

PLUNGING SOLITARY WAVES – A 3D NUMERICAL INVESTIGATION

by

Taisuk Kim

A thesis submitted to the Faculty of the University of Delaware in partial fulfillment of the requirements for the degree of Master of Civil Engineering

Spring 2014

© 2014 Taisuk Kim
All Rights Reserved

UMI Number: 1562390

All rights reserved

INFORMATION TO ALL USERS

The quality of this reproduction is dependent upon the quality of the copy submitted.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if material had to be removed, a note will indicate the deletion.



UMI 1562390

Published by ProQuest LLC (2014). Copyright in the Dissertation held by the Author.

Microform Edition © ProQuest LLC.

All rights reserved. This work is protected against unauthorized copying under Title 17, United States Code



ProQuest LLC.
789 East Eisenhower Parkway
P.O. Box 1346
Ann Arbor, MI 48106 - 1346

PLUNGING SOLITARY WAVES – A 3D NUMERICAL INVESTIGATION

by

Taisuk Kim

Approved:

Tian-Jian Hsu, Ph.D.
Professor in charge of thesis on behalf of the Advisory Committee

Approved:

Harry W. Shenton III, Ph.D.
Chair of the Department of Civil and Environmental Engineering

Approved:

Babatunde A. Ogunnaike, Ph.D.
Dean of the College of Engineering

Approved:

James G. Richards, Ph.D.
Vice Provost for Graduate and Professional Education

ACKNOWLEDGMENTS

This thesis is the product of the contribution of many talented and generous people around me. First, I would like to express my sincere gratitude and appreciation to my advisor, Dr. Tian-Jian Hsu. This thesis is the result of his continuous support of my research, and I could not have imagined completing it without his wisdom, patience, motivation, and guidance. In addition, I would like to thank the other members in our laboratory, Zhen cheng, Zheyu Zhou, and Jialin Chen for their instruction on the use of numerical modeling and data analysis tools, which were essential in completing my research. The inspiration and knowledge I have acquired working alongside the other faculty in the Center for Applied Coastal Research including Dr. Kirby, Dr. Kobayashi, and Dr. Puleo have also been unparalleled.

Finally, I would like to dedicate this thesis to my friends and family: to my parents, Yoon-hoi Kim and Soo-yeon Chai, who immigrated to the US in spite of significant hardships so that I could receive a quality education; to my sister, Eunha Kim, who has been by side through all the trying and joyous moments in my life; and to my friends, Margaret Ding, Yul Kim, and Eugene Han for their emotional support and valuable advice.

TABLE OF CONTENTS

LIST OF FIGURES	vi
ABSTRACT.....	x
Chapter	
1 INTRODUCTION	1
1.1 2011 Tohoku Earthquake and Tsunami and Potential Tsunami Risks in the United States	1
1.2 Experimental and Numerical Approach to Tsunami Research.....	2
2 NUMERICAL MODEL FORMULATION	7
2.1 3-D Large Eddy Simulation Model.....	7
2.2 Governing Equation	8
2.3 Sub-grid Closure	9
2.4 Boundary Conditions	10
2.5 Surface Tracking Method	13
3 NUMERICAL SIMULATION OF LABORATORY EXPERIMENT OF SUMER ET AL. [2011].....	15
3.1 Numerical Model Set-up.....	15
3.2 Numerical Simulation Overview	17
3.3 Numerical Model Validation	21
3.3.1 Wave Gauge Data Comparison.....	22
3.3.2 Bottom Shear Stress Comparison	24
3.4 Model Results and Discussion	27
3.4.1 Shoaling and Run-up.....	28
3.4.2 Run-down and Hydraulic Jumps.....	43
3.4.3 Separation of Boundary Layers during Hydraulic Jump	55
3.4.4 Fluctuation Component of BSS and turbulent intensity	61

4	DISCUSSION	68
4.1	Numerical Simulation of Experiment of <i>Synolakis</i> , [1986]	68
4.1.1	Numerical Model Set-up	68
4.1.2	Numerical Model Validation	69
4.1.3	Hydraulic Jump and Boundary Separation in <i>Synolakis</i> [1986]	73
4.2	Bottom Flow Acceleration in <i>Sumer et al.</i> , [2011]	78
5	CONCLUSION	88
	REFERENCES	90

LIST OF FIGURES

Figure 2.1:	The schematic plot of the numerical wave flume with boundary conditions and coordinates.....	11
Figure 3.1:	The schematic picture of the numerical wave tank based on <i>Sumer et al.</i> , [2011].....	15
Figure 3.2:	Surface elevation during the shoaling phase at (a) $t = 0.15$ s; (b) $t = 1.05$ s; (c) $t = 2.05$ s	17
Figure 3.3:	Same as Figure 3.2 except during the wave breaking at (a) $t = 2.35$ s; (b) $t = 2.45$ s; (c) $t = 2.55$ s; (d) $t = 2.75$ s; (e) $t = 3.05$ s.....	19
Figure 3.4:	Same as Figure 3.3 except during down-rush phase and the hydraulic jump at (a) $t = 5.45$ s; (b) $t = 6.45$ s; (c) $t = 6.95$ s; (d) $t = 7.05$ s; (e) $t = 8.05$ s	20
Figure 3.5:	The schematic plot of the locations of the wave gauges (GT, G1, G3, G5, and G8) and BSS measurements (G1, G2, G5, G6, and G8).....	21
Figure 3.6:	The wave gage data comparison at the toe, $x = -5.41$ m. Black line is the numerical data; red circle, the experimental data from <i>Sumer et al</i> [2011]. (a) GT, (b) G1, (c) G3, (d) G5, and (e) G8	23
Figure 3.7:	Squared friction velocity, $u\tau^2$ comparison at (a)G1, (b)G2, (c)G5, (d)G6, and G8.	26
Figure 3.8:	Snapshots of the contours of span-wise averaged (a) cross-shore velocity, u ; (b) vertical velocity, w ; (c) turbulence intensity, $2k$; and (d) dynamic pressure p_d at $t = 1.95$ s	31
Figure 3.9:	Same as Figure 3.8 except at $t = 2.25$ s.....	32
Figure 3.10:	Same as Figure 3.9 except at $t = 2.45$ s.....	33

Figure 3.11:	Same as Figure 3.10 except at $t = 2.55$ s.....	34
Figure 3.12:	Same as Figure 3.11 except at $t = 2.65$ s.....	35
Figure 3.13:	Same as Figure 3.12 except at $t = 2.95$ s.....	36
Figure 3.14:	Same as Figure 3.13 except at $t = 3.15$ s.....	37
Figure 3.15:	Same as Figure 3.14 except at $t = 3.32$ s.....	38
Figure 3.16:	Turbulent coherent structures during the wave breaking process. (a) $t = 2.45$ s; (b) $t = 2.55$ s; (c) $t = 2.55$ s; (d) $t = 2.95$ s; (e) $t = 3.15$ s; (f) $t = 3.32$ s.	40
Figure 3.17:	Span-wise averaged friction velocity in the snapshots in Figure 3.10-7.....	41
Figure 3.18:	The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 4.75$ s; (b) $t = 4.95$ s; (c) $t = 5.15$ s; (d) $t = 5.35$ s; (e) $t = 5.55$ s.....	44
Figure 3.19:	Snapshots of the contours of span-wise averaged (a) cross-shore velocity, u ; (b) vertical velocity, w ; (c) turbulence intensity, $2k$; and (d) dynamic pressure p_d at $t = 5.45$ s	47
Figure 3.20:	Same as Figure 3.19 except at $t = 6.45$ s.....	48
Figure 3.21:	Same as Figure 3.20 except at $t = 6.55$ s.....	49
Figure 3.22:	Same as Figure 3.21 except at $t = 6.75$ s.....	50
Figure 3.23:	Same as Figure 3.22 except at $t = 7.05$ s.....	51
Figure 3.24:	Same as Figure 3.23 except at $t = 7.75$ s.....	52
Figure 3.25:	Turbulent coherent structures under the hydraulic jumps. (a) $t = 6.45$ s; (b) $t = 6.55$ s; (c) $t = 6.65$ s; (d) $t = 6.75$ s; (e) $t = 7.05$ s; (f) $t = 7.75$ s.....	54
Figure 3.26:	The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 6.45$ s; (b) $t = 6.75$ s; (c) $t = 7.05$ s; (d) $t = 7.75$ s; (e) $t = 8.55$ s.....	57

Figure 3.27:	Froude number (first column) and friction velocity (second column) under the hydraulic jumps. (a) $t = 5.55$ s; (b) $t = 6.05$ s; (c) $t = 6.35$ s; (d) $t = 6.55$ s; (e) $t = 7.05$ s; (f) $t = 8.55$ s.	60
Figure 3.28:	BSS plot at $t = 2.65$ s. (a) span-wise averaged value; (b) fluctuating components	63
Figure 3.29:	Same as Figure 3.28 except at $t = 2.75$ s.....	64
Figure 3.30:	Same as Figure 3.29 except at $t = 3.55$ s.....	65
Figure 3.31:	Same as Figure 3.30 except at $t = 6.85$ s.....	66
Figure 3.32:	Same as Figure 3.31 except at $t = 7.75$ s.....	67
Figure 4.1:	The comparison between the experimental wave gauge data at $x = -3.19$ m (red circles) and the numerical data at -3.21 m (black lines).....	70
Figure 4.2:	The surface elevation comparison during the run-up and wave breaking phase at (a) $t = 1.43$ s; (b) $t = 2.14$ s; (c) $t = 2.86$ s; (d) $t = 3.14$ s; (e) $t = 3.57$ s.	71
Figure 4.3:	Same as Figure 4.2 except during the run-down phase and hydraulic jump at (a) $t = 4.28$ s; (b) $t = 7.14$ s; (c) $t = 8.57$ s; (d) $t = 9.28$ s; (e) $t = 9.99$ s.	72
Figure 4.4:	The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 5.74$ s; (b) $t = 6.24$ s; (c) $t = 6.74$ s; (d) $t = 7.24$ s; (e) $t = 7.74$ s.....	75
Figure 4.5:	Same as Figure 4.4 except at (a) $t = 8.24$ s; (b) $t = 8.74$ s; (c) $t = 9.24$ s; (d) $t = 9.74$ s; (e) $t = 10.24$ s.....	76
Figure 4.6:	The contour snapshots of the span-wise averaged turbulence intensity at (a) $t = 6.04$ s; (b) $t = 6.74$ s; (c) $t = 7.74$ s; (d) $t = 8.74$ s.....	77
Figure 4.7:	The maximum bottom flow acceleration $\partial u_b / \partial t$ over the entire tank versus time.	80

Figure 4.8(a):	The surface elevation (left), the span-wise averaged bottom flow acceleration (middle), and the top view of the contour of bottom flow acceleration at $t = 2.60$ s.....	82
Figure 4.8(b):	Same as Figure 4.8(a) except at $t = 2.70$ s	83
Figure 4.8(c):	Same as Figure 4.8(b) except at $t = 3.60$ s	84
Figure 4.8(d):	Same as Figure 4.8(c) except at $t = 6.50$ s	85
Figure 4.8(e):	Same as Figure 4.8(d) except at $t = 7.10$ s	86
Figure 4.8(f):	Same as Figure 4.8(e) except at $t = 8.00$ s	87

ABSTRACT

A 3D numerical investigation based on large-eddy simulation (LES) is carried out to study a plunging solitary wave over a slope. This study is motivated by recent field surveys of the aftermath of several tsunami disasters where significant amount of soil erosion and foundation failure seem to occur during the drawdown stage of the tsunami impact.

The mathematical formulation of the model is based on 3D filtered Navier-Stokes equations with a dynamic Smagorinsky closure. The model is solved numerically using on an open source CFD library of solvers called OpenFOAM, which was previously validated for a spilling solitary wave in a laboratory wave flume (*Sangermano*, [2013]). In this study, the numerical model is further validated with laboratory experiments of *Sumer et al.* [2011] and *Synolakis* [1986] for plunging wave condition.

After validation with laboratory data of *Sumer et al.* [2011], simulation results are further analyzed to understand the structures of flow velocity and turbulence during the run-up and drawdown stages of the plunging solitary wave and the resulting bottom stress and near bed flow acceleration. During run-up, wave breaking turbulence is not generated until the horizontal 2D rollers degenerated into smaller 3D structures due to the collapse of plunging breaker onto the bed.

During the drawdown stage, flow landward of the initial shoreline is dominated by boundary layer process, similar to those reported in the swash zone. However, at later stage of the drawdown process, hydraulic jump is observed in the laboratory experiment, which is also well-captured by the present numerical simulation. More detailed analysis on the simulation results further reveal the existence of boundary layer separation under the hydraulic jump. The separation coincides with the location of strong adverse pressure gradient, reversal of bottom shear stress, and intensive turbulence generation.

Peak near bed flow acceleration can reach as high as 30 m/s^2 which occurs at the initial shoreline during the impingement of plunging solitary wave. This peak acceleration is mainly associated with the mean component of the acceleration while the turbulent fluctuating components only contribute a minor portion. . During drawdown stage, flow acceleration can also exceed 10 m/s^2 , which is associated with the moving hydraulic jump and the corresponding flow separation. In this case, the fluctuating component contributes more to the total acceleration than the mean component. High flow acceleration increases the possibility on the occurrence of plug flow and the present simulation results suggest plug flow is likely to occur during the plunging of the solitary wave over a slope.

Examining simulation results of *Synolakis*, [1986] also indicates similar occurrence of boundary layer separation under the hydraulic jump, suggesting that

boundary layer separation may be common for plunging solitary wave. Future work should extend the existing simulation with sediment transport capability.

