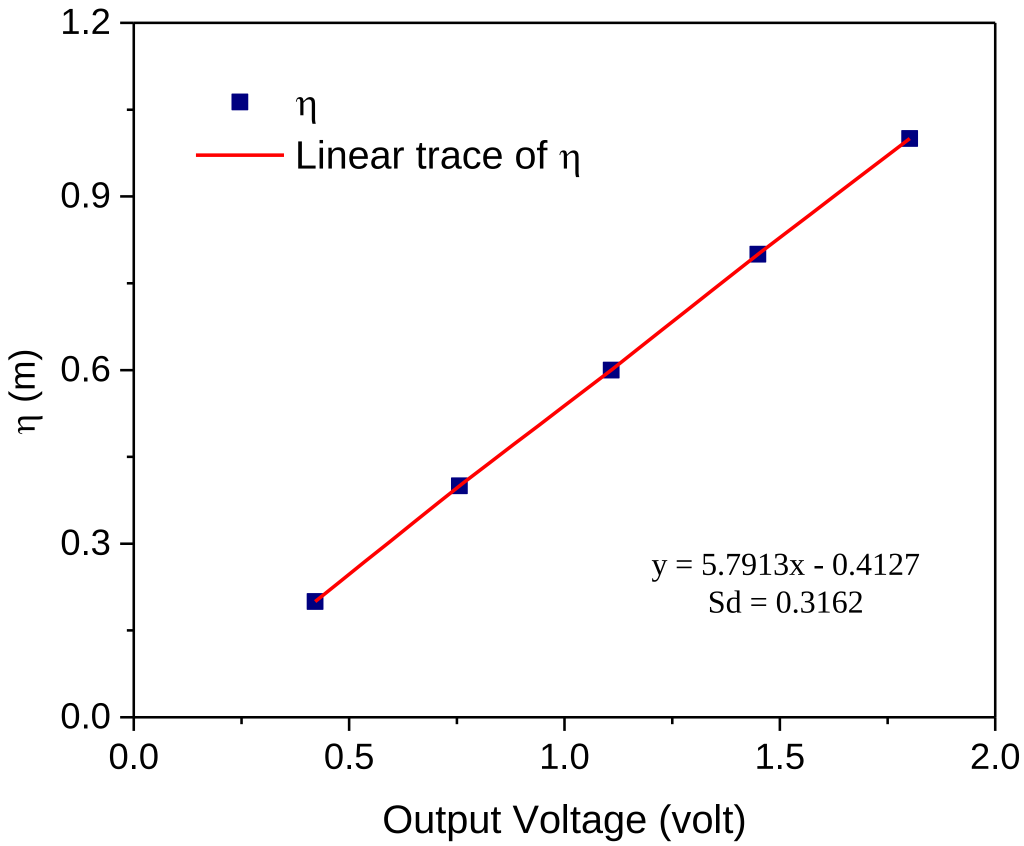
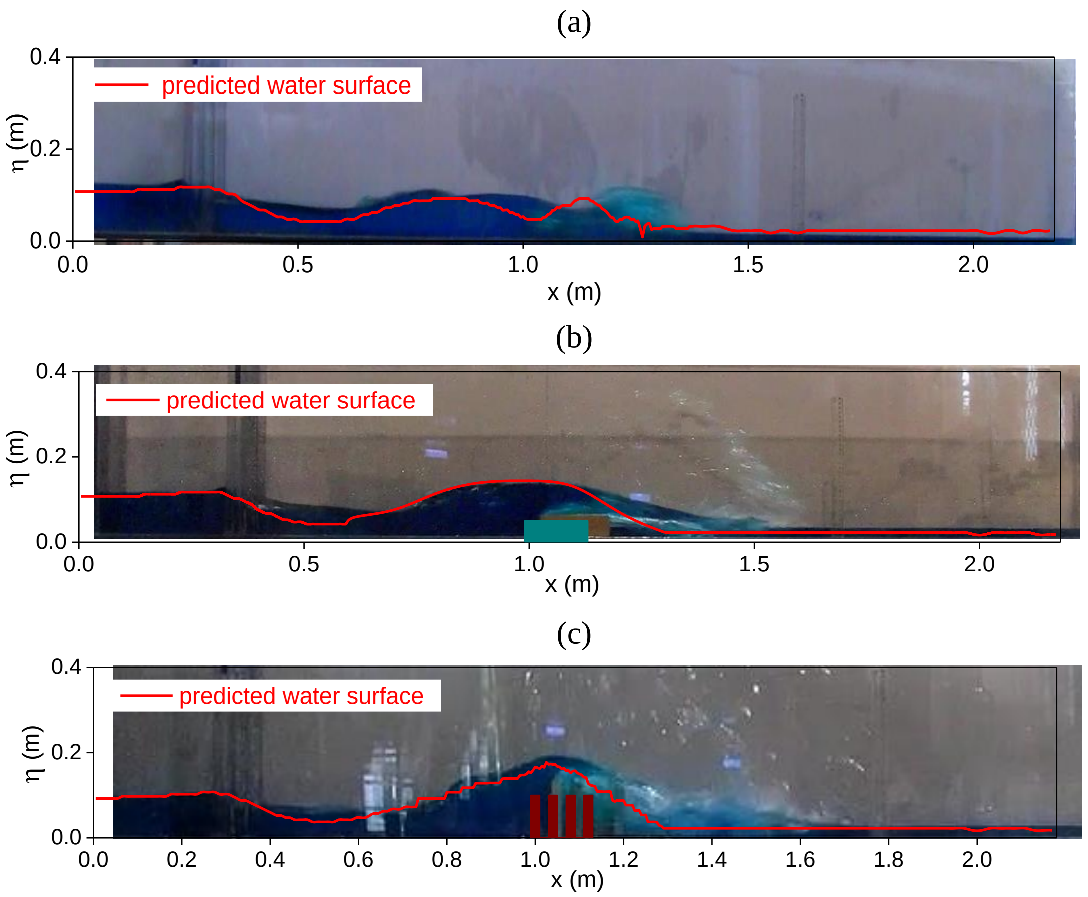


Figure 5. Calibration curve of the wave gauge.



(d)

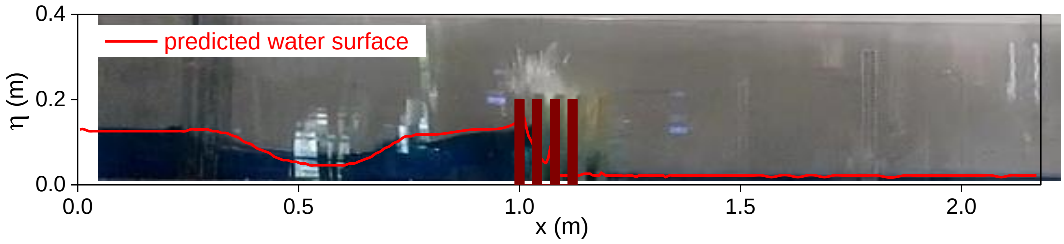


Figure 6. Photographs of the dambreak flows at time t = 0.60 sec. (a) Case 1 at t = 0.60 sec; (b) Case 2 at t = 0.60 sec; (c) Case 3 at t = 0.60 sec; (d) Case 4 at t = 0.60 sec.

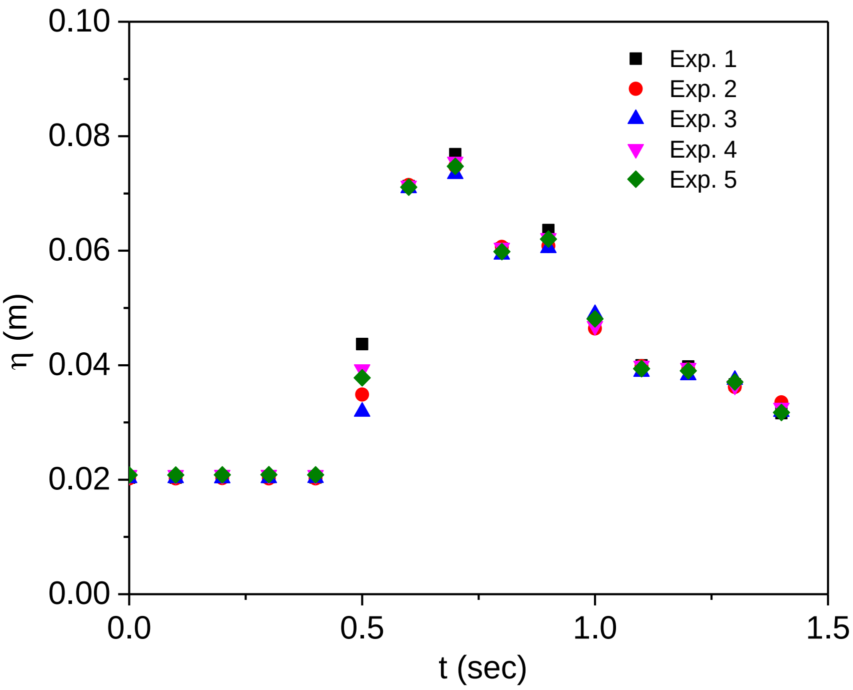


Figure 7. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 1 (without obstacle). The initial water depth behind the gate h_o = 0.26 m and initial water depth in flume h_1 = 0.02 m.

The experiments were repeated five times, and then calculated the average water level. Case 2 is the dam break flow with an impermeable, solid obstacle in the flume (see Figure 8). In the latter, dambreak flow through a porous medium in Cases 3 and 4. The difference between the Case 3 and Case 4 is the height of the obstacles, the obstacle height of Case 3 is h_f = 0.10 m and the measurement results can be seen in Figure 9. Case 4, the obstacle height h_f = 0.20 m and the measurement results can be seen in Figure 10. Figure 11 is the total of measurement results from all cases.

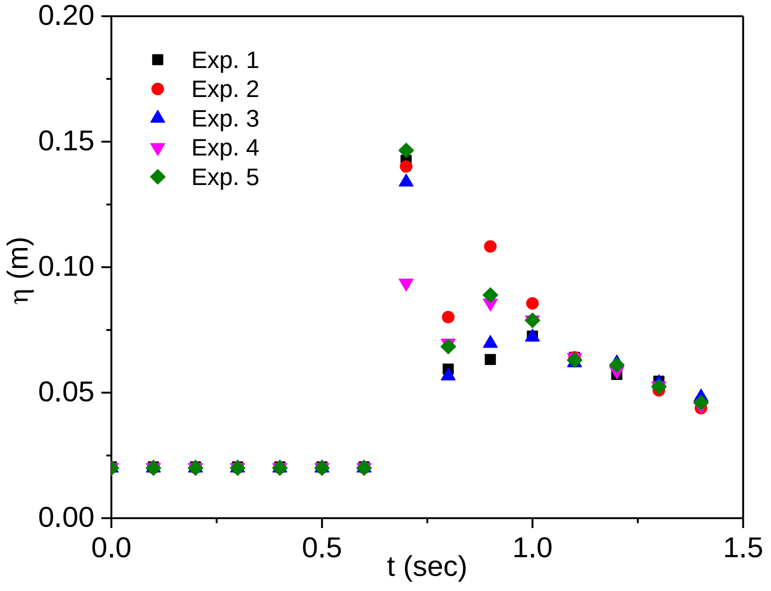


Figure 8. Measured wave heights at location x = 1.18 m and y = 0.08 m Case 2 (solid obstacle, h_f = 0.05 m).

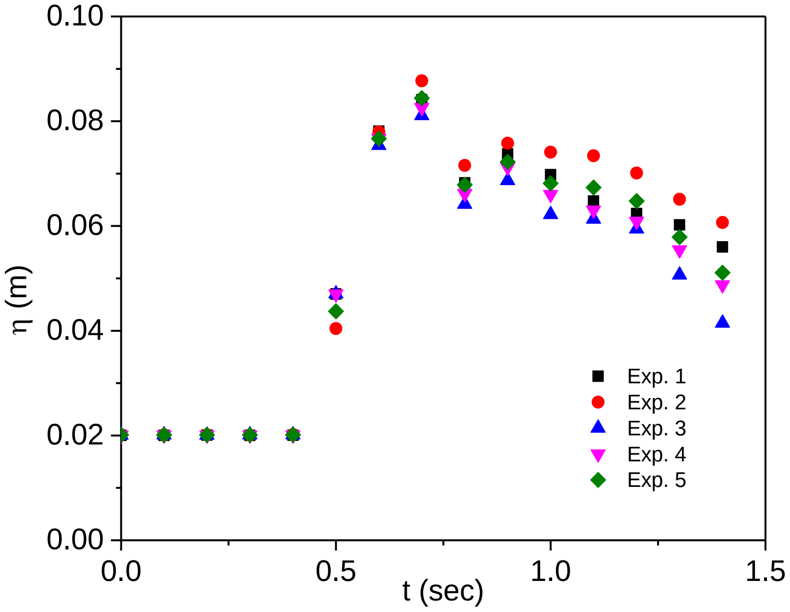


Figure 9. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 3 (short porous obstacles, h_f = 0.10 m).

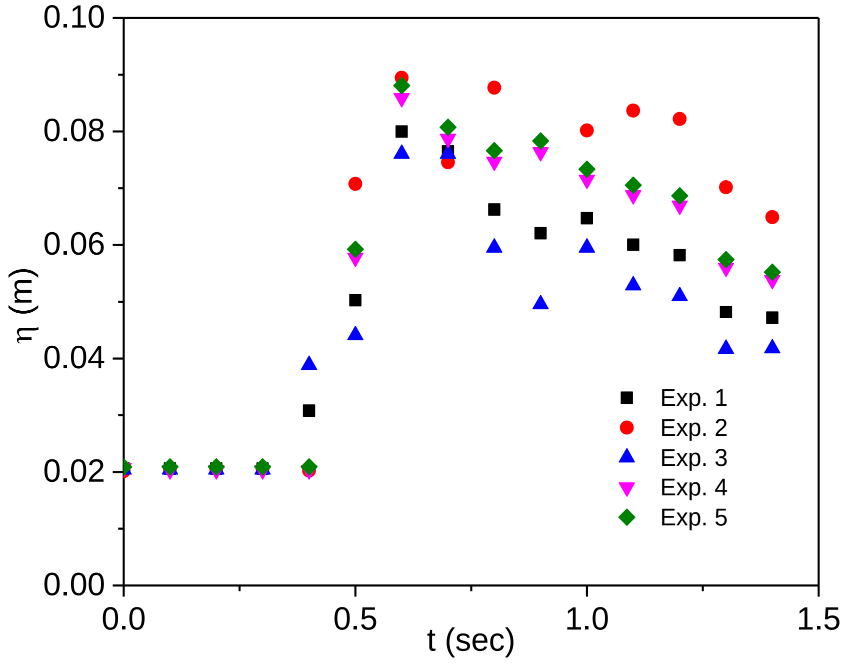


Figure 10. Measured wave heights at location x = 1.18 m and y = 0.08 m for Case 4 (tall porous obstacles, h_f = 0.20 m).

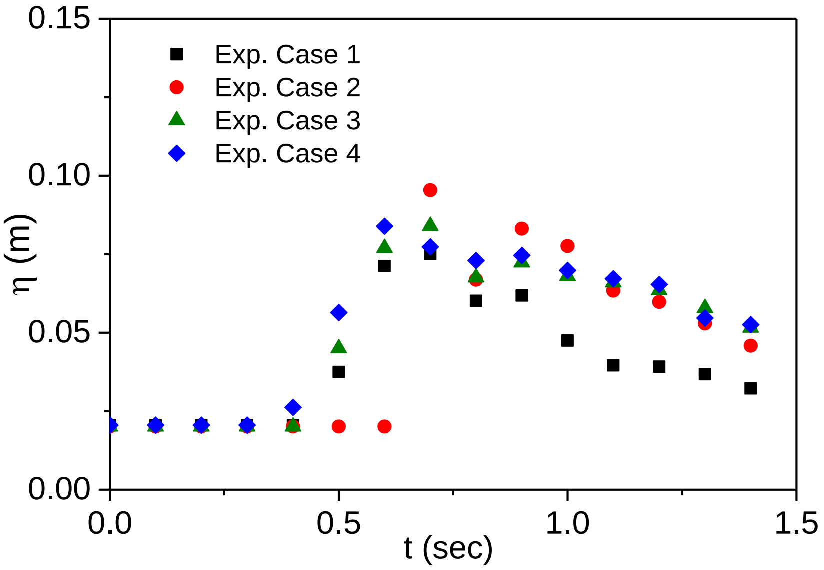


Figure 11. Comparison of averaged-measured wave heights at location x = 1.18 m and y = 0.08 m for all cases, initial water depth h_o = 0.26 m and h_1 = 0.02 m.

CHAPTER III: Formulation and Numerical Solution

3.1. Governing Equations

This study used a three-dimensional Large Eddy Simulation (LES) model to compute the flow field around an obstacle. The Volume of Fluid (VOF) algorithm was applied to track the motion of the water surface. The fluid motion was simulated by solving the continuity equation and the filtered Navier-Stokes equations:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial \rho \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \rho g \delta_{i3} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] + \rho f_{di} \tag{2}$$

subscripts i , j = 1, 2, 3 represent the x, y and z direction, respectively; t is time, $\bar{u}$ and $\bar{p}$ are the filtered velocity and pressure, ρ is the density of water, g is the gravitational acceleration, and μ_{eff} is the effective dynamic viscosity, defined as:

$$\mu_{eff} = \mu + \mu_{SGS} \tag{3}$$

where μ is the dynamic viscosity of water, and μ_{SGS} is the viscosity of sub-grid scale turbulence, defined as:

$$\mu_{SGS} = \rho(C_s \Delta_s)^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}} \tag{4}$$

where C_s is the Smagorinsky coefficient (Smagorinsky, 1963), and S_{ij} is the rate of strain:

$$\bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{5}$$

and Δ_s is the characteristic length of the computational cell, and can be calculated as:

$$\Delta_s = (\Delta x \Delta y \Delta z)^{1/3} \tag{6}$$

In this study, the value of the Smagorinsky coefficient $C_s = 0.15$ was chosen after comparing the simulation results with the experimental data. In addition, the projection method (DeLong, 1997) was used to solve the Poisson Pressure Equation (PPE) and to decouple the velocity and pressure in the Navier-Stokes equations. Further details of the numerical model can be found in Wu et al. (2014) and Chu et al. (2016).

The kinematics of the water surface was solved by the Volume of fluid (VOF) method (Hirt and Nichols, 1981). The volume fraction f_m occupied by the water in a grid cell can be described by:

$$\frac{\partial \boldsymbol{f_m}}{\partial \boldsymbol{t}} + \nabla \cdot (\boldsymbol{f_m u}) = 0 \tag{7}$$

The value of $f_m = 1$ represents the cell full of water and $0 < f_m < 1$ represents the cell partially occupied by water. The solid boundary, like the square cylinders, were brought into the numerical domain by the Partial-Cell Treatment (PCT). When a cell is partially occupied by the solid material, the effective cell volume is a fraction of the original cell volume V:

$$V_{eff} = (1 - f_{solid})V = \theta V \tag{8}$$

where V_{eff} is the effective cell volume of the fluid, θ is the effective volume fraction and $f_{solid}$is the volume fraction occupied by the solid in each cell. If a cell is fully occupied by the solid material, $V_{eff} = 0$, and the effective cell velocity is set to zero.

3.2. Numerical Setup

The present numerical model is based on the Truchas version 2.0.0 (Los Alamos National Laboratory, U.S.). For solving the spatially governing equations, the finite-volume method (FVM) and forward difference method are employed to discretize the space and time domain, respectively. The projection method is used to obtain accurate and stable solution. The Partial Cell Treatment (PCT) used by is modified to overcome this problem (Chu et al., 2016). The concept of PCT is that when a cell is partially occupied by the solids, the effective cell size is reduced to θV, in which θ is the volume fraction of the fluid and V is the cell volume. The modified Navier-Stokes equation can be written as:

$$\theta \frac{\partial(\mathbf{u})}{\partial t} + \nabla \cdot (\theta \mathbf{uu}) = -\frac{\theta}{\rho} \nabla p + \frac{\theta}{\rho} \nabla \cdot \tilde{\tau} + \theta \mathrm{g} \tag{9}$$

where **u** is the velocity, ρ is the fluid density and $\tilde{\tau}$ is the stress tensor. In the finite volume method, the above equation can be written in the integral form:

$$\theta \int \frac{\partial \mathbf{u}}{\partial t} dV + \theta \int d\mathbf{A} \cdot (\mathbf{uu}) = -\theta \int \frac{1}{\rho} \nabla p dV + \theta \int \frac{1}{\rho} d\mathbf{A} \cdot \tilde{\tau} + \theta \int \mathrm{g} dV \tag{10}$$

where **A** is the normal vector of the contral surface. The factor θ represents the blockage effect of the solid material inside the computational domain by reducing the mass and momentum fluxes between numerical cells.

In order to validate the present numerical model, the simulation results of the LES model were compared with the results of the flume experiments. The problem of dambreak waves interacting with porous obstacle can be separated into dambreak wave propagation and the interaction of wave and the porous media. The capability of the Large Eddy Simulation (LES) model for the prediction of the dambreak flows in the presence of porous obstacles was validated by the flume experiments.

This page is intentionally left blank

CHAPTER IV: Experimental and Numerical Results

4.1. Case 1: Dambreak Flow through without obstacle

Figure 12 shows the schematic diagram of the computational domain (length 1.90 m, width 0.16 m and height 0.40 m). For computational Grid 1: the mesh was 268 × 40 × 80 in x, y and z diection, respectively.

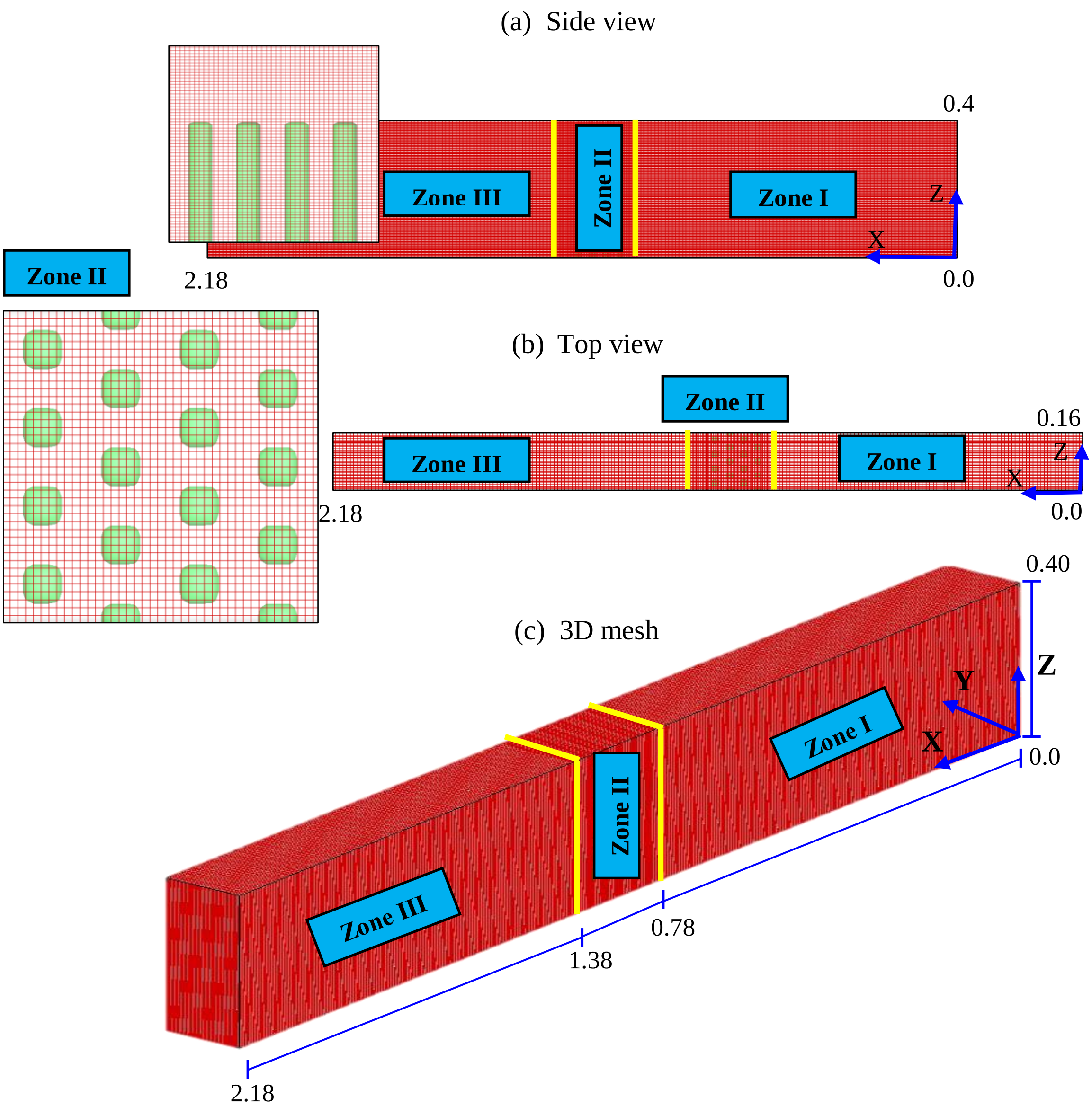


Figure 12. Computational domain and grid arrangement of grid A and the obstacle installed in zone II. (smallest grid size: Δx = 4 mm; Δy = 4 mm; Δz = 5 mm in zone II).

The stretching ratios equal to 1.0 for all three zones (uniform, orthogonal grid in all three zones). The smallest grid size was in zone II: $\Delta x = 4$ mm, $\Delta y = 4$ mm and $\Delta z = 5$ mm. The average time step was $\Delta t = 5 \times 10^{-4}$ sec. The grid sensitivity was checked by comparing the simulation results and computational times of three different grids for Case 1 (Table 1).

Table 1. Flow parameters of the flume experiments.

Case	h_o (m)	h_1(m)	C_s	Porous models	h_f (m)
1	0.26	0.02	0.15	-	-
2	0.26	0.02	0.15	Solid obstacle	0.05
3	0.26	0.02	0.15	1. Solid obstacle 2. Carman-Koseny 3. Forchheimer	0.10
4	0.26	0.02	0.15	Forchheimer	0.20

The length, width and height of the flume are 2.18, 0.16 m and 0.40 m. The staggered obstacle with 0.16 m by 0.16 m at located centrally at x = 0.78 m.

The no-slip boundary condition is applied at the bottom of the flume. The free-slip boundary conditions is applied at side walls of the flume. On the free surface, the zero pressure boundary condition is employed. In addition, no flux boundary condition is set at the outlet of the computational domain. The initial water depth behind the sliding gate was $h_o = 0.26$ m, and the initial water depth in the flume was $h_1 = 0.02$ m.

The difference Δ is defined as:

$$\Delta = \left| \frac{\eta_p - \eta_m}{\eta_p} \right| \times 100\% \tag{11}$$

where η_m is the measured wave height, η_p is the predicted wave height.

The maximum η_{max} and time-averaged wave heights $\bar{\eta}$ (measured at a distance and 40 mm to the sidewalls) were computed from the simulation results between 0 ~ 1.4 sec.

The time-averaged wave height is calculated as:

$$\bar{\eta}=\frac{1}{n}\sum_{i=1}^{n}\eta_{p,i} \tag{12}$$

where $\eta_{p,i}$ is the predicted wave height at time t = 0.1 x i sec at location x = 1.18 m and the total data number n = 14.

Table 2. Simulation results of different computational grids for Case 1.

		Grid A	**Grid B**	**Grid C**
smallest grid size (mm)		Δx = 4 Δy = 4 Δz = 5	Δx = 3 Δy = 4 Δz = 4	Δx = 4 Δy = 4 Δz = 2
Total grid number		857,600 268×40×80	1,144,800 318×40×90	1,500,800 268×40×140
Wave height	Measured $\bar{\eta}$, η_{max} (m)	0.0411, 0.0751		
	Predicted $\bar{\eta}$, η_{max} (m)	0.0439, 0.0800	0.0439, 0.0826	0.0434, 0.0810
	Δ	**4.9%, 6.2%**	**6.5%, 9.1%**	**5.2%, 7.4%**
$\overline{\Delta t}$ (sec)		0.0005	0.0006	0.0004
Computational time		10.4 hr	16.0 hr	43.2 hr

As can be seen in Table 2, the difference between the simulated and measured maximum wave heights η of Grid A, Grid B, Grid C, were Δ = 6.2%, 9.1%, 7.4%, respectively. Therefore, Grid C (268×40×140) was used for the simulation, and the wave heights were shown in Figure 13.

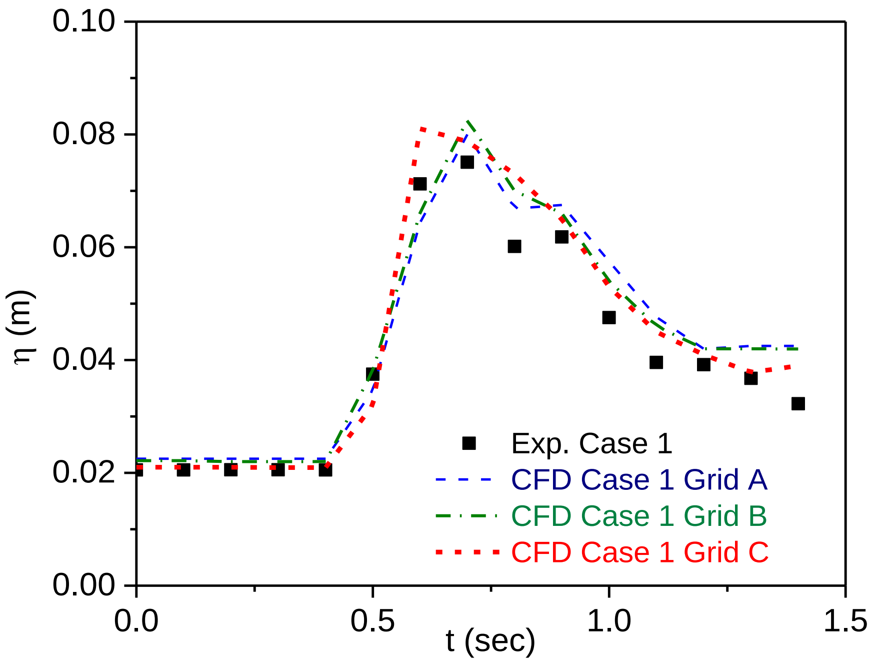


Figure 13. Comparison of measured and simulated wave heights of different computational grids for Case 1 (without obstacle).

Figure 14 and Table 3 compare the simulated wave heights of different Courant numbers. The Courant number is defined as:

$$\mathrm{Cr} = \frac{V \cdot \Delta t}{\Delta x} \tag{13}$$

where Δt is the time step, Δx is the spacing of the grid in the numerical model, and V is the fluid velocity.

The differences Δ between the measured and simulated average wave heights η of Cr = 0.5, Cr = 0.7, Cr = 0.9 were Δ = 8.8%, 7.1%, 6.1%, respectively. The differences between the measured and simulated maximum wave heights were Δ = 9.6%, 6.2% and 6.2% for Cr = 0.5, Cr = 0.7, Cr = 0.9, respectively. The Courant number Cr = 0.9 was used for all cases of the simulation to obtain a sufficient precision and to save the computational time.

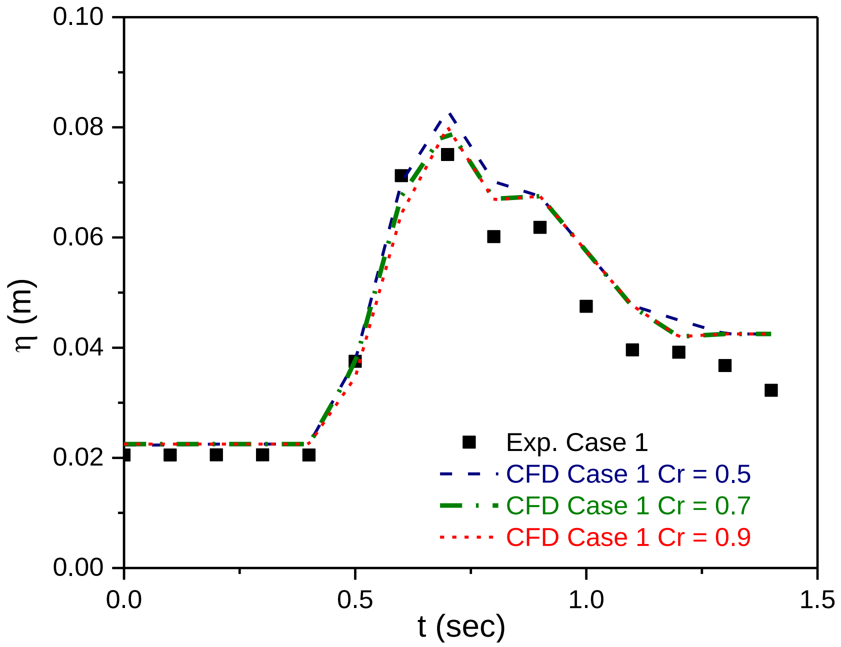


Figure 14. Simulation results of different Courant numbers for Case 1 without obstacle. The initial water depth behind the gate h_o = 0.26 m; in flume h_1 = 0.02 m.

Table 3. Simulation results of different Courant numbers for Case 1.

Case		**1**	**1**	**1**
Cr		0.5	0.7	0.9
Total grid number		857,600 268×40×80	857,600 268×40×80	857,600 268×40×80
Wave height	Measured $\bar{\eta}$, η_{max} (m)		0.0411, 0.0751	
	Predicted $\bar{\eta}$, η_{max} (m)	0.0450, 0.0831	0.0443, 0.0800	0.0439, 0.0800
	Δ	**8.8%, 9.6%**	**7.1%, 6.2%**	**6.3%, 6.2%**
$\overline{\Delta t}$ (sec)		0.0004	0.0005	0.0005
Computational time		17.0 hr	12.3 hr	10.4 hr

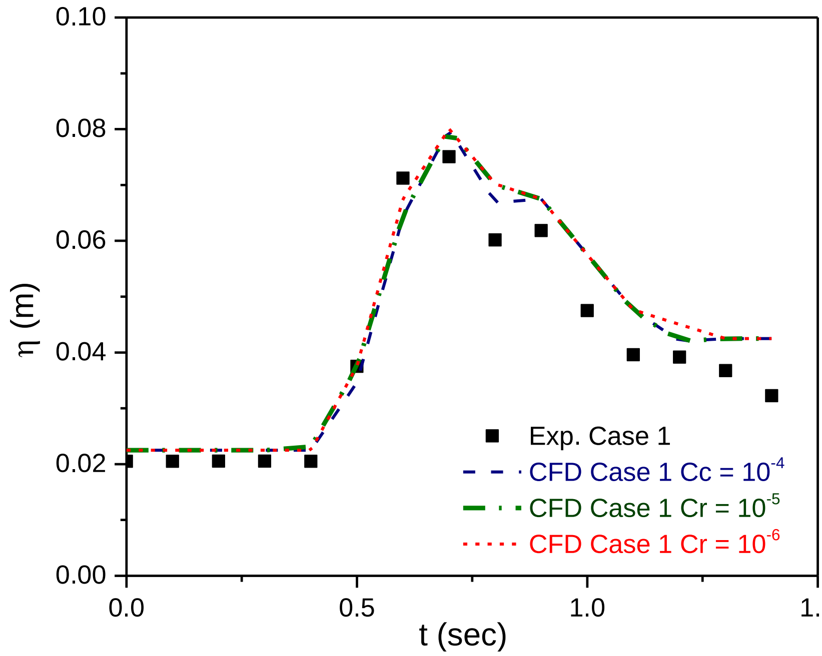


Figure 15. Simulation results of different convergence criterion for Case 1.

Table 4. Simulation results of different convergence criterion for Case 1.

		Case 1 Grid A	**Case 1 Grid A**	**Case 1 Grid A**
Convergence Criterion		1.0e-4	1.0e-5	1.0e-7
Total grid number		857,600 268×40×80	857,600 268×40×80	857,600 268×40×80
Cr		0.9	0.9	0.9
Wave height	Measured $\bar{\eta}$, η_{max} (m)		0.0411, 0.0751	
	Predicted $\bar{\eta}$, η_{max} (m)	0.0439, 0.0800	0.0443, 0.0800	0.0447, 0.0800
	Δ	**6.3%, 6.2%**	**7.2%, 6.2%**	**8.0%, 6.2%**
$\overline{\Delta t}$ (sec)		0.0005	0.0005	0.0007
Computational time		10.4 hr	11.3 hr	9.4 hr

Figure 15 compares the simulated wave heights using different convergence criterion. The differences (see Table 4) between the measured and simulated average wave heights η for Cc = 1.0 x 10^{-4}, 1.0 x 10^{-5}, 1.0 x 10^{-6} were Δ = 6.3%, 7.2%, 8.0%, respectively. Therefore, the convergence criterion Cc = 1.0 x 10^{-4} was used for all cases of the simulation. The Smagorinsky constant Cr = 0.15 was used for all cases of the simulation.