Chapter 1

INTRODUCTION

1.1 2011 Tohoku Earthquake and Tsunami and Potential Tsunami Risks in the United States

The series of recent earthquakes and their associated tsunami events have alarmed governments, science and engineering communities, and media as well as general public, about the predictable, yet hardly preventable, natural disasters. Of the major tsunami occurrences in the past decade, the 2011 Tohoku Earthquake Tsunami (April 7, 2011) was the most devastating. The tsunami affected more than 2000 km stretch of Japan's Pacific coast, flooded more than 400 km² of land, and damaged more than 200,000 critical infrastructures including Daiichi Nuclear Plant in Fukushima (*Mori et al.*, [2012]). The total economic cost caused by the tsunami alone was initially estimated between \$176 and \$258 billion (*Daniell et al.*, [2011]). But the total cost will keep accumulating because the rippled economic effect is insurmountable, and the leakage of the radiation from the melted down nuclear reactor has yet been fully contained.

The occurrence of the tsunami in Japan was not completely surprising. Sitting on top of the zone where four tectonic plates – Eurasia Plate, North America Plate, Pacific Plate, and Philippine Sea Plate – the Japanese Island is prone to

earthquakes and subsequently to high risk of tsunamis. For long, the Japanese government has been investing much effort and resources to develop protective and preventive measures for the earthquake and tsunami disasters. However, as reflected from the aftermath of the 2011 Tohoku Tsunami, the current counter-measures need further reinforcement, and the knowledge in the characteristics of the tsunami inundation in built-environment is still insufficient (*Mori et al.*, [2011]).

The coastlines in United States, especially along the Pacific Coast, are also prone to the underwater earthquakes and subsequently to the tsunami related disasters (*Stover and Coffman*, [1993]; *Ross et al.*, [2004]). Although any major urban cities in the United States have not been hit by tsunamis in the recorded history, many investigations in the geological records identified the layers of offshore deposits along the Pacific Rim, which might have been carried by either large storms or tsunamis (*Atwater*, [1987]; *Dawson*, [1994]; *Dawson and Stewart*, [2007]). Furthermore, the 1964 Alaska Earthquake triggered a 8.2 m tsunami that caused around \$311 million in property damage and took 110 lives (*Stover and Coffman*, [1993]).

1.2 Experimental and Numerical Approach to Tsunami Research

The very first step to enhance the countermeasures for tsunami disasters requires better understanding of the near-shore hydrodynamic processes associated with

the tsunamis waves. A solitary wave is a unique type of a long wave that consists of a single wave crest. The characteristics of a solitary wave such as wave speed and crest shape depend on a water depth and wave height. Due to their physical resemblance to propagating tsunami waves and analytical simplicity, solitary waves and bores have long been used in modeling tsunami uprushes in laboratory settings (*Hall and Watts*, [1953]; *Synolakis*, [1986]; *Li and Raichlen*, [2001]; *Hsiao et al.*, [2008]; *Baldock et al.*, [2009]; *Baldock et al.*, [2011]). The implementation of a solitary wave has also aided to further understand the complex interaction between sediments and a single breaking wave crest (*Sumer et al.*, [2011]). In addition, *Grilli et al.*, [1997] used the slope parameter, a self-similarity parameter relating wave conditions and beach slopes, to categorize the solitary wave breaker types into plunging, spilling, and surging.

The early experimental and numerical studies of solitary waves primarily focused on the propagation, the evolution of surface elevation, the inundation characteristics, and the breaker-type characterization. The pioneering work of *Synolakis*, [1986] provided important information regarding the wave propagation, the surface elevation evolution, and the uprush/down-rush phase of the plunging solitary waves. More recently, with the laboratory techniques such as particle image velocimetry (PIV) (*Liu et al.*, [2007]; *Ting*, [2006, 2008]; *Ting et al.*, [2011]), particle tracking velocimetry (PTV) (*Park et al.*, [2012]), acoustic Doppler velocimetry (ADV) (*Ting*, [2006]; *Baldock et al.*, [2009]), and direct bottom

stress measurement (*Sumer et al.*, [2011]), more detailed information in breaking solitary wave kinetics has become available. But, the complete picture of the 3-D nature of the turbulence mechanism is difficult to achieve through the experiments alone because the observing window in the experiments is technically limited. Both PTV and PIV are only capable of capturing 2-D images of the flow structures in a selected section. The ADV only records the time history of velocity at a fixed location, and its installation is intrusive to small-scale hydrodynamic processes. Also, the scope of the direct bottom stress measurement is limited to selected locations within the near-bed regions.

Numerical modeling/simulation, once validated, can be a useful tool that can lift such observational restrictions and bolster the explanation regarding the phenomena observed in the experiments. Equipped with sufficient data storage capacity and data processing power, the users of the numerical simulations can control the scopes of the observation and extract any desired datasets. *Lin et al.*, [1999] implemented the numerical model based on the Reynolds Averaged Navier-Stokes equations (RANS) and the k - ϵ equations to investigate the flow structures under plunging solitary wave, originally experimented by *Synolakis*, [1986]. This numerical investigation adds valuable information regarding the water particle motion in the collapsing wave front. In addition, not only does the model predict the surface elevation evolution, but also it allows observing the turbulent kinetic energy (TKE) distribution under the breaking solitary wave. *Zhang and Liu*,

[2008] used the similar model to examine the run-up and run-down processes of dam-break generated bores. With their numerical data, combined with the experimental data of *Yeh and Ghazali*, [1988] and *Yeh et al.*, [1989], they provided valuable details in the bore propagation and the vorticity generation as well as the violent interaction between two opposite currents during the down-wash phase.

Recently, *Sangermano*, [2013] performed a 3-D Large Eddy Simulation (LES) study, with the dynamic Smagorinsky closure, to examine the spilling solitary waves based on the experimental data of *Ting* [2006; 2007]. Besides validating the model, he visualized the 2-D rollers generated by initial wave overturning and their evolution to the 3-D turbulent coherent structures. The numerical results gave intriguing insight that the breaking surface generated counter-rotating vortex, similar to hairpin vortices, descending downward and impinging the bed.

The primary goal of this study is to investigate the hydrodynamic processes under plunging solitary waves on smooth plane beaches by using the LES approach of *Sangermano*, [2013]. Two numerical simulations are performed in this study: one based on the experiment of *Sumer et al.*, [2011]; another, on the experiment of *Synolakis*, [1986]. In Chapter 2, a general description of the theoretical background of the numerical model is presented. In Chapter 3, the numerical model set-up and the results of the simulation based on the experiment of *Sumer et al.*, [2011] will be presented followed by a thorough analysis on the simulation results, specifically the impingement of the plunging breaker during the run-up

stage and the generation of hydraulic jump and flow separation during the draw-down stage. The simulation based on the experiment of *Synolakis*, [1986] is discussed in Chapter 4 along with a discussion on the near-bed flow acceleration in order to evaluate the occurrence of plug flow. Finally, a summary of the conclusions will be discussed in Chapter 5.

Chapter 2

NUMERICAL MODEL FORMULATION

2.1 3-D Large Eddy Simulation Model

The present study implements the large-eddy simulation (LES) to investigate plunging solitary wave over a slope. The numerical model and similar implementation of the numerical wave flume is validated by *Sangermano*, [2013] for spilling solitary wave condition. The filtering operation in the LES decomposes the velocity field $u_i(x_i, t)$ into the resolved and the unresolved components. The large scale motions in the resolved velocity field are explicitly computed by the filtered Navier-Stokes equations that contain sub-grid scale (SGS) stress tensors. The turbulence in the unresolved field is parameterized by the SGS closures such as the dynamic Smagorinsky closure (*Germano*, [1991]) employed in this study.

OpenFOAM, an open-source library of C++ solvers for Computational Fluid Dynamics (CFD), is utilized to solve the model numerically. This package includes a Navier-Stokes equation solver for two immiscible and incompressible fluids called interFoam (*OpenCFD Limited*, [2011]). The interface between the two immiscible fluids is obtained by the volume of fluid (VOF) method. The nu-

merical wave flume is established with boundary-fit domains and based on the finite volume scheme.

2.2 Governing Equation

The motion of an incompressible fluid can be described by the Navier-Stokes equations in tensor notation as:

$$\frac{\partial u_i}{\partial t} + u_i \frac{\partial u_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + g_i \quad (2.2.1)$$

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2.2.2)$$

where $i, j = 1, 2, 3$, u_i is flow velocity, ρ is the fluid density, p is pressure, g_i is the gravity, and ν is the kinematic viscosity of the fluid.

For the filtering operation in the LES, the filter length is defined as

$$\Delta = (\Delta x \cdot \Delta y \cdot \Delta z)^{\frac{1}{3}} \quad (2.2.3)$$

in which Δx , Δy , Δz are the grid size in each corresponding direction and hence Δ is the characteristic length scale of the grid size. The fluid motions larger than the filter length scale are solved directly by the filtered Navier-Stokes equations:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_i \frac{\partial \bar{u}_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} + g_i + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2.2.4)$$

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \quad (2.2.5)$$

in which $\overline{(\cdot)}$ represents a filtered quantity, and τ_{ij} is the sub-grid stress tensor defined as

$$\tau_{ij} = \overline{u}_i \overline{u}_j - \overline{u_i u_j} \quad (2.2.6)$$

The sub-grid stress tensor term contains the effect of the fluid motion smaller than the filter length and requires an appropriate closure for accurate simulation of turbulent flow. The further details of the sub-grid closure used in this study will be discussed in the next section.

2.3 Sub-grid Closure

In this study, the dynamic Smagorinsky closure, based on the work of *Germano* [1991] and the modification of *Lilly* [1992], is used as the sub-grid scale closure. In the standard Smagorinsky closure, the sub-grid scale stress tensor is calculated with the following assumption:

$$\tau_{ij} - \frac{1}{3} \delta_{ij} \tau_{kk} = 2(C_S \Delta)^2 |\overline{S}_{ij}| \overline{S}_{ij} \quad (2.3.1)$$

where C_S is the Smagorinsky coefficient, specified with a constant value, 0.167;

$\overline{S}_{ij} = \frac{1}{2} \left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right)$ is the strain rate tensor; $|\overline{S}_{ij}| = (2\overline{S}_{ij}\overline{S}_{ij})^{\frac{1}{2}}$ is the magnitude of

the strain rate. The sub-grid scale viscosity, ν_{ts} is defined as (*Pope*, [2000])

$$\nu_{ts} = (C_S \Delta)^2 |\bar{S}_{ij}| \quad (2.3.2)$$

Modified from the standard closure, the dynamic closure introduces a test scale stress tensor by applying a second test filter $\hat{\Delta} = 2\Delta$ to Equation (2.3.1). Instead of using the constant value for C_S , the dynamic closure chooses the value of C_S that minimizes the discrepancy between the sub-grid scale stress tensor and the test scale tensor. The previous investigation of *Sangermano*, [2013] has proven that the numerical result with the dynamic closure shows better agreement with the measured data.

2.4 Boundary Conditions

Figure 2.1 shows the origin of coordinate $(x,y,z) = (0,0,0)$ and the boundary conditions used in the numerical wave flume. The origin of the coordinate is defined at the initial still-water shoreline, and x , y , z axes are labeled as the cross-shore, span-wise and vertical direction, respectively. Along the cross-shore axis, positive x is defined as the wave propagation direction.

The solitary wave is generated by a user-defined function for wave generation, *groovyBC*, implemented in the inlet boundary [*Gschaider*, 2009]. This function inputs water wave free surface elevation and velocity profiles based on the analytical solutions. The present solitary wave formulation is governed by the following first-order equations presented in *Lee, et al.*, [1982]:

$$\eta(x, t) = H \sec^2 \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \quad (2.4.1)$$

$$u(x, t) = \sqrt{gh} \frac{H}{h} \sec^2 \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \left\{ 1 - \frac{H}{4h} \sec^2 \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \right\} \quad (2.4.2)$$

$$w(x, t) = \sqrt{gh} \frac{Hz}{h} \left\{ 1 - \frac{H}{2h} \sec^2 \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \right\} \left\{ 2H \sqrt{\frac{3}{4} \frac{H}{h^3}} \tanh \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \sec^2 \left[\sqrt{\frac{3}{4} \frac{H}{h^3}} (x_s - ct) \right] \right\} \quad (2.4.3)$$

$$c = \sqrt{g(h + H)} \quad (2.4.4)$$

where H is the wave amplitude, h is the initial still water depth, t is time, z is the vertical position from the free surface, c is the wave celerity, x_s is a constant that

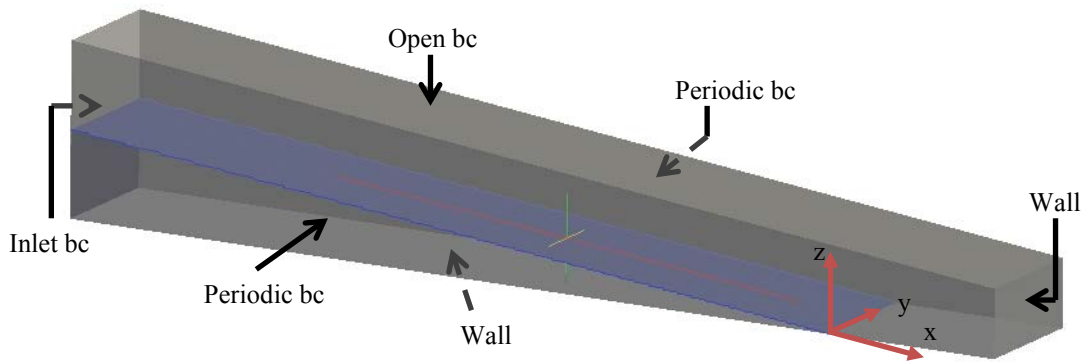


Figure 2.1: The schematic plot of the numerical wave flume with boundary conditions and coordinates.

defines the origin and effective length of the solitary wave. In theory, the solitary wave is infinitely long. Hence, the constant x_s is used to set the effective length of the wave in the numerical wave flume.