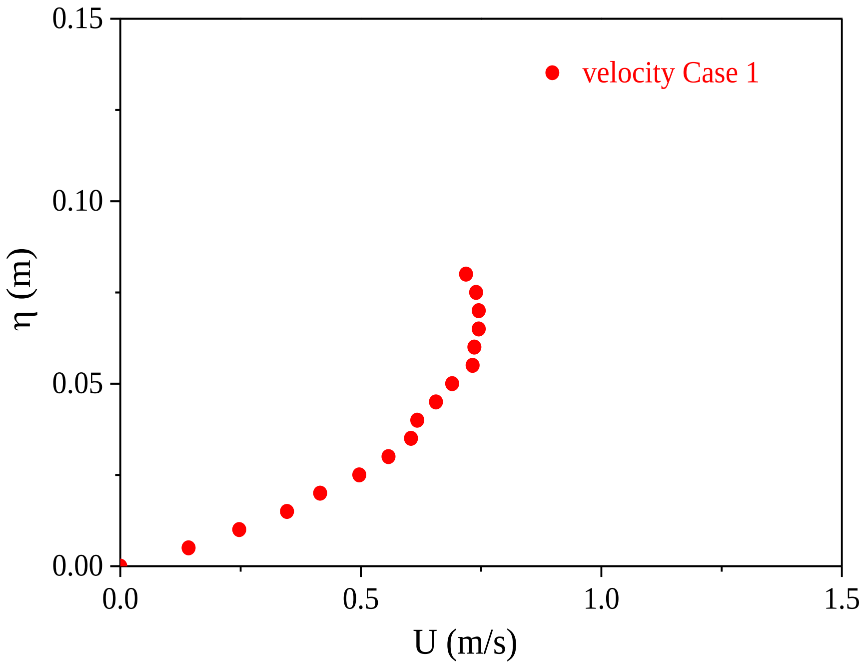


Figure 16. Profiles of instantaneous velocity on the central plane at location x = 1.18 m, y = 0.08 m when t = 0.70 sec for Case 1 (without obstacle).

Figure 16 shows the simulated instantaneous velocity at time t = 0.7 sec and location x = 1.18 m, y = 0.08 m for Case 1. The velocity at the channel bed U = 0.0 m/s, this is because the boundary conditions of the channel bed is a no-slip boundary. The maximum velocity between depth z = 0.0 ~ 0.080 m was about 0.75 m/s. Because Case 1 is dambreak flow without obstacle, the characteristic velocity of the dambreak flow is $U_o = \sqrt{gh_o}$. When the initial water depth h_o = 0.26 m, and the characteristic velocity U_o = 1.60 m/s. The simulated velocity is smaller than the characteristic velocity due to the bed friction. Figure 17 illustrated velocity vectors on the central plane (y = 0.08 m) of the flume at six different times from t = 0.5~1.0 sec.

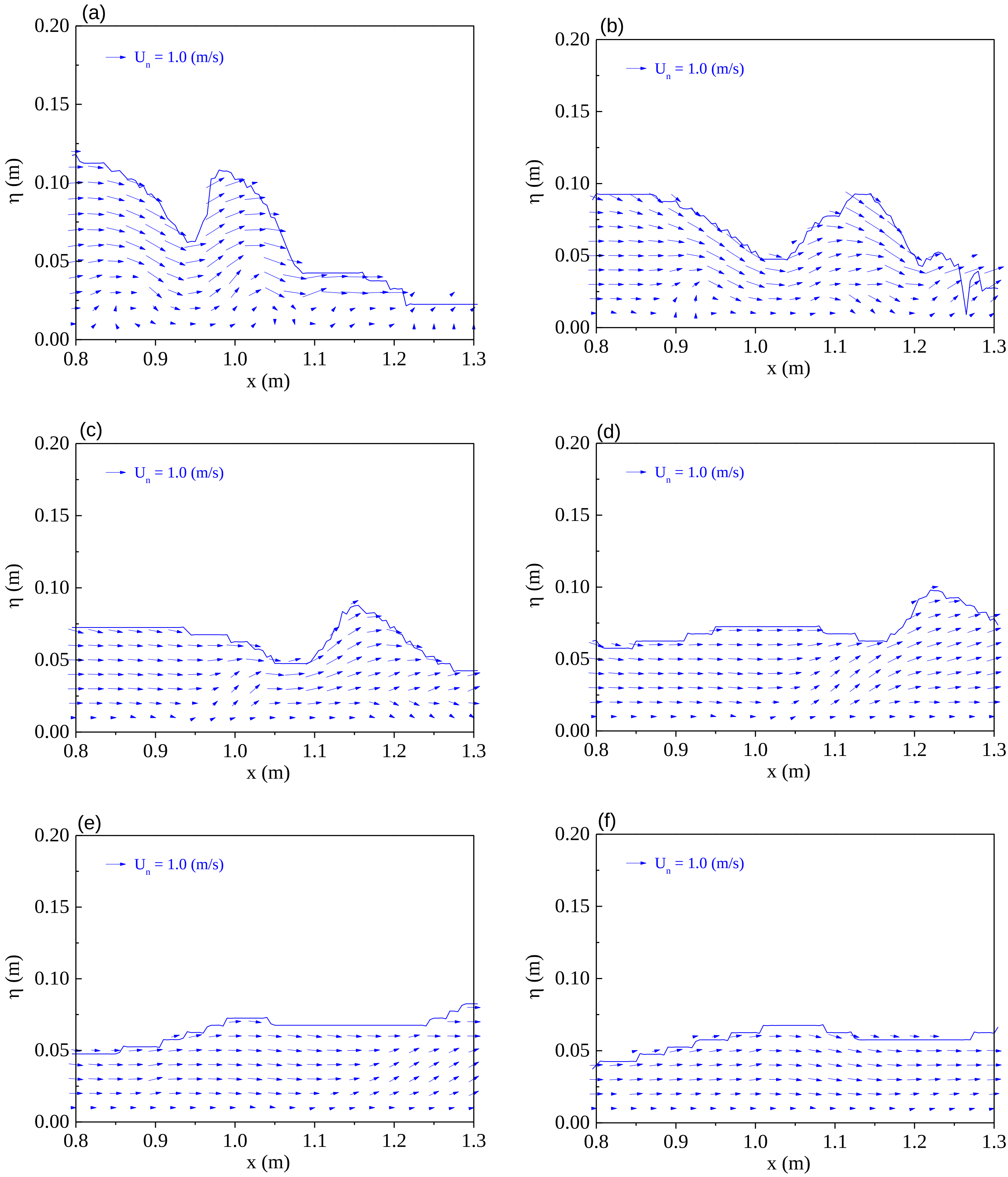


Figure 17. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 1 (without obstacle). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; and (f) t = 1.0 s.

4.2. Case 2: Dambreak Flow through a solid obstacle

The initial water depth behind the gate was h_0 = 0.26 m, in the flume was h_1 = 0.02 m. The length of the obstacle was 0.16 m, and obstacle height was 0.05 m. The computational mesh was 268 × 40 × 80. The smallest grid size was in Zone II: Δx = 4 mm, Δy = 4 mm and Δz = 5 mm. The stretching ratio in Zone I was 1.0, Zone II was 1.0, Zone III was 1.0. The grid sensitivity was checked by comparing the simulation results of three different grids the same flow condition. The average time step was Δt = 5×10^{-4} sec.

The flow parameters of the validation cases are listed in Table 5. The wave height η were calculated from the simulation results between 0 ~ 1.4 sec. As can be seen in Table 6, the relative differences between the simulated and measured average wave heights η of Grid A, Grid B, Grid C were Δ = 7.1%, 12.8%, and 4.7%, respectively. In Case 2, grid C showed good result with the least error value (see Figure 18).

Table 5. Information of parameters used for simulation.

Parameters	Values
output dt (sec)	0.1 sec
Courant number	0.90
Convergence criterion	1.0×10^{-4}
dt_init, dt_grow, dt_min, dt_max	1.0×10^{-6}, 1.1, 1.0×10^{-7}, 1.0
water density (kg/m^3)	1000
inviscid, turbulence_model	false, LES
Smagorinsky coefficient (Cs)	0.15
porous_flow, permeability constant C, α, β	true, 3000, 500, 1.1

Table 6. Simulation results of different computational grids for Case 2.

		Grid C
smallest grid size (mm)		Δx = 4 Δy = 4 Δz = 2
Total grid number		1,500,800 268×40×140
Wave height	Measured $\bar{\eta}$, η_{max} (m)	0.0481, 0.1314
	Predicted $\bar{\eta}$, η_{max} (m)	0.0501, 0.1378
	Δ	**4.0%, 4.7%**
$\overline{\Delta t}$ (sec)		0.0002
Computational time		53.6 hr

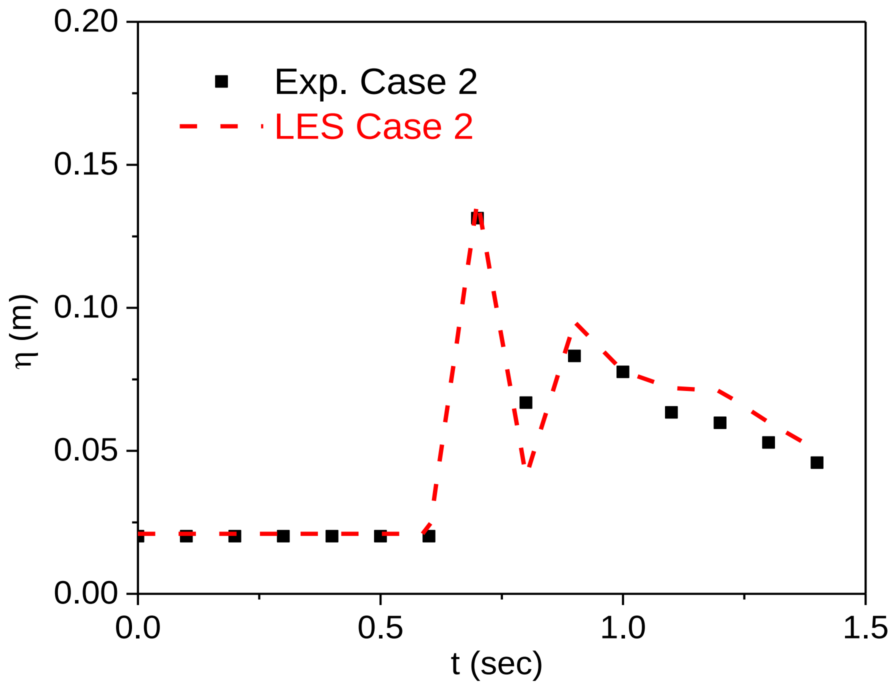


Figure 18. Comparison of measured and simulated wave heights of Case 2.

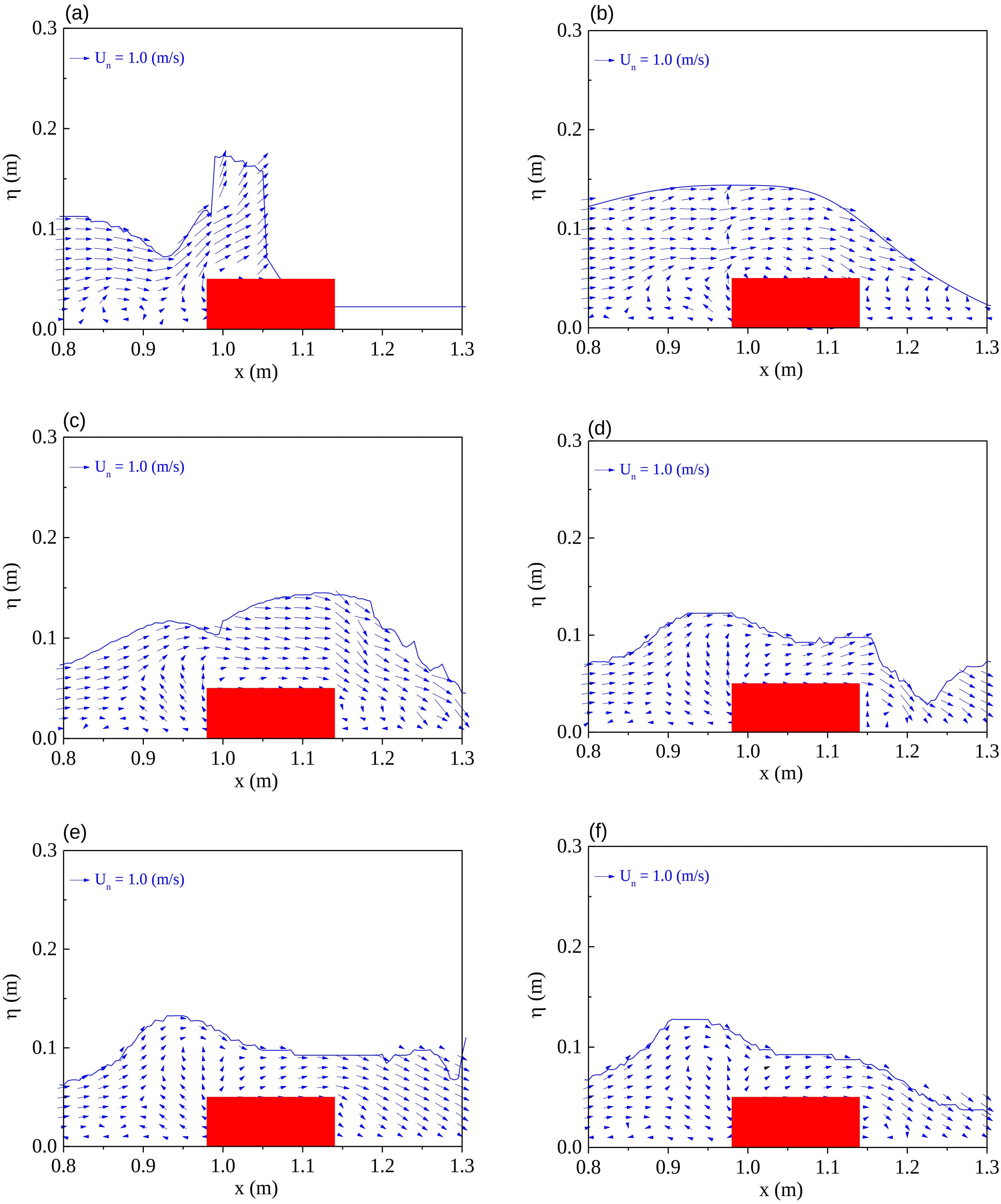


Figure 19. Velocity vectors on the central plane (y = 0.08 m) for Case 2 (solid obstacle, h_f = 0.05 m). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s.

Figure 19 illustrates the simulated velocity vectors on the central plane (y = 0.08 m) of the flume at six different times from t = 0.5 ~ 1.0 sec. Figure 20 shows profiles of instantaneous horizontal velocity after obstacle on the central plane of the flume (x = 1.18 m, y = 0.08 m) at time t = 0.7 sec. Case 2 is the dambreak flow with a solid obstacle, when the wave hit the obstacle, then a hydraulic jump appeared at the downstream side. This phenomenon caused the velocity on the downstream increased rapidly.

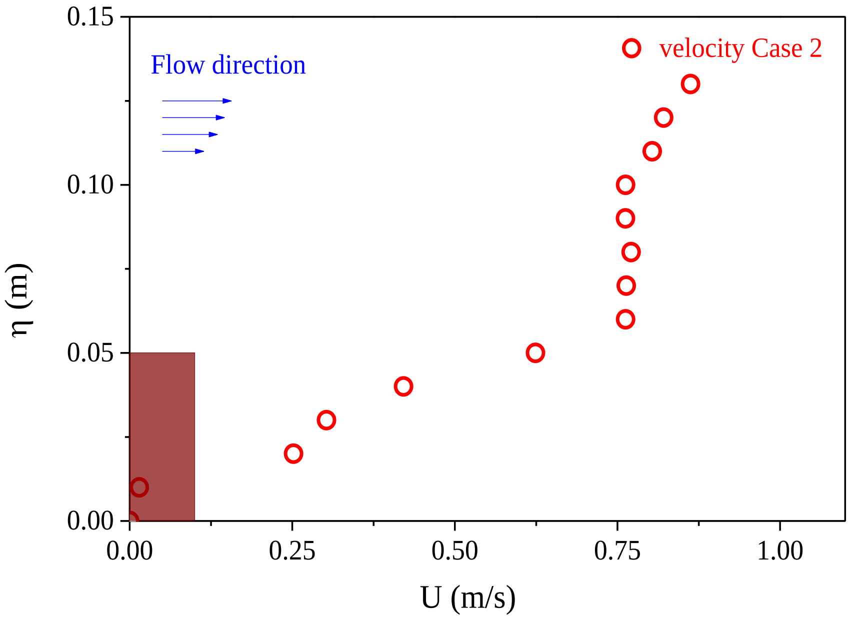


Figure 20. Profiles of instantaneous velocity after obstacle on the central plane of the flume (x = 1.18 m, y = 0.08 m) for Case 2 (solid obstacle).

Figure 21 shows the profile of horizontal velocity upstream of the obstacle on the central plane of the flume (x = 1.18 m, y = 0.08 m) at time t = 0.7 sec. The upstream velocity changed before obstacle does not occur, only a little effect on the flow behind the wave before the wave front hit the obstacle.

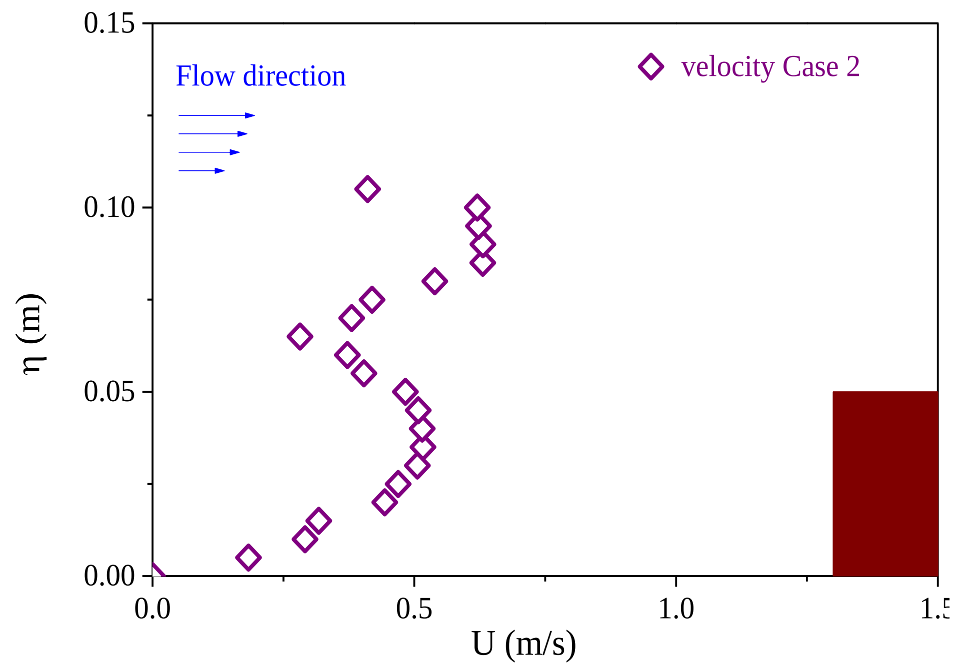


Figure 21. Profiles of instantaneous velocity before obstacle on the central plane of the flume (x = 0.92 m, y = 0.08 m) for Case 2 (solid obstacle).