The near-wall modeling is applied in the bottom boundary to compensate with the insufficient numerical resolution close to the bed. Typically, the numerical resolution is too coarse to resolve the velocity profiles in the thin viscous sub-layers and buffer layers. The bottom boundary in the present study incorporates the semi-empirical profile by Spalding, [1961] to approximate the near-wall velocity profiles:

$$z^+ = u^+ + \frac{1}{E} \left[e^{\kappa u^+} - 1 - \kappa u^+ - \frac{1}{2} (\kappa u^+)^2 - \frac{1}{6} (\kappa u^+)^3 \right] \quad (2.4.5)$$

in which $E = 9.8$, $\kappa = 0.41$, $z^+ = z u_\tau / \nu$, $u^+ = U / u_\tau$, U is the velocity magnitude, u_τ is friction velocity. Sub-grid scale viscosity for the bottom boundary condition is

$$\nu_{ts} = \frac{u_\tau^2}{\partial \bar{u} / \partial z} - \nu \quad (2.4.6)$$

The value of u_τ estimated obtained from Equation 2.4.6 is later used in Chapter 3 to represent the bottom shear stress (BSS) based on the definition of the friction velocity (*Pope*, [2000]):

$$u_{\tau}^2 = \frac{\tau_w}{\rho} \quad (2.4.7)$$

in which τ_w is wall shear stress.

The periodic boundary conditions are applied in two lateral boundaries so that the span-wise averaged turbulent quantities can be handled as the ensemble-averaged ones.

2.5 Surface Tracking Method

The main advantage of interFoam is that it is capable to describe the interface between two immiscible fluids (the water-air interface in the present study) via the Volume of Fluid (VOF) method (*Hirt and Nicholas*, [1981]). The general representation of the density, ρ in this method is:

$$\rho = \alpha_1 \rho_1 + (1 - \alpha_1) \rho_2 \quad (2.5.1)$$

where the indices 1 and 2 indicate water and air, respectively; $\rho_1 = 1000 \text{ kg/m}^3$ is the water density; $\rho_2 = 1 \text{ kg/m}^3$ is for the air density.

The governing equation of the volume fraction α_1 in an immiscible two-fluid system is written as (*Hirt and Nicholas*, [1981]):

$$\frac{\partial \alpha_1}{\partial t} + \frac{\partial}{\partial x_j} (\alpha_1 \bar{u}_{1j}) = 0 \quad (2.5.2)$$

$$\alpha_1 \begin{cases} = 1 & \text{for water phase} \\ = 0 & \text{for air phase} \end{cases} \quad (2.5.3)$$

in which $\bar{u}_{1i}$ is the velocity of water. $\bar{u}_{1i}$ is estimated with a surface compression method adopted from *Klostermann, et al.*, [2012]. Hence, the governing equation becomes:

$$\frac{\partial \alpha_1}{\partial t} + \nabla \cdot (\alpha_1 \bar{u}_{1i}) + \nabla \cdot [\alpha_1 \bar{u}_{ri}(1 - \alpha_1)] = 0 \quad (2.5.4)$$

where $\bar{u}_{ri} = \bar{u}_{1i} - \bar{u}_{2i}$ is the relative velocity between the two phases.

The free water surface is simply defined as the interpolated location of the grid cells that contain only half water, i.e. $\alpha_1 = 0.5$.

Chapter 3

NUMERICAL SIMULATION OF LABORATORY EXPERIMENT OF SUMER ET AL. [2011]

3.1 Numerical Model Set-up

Figure 3.5 presents the numerical wave flume similar to the laboratory experiment of *Sumer et al.* [2011]. The wave tank dimensions and wave conditions follow the experimental condition reported in *Sumer et al.*, [2011]. The total cross-shore length is reduced to 13.63 m from 19.0 m, and the total tank height is reduced to 0.7 m from 0.8 m in order to relieve computational cost. The width is kept the same as that in the experiment, 0.6 m. The numerical tank consists of three sec-

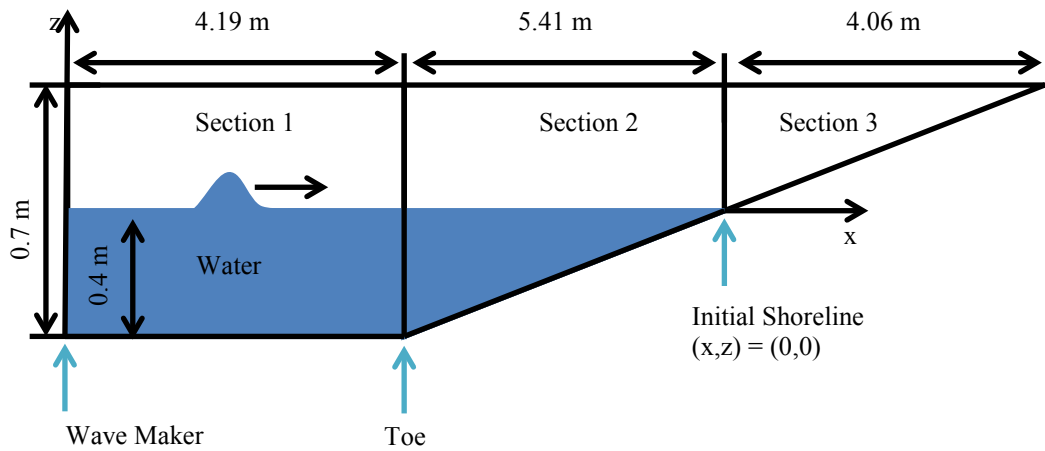


Figure 3.1: The schematic picture of the numerical wave tank based on *Sumer et al.*, [2011]

tions labeled as Section 1 between the wave maker to the toe; as Section 2, between the toe to the initial shoreline; and as Section 3, between the initial shoreline and the rear of the tank. The cross-shore length of each section is included in Figure 3.1. The slope starts at the toe located 4.19 m away from the wave maker and continues to the rear of the tank. The initial shoreline is 5.41 m away from the toe, and it is used as the reference point, $(x, z) = (0 \text{ m}, 0 \text{ m})$. The water depth in the flat bottom region (Section 1) is 0.4 m. The beach slope, therefore, is 0.0739, approximately 1/14. The wave height of the solitary wave is set to be 0.071 m.

A total number of 7,040,000 grids are used for the simulation, which includes 1100 in the x direction; 80 in both y and z directions. Each section is provided with different number of cross-shore grids. Section 2 and Section 3 are given with the same number of total cross-shore grids of 400, and Section 1 is given as 300. In all sections, the vertical grids are non-uniformly distributed by 1:10 ratio to allow dense grids in the bottom, but the span-wise grids are uniformly distributed. Hence, the near-bottom Δz is 0.2 cm in the flat region and 0.13 cm around the initial shoreline. The cross-shore axis in the Section 1 has uniform a grid distribution of $\Delta x = 1.4 \text{ cm}$. Both Section 2 and Section 3 have non-uniform cross-shore grid distributions with 1:10 ratio converging around the initial shoreline with the finest $\Delta x = 0.26 \text{ cm}$.

3.2 Numerical Simulation Overview

The solitary wave appears to maintain its symmetric shape until it passes the half point of the slope around $x=-3$ m (see Figure 3.2(a), (b)). The wave shape then evolves into an asymmetric shape – steep bore front and elongated tail – and a slight surface depression occurs between the wave front and the initial shoreline (see Figure 3.2(c)). Around $t=2.35$ s (Figure 3.3(a)), the wave height reaches the maximum (about 0.08 m). As the wave further approaches to the initial shoreline

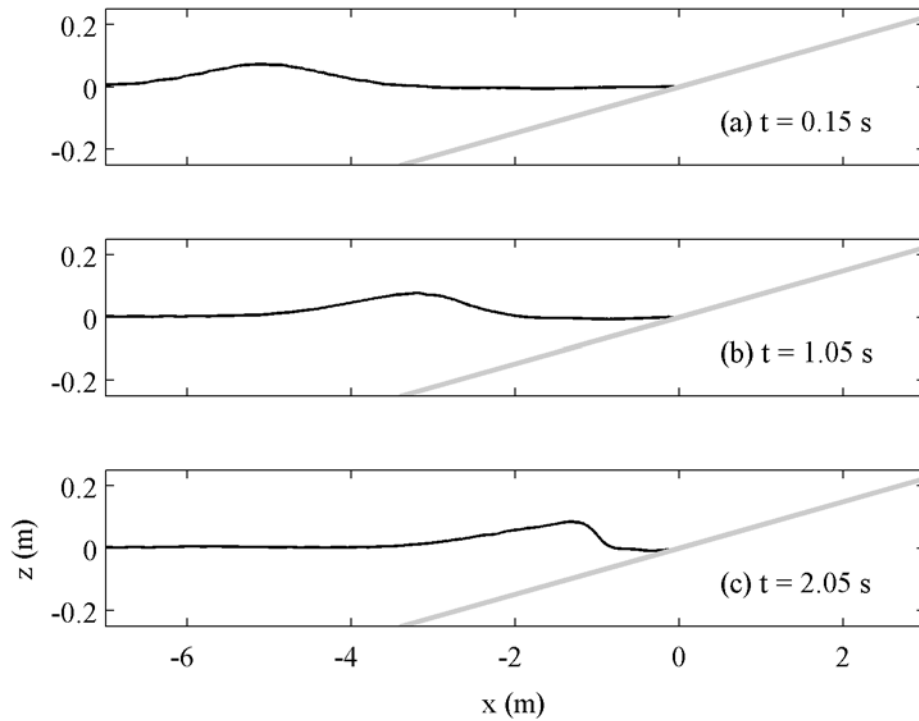


Figure 3.2: Surface elevation during the shoaling phase at (a) $t = 0.15$ s; (b) $t = 1.05$ s; (c) $t = 2.05$ s

at $x=0$ m, the steep wave front curls forming a tongue-like feature and trapping a large air bubble (see Figure 3.3 (b), (c)). When the extended tongue impinges the water below itself, a secondary and a tertiary tongues extend from the impinging front, trapping more air bubbles and crumbling the water surface (Figure 3.3(d)).

The run-up process continues until the tip reaches to $x=3.66$ m ($z = 0.27$ m) at $t=5.45$ s (Figure 3.4(a)), and the run-down process begins. As the run-down process further proceeds, the surface elevation gets suppressed between $x=-0.5$ m and -0.6 m, and a bore-like feature forms at $x=-0.5$ m (see Figure 3.4(b)). After $t=6.45$ s, the bore front collapses and the water surface around this region becomes unstable (Figure 3.4(c)). As the time progresses to $t=7.05$ s, the surface stabilizes, but the bore-like feature starts forming again (Figure 3.4(d)), and migrates further offshore to $x=-0.7$ m. This new jump appears stronger than the previous one (see Figure 3.4(e)).

In the following sections, the numerical model will be validated with the experimental data available in *Sumer et al.*, [2011]. After the validation, the numerical model results will be presented in more details in order to study flow and turbulence structures under a plunging solitary wave.

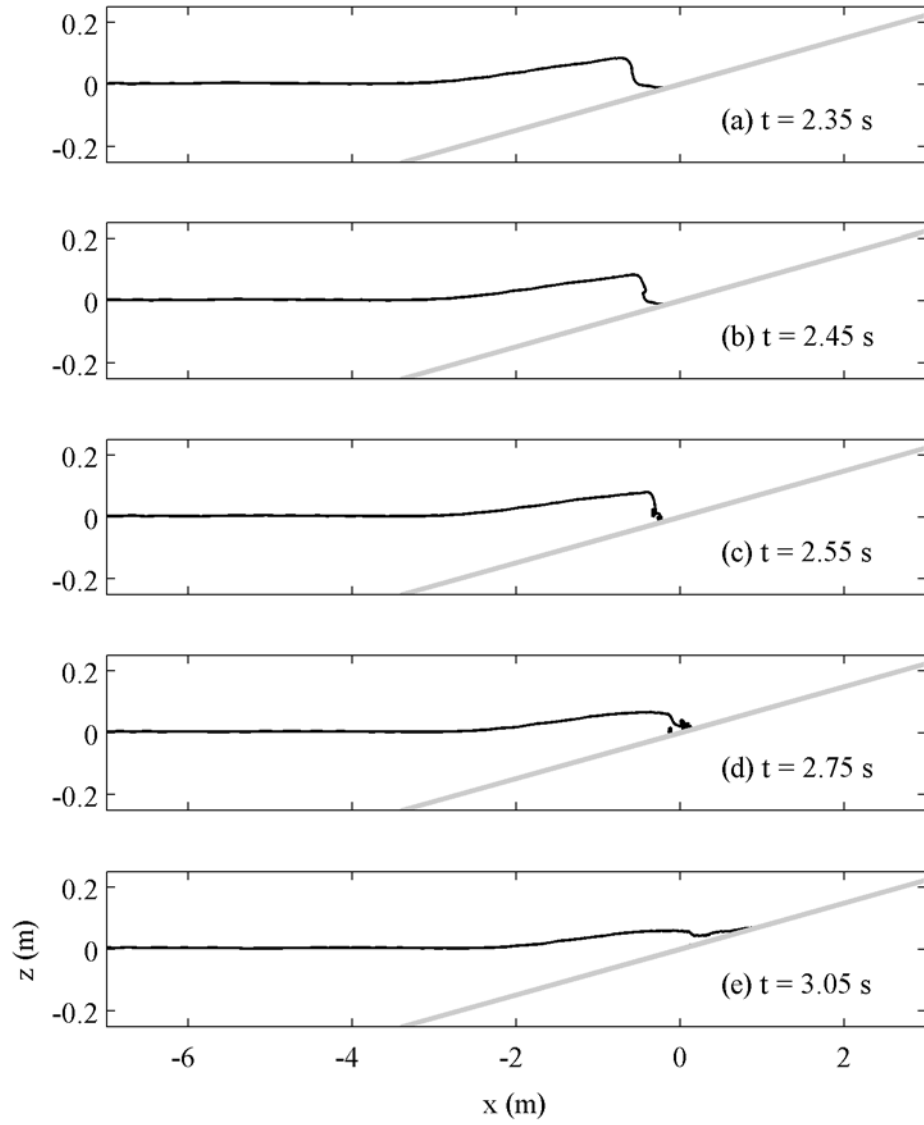


Figure 3.3: Same as Figure 3.2 except during the wave breaking at
(a) $t = 2.35$ s; (b) $t = 2.45$ s; (c) $t = 2.55$ s; (d) $t = 2.75$ s;
(e) $t = 3.05$ s

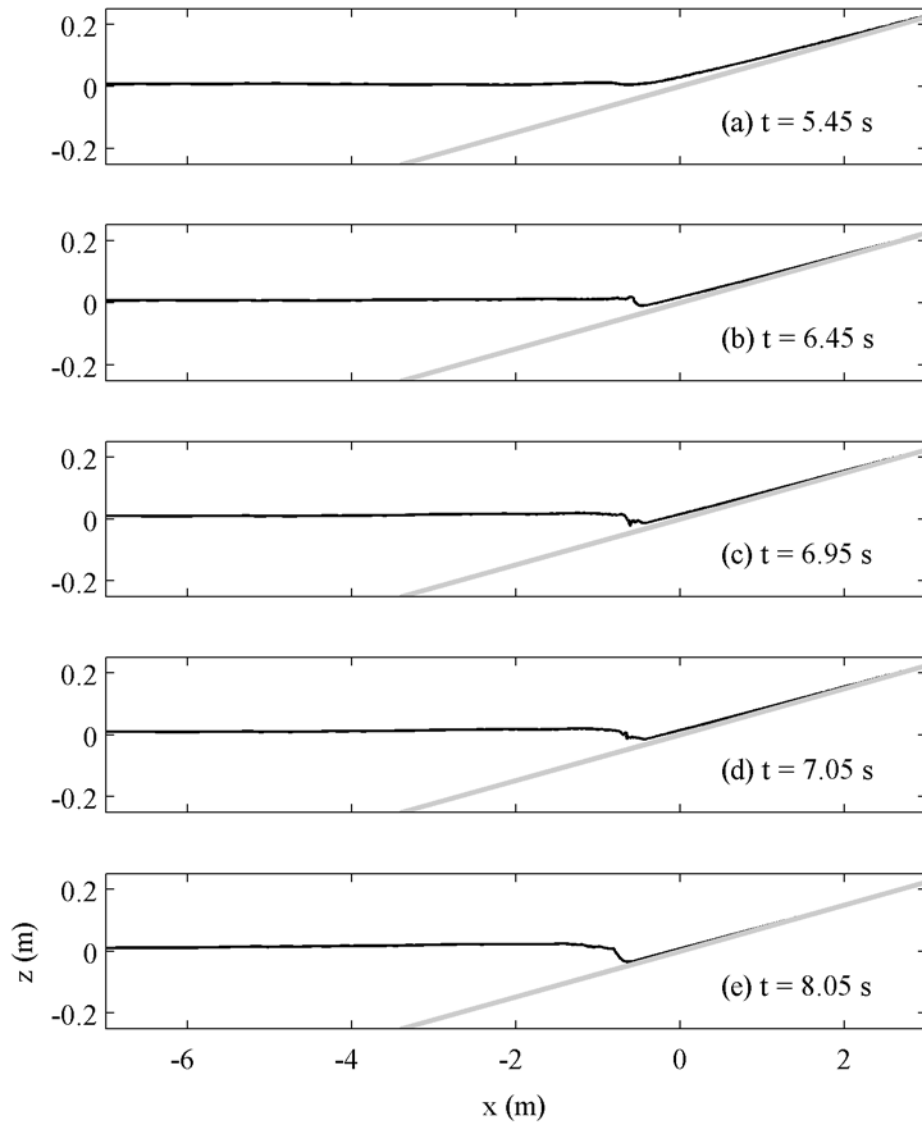


Figure 3.4: Same as Figure 3.3 except during down-rush phase and the hydraulic jump at (a) $t = 5.45$ s; (b) $t = 6.45$ s; (c) $t = 6.95$ s; (d) $t = 7.05$ s; (e) $t = 8.05$ s

3.3 Numerical Model Validation

Sumer et al., [2011] measure their experimental data in 7 locations over the slope:

Toe (GT; $x = -5.41$ m), Gauge 1 (G1; $x = -0.78$ m), Gauge 2 (G2, $x = -0.72$ m),

Gauge 3 (G3, $x = -0.54$ m), Gauge 5 (G5, $x = -0.06$ m), Gauge 6 (G6, $x = 0.18$ m),

and Gauge 8 (G8, $x = -0.44$). The wave gauge data are available in Sumer et al.

[2011] in GT, G1, G3, G5, and G8. The bottom shear stresses (BSS) data are also available in G1, G2, G5, G6, and G6. Figure 3.5 shows the schematic description

of the measurement locations.

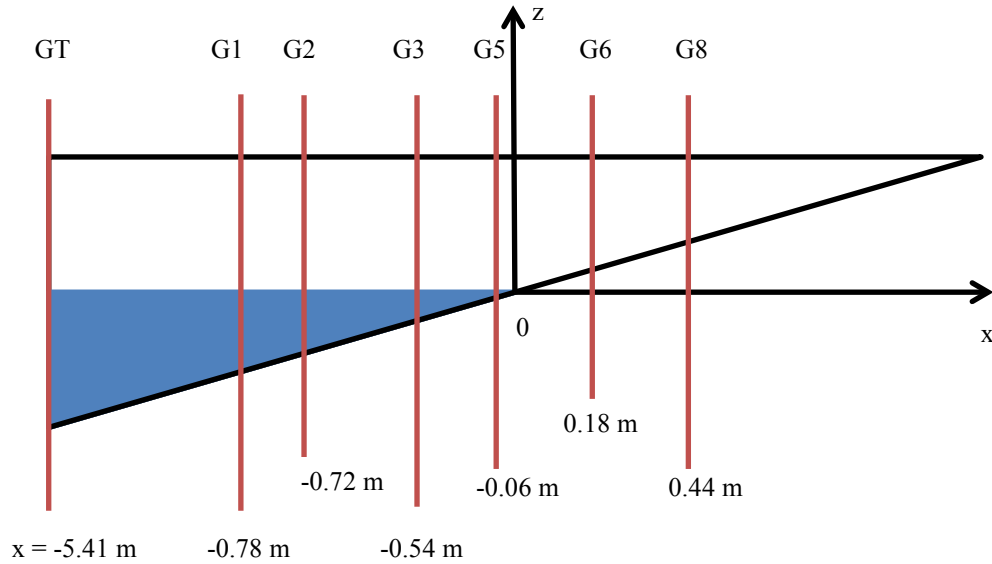


Figure 3.5: The schematic plot of the locations of the wave gauges (GT, G1, G3, G5, and G8) and BSS measurements (G1, G2, G5, G6, and G8)

3.3.1 Wave Gauge Data Comparison

As shown in Figure 3.3(a), the free surface elevation in the toe, specifically in the wave front, matches well with the experimental data. However, a small discrepancy arises after the wave crest passes the toe. The numerical simulation appears to add about 0.5~1 cm of the mean surface elevation tailing behind the solitary wave. It is confirmed that this discrepancy is caused by the difference between the physical wave maker and the numerical wave generator. Experimental wave makers generate solitary waves with a single push without adding water into the flume. Hence, the total volume of water is the same throughout the experiment. On the other hand, the numerical wave maker used in this simulation sends in, and therefore adds, a volume of water into the model domain determined by the analytical solution of solitary waves. The shorter cross-shore length of the numerical wave tank can further contribute to the increase of water level. Despite such difference, the general trend of the surface elevation is well captured in the numerical results.

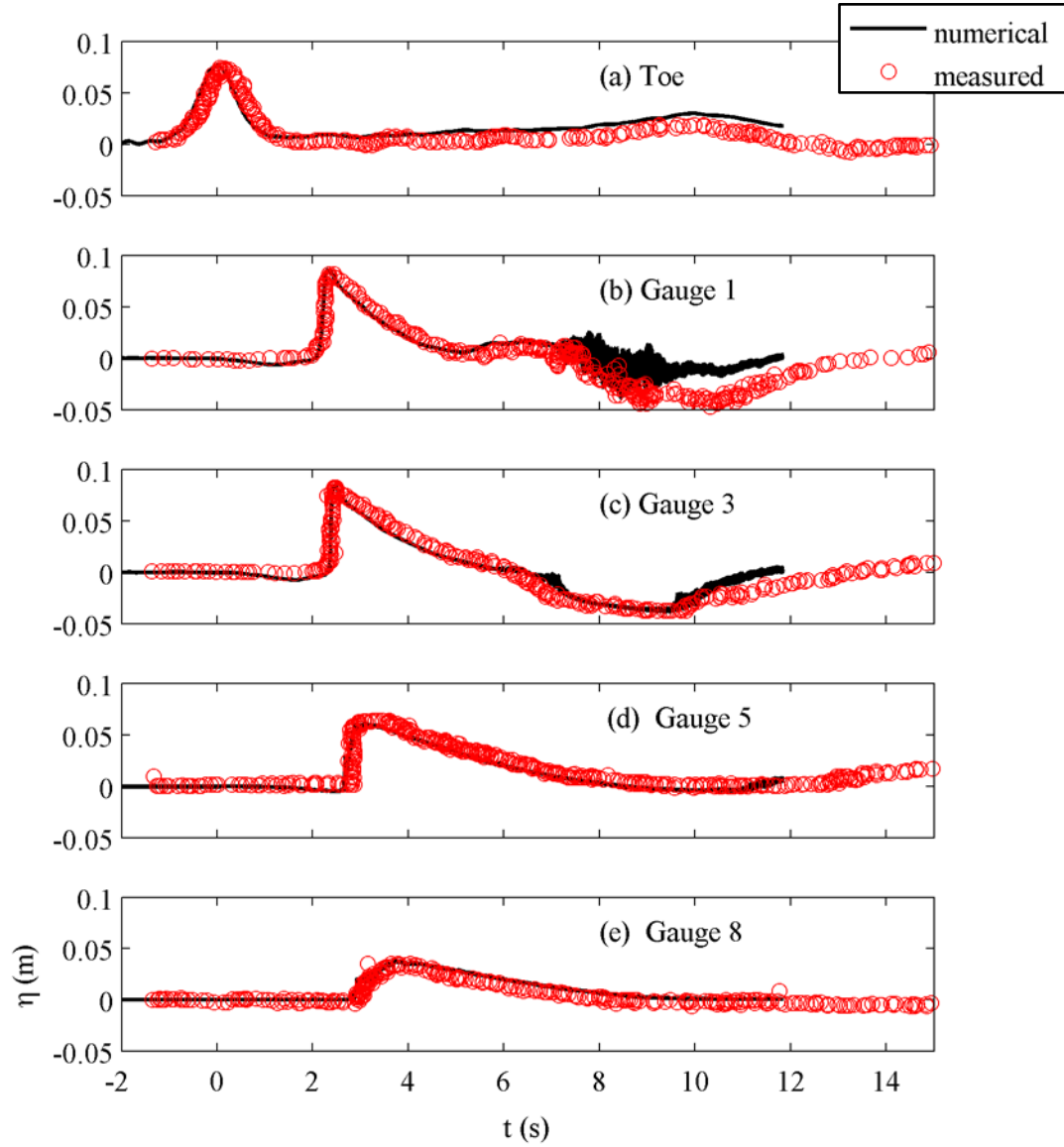


Figure 3.6: The wave gage data comparison at the toe, $x = -5.41$ m. Black line is the numerical data; red circle, the experimental data from *Sumer et al* [2011]. (a) GT, (b) G1, (c) G3, (d) G5, and (e) G8

The wave height comparisons in the other onshore locations are shown in Figure 3.6(b-e). The wave run-ups recorded in G1, G3 and G5 are well predicted in the numerical simulation. Also, G8, located in the swash zone, also shows fairly good prediction for the run-down process occurring between 4 to 8 seconds. The noticeable discrepancies are observed at around 7.0 s in G1 and G3. In G1, the experimental data show greater and longer depression in the water surface than the numerical prediction. The depression in G3 in the experiment occurs a few second earlier and lasts a few seconds longer than that in the numerical simulation. As we will discuss later, such discrepancy can be caused by a slight error in the predicted seaward migration of hydraulic jump.