4.3. Case 3 and Case 4: Dambreak flow through porous obstacles

This section investigates a three-dimensional dambreak flow interacting with porous obstacles. However, few experiments have been conducted for these situations. Similar to the Case 1 and 2, there was no difference in the boundary condition. The Smagorinsky coefficient was set as Cs = 0.15. The initial water depth was h_0 = 0.26 m, obstacle height was 0.10 m (Case 3) and 0.20 m (Case 4). The computational mesh was 268 × 40 × 80. The smallest grid size was in Zone II: Δx = 4 mm, Δy = 4 mm and Δz = 5 mm.

The porosity is defined as:

$$n = \frac{Area\ of\ solid}{Cross\text{-}sectional\ area\ of\ channel\ flow} \tag{14}$$

For Case 3 and 4, the porosity of the obstacles is 0.5.

The wave height η were calculated from the simulation results between 0 ~ 1.4 sec. Figure 22 compares the measured and simulated wave heights of Case 3 that uses solid method. In this simulation, the obstacles were solid, square cylinders and were installed one by one in accordance with their actual positions.

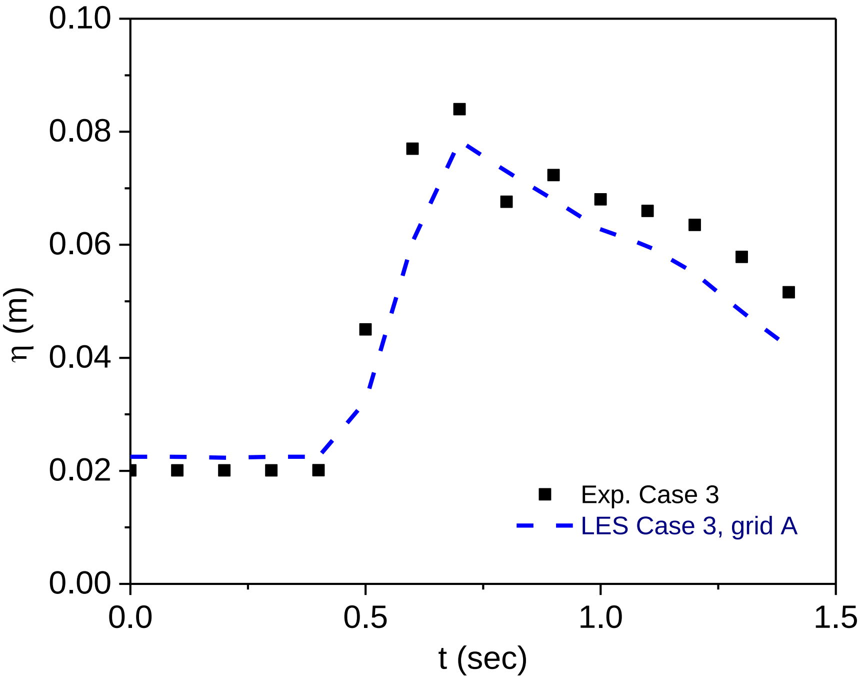


Figure 22. Comparison of measured and simulated wave heights uses solid method for Case 3 at x = 1.18 m and y = 0.08 m, h_f = 0.10 m. The square cylinders are installed in their actual positions.

This method uses grid A and grid B. The relative differences between the simulated and measured of maximum wave heights η_{max} of Grid A and Grid B were Δ = 28.4%, and 21.0%, respectively. The differences between the simulated and measured average wave heights η of Grid A and Grid B were Δ = 36.3%, and 33.4%, respectively. The results indicate a major fault, the fault is probably caused by insufficient resolution of the computational grid. Therefore, the effect of obstacles on the dambreak flow was simulated by a porous drag model.

The drag force, f_d, in the direction of the flow can be calculated by the Darcy law (Carman-Kozeny, 1937):

$$\vec{f}_d = -C \cdot \frac{(1-n)^2}{n^3} \vec{u}_n \tag{15}$$

where n is the porosity of the obstacle; u_n is the directional velocity; and C is the permeability constant.

Figure 23 and Table 7 compare the measured and simulated wave heights of Case 3 by the drag model of Carman-Kozeny (1937) with the permeability constant C = 1800 and 3000. The porous obstacle can be simulated by coarse grid, so the grid A was used for the simulation. The differences between the simulated and measured average wave heights η was Δ = 4.5% for C = 1800, Δ = 1.1% for C = 3000. The differences between the simulated and measured maximum wave heights η was Δ = 13.7% for C = 1800, Δ = 10.1% for C = 3000.

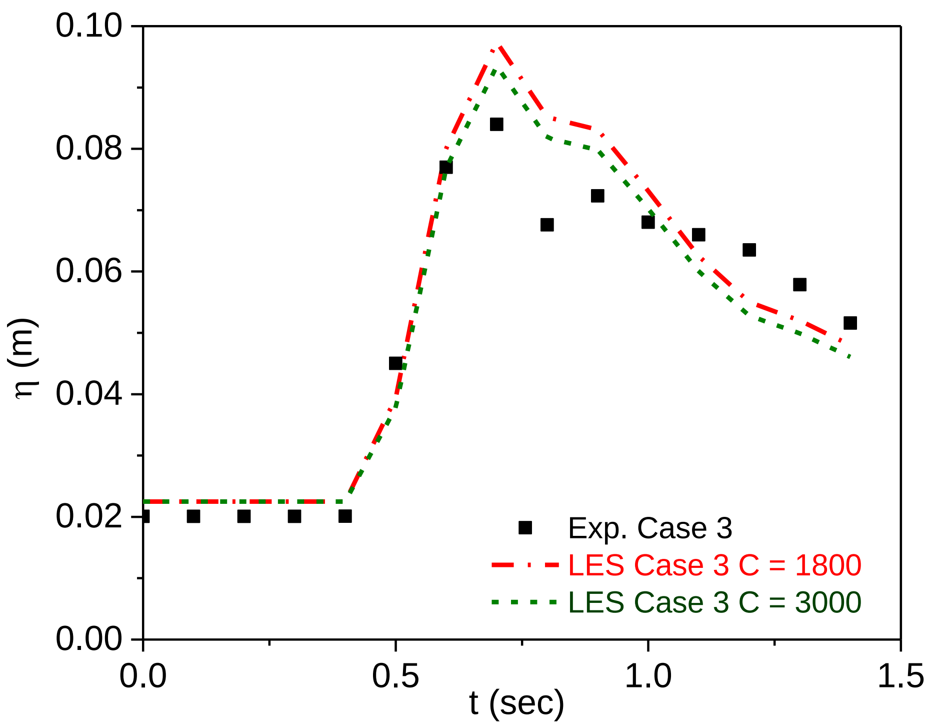


Figure 23. Comparison of measured and simulated wave heights for Case 3 (short porous obstacles) at location x = 1.18 m and y = 0.08 m using the drag function of Carman-Kozeny (1937).

Table 7. Simulation results of short porous obstacle Case 3 with Caman-Kozeny method.

		Grid A	Grid A
smallest grid size (mm)		Δx = 4 Δy = 4 Δz = 5	Δx = 4 Δy = 4 Δz = 5
Total grid number		857,600 268×40×80	857,600 268×40×80
Permebaility constant C		1800	3000
Wave height	Measured $\bar{\eta}$, η_{max} (m)	0.0502, 0.0840	
	Predicted $\bar{\eta}$, η_{max} (m)	0.0526, 0.0974	0.0508, 0.0935
	Δ	**4.5%, 13.7%**	**1.1%, 10.1%**
$\overline{\Delta t}$ (sec)		0.0004	0.0004
Computational time		21.9 hr	22.8 hr

Forchheimer (1901) added an inertia term to Darcy's equation for high Reynolds number flows in porous media:

$$\vec{f}_d = -\alpha \cdot \frac{(1-n)^2}{n^3}\vec{u}_n - \beta \cdot \frac{(1-n)}{n^3}\vec{u}_n^2 \tag{16}$$

where α is the permeability constant where β is the inertia factor (dimensionless), and the other formula can be seen in Table 8.

Figure 24 compares the measured and simulated wave heights using the drag model of Forchheimer (1901) for Case 3 at location x = 1.18 m and y = 0.08 m. There are several references for the values of α and β. This study set the permeability constant α = 500 and the inertia factor β = 1.1 by comparing with the measured water surfaces of Case 3. The difference between the simulated and measured average wave heights η was Δ = 5.6% at location x = 1.18 m (see Table 9 for detail). The difference between the simulated and measured maximum wave heights η was Δ = 4.0% at time t = 0.6 sec.

Table 8. Some reference to determine the value of porous drag (f_d)

Perry, Gree, and Maloney (1984)	$\vec{f}_d = \frac{1}{C^2}\frac{\left(A_p / A_f\right)^2 - 1}{t}$
Miguel et al. (1997)	$\vec{f}_d = \frac{1}{\rho}\frac{\partial P}{\partial x} = \frac{\nu}{K}\vec{u}_n + \left(\frac{Y}{\sqrt{K}}\right)\vec{u}_n^2$
Liu et al., 1999 van Gent, 1995 Lara et al., 2011 Wu and Hsiao, 2013	$f_d = -\alpha\nu\cdot\frac{(1-n)^2}{n^3 D_{50}^2}u_n - \beta\cdot\frac{(1-n)}{n^3 D_{50}^2}u_n^2$
Pramukti (2017)	$\vec{f}_d = -\alpha\cdot\frac{(1-n)^2}{n^3}\vec{u}_n - \beta\cdot\frac{(1-n)}{n^3}\vec{u}_n^2$

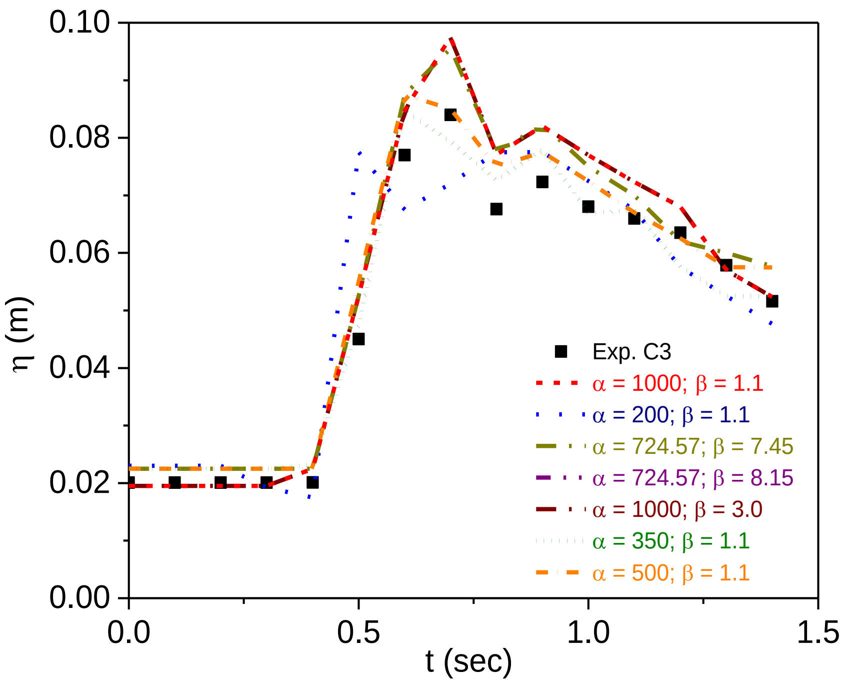


Figure 24. Comparison of measured and simulated wave heights for Case 3 (short porous obstacles) using Forchheimer model at location x = 1.18 m and y = 0.08 m.

Table 9. Simulation results of short porous obstacle Case 3 with Forchheimer method.

		Liu et al. (1999)	Van Gent (1995)	Lara et al. (2011)	Lara et al. (2011)	Wu and Hsiao (2013)	Pramukti (2017)
smallest grid size (mm)		Δx = 4 Δy = 4 Δz = 5					
Total grid number		268×40×80 = 857,600					
α , β		**200 , 1.1**	**1000 , 1.1**	**724.57 , 7.45**	**724.57 , 8.15**	**1000 , 3.0**	**500 , 1.1**
Wave height	Measured $\bar{\eta}$, η_{max} (m)	0.0502, 0.0840					
	Predicted $\bar{\eta}$, η_{max} (m)	0.0517, 0.0775	0.0548, 0.0975	0.0555, 0.0955	0.0555, 0.0955	0.0548, 0.0975	0.0540, 0.0875
	Δ	**1.4%, 8.4%**	**6.9%, 13.9%**	**8.2%, 12.1%**	**8.2%, 12.1%**	**6.9%, 13.9%**	**5.6%, 4.0%**
$\overline{\Delta t}$ (sec)		0.0007	0.0005	0.0005	0.0005	0.0004	0.0004
Computational time		26.8 hr	18.7 hr	17.9 hr	17.8 hr	18.7 hr	19.6 hr

Figure 25 compares the measured and simulated wave heights by the Forchheimer model for Case 4 at location x = 1.18 m and y = 0.08 m. The permeability constant α = 500 and the inertia factor β = 1.1 was set for Case 4. The difference between the simulated and measured average wave heights η was Δ = 1.2%. The difference between the simulated and measured maximum wave heights η was Δ = 6.7% (see Table 10). The simulation shows good results and can represent the results of experiments.

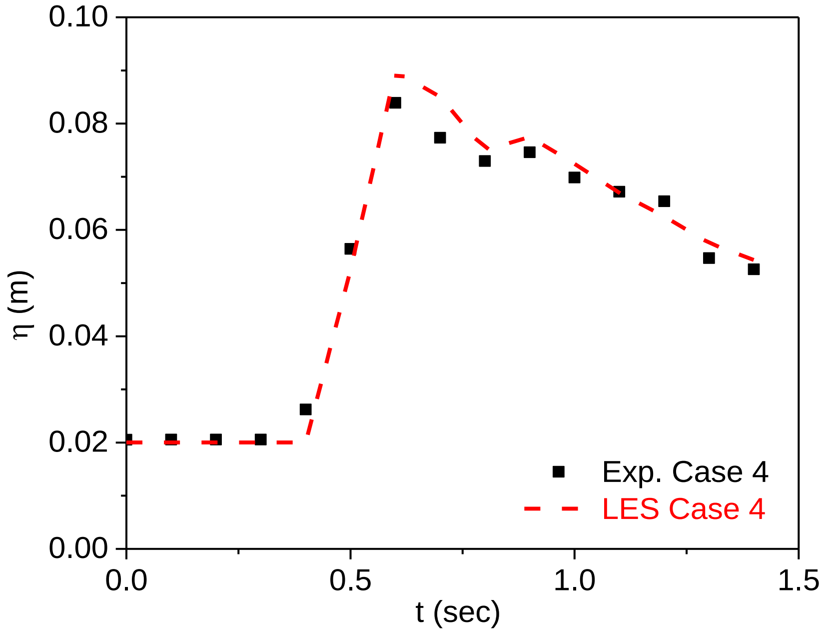


Figure 25. Comparison of measured and simulated wave heights for Case 4 (tall porous obstacles, h_f = 0.20 m, porosity n = 0.5) with the coefficients α = 500 and β = 1.1.

Table 10. Simulation results of short porous obstacle Case 4 with Forchheimer method.

		Grid A
smallest grid size (mm)		Δx = 4 Δy = 4 Δz = 5
Total grid number		857,600 268×40×80
α , β		500 , 1.1
Wave height	Measured $\bar{\eta}$, η_{max} (m)	0.0522, 0.0839
	Predicted $\bar{\eta}$, η_{max} (m)	0.0529, 0.0899
	Δ	**1.2%, 6.7%**
$\overline{\Delta t}$ (sec)		0.0005
Computational time		24.1 hr

Figure 26 and Table 11 compares the simulated wave heights for all cases at location x = 1.18 m and y = 0.08 m. The wave height of Case 2 is higher than others. It is because the wave hit the solid obstacle and jump up. In addition, the phenomena cause the uncertainity in the measured wave height.

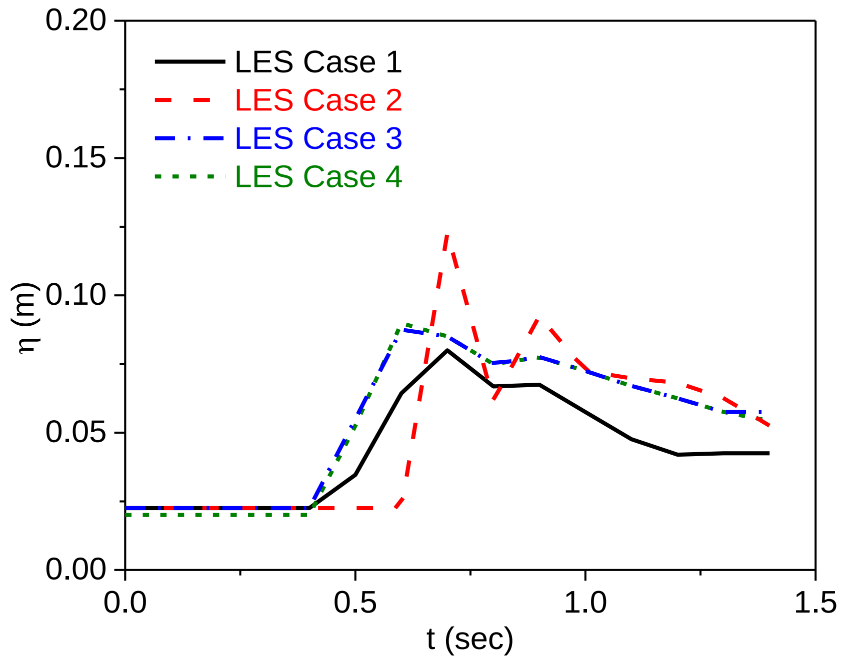


Figure 26. Comparison of simulated wave heights at location x = 1.18 m and y = 0.08 m for all cases.

Table 11. Simulation results of maximum and average wave height for all cases.