3.3.2 Bottom Shear Stress Comparison

In the G1 and G2 (Figure 3.7(a), (b)), the numerical simulation performed well in the prediction of the shape and magnitude of the measured BSS during the run-up phase. Notice that wave breaking occurs near the initial shoreline ($x=0$ m) and hence the flow is not turbulent at this stage. However, in the highly turbulent phases during the run-down stage, the numerical simulation under predicted the magnitude of the BSS. In the gauges near the initial shoreline (G5, see Figure 3.7(c)) and in the swash zone (G6 and G8, see Figure 3.7(d), (e)), the numerical model predicts a much larger spike of bottom stress associated with the plunging breaker impinges to the bed. The BSS during the run-down in these locations,

however, are significantly under predicted. The discrepancies may be due to the coarse grid sizes near the bed and the limitation of near-wall modeling. To improve the numerical model in this aspect, very fine resolution near the bed is necessary in order to use no-slip bottom boundary condition without near wall modeling (the location of first grid point above the bed is much smaller than 1 in terms of wall unit). At this point, we cannot afford to carry out such high resolution simulation.

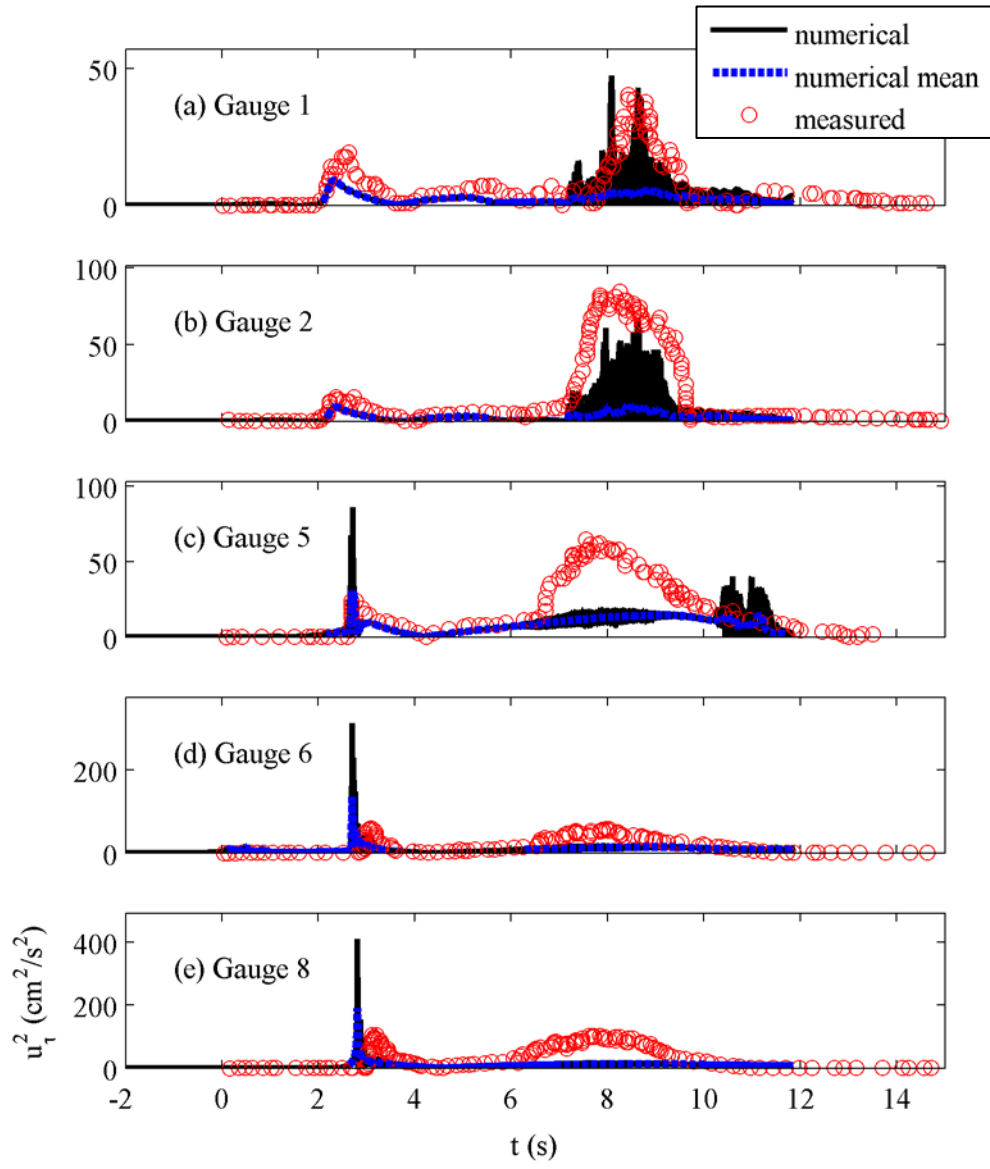


Figure 3.7: Squared friction velocity, u_τ^2 comparison at (a)G1, (b)G2, (c)G5, (d)G6, and G8.

3.4 Model Results and Discussion

This section is devoted to discussing the numerical model results in the terms of critical physical quantities such as horizontal velocity, vertical velocity, turbulence intensity, and dynamic pressure as well as their evolution throughout different hydrodynamic phases. Because the numerical simulation only computes the primary flow quantities, e.g., velocity and total pressure, the TKE and dynamic pressure are computed in post-processing.

TKE, k is defined as the sum of the squared values of the fluctuation components of velocities, u' , v' , and w' .

$$k = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \quad (3.8)$$

in which the bar, $\overline{(\cdot)}$ denotes a span-wise averaged quantity; $u' = u - \bar{u}$; $v' = v - \bar{v}$; $w' = w - \bar{w}$

In the following sections the intensity of TKE will be mainly shown via turbulence intensity, KI , which has a dimension identical to velocity

$$KI = \sqrt{2k} \quad (3.9)$$

The dynamic pressure is obtained by subtracting the hydrostatic pressure along the cross-shore axis from the total pressure. The hydrostatic pressure is estimated as the pressure caused by the stack of water column above the point of interest.

$$P_d = P_{total} - \rho g h_{stacked} \quad (3.10)$$

$$h_{stacked} = \sum_{i=1}^N \alpha_i \Delta z_{i-1} \quad (3.11)$$

where N represent the total number of grid point in the vertical direction at a given x-y location. This method of stacking the water columns is only an approximation as it neglects the existence of air bubbles in the water columns. Although this method can include some errors under unstable water surface, the strong dynamic pressure in the shallow water depth in this simulation can compensate with such errors.

3.4.1 Shoaling and Run-up

As the wave crest approaches toward the initial shoreline, the small velocity near the initial shoreline directs seaward causing slight surface depression (see Figure 3.8 (a)). The flow below the wave crest shows notable shoreward velocities and adverse dynamic pressure gradient. While the crest still retains a symmetric solitary wave shape, the water column beneath the crest maintains fairly uniform horizontal velocity throughout the water column, and its vertical velocity increases from zero at the bottom to about 0.2 m/s the near surface. The shallow water body in front of the wave front shows relatively slow velocity and positive dynamic pressure. This water body appears to behave like an imaginary wall (between -0.5 and -0.6 m in Figure 3.9(d)) that hinders the propagation of the wave crest. The wave front, as a result, tends to move upward and the wave height increases.

When the wave crest reaches the maximum height (Figure 3.10), the vertical velocity changes its direction downward (see (b)), and the horizontal velocity increases near the tip of the wave front. The relatively fast wave front tip extends out from the wave front body and forms a tongue-like feature. The generation mechanism of the tongue-like feature was well reported in the previous numerical studies. In their 2-DV numerical simulations on a plunging solitary wave, Lin et al (1999) observed that the relatively fast water particles under the wave front shoot out and generate a tongue-like feature. The extended tongue impinges immediately into the water under the wave front, which generates a roller, traps a large air bubbles, and triggers smaller secondary impingements (Figure 3.11). Shortly after the first impact, when the secondary impingements hit the dry portion of the slope, significant amount of turbulence is generated at the impinging tip (see Figure 3.12). At this instant, we begin to see some notable level of turbulence generated from the roller created around the first large air bubble. In Figure 3.13, as it hits the bottom, the roller degenerates into many smaller structures. Strong turbulent intensity starts to form around the smaller structures. The peak turbulent intensity in the core of these structures exceeds 0.5 m/s.

Figure 3.14 and 3.15 shows the uprush phase following the wave breaking process. The run-up tip leads up the run-up flow along the slope, showing the greatest horizontal and vertical velocities. The horizontal velocity during run-up exceeds 1.5 m/s. In Figure 3.14(c), the turbulence intensity is spotted in three dif-

ferent locations: $x = 0.4 \text{ m}$, 0.6 m , and 1.0 m . A comparison between this figure and the earlier snapshots in Figure 3.13(c) shows that the first two turbulent spots are generated by the disintegrating large rollers, and the last one is generated by the wave impingements. Turbulence intensity at first appears confined near the bed and around the run-up tip. As the run-up flow continues moving up along the slope, the turbulence intensity disperses and degenerates upward. As the run-up flow propagates further into the dry portion of the slope (Figure 3.15), the previously generated turbulence intensity accumulates near the run-up tip, but the trailing turbulence weakens. The previous studies in the run-up of solitary waves (*Lin et al.*, 2002; *Sumer et al.*, 2011) observe similar trends. The weakening of turbulence is caused by the lack for strong turbulence generation mechanism after the impingement. This implies that the wave breaking process is the dominant turbulence generation mechanism during the run-up. In this simulation, no significant turbulence generation from the bed during this stage is observed possibly also because the grid size near the bottom is too coarse to resolve the thin boundary layer.

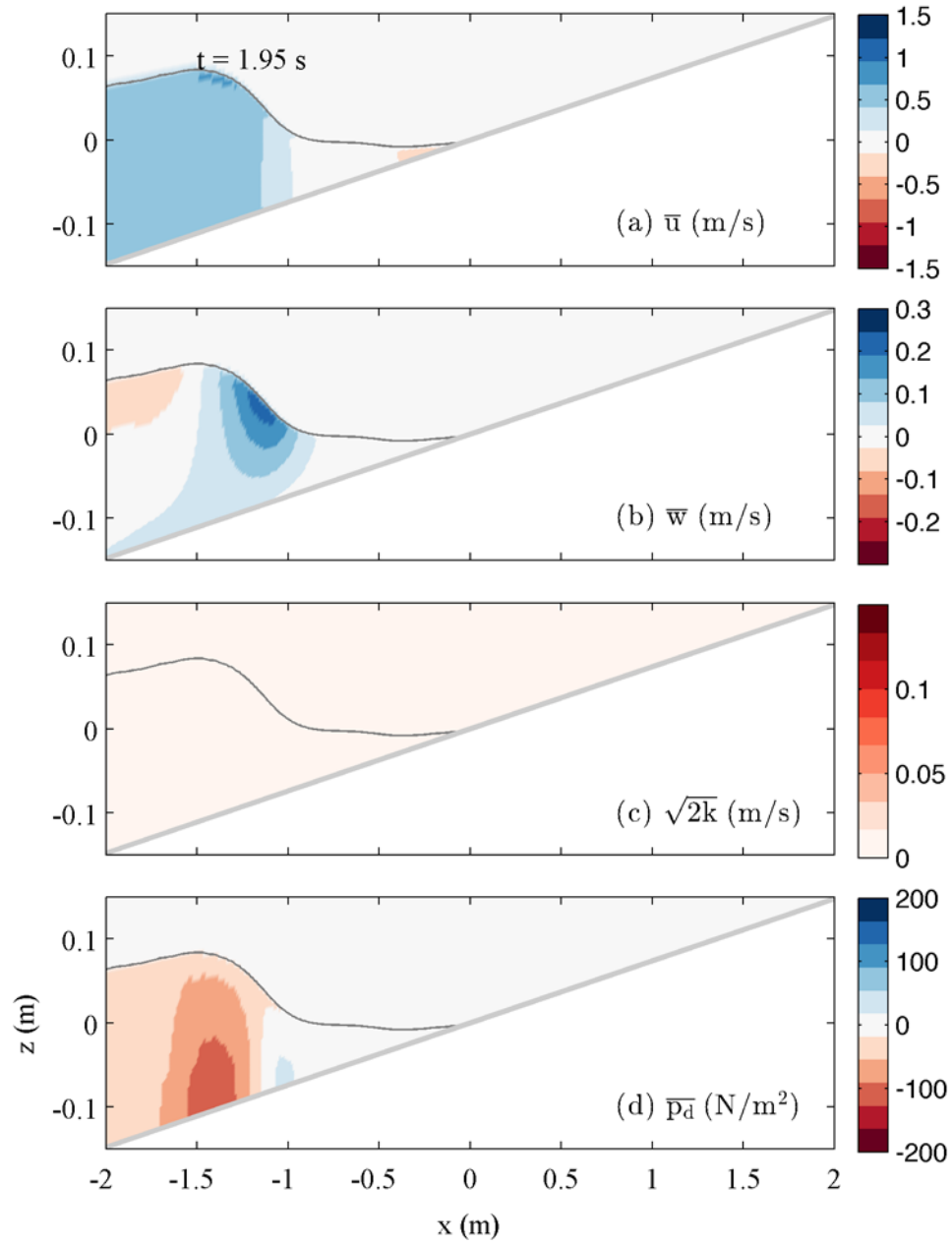


Figure 3.8: Snapshots of the contours of span-wise averaged (a) cross-shore velocity, $\bar{u}$; (b) vertical velocity, $\bar{w}$; (c) turbulence intensity, $\sqrt{2k}$; and (d) dynamic pressure $\bar{p}_d$ at $t = 1.95$ s

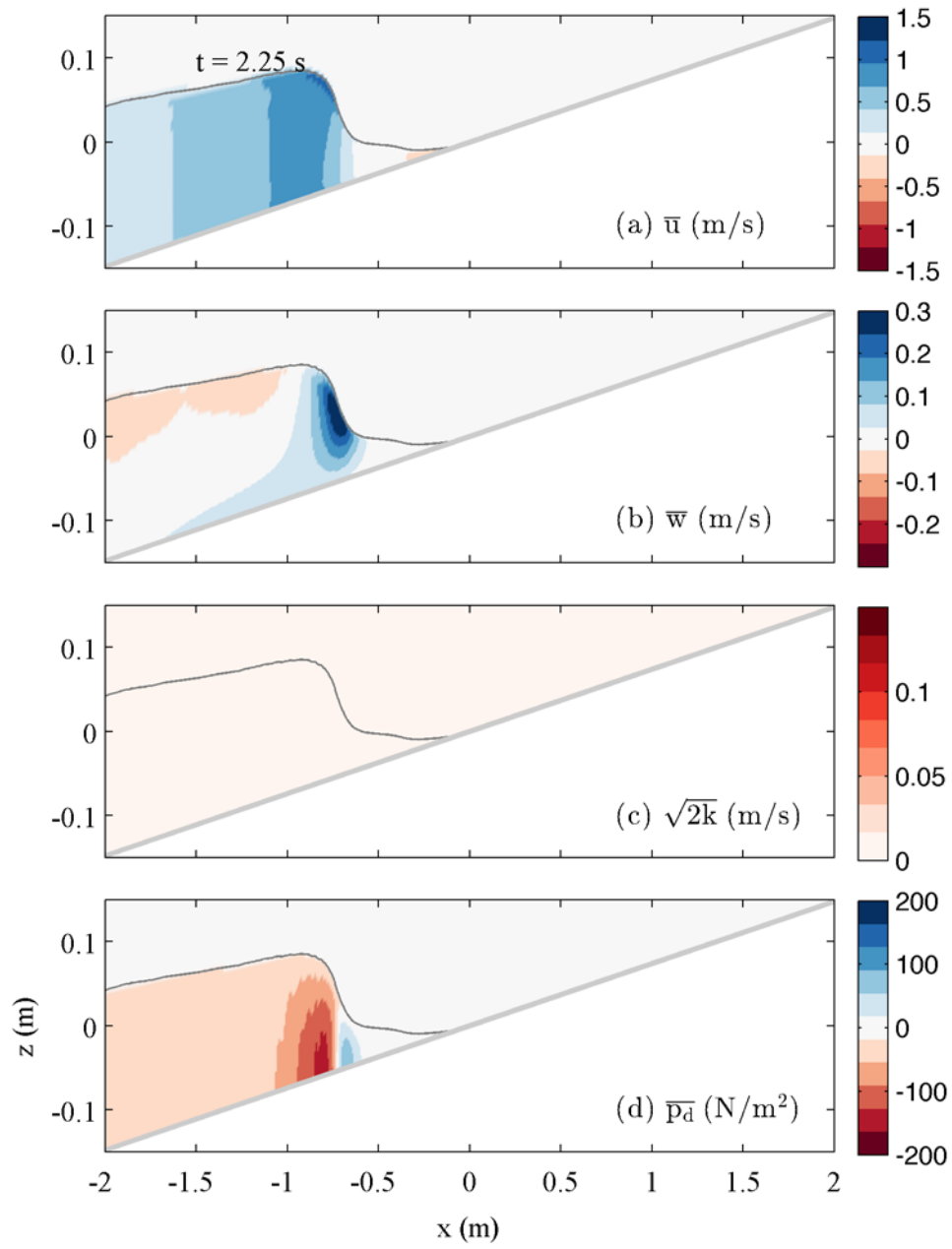


Figure 3.9: Same as Figure 3.8 except at $t = 2.25$ s

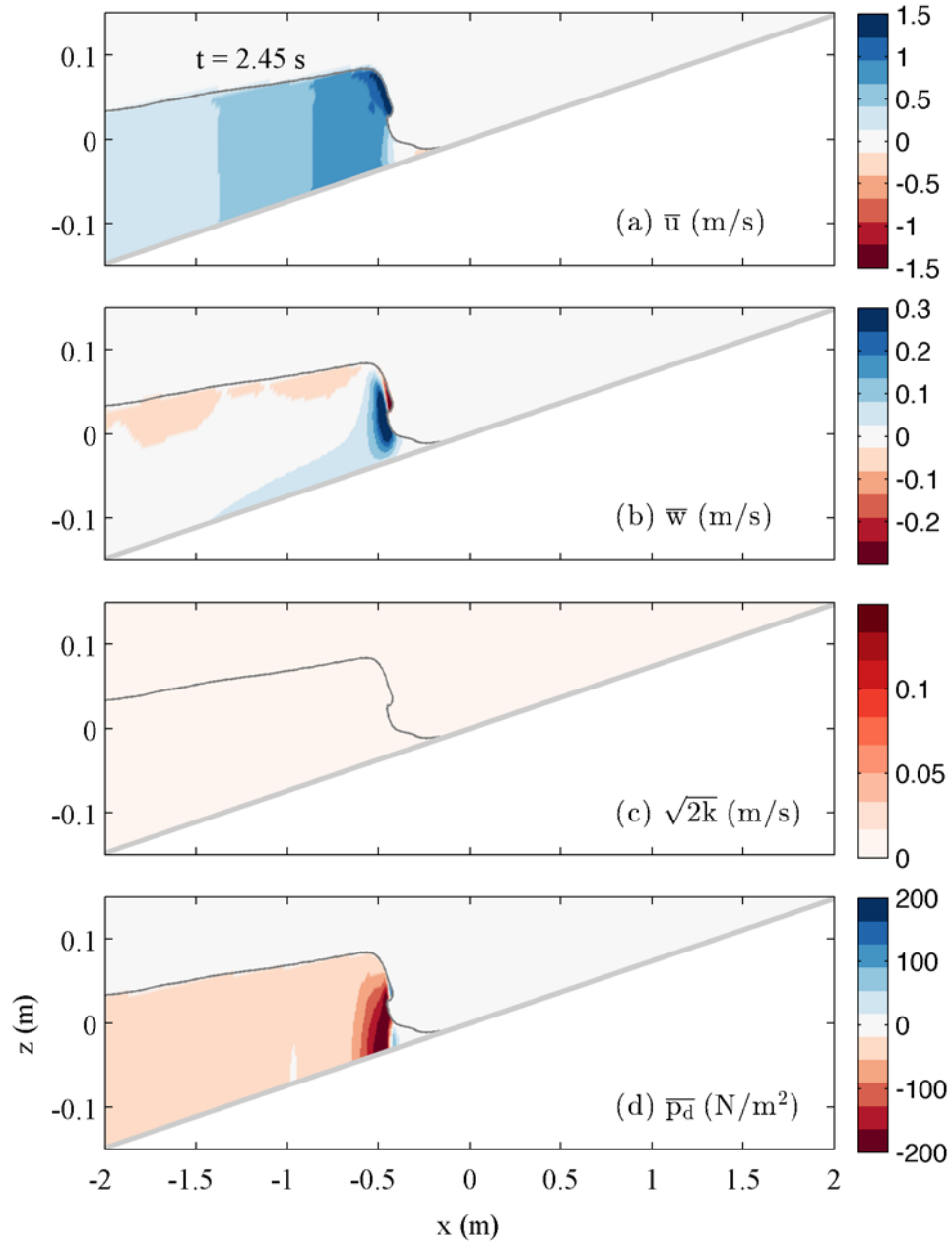


Figure 3.10: Same as Figure 3.9 except at $t = 2.45$ s

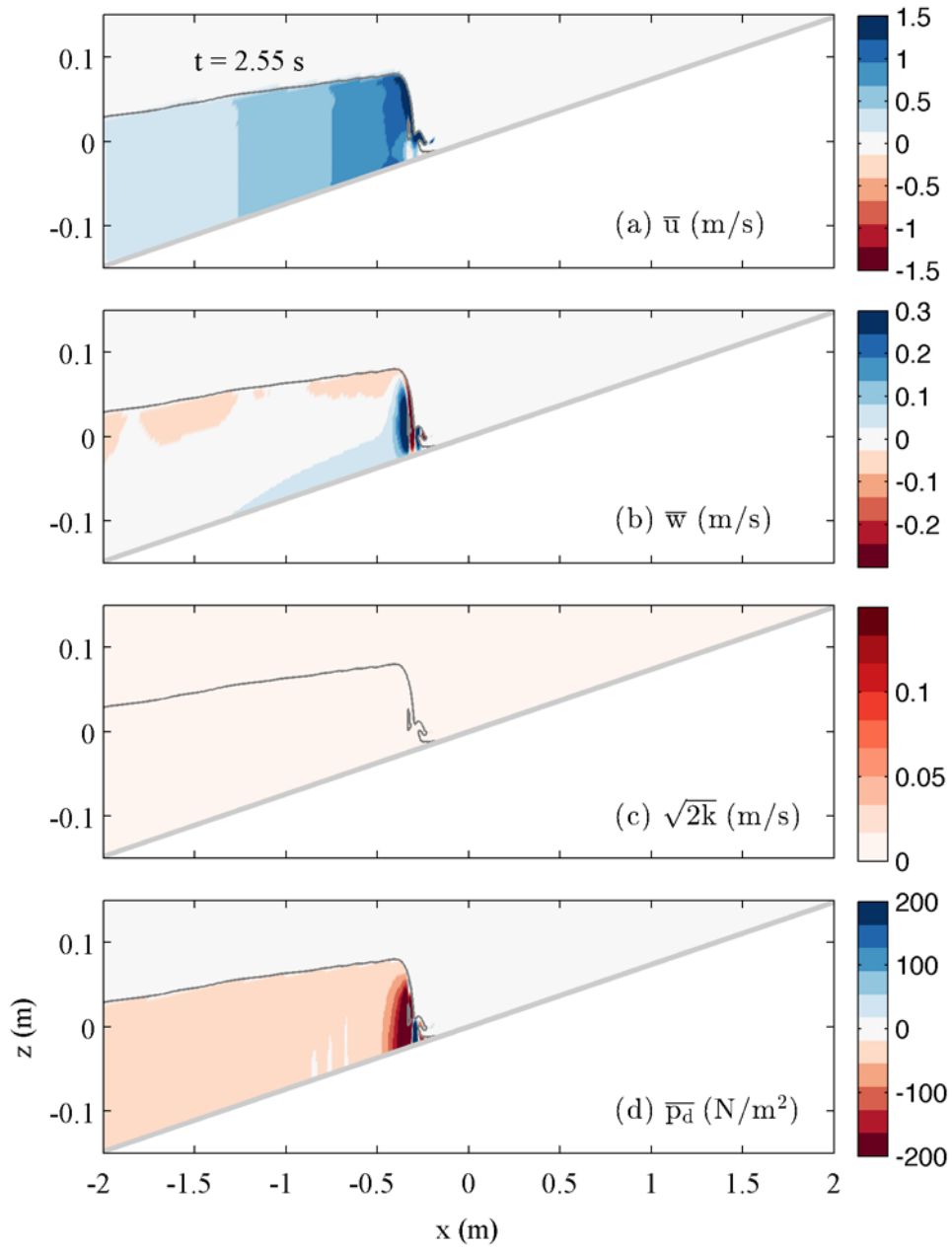


Figure 3.11: Same as Figure 3.10 except at $t = 2.55$ s

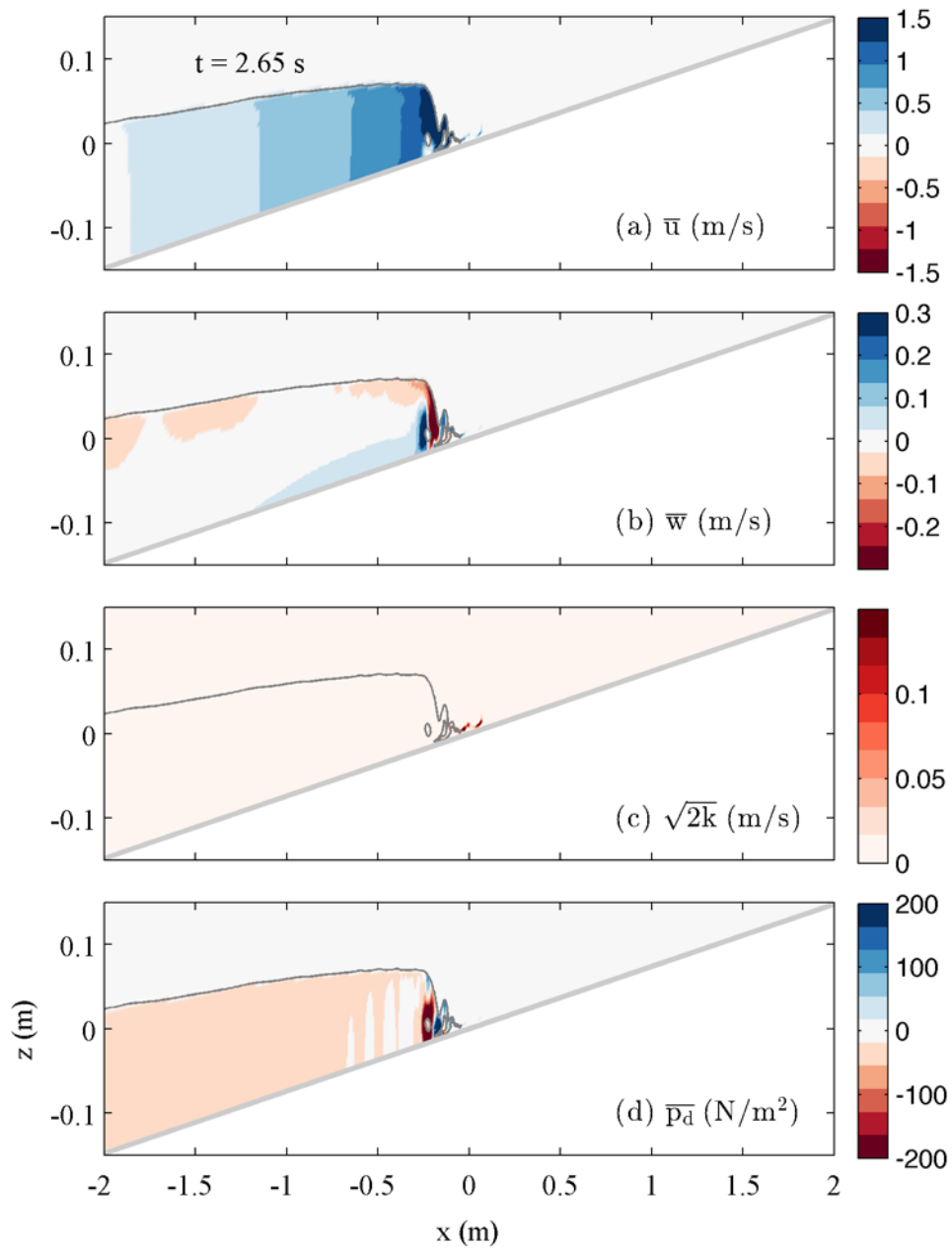


Figure 3.12: Same as Figure 3.11 except at $t = 2.65$ s

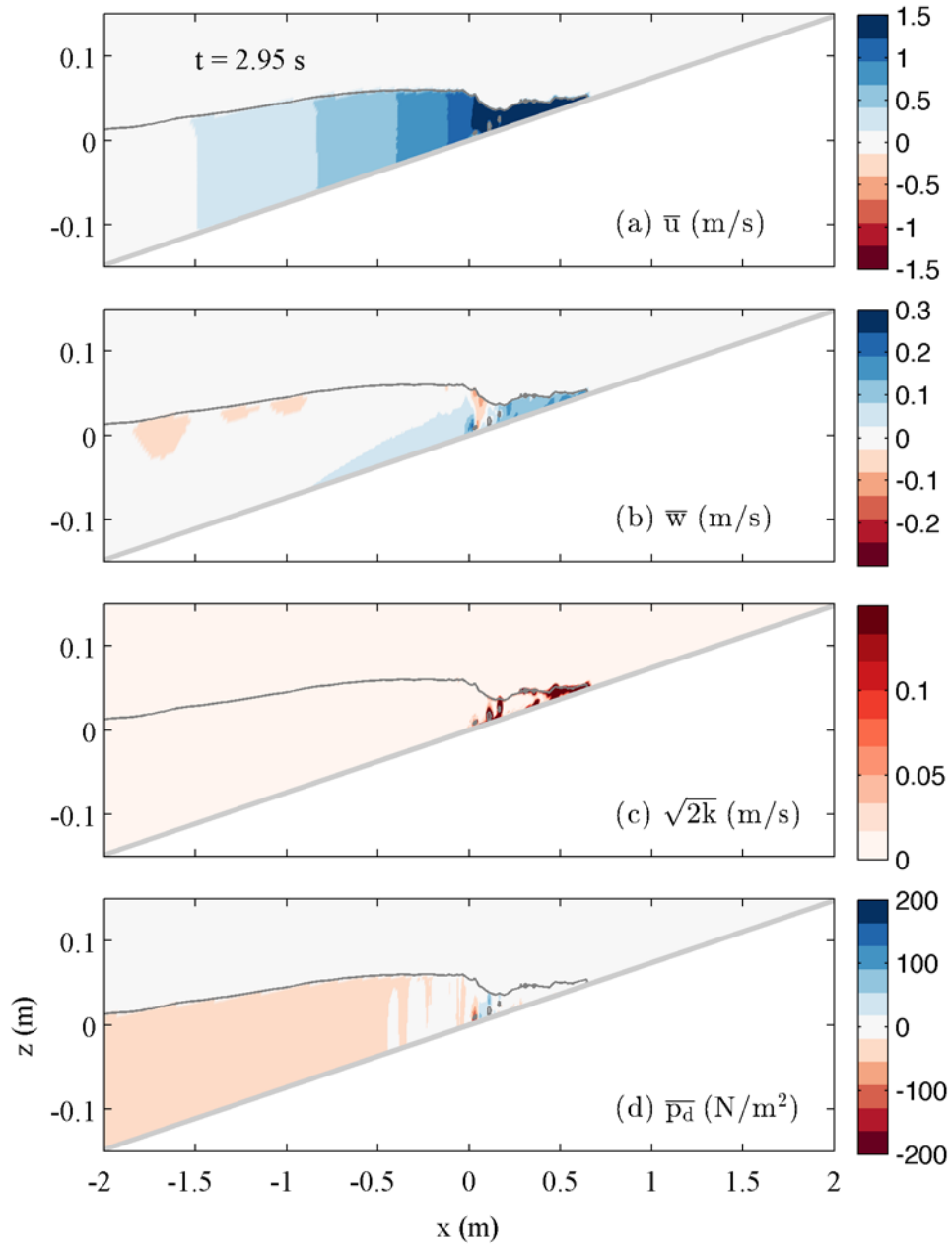


Figure 3.13: Same as Figure 3.12 except at $t = 2.95$ s

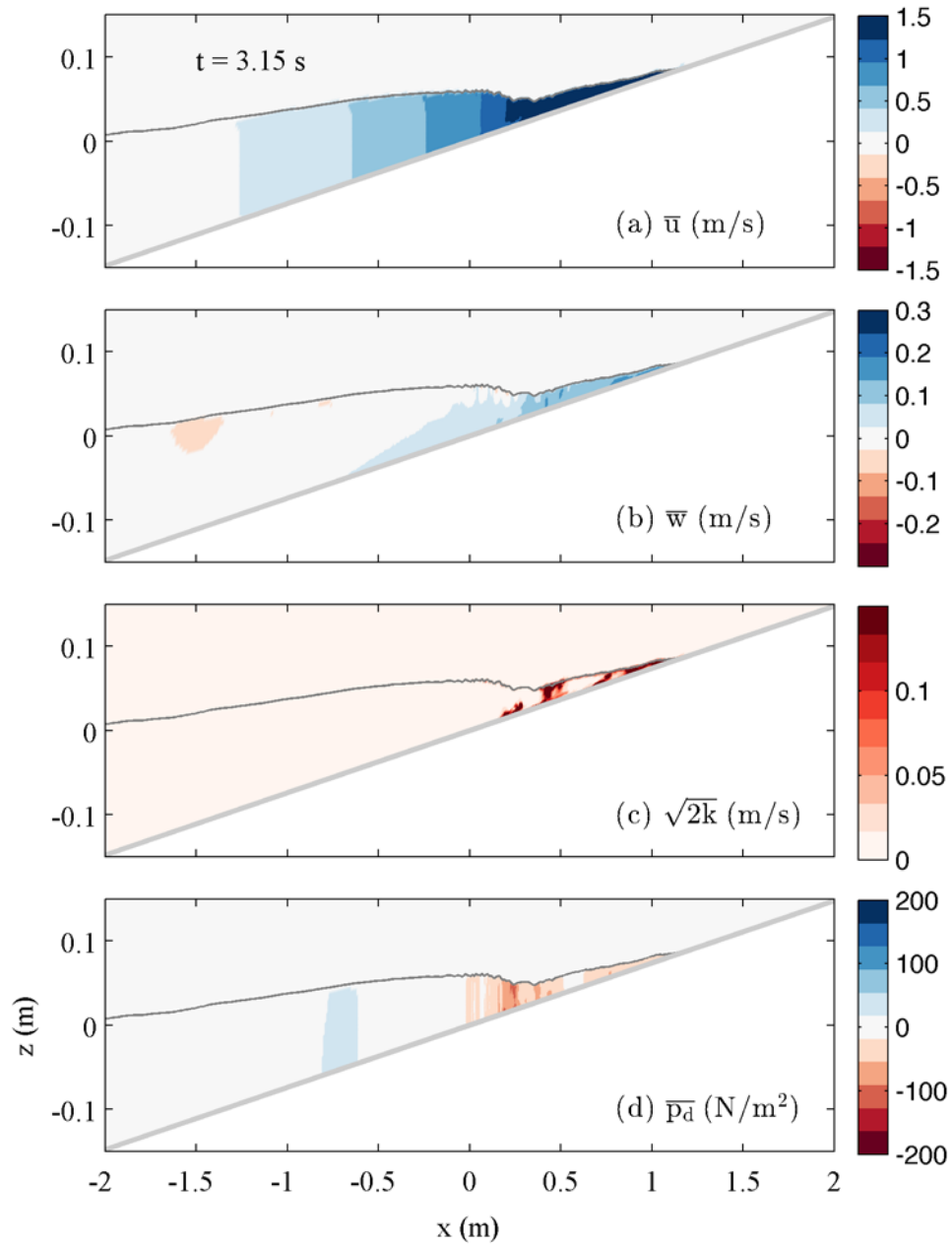


Figure 3.14: Same as Figure 3.13 except at $t = 3.15$ s

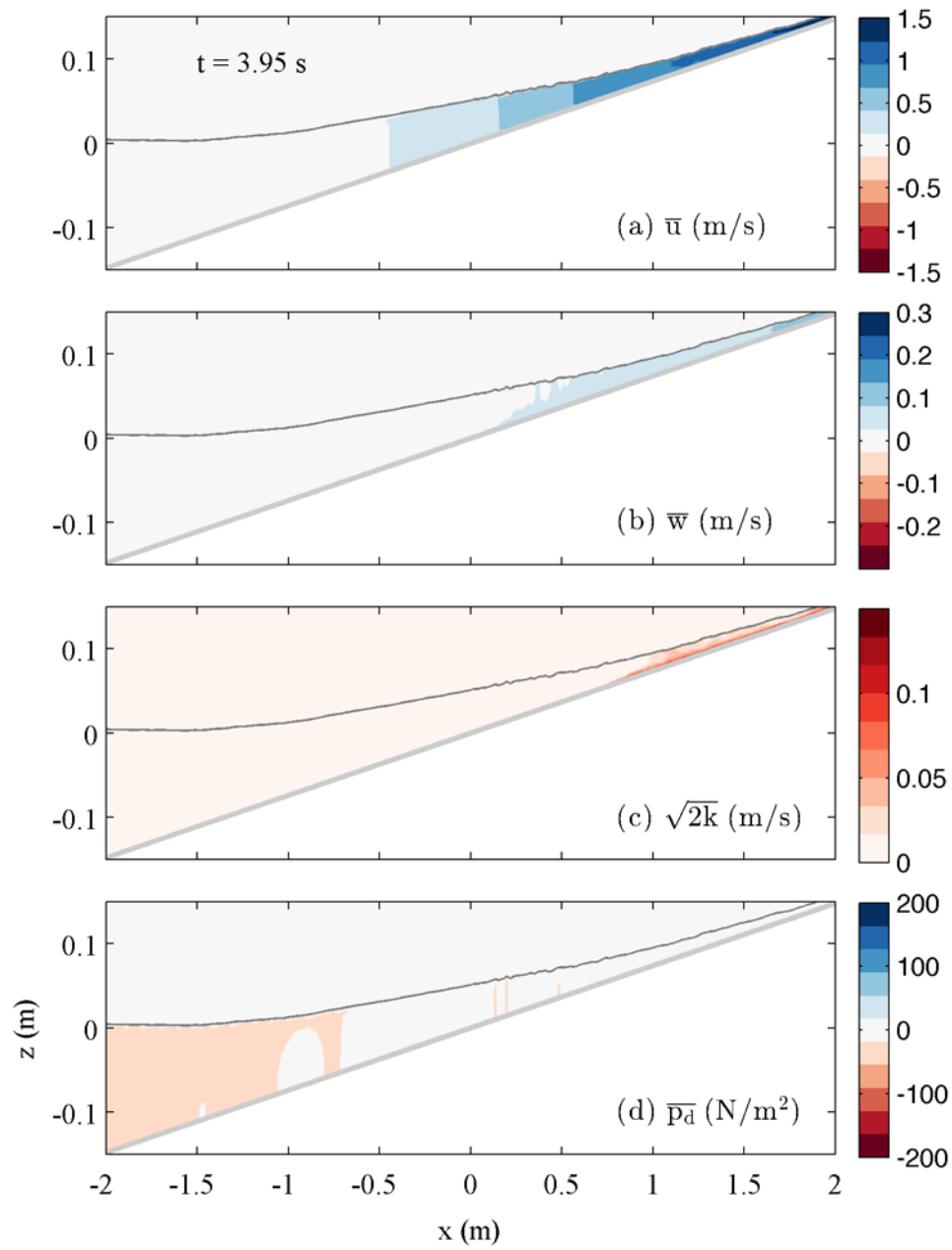


Figure 3.15: Same as Figure 3.14 except at $t = 3.95$ s

To further illustrate the production of turbulence, the evolution of turbulent coherent structures during the run-up stage under the plunging wave is shown in Figure 3.16. The turbulent coherent structures are visualized by using the λ -2 contour. λ -2 method defines vortex cores in terms of the eigenvalues of symmetric tensors consisting of symmetric and antisymmetric parts of velocity gradient tensors (*Jeong and Hussain*, [1994]). Contoured by an appropriate critical eigenvalue, the λ -2 is used to identify the vortex cores and to visualize the turbulent coherent structures.