		Case 1	**Case 2**	**Case 3**	**Case 4**
Total grid number		268×40×80 = 857,600			
Wave height	Measured η_{max}, $\bar{\eta}$ (m)	0.0751, 0.0417	0.0954, 0.0457	0.0840, 0.0502	0.0838, 0.0522
	Predicted η_{max}, $\bar{\eta}$ (m)	0.0800, 0.0439	0.1378 , 0.0501	0.0875, 0.0540	0.0899, 0.0528
	Δ	**6.2%, 4.9%**	**4.7%, 4.0%**	**4.0%, 5.6%**	**6.7%, 1.2%**
$\overline{\Delta t}$ (sec)		0.0005	0.0006	0.0004	0.0004
Computational time		10.4 hr	13.5 hr	19.6 hr	24.1 hr

Table 12 compares the average error at location x = 1.18 m for all cases. The average error of the maximum wave height is defined as:

$$E(\%) = \frac{1}{n}\sum_{i=1}^{n}\frac{\left|\eta_{p,i} - \eta_{m,i}\right|}{\eta_{m,i}} \tag{17}$$

where $\eta_{p,i}$ and $\eta_{m,i}$ are the predicted and measured wave height at time t = 0.1 x i sec, and the total data number n = 14. The average error for Case 1, 2, 3 and 4 were 12.7%, 10.0%, 7.8% and 7.1%, respectively.

Table 12. Prediction errors for all four cases at location x = 1.18 m.

t	**Case 1**		**Case 2**		**Case 3**		**Case 4**	
(sec)	η_P	η_m	η_P	η_m	η_P	η_m	η_P	η_m
0.1	0.0225	0.0205	0.021	0.0201	0.0225	0.0201	0.0200	0.0206
0.2	0.0225	0.0205	0.021	0.0201	0.02253	0.0201	0.0200	0.0206
0.3	0.0225	0.0205	0.021	0.0201	0.0225	0.0201	0.0225	0.0206
0.4	0.0225	0.0205	0.021	0.0201	0.02314	0.0201	0.0165	0.0262
0.5	0.0346	0.0375	0.021	0.0201	0.0475	0.0450	0.0525	0.0564
0.6	0.0644	0.0712	0.021	0.0201	0.0847	0.0770	0.0899	0.0839
0.7	0.0800	0.0751	0.1378	0.1314	0.0794	0.0840	0.0850	0.0773
0.8	0.0668	0.0602	0.0409	0.0669	0.0726	0.0676	0.0750	0.0730
0.9	0.0675	0.0618	0.0951	0.0831	0.0780	0.0723	0.0775	0.0746
1	0.0575	0.0475	0.0778	0.0776	0.0670	0.0680	0.0725	0.0699
1.1	0.0476	0.0396	0.0720	0.0634	0.0674	0.0660	0.0670	0.0672
1.2	0.0420	0.0392	0.0710	0.0598	0.0576	0.0635	0.0625	0.0654
1.3	0.0425	0.0368	0.0599	0.0529	0.0525	0.0579	0.0575	0.0547
1.4	0.0425	0.0323	0.0503	0.0458	0.0525	0.0516	0.05435	0.0526
E (%)	**12.7**		**10.0**		**7.8**		**7.1**	

Figure 27, 28, and 29 illustrates the simulated velocity vectors on the central plane (y = 0.08 m) of the flume at six different times from t = 0.5 ~ 1.0 sec. The spatial distributions of the free surface elevation corresponding velocity fields. When the wave hit the obstacles, strong turbulent flows occur and lift up the water level.

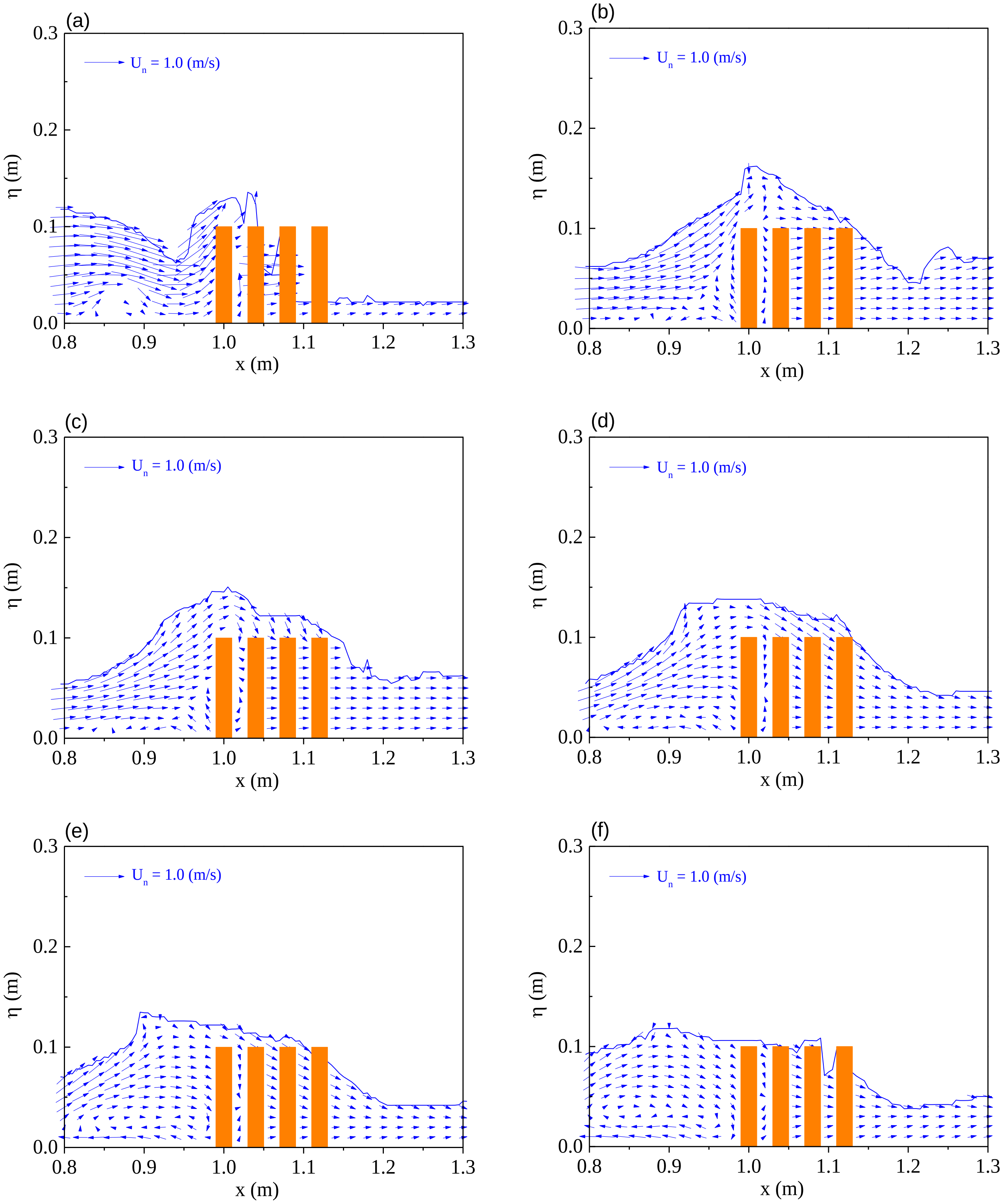


Figure 27. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 3 (short porous obstacles) uses solid method, $h_o = 0.26$ m; $h_1 = 0.02$ m; $h_f = 0.10$ m. (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s.

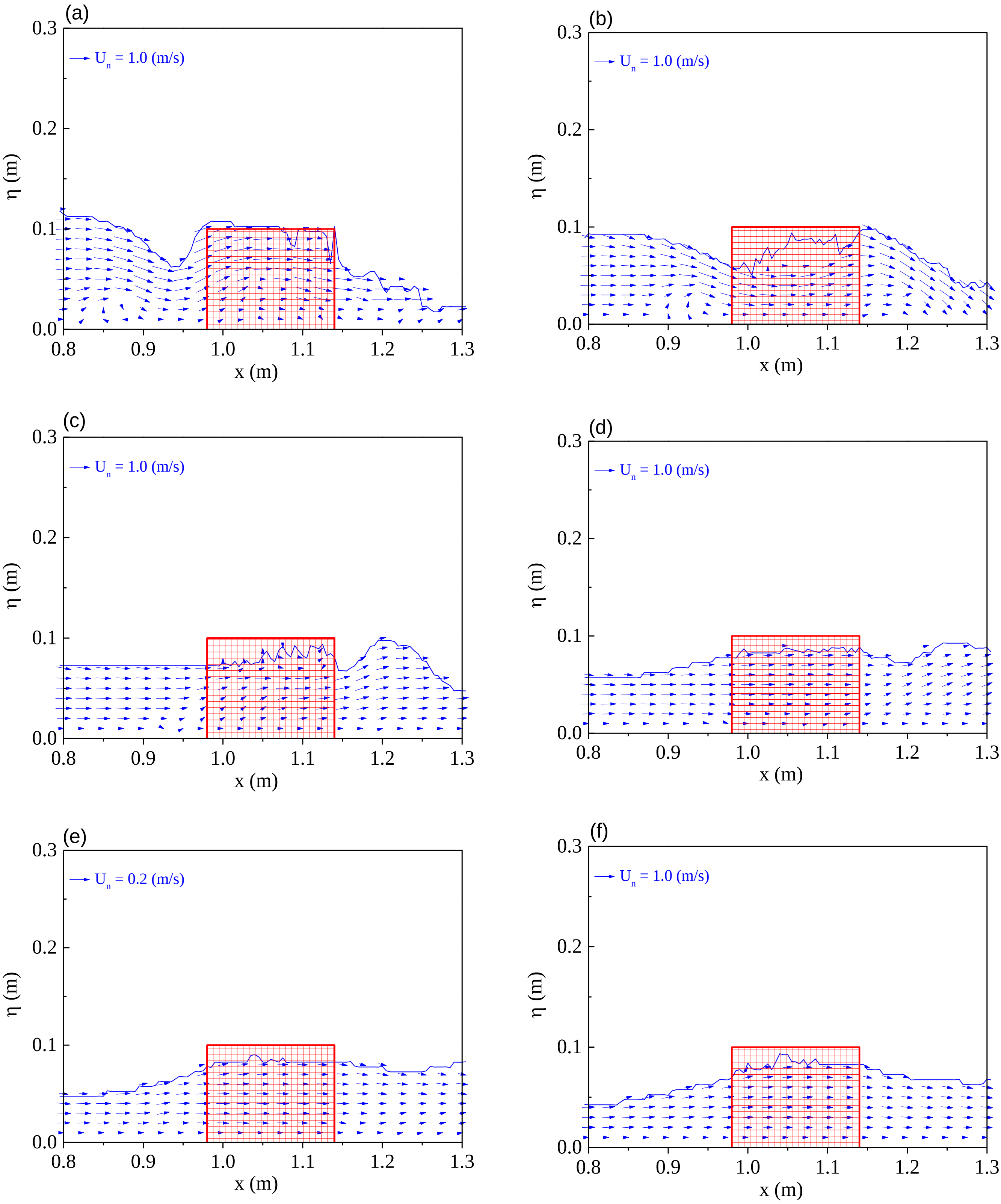


Figure 18. Velocity vectors on the central plane (y = 0.08 m) of the flume for Case 4 (tall porous obstacles, h_f = 0.10 m). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s.

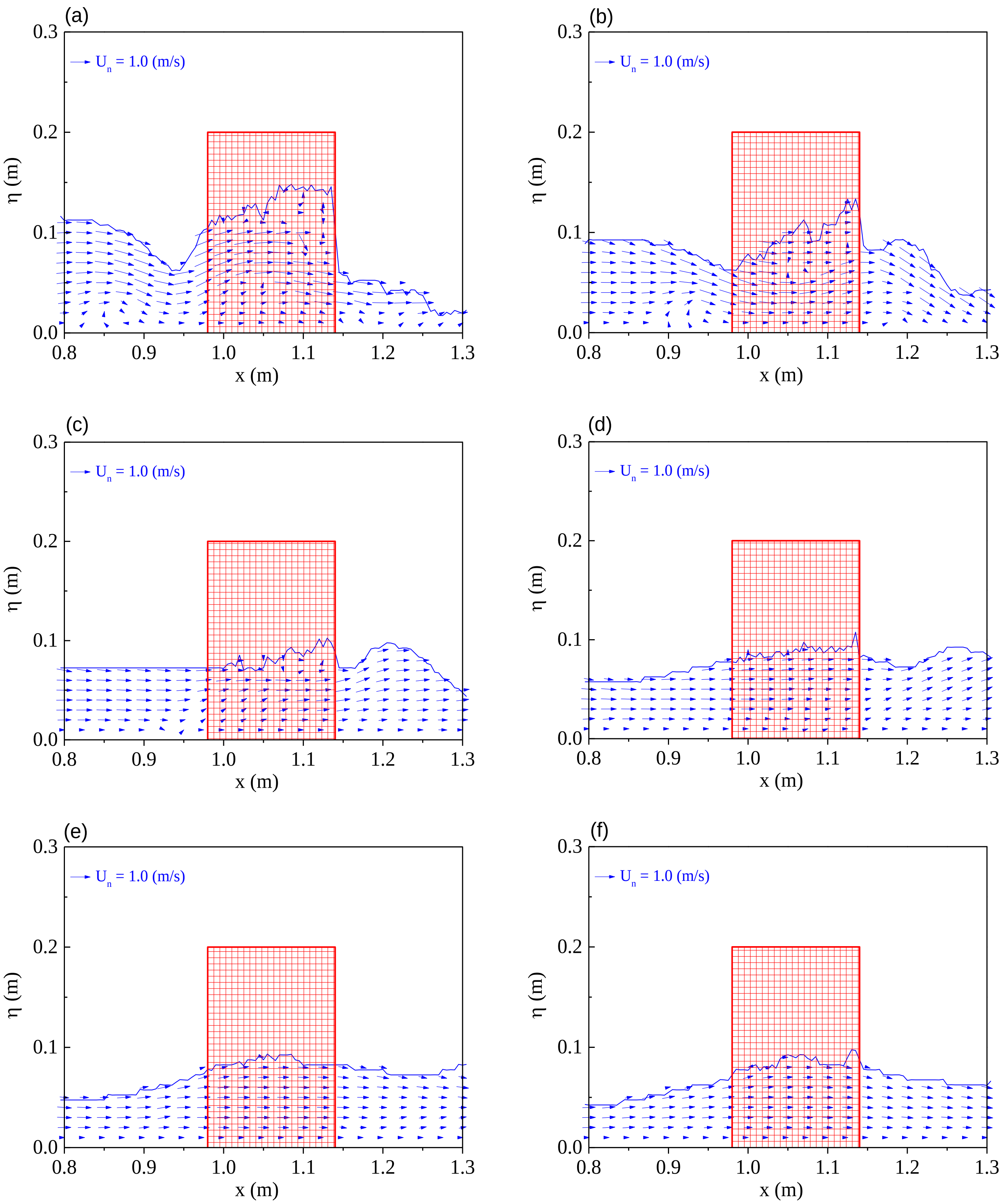


Figure 229. Velocity vectors on the central plane of the flume for Case 4 (tall porous obstacles). (a) t = 0.5 s; (b) t = 0.6 s; (c) t = 0.7 s; (d) t = 0.8 s; (e) t = 0.9 s; (f) t = 1.0 s.

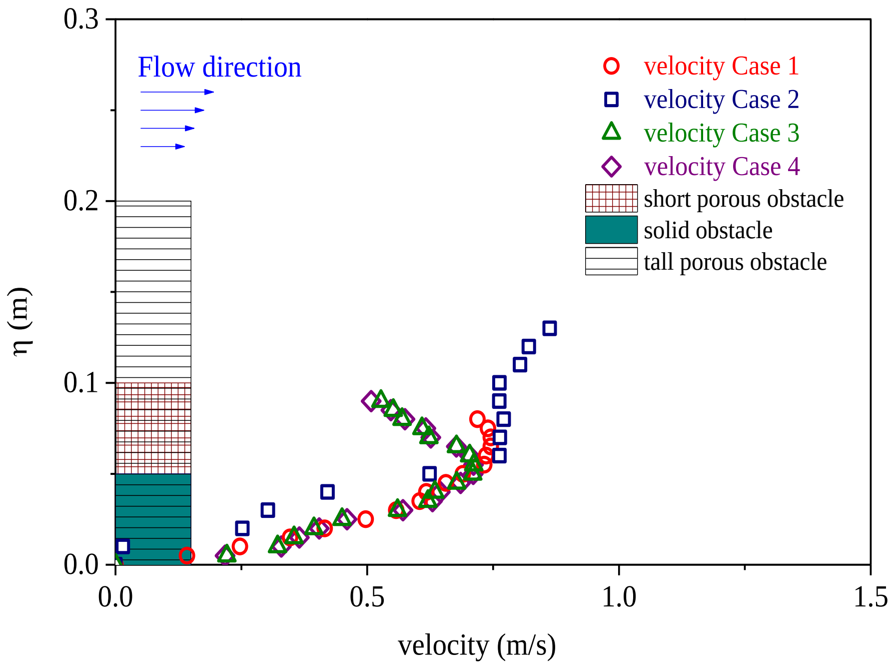


Figure 30. Comparison of instantaneous velocity behind the obstacles on the central plane of the flume at location x = 1.18 m at the same time t = 0.7 sec.

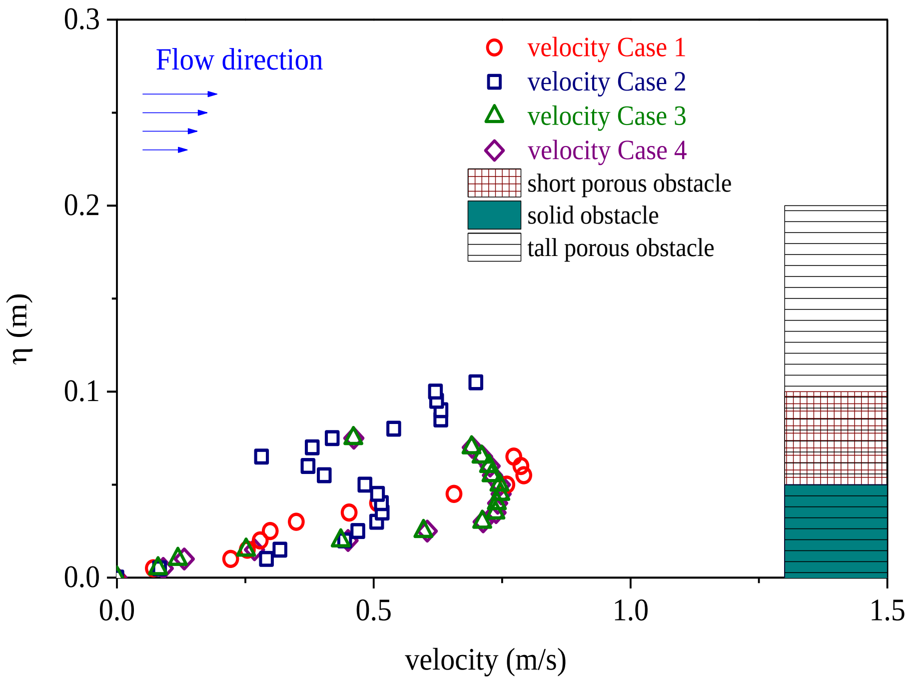


Figure 31. Comparison of instantaneous velocity in front of the obstacles on the central plane of the flume at location x = 0.92 m at the same time t = 0.7 sec.

Figure 30 and 31 compares the simulated instantaneous velocities at location x = 1.18 m for all cases at the same time. For Figure 30, shows the velocity distribution behind the obstacles. Case 1 obviously has the largest velocity, the maximum velocity is U = 0.75 m/s. The maximum velocity Case 2 is U = 1.01 m/s. The velocity distributions of Cases 3 and 4 are similar, and the maximum velocities were around U = 0.70 m/s. In other words, the porous obstacle can reduce the velocity effectively and Figure 31 shows the velocities distributions in front of the obstacle for all four cases. The velocity profiles were very similar and yet there is a change.

4.4. Drag Coefficient (C_D)

The validated Large Eddy Simulation model was used to investigate the interaction of free surface flow and porous obstacle. A series of numerical simulation were carried out to evaluate the effect of solid and porous obstacles on the dambreak flow. The simulated results of average and maximum wave heights are discussed.

Momentum integration method can be used to calculate the force acting on the object in a flow field. It is based on the momentum equation, and it has been used in wake flow, boundary layer flow and jet flow. In steady flow, the momentum equation can be simplified as:

$$F_x = \rho_{water}\left[\int_{up} U^2\, dA - \int_{down} U^2\, dA\right] \tag{18}$$

where F_x is the drag experienced by the water flow, U is velocity and $A = bh_f$ is the frontal area of the obstacle.

$$F_x = \rho_{water} b\left[\int_0^{h_f} u_{up}^2(z)\, dz - \int_0^{h_f} u_{down}^2(z)\, dz\right] \tag{19}$$

where $\rho_{water} = 1000$ kg/m^3 is the density of water, u_{up} and u_{down} are the velocity upstream and downstream of the obstacles, respectively.

The simulation results show that the flow through porous media is very well approximated by Forchheimer model (1901). The drag force is caused by the fluid impinging upon the obstacles. The drag force is a function of the fluid density and velocity, and the dimensionless drag coefficient is defined as:

$$C_D = \frac{F_x}{0.5\rho_{water}U_o^2 A} = \frac{F_x}{0.5\rho_{water}U_o^2(b \cdot h_f)} \qquad (20)$$

where $U_o = (gh_o)^{0.5} = 1.60$ m/s is a characteristic velocity of the dambreak flow.

Figure 32 shows the drag coefficients of dambreak flow through obstacles for Case 2, Case 3 and Case 4. Case 3 and Case 4, the value of drag coefficient is relatively similar. The highest values occurred at t = 0.80 sec, $C_d = 0.28$ for Case 3 and $C_d = 0.14$ for Case 4. There are some negative values in the Case 2, it is due to upstream point is too close to the obstacle. So, the velocity in the upstream tend to be smaller than in the downstream, due to the backflow.

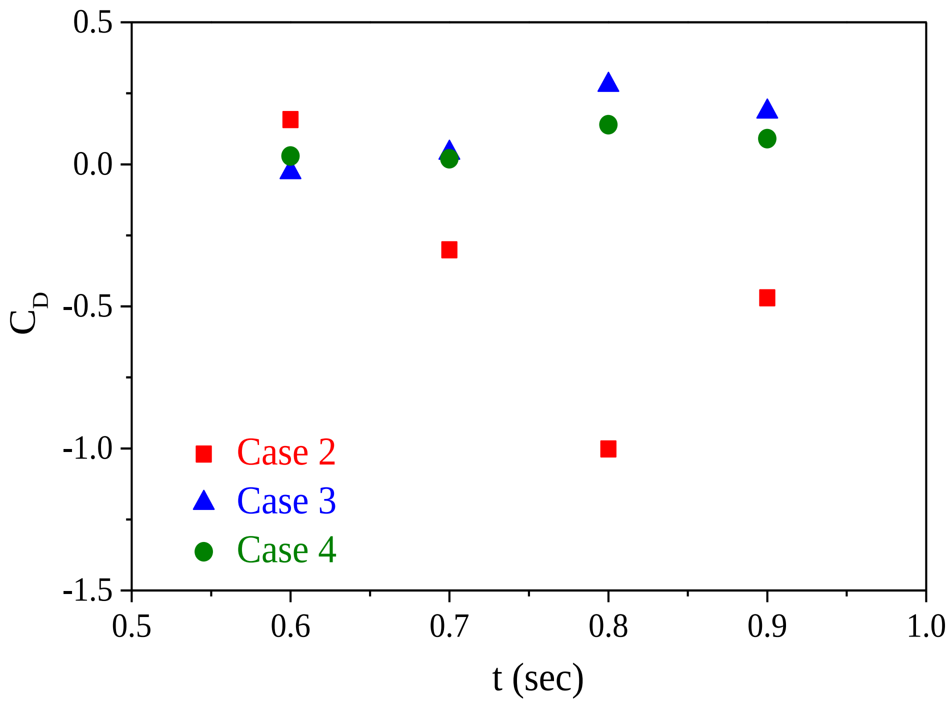


Figure 32. Drag coefficients of dambreak flow through obstacles at different times.

This page is intentionally left blank

CHAPTER V: Conclusions

This study used a three-dimensional Large Eddy Simulation (LES) model and Volume of fluid (VOF) method to investigate dambreak flows in the presence of porous obstacles in the channel. The porous obstacles could influence the dynamics of the dambreak flow. This study focused on the formation of turbulent flows around obstacles.

The close agreement between predicted water surfaces by the LES model and measured water levels in flume experiments domenstrates the capability of the present LES model in capturing the temporal variation of water depth and velocity. The downstream flow velocity was reduced by 30% in the presence of porous media on the flume.

The simulation of dambreak flow through porous media by using the model of Forchheimer (1901) shown great results. The values of permeability coefficient α and inertia factor β are essential for the simulation of the flow through the porous media. In this study, the permeability coefficient $\alpha = 500$ and inertia factor $\beta = 1.1$ respectively. In addition, the value of porosity is also very important in the prediction of flow through porous media.

Bibliography

[1] Aureli, F., Mignosa, P. and Tomirotti, M. (2000) Numerical simulation and experimental verification of dam-break flows with shocks, *J. of Hydraulic Research,* Vol. 38, No. 3, 197-206.

[2] Aureli, F., Maranzoni, A, Mignosa P., Ziveri, C. (2008) Dam-break flows: acquisition of experimental data through an imaging technique and 2D numerical modeling, *J. of Hydraulic Eng.,* Vol. 134, No. 8, 197-206.

[3] Bell, S.W., Elliot, R.C. and Chaudhry, M. (1992) Experimental results of two-dimensional dam-break flows, *J. of Hydraulic Research,* Vol. 30, No. 2, 225-252.

[4] Bellos, C.V., Soulis, V. and Sakkas, J.G. (1992) Experimental investigation of two-dimensional dam-break induced flows, *J. of Hydraulic Research*, Vol. 30, No. 1, 47-63.

[5] Cagatay, H. and Kocaman, S. (2010) Dam-break flows during initial stage using SWE and RANS approaches, *J. of Hydraulic Research,* Vol. 48, No. 5, 603-611.

[6] Cagatay, H. and Kocaman, S. (2011) Dam-break flow in the presence of obstacle: experimental and CFD simulation, *J. of Hydraulic Research,* Vol. 5, No. 4, 541-552.

[7] Chanson, H. (2009) Application of the method of characteristics to the dam break wave problem, *J. of Hydraulic Research,* Vol. 47, No. 1, 41-49.

[8] Chu, C.-R., Chung, C.-H., Wu, T.-R., Wang, C.-Y. (2016) Numerical simulation of hydrodynamic loading on fully submerged bridge decks, *J. of Hydraulic Eng. ASCE*. 143, 10.1061/(ASCE)HY.1943-7900.0001177.

[9] Eaket J., Hicks, F. and Peterson, A. (2005) Use of stereoscopy for Dam Break flow measurement. *J. of Hydraulic Eng. ASCE.* 10.1061/(ASCE) 0733-9429 (2005) 131:1 (24), 24-29.

[10] Elliot, R.C. and Chaudhry, M. (1992) A wave propagation model for two-dimensional dam-break flows, *J. of Hydraulic Research,* Vol. 30, No. 4, 467-483.

[11] Hu, K.-C., Hsiao, S.-C., Hwung, H.-H. and Wu, T.-R. (2012) Three-dimensional numerical modeling of the interaction of dam-break waves and porous media, *Advances in Water Resources,* No. 47 (2012), 14-30.

[12] Hu et al. (2014) Laboratory study on wave dissipation by vegetation in combined current-wave flow, *Coastal Engineering*, No. 88, 131 – 142.

[13] Hunt, B. (1987) An inviscid dam-break solution, *J. of Hydraulic Research,* Vol. 25, No. 3, 313-327.

[14] Larocque, L.A., Imran, J. and Chaudhry, M. (2013) 3D numerical simulation of partial breach dam-break flow using the LES and k-ε turbulence models, *J. of Hydraulic Research,* Vol. 51, No. 2, 145-157.

[15] Li, C.W. and Wang, J.H. (2002) Large Eddy Simulation of dispersion in free surface shear flow, *J. of Hydraulic Research,* Vol. 40, No. 3, 351-358.

[16] Lin, G.-F., Lai, J.-S. and Guo, W.-D. (2005) High-resolution TVD schemes in finite volume method for hydraulic shock wave modeling, *J. of Hydraulic Research,* Vol. 43, No. 4, 376-389.

[17] Miller, S., and Chaudhry, M.H. (1989) Dam-break flows in curved channel. *J. Hydraulics Eng*. 115(11), 1465-1478.

[18] Ritter A., (1992) Die Fortplanzung der Wasserwellen, *Zeitschriftdes Vereines Deutscher Ingeniere*, 36(33), 947-954.

[19] Soares-Frazao, S. (2007) Experiments of dam-break wave over a triangular bottom sill, *J. of Hydraulic Research,* Vol. 45, No. 1, 19-26.

[20] Soares-Frazao, S. and Zech, Y. (2007) Experimental study of dam-break flow against an isolated obstacle, *J. of Hydraulic Research*, Vol. 345 No. 1, 27-36.

[21] Soares-Frazao, S. and Zech, Y. (2008) Dam-break flow through an idealized city, *J. of Hydraulic Research,* Vol. 46, No. 5, 648-658.

[22] Stansby, P.K., et al. (1998) The initial stages of dam-break flow. *J. Fluid Mech*. 374, 407-424.

[23] Stoesser, T. (2014) Large-eddy simulation in hydraulics: Quo Vadis?, *J. of Hydraulic Research,* Vol. 52, No. 4, 441-452.

[24] Tseng, M.H., Hsu, C.A. and Chu, C.-R. (2001) Channel routing in open-channel flows with surges, *J. of Hydraulic Eng.*, ASCE, Vol. 127, No. 2, 115-122.

[25] Tseng, M.H. and Chu, C.-R. (2000) Simulation of dam-break flows by an improved Predictor-Corrector TVD Scheme, *Advances in Water Resources,* No. 23 (6), 637-643.

[26] Tseng, M.H. and Chu, C.-R. (2000) Two-dimensional shallow water flows simulation using TVD-MacCormack scheme, *J. of Hydraulic Research*, Vol. 38, No. 2, 123-131.

[27] Wu, Y.-T, Hsiao, S.-C. (2013) Propagation of solitary waves over a submerged permeable breakwater, *Coastal Engineering,* No. 81, 1 – 18.

[28] Wu, T-R., Chu, C.-R., Huang, C.-J., Wang, C.-Y., Chien, S.-Y. Chen M.-Z. (2014) A two-way coupled simulation of moving solids in free-surface flows, *Computers and Fluids.* Vol.100, 347-355.

[29] Xie, Z., Lin, B., Falconner, R.A. and Maddux, T.B. (2013) Large-eddy simulation of turbulent open-channel flow over three-dimensional dunes, *J. of Hydraulic Research,* Vol. 51, No.52, 494-505.

[30] Yang, C. Lin, B., Jiang, C. and Liu, Y. (2010) Predicting near-field dam-break flow and impact force using a 3D model, *J. of Hydraulic Research,* Vol. 48, No. 6, 784-792.

Appendix A

Input Files of Truchas

Case 1: Dambreak flow without obstacle

```
&MESH
   Ncell          = 78, 40, 40,
                    150, 0, 40,
                    40, 0, 0,
   Coord         = 0.0, 0.0, 0.0,
                   0.78, 0.16, 0.20,
                   1.38, 0.16, 0.40,
                   2.18, 0.16, 0.40,
   Fuzz          = 3*0.0
   Ratio         = 1.0, 1.0, 1.0,
                   1.0, 1.0, 1.0,
                   1.0, 1.0, 1.0,
   Partitions_Per_Process  = 1
/
```

```
&OUTPUTS
   Output_T                        = 0.0, 1.4
   Output_Dt                       = 0.01
   Int_Output_Dt_Multiplier      = 0
   Gra_Output_Dt_Multiplier     = 1
   Short_Output_Dt_Multiplier   = 1
   Long_Output_Dt_Multiplier    = 0
   User_Output_Dt_Multiplier    = 0
   Graphics_Format                = 'gmv'
/
```

```
&LINEAR_SOLVER
   name                          = 'projection'
   method                        = 'fgmres'
   preconditioning_method        = '2level'
   preconditioning_preconditioner = 'ssor'
   convergence_criterion         = 1.0e-4
   relaxation_parameter          = 1.4
   preconditioning_steps         = 2
   maximum_iterations            = 200
/
```

```
&NUMERICS
   volume_track_interfaces        = .true.
   volume_track_brents_method  = .true.
   volume_track_iter_tol           = 1.0e-8
   projection_linear_solution='projection'
   discrete_ops_type          = 'ortho'
   courant_number             = 0.90
   dt_init                    = 1.0e-6
   dt_grow                    = 1.1
   dt_min                     = 1.0e-7
   dt_max                     = 1.0
/
```

```
&PHYSICS
   Body_Force            = 0.0, 0.0, -9.81
   fluid_flow            = .true.
   heat_conduction     = .false.
   stokes                = .false.
   surface_tension_model  = 'off'
   csf_normal            = .false.
   csf_tangential       = .false.
   phase_change         = .false.
   inviscid              = .false.
   porous_flow         = .false.
   turbulence_model   = 'LES1'
   two_d                 = .false.
   turbulence_CS      = 0.15
/
```

```
&INTERFACES
   int_particles   = 5
   vof_particles  = 10
/
```

X1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_X = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /	X2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_X = 1.0 Conic_Constant = -2.18 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /
Y1 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /	Y2 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.16 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /
Z1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'no-slip' /	Z2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.4 Conic_Tolerance = 1.0e-6 BC_Variable = 'pressure' BC_Type = 'dirichlet' BC_Value = 0.0 /
1. Solid Body left &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
bottom &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside'	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02

Length = 0.02, 0.12, 0.02 Translation_Pt = 0.29, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	Translation_Pt = 0.33, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
right &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
2. INITIAL WATER DEPTH &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.30, 0.16, 0.260 Translation_Pt = 0.15, 0.08, 0.130 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /	INITIAL WATER FLUME &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 1.86, 0.16, 0.020 Translation_Pt = 1.25, 0.08, 0.010 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /
3. air &BODY Material_number = 3 Surface_Name = 'background' Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	Material_number = 1 : solid Material_number = 2 : water Material_number = 3 : air &MATERIAL Material_Name = 'solid' Material_Number = 1 priority = 1 Immobile = .true. Density = 1800.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Permeability_constant = 1800.0, 1800.0, 1800.0 /
&MATERIAL Material_Name = 'water' Material_Number = 2	&MATERIAL Material_Name = 'air' Material_Number = 3

```
   priority                  = 2
   Density                   = 1000.0
   Cp_Relation               = 'constant'
   Cp_Constants              = 1.0
   Cp_Exponents              = 0.0
   Conductivity_Relation     = 'constant'
   Conductivity_Constants    = 1.0
   Conductivity_Exponents    = 0.0
   Viscosity_Relation        = 'constant'
   Viscosity_Constants       = 1.0e-3
   Viscosity_Exponents       = 0.0
/
```

```
   Material_Feature          = 'background'
   priority                  = 3
   Density                   = 0.0
   Cp_Relation               = 'constant'
   Cp_Constants              = 1.0
   Cp_Exponents              = 0.0
   Conductivity_Relation     = 'constant'
   Conductivity_Constants    = 1.0
   Conductivity_Exponents    = 0.0
/

&PARALLEL_PARAMETERS
   Partitioner  = 'automatic',
/
```

Case 2: Dambreak flow with solid obstacle (h_f = 0.05 m)

```
&MESH
   Ncell          = 78, 40, 40,
                    150, 0, 40,
                    40, 0, 0,
   Coord          = 0.0, 0.0, 0.0,
                    0.78, 0.16, 0.20,
                    1.38, 0.16, 0.40,
                    2.18, 0.16, 0.40,
   Fuzz           = 3*0.0
   Ratio          = 1.0, 1.0, 1.0,
                    1.0, 1.0, 1.0,
                    1.0, 1.0, 1.0,
   Partitions_Per_Process  = 1
/
```

```
&OUTPUTS
   Output_T                        = 0.0, 1.4
   Output_Dt                       = 0.01
   Int_Output_Dt_Multiplier        = 0
   Gra_Output_Dt_Multiplier        = 1
   Short_Output_Dt_Multiplier      = 1
   Long_Output_Dt_Multiplier       = 0
   User_Output_Dt_Multiplier       = 0
   Graphics_Format                 = 'gmv'
/
```

```
&LINEAR_SOLVER
   name                               = 'projection'
   method                             = 'fgmres'
   preconditioning_method             = '2level'
   preconditioning_preconditioner = 'ssor'
   convergence_criterion              = 1.0e-4
   relaxation_parameter               = 1.4
   preconditioning_steps              = 2
   maximum_iterations                 = 200
/
```

```
&NUMERICS
   volume_track_interfaces           = .true.
   volume_track_brents_method  = .true.
   volume_track_iter_tol             = 1.0e-8
   projection_linear_solution='projection'
   discrete_ops_type            = 'ortho'
   courant_number               = 0.90
   dt_init                      = 1.0e-6
   dt_grow                      = 1.1
   dt_min                       = 1.0e-7
   dt_max                       = 1.0
/
```

```
&PHYSICS
   Body_Force            = 0.0, 0.0, -9.81
   fluid_flow            = .true.
   heat_conduction       = .false.
   stokes                = .false.
   surface_tension_model  = 'off'
   csf_normal            = .false.
   csf_tangential        = .false.
   phase_change          = .false.
   inviscid              = .false.
   porous_flow           = .false.
   turbulence_model      = 'LES1'
   two_d                 = .false.
   turbulence_CS         = 0.15
/
```

```
&INTERFACES
   int_particles   = 5
   vof_particles   = 10
/
```

```
X1
&BC
   Surface_Name      = 'conic'
   Conic_Relation    = '='
   Conic_X           = 1.0
```

```
X2
&BC
   Surface_Name      = 'conic'
   Conic_Relation    = '='
   Conic_X           = 1.0
```

Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /	Conic_Constant = -2.18 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /
Y1 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /	Y2 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.16 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /
Z1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'no-slip' /	Z2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.4 Conic_Tolerance = 1.0e-6 BC_Variable = 'pressure' BC_Type = 'dirichlet' BC_Value = 0.0 /
1. Solid Body left &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
bottom &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.29, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.33, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /

right &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
2. Solid Obstacle &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.16, 0.16, 0.05 Translation_Pt = 1.06, 0.08, 0.025 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	3. INITIAL WATER DEPTH &BODY Material_number = 3 Surface_Name = 'box' Fill = 'inside' Length = 0.30, 0.16, 0.260 Translation_Pt = 0.15, 0.08, 0.130 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /
INITIAL WATER FLUME &BODY Material_number = 3 Surface_Name = 'box' Fill = 'inside' Length = 0.66, 0.16, 0.020 Translation_Pt = 0.63, 0.08, 0.010 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /	&BODY Material_number = 3 Surface_Name = 'box' Fill = 'inside' Length = 1.04, 0.16, 0.020 Translation_Pt = 1.66, 0.08, 0.010 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /
4. air &BODY Material_number = 3 Surface_Name = 'background' Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	Material_number = 1 : solid Material_number = 2 : obstacle Material_number = 3 : water Material_number = 4 : air &MATERIAL Material_Name = 'solid' Material_Number = 1 priority = 1 Immobile = .true. Density = 1800.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0

	Permeability_constant = 1800.0, 1800.0, 1800.0 /
&MATERIAL Material_Name = 'obstacle' Material_Number = 2 priority = 2 Immobile = .true. Density = 1800.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Permeability_constant = 1800.0, 1800.0, 1800.0 /	&MATERIAL Material_Name = 'water' Material_Number = 3 priority = 3 Density = 1000.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Viscosity_Relation = 'constant' Viscosity_Constants = 1.0e-3 Viscosity_Exponents = 0.0 /
&MATERIAL Material_Name = 'air' Material_Number = 4 Material_Feature = 'background' priority = 4 Density = 0.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 /	&PARALLEL_PARAMETERS Partitioner = 'automatic', /

Case 3: Dambreak flow through porous obstacle (h_f = 0.10 m)

A. Solid Method

```
&MESH
   Ncell          = 78, 40, 80,
                    150, 0, 0,
                    40, 0, 0,
   Coord          = 0.0, 0.0, 0.0,
                    0.78, 0.16, 0.40,
                    1.38, 0.16, 0.40,
                    2.18, 0.16, 0.40,
   Fuzz           = 3*0.0
   Ratio          = 0.979, 1.0, 1.0,
                    1.0, 1.0, 1.0,
                    1.07, 1.0, 1.0,
   Partitions_Per_Process  = 1
/
```

```
&OUTPUTS
   Output_T                       = 0.0, 1.4
   Output_Dt                      = 0.01
   Int_Output_Dt_Multiplier       = 0
   Gra_Output_Dt_Multiplier       = 1
   Short_Output_Dt_Multiplier     = 1
   Long_Output_Dt_Multiplier      = 0
   User_Output_Dt_Multiplier      = 0
   Graphics_Format                = 'gmv'
/
```

```
&LINEAR_SOLVER
   name                             = 'projection'
   method                           = 'fgmres'
   preconditioning_method           = '2level'
   preconditioning_preconditioner = 'ssor'
   convergence_criterion            = 1.0e-4
   relaxation_parameter             = 1.4
   preconditioning_steps            = 2
   maximum_iterations               = 200
/
```

```
&NUMERICS
   volume_track_interfaces          = .true.
   volume_track_brents_method  = .true.
   volume_track_iter_tol            = 1.0e-8
   projection_linear_solution='projection'
   discrete_ops_type            = 'ortho'
   courant_number               = 0.90
   dt_init                          = 1.0e-6
   dt_grow                          = 1.1
   dt_min                           = 1.0e-7
   dt_max                           = 1.0
/
```

```
&PHYSICS
   Body_Force             = 0.0, 0.0, -9.81
   fluid_flow             = .true.
   heat_conduction        = .false.
   stokes                 = .false.
   surface_tension_model  = 'off'
   csf_normal             = .false.
   csf_tangential         = .false.
   phase_change           = .false.
   inviscid               = .false.
   porous_flow            = .false.
   turbulence_model       = 'LES1'
   two_d                  = .false.
   turbulence_CS          = 0.15
/
```

```
&INTERFACES
   int_particles   = 5
   vof_particles   = 10
/
```

```
X1
&BC
   Surface_Name     = 'conic'
   Conic_Relation   = '='
```

```
X2
&BC
   Surface_Name     = 'conic'
   Conic_Relation   = '='
```

Conic_X = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /	Conic_X = 1.0 Conic_Constant = -2.18 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /
Y1 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /	Y2 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.16 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /
Z1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'no-slip' /	Z2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.4 Conic_Tolerance = 1.0e-6 BC_Variable = 'pressure' BC_Type = 'dirichlet' BC_Value = 0.0 /
1. Solid Body left &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.29, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.33, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /

right &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.15, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
2. Porous Obstacle plane &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.16, 0.16, 0.0050 Translation_Pt = 1.06, 0.08, 0.00250 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	1 the first &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.00, 0.155, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
2 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.00, 0.12, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	3 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.00, 0.08, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
4 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.00, 0.04, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0	5 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.00, 0.005, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0

Temperature = 273.0 /	Temperature = 273.0 /
6 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.04, 0.14, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	7 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.04, 0.10, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
8 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.04, 0.06, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	9 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.04, 0.02, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
10 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.08, 0.155, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	11 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.08, 0.12, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
12 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10	13 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10

Translation_Pt = 1.08, 0.08, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	Translation_Pt = 1.08, 0.04, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
14 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.08, 0.005, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	15 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.12, 0.14, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
16 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.10 Translation_Pt = 1.12, 0.10, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	17 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.12, 0.06, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /
18 &BODY Material_number = 2 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.01, 0.10 Translation_Pt = 1.12, 0.02, 0.055 Rotation_Pt = 0.0, 0.0, 0.0 Rotation_Angle = 0.0, 0.0, 0.0 Velocity = 0.0 Temperature = 273.0 /	3. INITIAL WATER DEPTH &BODY Material_number = 3 Surface_Name = 'box' Fill = 'inside' Length = 0.30, 0.16, 0.260 Translation_Pt = 0.15, 0.08, 0.130 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /
INITIAL WATER FLUME	4. air

&BODY Material_number = 3 Surface_Name = 'box' Fill = 'inside' Length = 1.86, 0.16, 0.020 Translation_Pt = 1.25, 0.08, 0.010 Velocity = 0.0, 0.0, 0.0 Temperature = 293.0 /	&BODY Material_number = 3 Surface_Name = 'background' Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
Material_number = 1 : solid Material_number = 2 : obstacle Material_number = 3 : water Material_number = 4 : air &MATERIAL Material_Name = 'solid' Material_Number = 1 priority = 1 Immobile = .true. Density = 1800.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Permeability_constant = 1800.0, 1800.0, 1800.0 /	&MATERIAL Material_Name = 'obstacle' Material_Number = 2 priority = 2 Immobile = .true. Density = 1800.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Permeability_constant = 1800.0, 1800.0, 1800.0 /
&MATERIAL Material_Name = 'water' Material_Number = 3 priority = 3 Density = 1000.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Viscosity_Relation = 'constant' Viscosity_Constants = 1.0e-3 Viscosity_Exponents = 0.0 /	&MATERIAL Material_Name = 'air' Material_Number = 4 Material_Feature = 'background' priority = 4 Density = 0.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 / &PARALLEL_PARAMETERS Partitioner = 'automatic', /

Case 3: Dambreak flow through porous obstacle (h_f = 0.10 m)

B. Porous Method

&MESH Ncell = 78, 40, 80, 150, 0, 0, 40, 0, 0, Coord = 0.0, 0.0, 0.0, 0.78, 0.16, 0.40, 1.38, 0.16, 0.40, 2.18, 0.16, 0.40, Fuzz = 3*0.0 Ratio = 0.979, 1.0, 1.0, 1.0, 1.0, 1.0, 1.07, 1.0, 1.0, Partitions_Per_Process = 1 /	&OUTPUTS Output_T = 0.0, 1.4 Output_Dt = 0.01 Int_Output_Dt_Multiplier = 0 Gra_Output_Dt_Multiplier = 1 Short_Output_Dt_Multiplier = 1 Long_Output_Dt_Multiplier = 0 User_Output_Dt_Multiplier = 0 Graphics_Format = 'gmv' /
&LINEAR_SOLVER name = 'projection' method = 'fgmres' preconditioning_method = '2level' preconditioning_preconditioner = 'ssor' convergence_criterion = 1.0e-4 relaxation_parameter = 1.4 preconditioning_steps = 2 maximum_iterations = 200 /	&NUMERICS volume_track_interfaces = .true. volume_track_brents_method = .true. volume_track_iter_tol = 1.0e-8 projection_linear_solution='projection' discrete_ops_type = 'ortho' courant_number = 0.90 dt_init = 1.0e-6 dt_grow = 1.1 dt_min = 1.0e-7 dt_max = 1.0 /
&PHYSICS Body_Force = 0.0, 0.0, -9.81 fluid_flow = .true. heat_conduction = .false. stokes = .false. surface_tension_model = 'off' csf_normal = .false. csf_tangential = .false. phase_change = .false. inviscid = .false. porous_flow = .false. turbulence_model = 'LES1' two_d = .false. turbulence_CS = 0.15 /	&INTERFACES int_particles = 5 vof_particles = 10 /
X1 &BC Surface_Name = 'conic' Conic_Relation = '='	X2 &BC Surface_Name = 'conic' Conic_Relation = '='

Conic_X = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /	Conic_X = 1.0 Conic_Constant = -2.18 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /
Y1 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /	Y2 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.16 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /
Z1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'no-slip' /	Z2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.4 Conic_Tolerance = 1.0e-6 BC_Variable = 'pressure' BC_Type = 'dirichlet' BC_Value = 0.0 /
1. Solid Body left &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.29, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.33, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /

```
right
&BODY
     Material_number     = 1
     Surface_Name        = 'box'
     Fill                = 'inside'
     Length              = 0.02, 0.02, 0.4
     Translation_Pt      = 0.29, 0.15, 0.2
     Velocity            = 0.0, 0.0, 0.0
     Temperature         = 273.0
/
```

```
&BODY
     Material_number     = 1
     Surface_Name        = 'box'
     Fill                = 'inside'
     Length              = 0.02, 0.02, 0.4
     Translation_Pt      = 0.33, 0.15, 0.2
     Velocity            = 0.0, 0.0, 0.0
     Temperature         = 273.0
/
```

```
2. Porous Obstacle
plane
&BODY
     Material_number     = 2
     Surface_Name        = 'box'
     Fill                = 'inside'
     Length              = 0.16, 0.16, 0.10
     Translation_Pt      = 1.06, 0.08, 0.05
     Rotation_Pt         = 0.0, 0.0, 0.0
     Rotation_Angle      = 0.0, 0.0, 0.0
     Velocity            = 0.0
     Temperature         = 273.0
     Porosity            = 0.5
/
```

```
3. INITIAL WATER DEPTH
&BODY
     Material_number     = 3
     Surface_Name        = 'box'
     Fill                = 'inside'
     Length              = 0.30, 0.16, 0.260
     Translation_Pt      = 0.15, 0.08, 0.130
     Velocity            = 0.0, 0.0, 0.0
     Temperature         = 293.0
/
```

```
INITIAL WATER FLUME
&BODY
     Material_number     = 3
     Surface_Name        = 'box'
     Fill                = 'inside'
     Length              = 1.86, 0.16,
0.020
     Translation_Pt      = 1.25, 0.08,
0.010
     Velocity            = 0.0, 0.0, 0.0
     Temperature         = 293.0
/
```

```
4. air
&BODY
     Material_number     = 3
     Surface_Name        = 'background'
     Velocity            = 0.0, 0.0, 0.0
     Temperature         = 273.0
/
```

```
Material_number = 1 : solid
Material_number = 2 : obstacle
Material_number = 3 : water
Material_number = 4 : air

&MATERIAL
   Material_Name         = 'solid'
   Material_Number       = 1
   priority              = 1
   Immobile              = .true.
   Density               = 1800.0
   Cp_Relation           = 'constant'
   Cp_Constants          = 1.0
```

```
&MATERIAL
   Material_Name            = 'obstacle'
   Material_Number          = 2
   priority                 = 2
   Immobile                 = .true.
   Density                  = 1800.0
   Cp_Relation              = 'constant'
   Cp_Constants             = 1.0
   Cp_Exponents             = 0.0
   Conductivity_Relation    = 'constant'
   Conductivity_Constants   = 1.0
   Conductivity_Exponents   = 0.0
```

Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Permeability_constant = 1800.0, 1800.0, 1800.0 /	Permeability_constant = 500.0, 10e5, 10e5 /
&MATERIAL Material_Name = 'water' Material_Number = 3 priority = 3 Density = 1000.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 Viscosity_Relation = 'constant' Viscosity_Constants = 1.0e-3 Viscosity_Exponents = 0.0 /	&MATERIAL Material_Name = 'air' Material_Number = 4 Material_Feature = 'background' priority = 4 Density = 0.0 Cp_Relation = 'constant' Cp_Constants = 1.0 Cp_Exponents = 0.0 Conductivity_Relation = 'constant' Conductivity_Constants = 1.0 Conductivity_Exponents = 0.0 / &PARALLEL_PARAMETERS Partitioner = 'automatic', /

Case 4: Dambreak flow through porous obstacle (h_f = 0.20 m)

A. Porous Method

```
&MESH
   Ncell          = 78, 40, 80,
                    150, 0, 0,
                    40, 0, 0,
   Coord          = 0.0, 0.0, 0.0,
                    0.78, 0.16, 0.40,
                    1.38, 0.16, 0.40,
                    2.18, 0.16, 0.40,
   Fuzz           = 3*0.0
   Ratio          = 0.979, 1.0, 1.0,
                    1.0, 1.0, 1.0,
                    1.07, 1.0, 1.0,
   Partitions_Per_Process  = 1
/
```

```
&OUTPUTS
   Output_T                        = 0.0, 1.4
   Output_Dt                       = 0.01
   Int_Output_Dt_Multiplier        = 0
   Gra_Output_Dt_Multiplier        = 1
   Short_Output_Dt_Multiplier      = 1
   Long_Output_Dt_Multiplier       = 0
   User_Output_Dt_Multiplier       = 0
   Graphics_Format                 = 'gmv'
/
```

```
&LINEAR_SOLVER
   name                            = 'projection'
   method                          = 'fgmres'
   preconditioning_method          = '2level'
   preconditioning_preconditioner  = 'ssor'
   convergence_criterion           = 1.0e-4
   relaxation_parameter            = 1.4
   preconditioning_steps           = 2
   maximum_iterations              = 200
/
```

```
&NUMERICS
   volume_track_interfaces         = .true.
   volume_track_brents_method      = .true.
   volume_track_iter_tol           = 1.0e-8
   projection_linear_solution='projection'
   discrete_ops_type               = 'ortho'
   courant_number                  = 0.90
   dt_init                         = 1.0e-6
   dt_grow                         = 1.1
   dt_min                          = 1.0e-7
   dt_max                          = 1.0
/
```

```
&PHYSICS
   Body_Force             = 0.0, 0.0, -9.81
   fluid_flow             = .true.
   heat_conduction        = .false.
   stokes                 = .false.
   surface_tension_model  = 'off'
   csf_normal             = .false.
   csf_tangential         = .false.
   phase_change           = .false.
   inviscid               = .false.
   porous_flow            = .false.
   turbulence_model       = 'LES1'
   two_d                  = .false.
   turbulence_CS          = 0.15
/
```

```
&INTERFACES
   int_particles   = 5
   vof_particles   = 10
/
```

```
X1
&BC
   Surface_Name     = 'conic'
   Conic_Relation   = '='
```

```
X2
&BC
   Surface_Name     = 'conic'
   Conic_Relation   = '='
```

Conic_X = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /	Conic_X = 1.0 Conic_Constant = -2.18 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'dirichlet' BC_Value = 0.0 /
Y1 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /	Y2 free-slip &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Y = 1.0 Conic_Constant = -0.16 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'free-slip' /
Z1 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.0 Conic_Tolerance = 1.0e-6 BC_Variable = 'velocity' BC_Type = 'no-slip' /	Z2 &BC Surface_Name = 'conic' Conic_Relation = '=' Conic_Z = 1.0 Conic_Constant = -0.4 Conic_Tolerance = 1.0e-6 BC_Variable = 'pressure' BC_Type = 'dirichlet' BC_Value = 0.0 /
1. Solid Body left &BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.29, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.02, 0.4 Translation_Pt = 0.33, 0.01, 0.2 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /
&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.29, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /	&BODY Material_number = 1 Surface_Name = 'box' Fill = 'inside' Length = 0.02, 0.12, 0.02 Translation_Pt = 0.33, 0.08, 0.01 Velocity = 0.0, 0.0, 0.0 Temperature = 273.0 /

```
right
&BODY
    Material_number     = 1
    Surface_Name        = 'box'
    Fill                = 'inside'
    Length              = 0.02, 0.02, 0.4
    Translation_Pt      = 0.29, 0.15, 0.2
    Velocity            = 0.0, 0.0, 0.0
    Temperature         = 273.0
/
```

```
&BODY
    Material_number     = 1
    Surface_Name        = 'box'
    Fill                = 'inside'
    Length              = 0.02, 0.02, 0.4
    Translation_Pt      = 0.33, 0.15, 0.2
    Velocity            = 0.0, 0.0, 0.0
    Temperature         = 273.0
/
```

```
2. Porous Obstacle
plane
&BODY
    Material_number     = 2
    Surface_Name        = 'box'
    Fill                = 'inside'
    Length              = 0.16, 0.16, 0.20
    Translation_Pt      = 1.06, 0.08, 0.10
    Rotation_Pt         = 0.0, 0.0, 0.0
    Rotation_Angle      = 0.0, 0.0, 0.0
    Velocity            = 0.0
    Temperature         = 273.0
    Porosity            = 0.5
/
```

```
3. INITIAL WATER DEPTH
&BODY
    Material_number     = 3
    Surface_Name        = 'box'
    Fill                = 'inside'
    Length              = 0.30, 0.16, 0.260
    Translation_Pt      = 0.15, 0.08, 0.130
    Velocity            = 0.0, 0.0, 0.0
    Temperature         = 293.0
/
```

```
INITIAL WATER FLUME
&BODY
    Material_number     = 3
    Surface_Name        = 'box'
    Fill                = 'inside'
    Length              = 1.86, 0.16,
0.020
    Translation_Pt      = 1.25, 0.08,
0.010
    Velocity            = 0.0, 0.0, 0.0
    Temperature         = 293.0
/
```

```
4. air
&BODY
    Material_number     = 3
    Surface_Name        = 'background'
    Velocity            = 0.0, 0.0, 0.0
    Temperature         = 273.0
/
```

```
Material_number = 1 : solid
Material_number = 2 : obstacle
Material_number = 3 : water
Material_number = 4 : air

&MATERIAL
   Material_Name        = 'solid'
   Material_Number      = 1
   priority             = 1
   Immobile             = .true.
   Density              = 1800.0
   Cp_Relation          = 'constant'
   Cp_Constants         = 1.0
```

```
&MATERIAL
   Material_Name             = 'obstacle'
   Material_Number           = 2
   priority                  = 2
   Immobile                  = .true.
   Density                   = 1800.0
   Cp_Relation               = 'constant'
   Cp_Constants              = 1.0
   Cp_Exponents              = 0.0
   Conductivity_Relation     = 'constant'
   Conductivity_Constants    = 1.0
   Conductivity_Exponents    = 0.0
```

```
   Cp_Exponents            = 0.0
   Conductivity_Relation     = 'constant'
   Conductivity_Constants   = 1.0
   Conductivity_Exponents  = 0.0
   Permeability_constant      = 1800.0,
  1800.0, 1800.0
/
```

```
   Permeability_constant    = 500.0, 10e5,
    10e5
/
```

```
&MATERIAL
   Material_Name           = 'water'
   Material_Number         = 3
   priority                        = 3
   Density                        = 1000.0
   Cp_Relation                = 'constant'
   Cp_Constants               = 1.0
   Cp_Exponents              = 0.0
   Conductivity_Relation     = 'constant'
   Conductivity_Constants   = 1.0
   Conductivity_Exponents  = 0.0
   Viscosity_Relation     = 'constant'
   Viscosity_Constants              = 1.0e-3
   Viscosity_Exponents             = 0.0
/
```

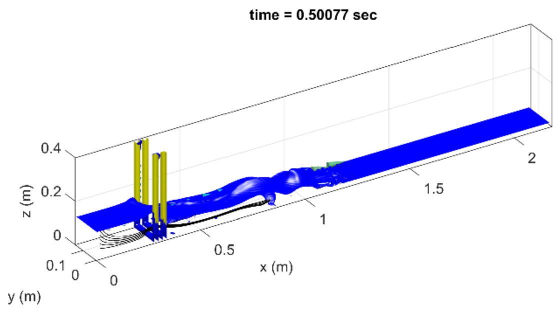
```
&MATERIAL
   Material_Name           = 'air'
   Material_Number        = 4
   Material_Feature        = 'background'
   priority                        = 4
   Density                       = 0.0
   Cp_Relation               = 'constant'
   Cp_Constants              = 1.0
   Cp_Exponents             = 0.0
   Conductivity_Relation   = 'constant'
   Conductivity_Constants  = 1.0
   Conductivity_Exponents  = 0.0
/

&PARALLEL_PARAMETERS
   Partitioner  = 'automatic',
/
```

Appendix B

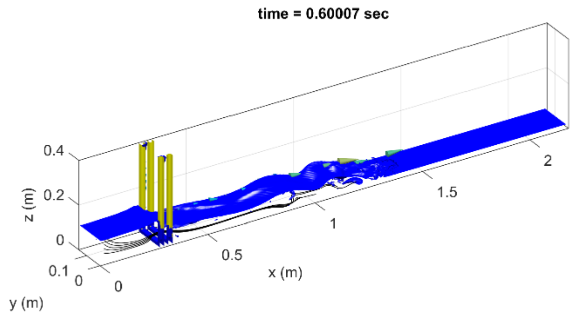
Snapshots of Dambreak

Case 1: Dambreak flow without obstacle

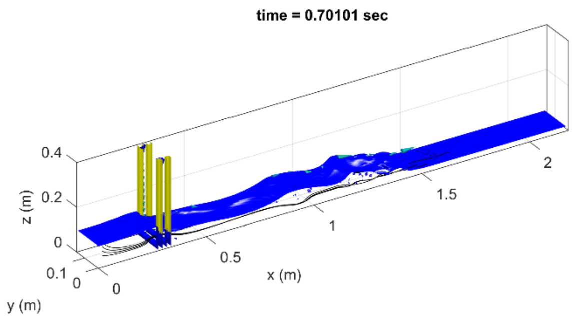
(a)



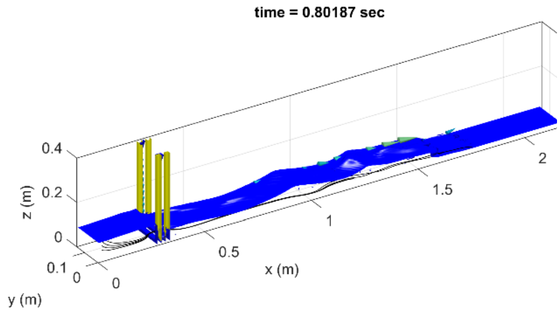
(b)



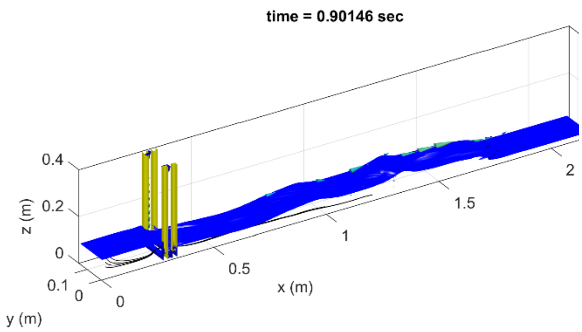
(c)



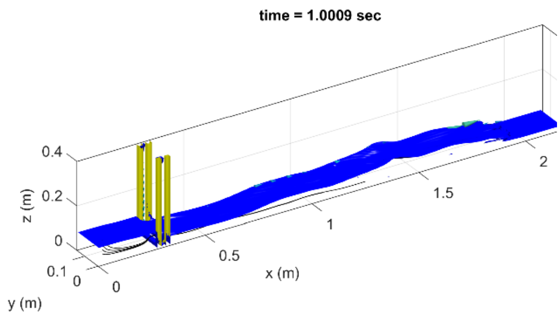
(d)



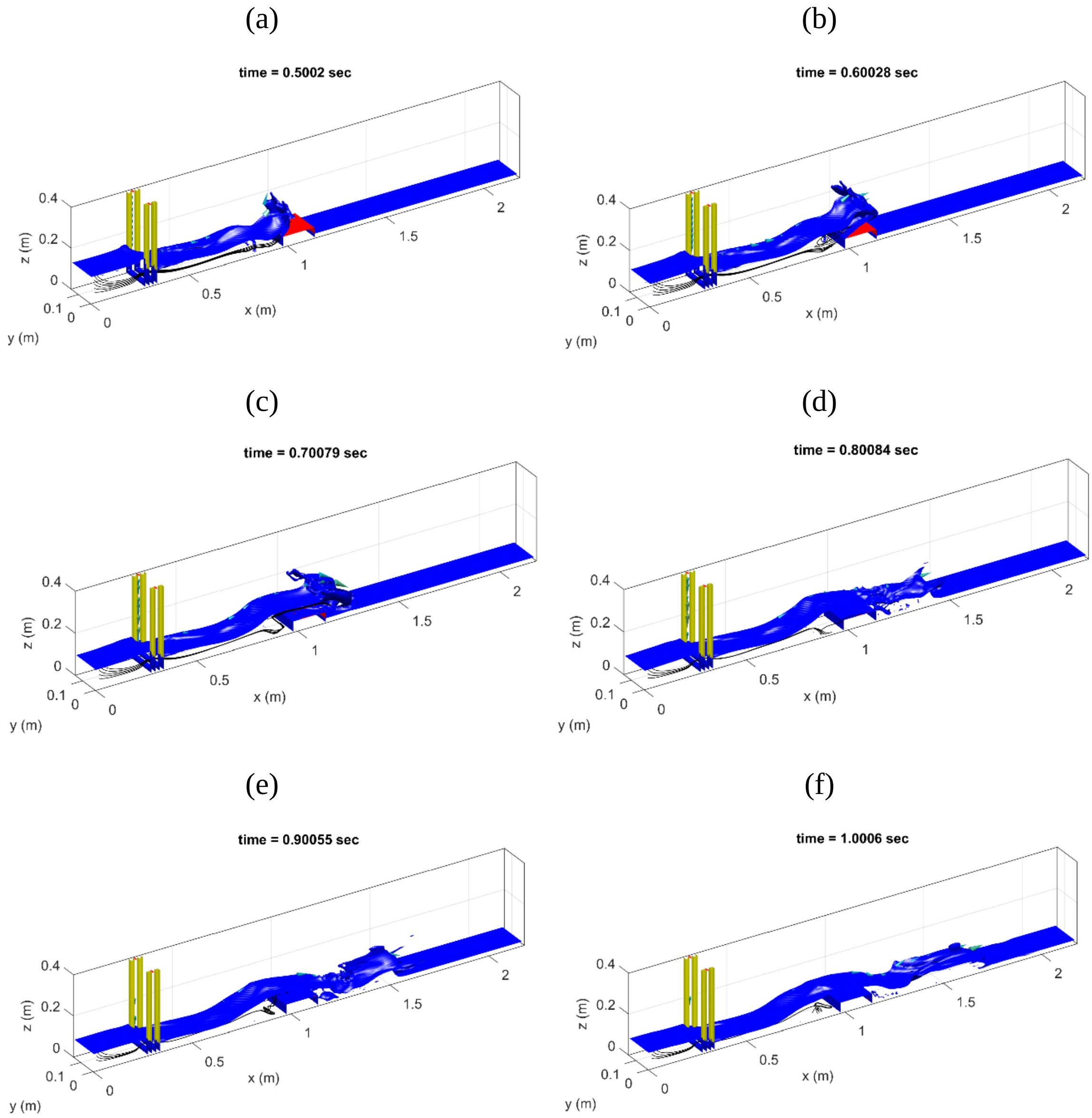
(e)



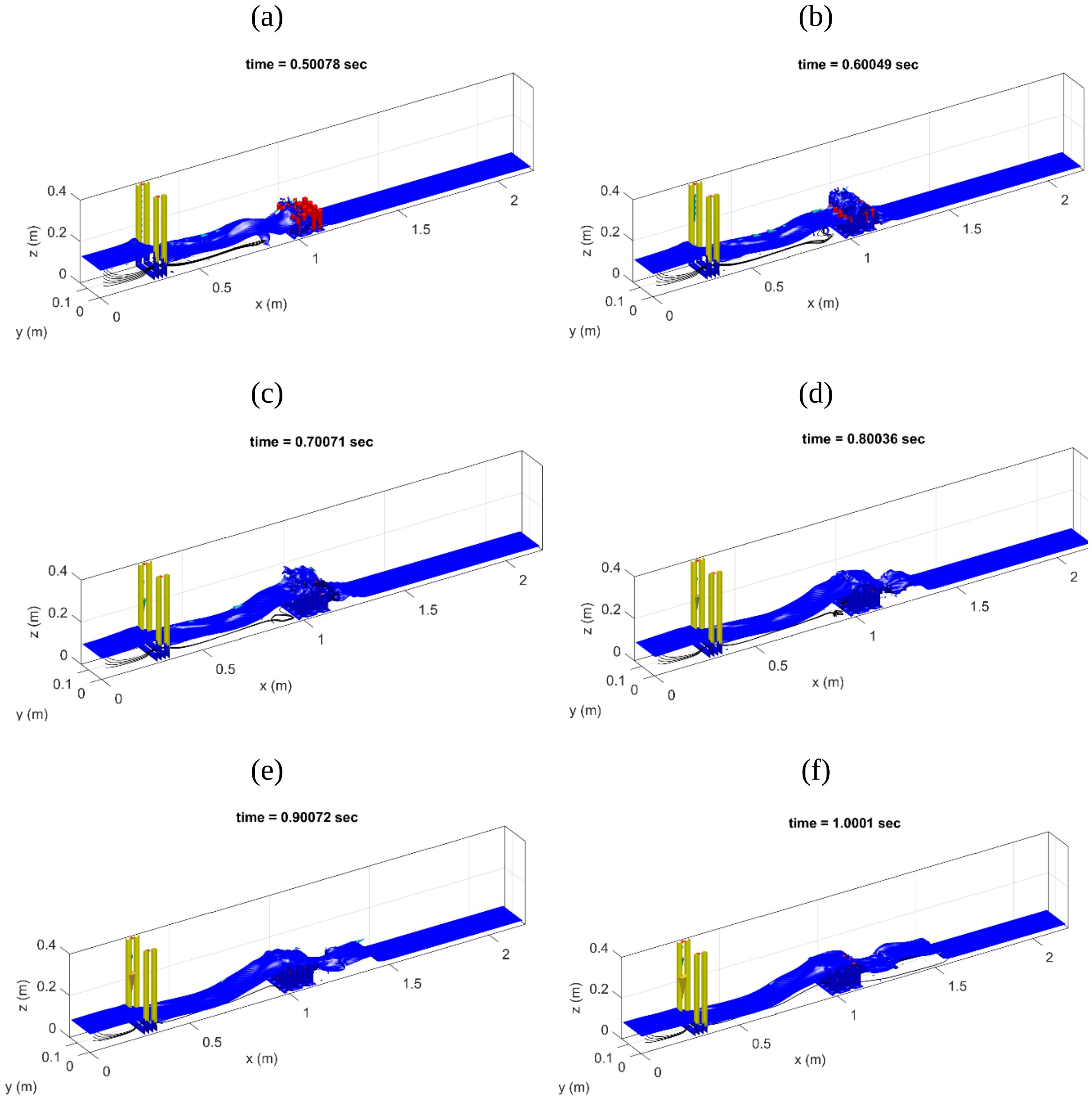
(f)



Case 2: Dambreak flow with solid obstacle

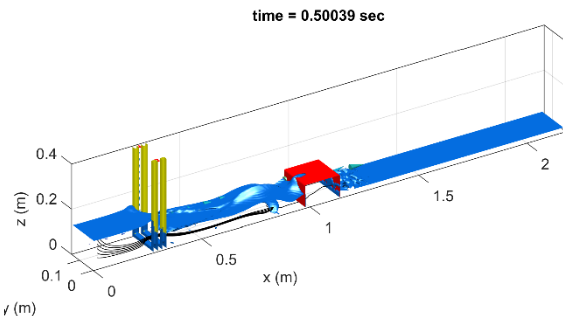


Case 3: Dambreak flow through porous obstacle (solid method)

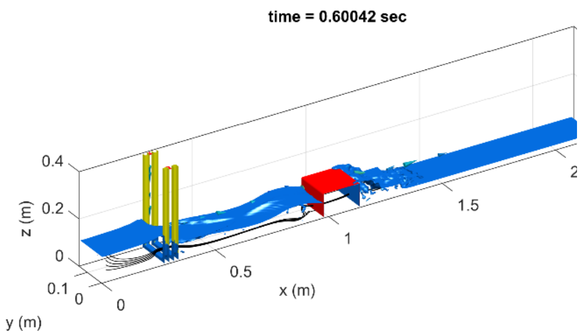


Case 3: Dambreak flow through porous obstacle (porous method)

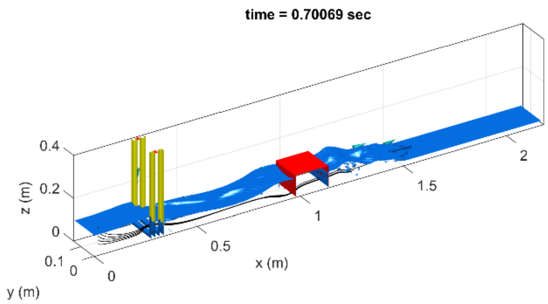
(a)



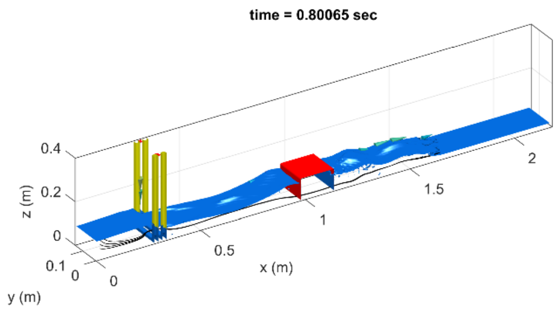
(b)



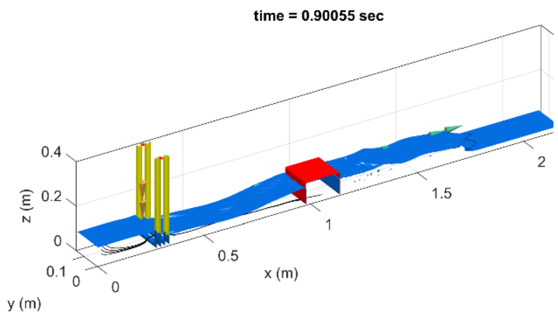
(c)



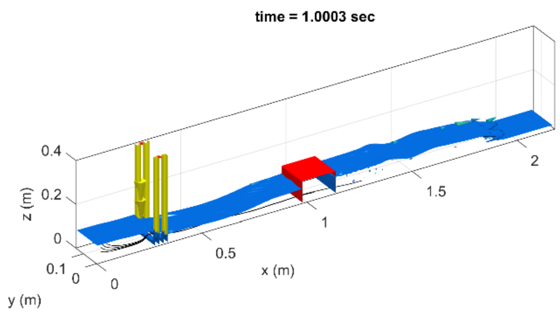
(d)



(e)



(f)



Case 4: Dambreak flow through porous obstacle (porous method)