In this simulation, the λ -2 data contoured with -250 demonstrates the valuable information regarding the evolution of the 2-D rollers and the generation and evolution of turbulence. In Figure 3.16(a), when the front of the free-surface starts curling, the first roller forms behind the wave front. In Figure 3.16(b), when the first impingement of the curling wave triggers secondary impingements, another roller forms in the similar manner. The second roller soon separates into smaller ones as they move onshore, but the 2-D shape is still intact. In Figure 3.16(c), we begin to see the smaller 2-D rollers shatter into smaller 3-D structures. The shattering of the coherent structures is congruent with the generation turbulence intensity in Figure 3.12(c).

During the uprush phase, the shattered coherent structures break into further smaller pieces (see Figure 3.11(d)) and finally dissipate away. No new rollers emerge after the wave is completely broken. The run-up tip is much more

dense with the broken coherent structures than is the following run-up body. This observation agrees with the degenerating trend of turbulence intensity in Figure 3.14c and Figure 3.15c. Like the turbulence intensity, the coherent structures are mostly generated during the wave plunging process, and the contribution of

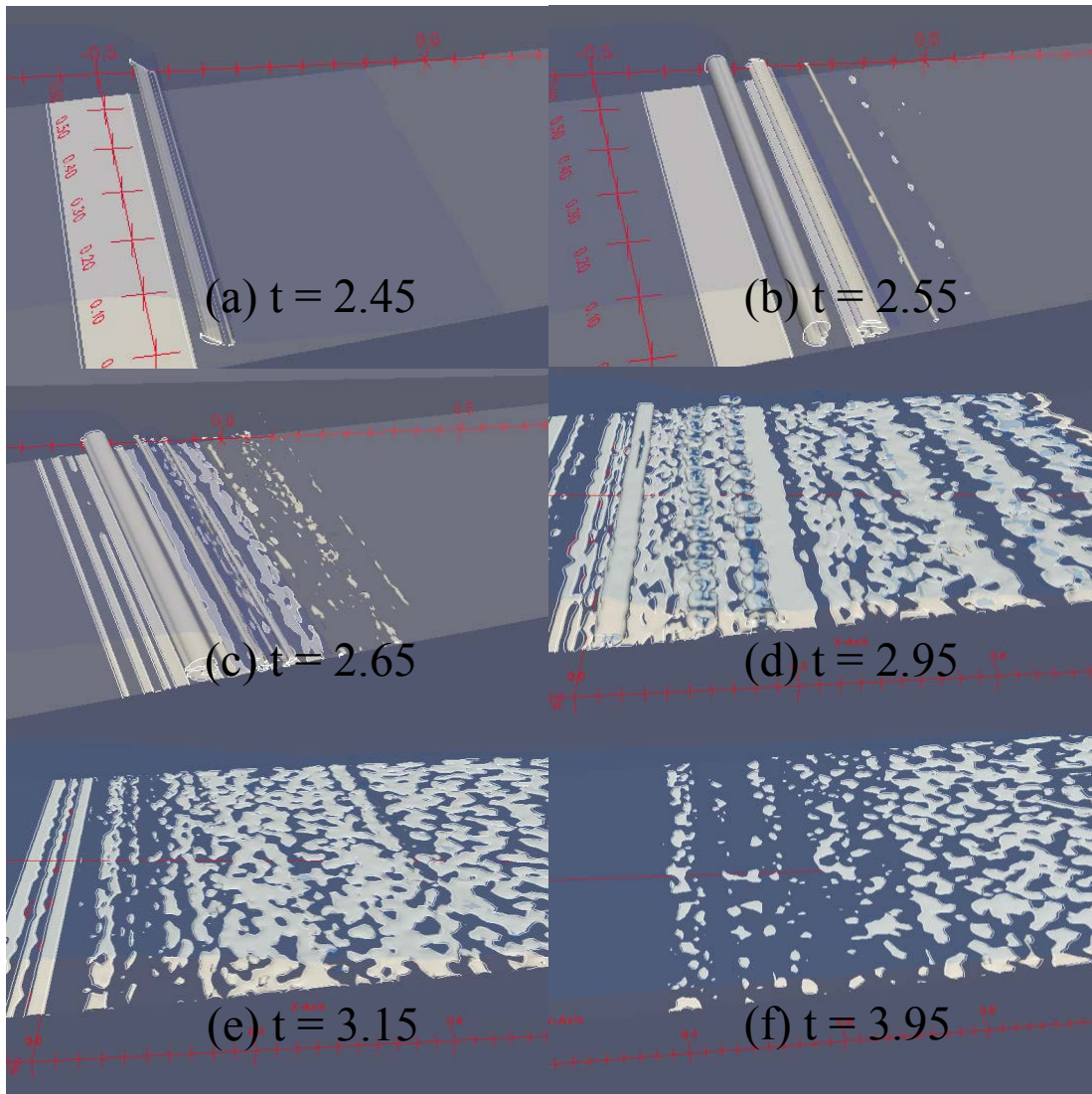


Figure 3.16: Turbulent coherent structures during the wave breaking process. (a) $t = 2.45$ s; (b) $t = 2.55$ s; (c) $t = 2.55$ s; (d) $t = 2.95$ s; (e) $t = 3.15$ s; (f) $t = 3.95$ s.

turbulence generation during the up-rush process appears non if not minimal in the present simulation.

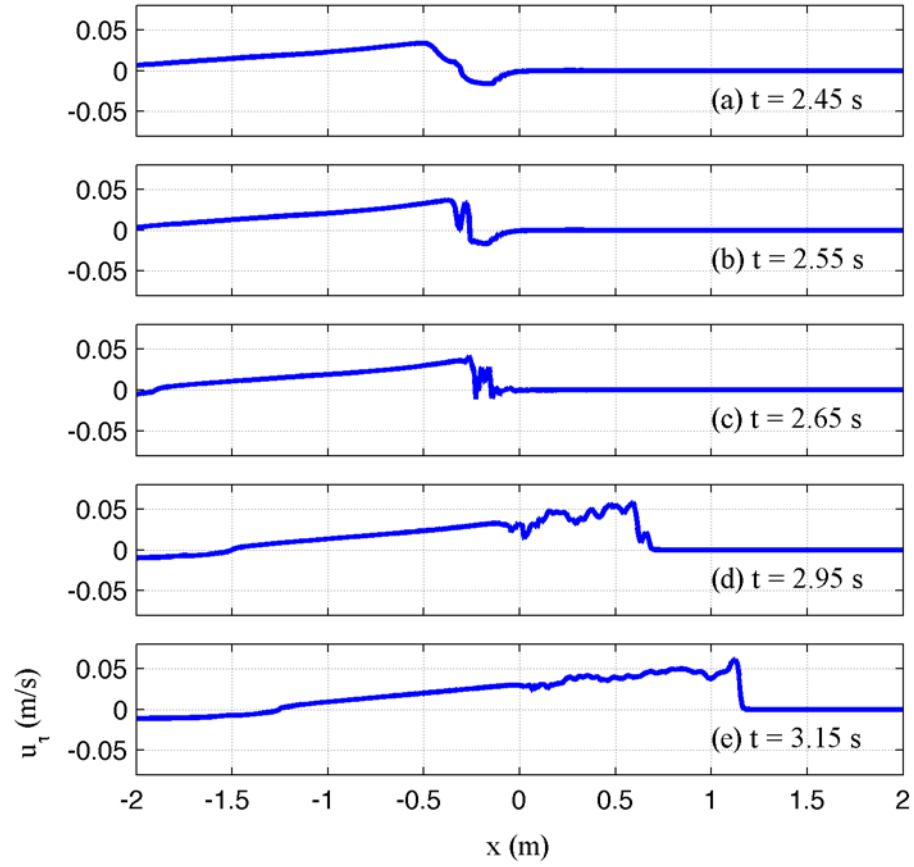


Figure 3.17: Span-wise averaged friction velocity in the snapshots in Figure 3.10-14.

Figure 3.17 shows the span-wise averaged bottom shear stress (BSS), expressed in terms of friction velocity, u_τ . The direction of u_τ is assigned by the direction of cross-shore velocity at the nearest resolved grid point above the bed. As shown in Figure 3.11(a), before the plunging surface impinges onto the bed, the cross-shore distribution of averaged bed friction velocity appears fairly smooth (see Figure 3.10). The peak friction velocity is around 0.04 m/s, which coincide with the location where the wave shape become the steepest. In Figure 3.17(b), the friction velocity shows a sudden drop to 0 around -0.3 m and then immediately followed by another peak. This location corresponds to the location of first impingement of plunging breaker with the large dynamic pressure gradient (see Figure 3.11). Noticeable fluctuations in the friction velocity are observed in Figure 3.17(c), which is associated with the second impingement shown in Figure 3.12. The bed friction velocity shows more fluctuation (limited by the spanwise average in which only limited independent ensemble is available) between 0 and 0.6 m in Figure 3.17(d) right after the plunging breaker completes the impingement process. This observation is congruent with the snapshot in Figure 3.13(d) in which turbulent intensity grows right after the wave impinges to the dry surface of the slope. The fluctuation then decreases in Figure 3.17(e). The overall shape of bottom friction velocity distribution is similar to the run up of the bore.

3.4.2 Run-down and Hydraulic Jumps

The uprush phase gradually evolves into the down-rush phase as the run-up tip hits the maximum run-up heights ($z = 0.27$ m) and the flow reverses its direction seaward. Figure 3.18 shows the transition in the velocity profiles in the mid-section of the swash zone right before and right after the down-rush phase begins.

According to Figure 3.18(b), and (c), it is evident that the flow velocity profile reversal (toward offshore direction) is initiated from the bottom and then migrate to the top because of the presence of no-slip boundary condition. The abrupt change of the velocity directions along the depth causes inflection points near the bottom in the velocity profile along the depth. As the flow moves further down (Figure 3.18(c) and (d)), the inflection point moves slightly upward and disappears later when the run-down flow becomes fairly uniform. The flow reversal process causes strong flow shear and turbulence production (see next).

Figure 3.19 show the snapshot of the several flow quantities when the run-up tip reaches the maximum run-up height. The turbulence intensity (Figure 3.19(c)) is less intense comparing to that in the run-up stage after the plunging breaker impinging to the bed. Turbulence generated here is partly associated with boundary layer turbulence caused by the shear flow presented in Figure 3.18. The horizontal velocity (Figure 3.19(a)) is directed seaward with higher magnitude between $x = -0.5$ and 0 m, the location where the shallow upstream flow acceler-

ates sufficiently due to downslope gravitational effect on the slope but eventually meets the thick downstream water body.

As the downrush phase further progresses as shown in Figure 3.20, the ve-

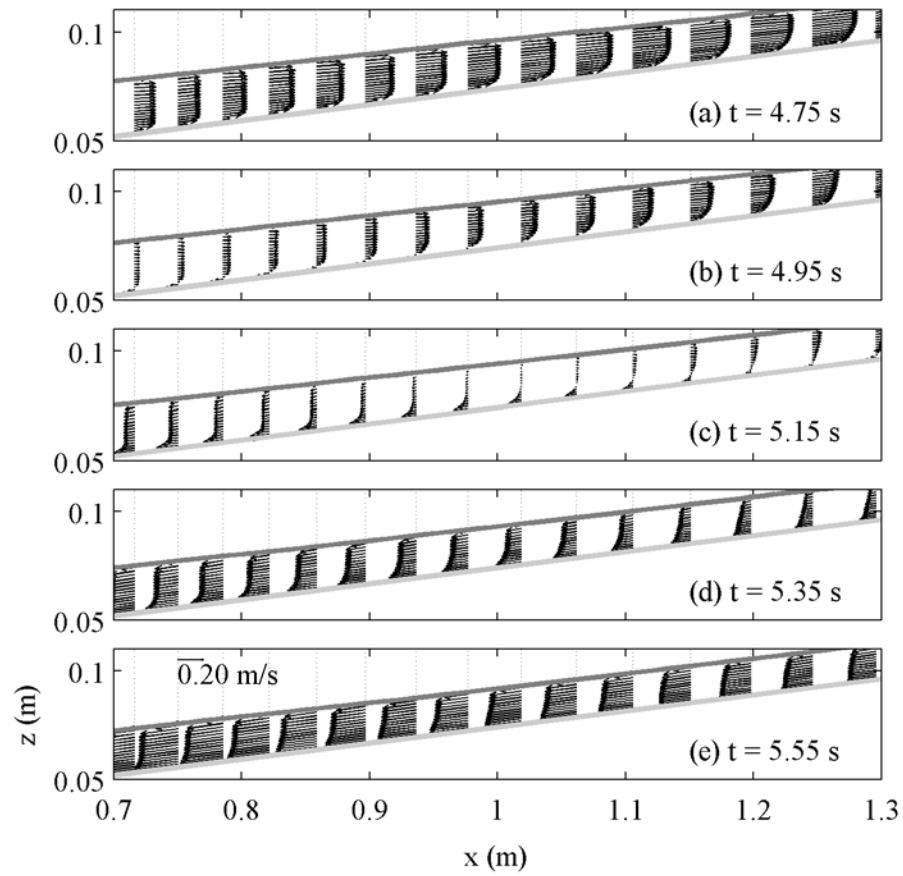


Figure 3.18: The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 4.75$ s; (b) $t = 4.95$ s; (c) $t = 5.15$ s; (d) $t = 5.35$ s; (e) $t = 5.55$ s.

locity around the initial shoreline ($x = 0$) becomes greater than 1 m/s. This relatively strong flow behaves like a jet impinges into the relatively stationary and thick downstream water body located at seaward of $x=-0.5$ m. At this instant, upward vertical velocity intensifies slightly, and the first yet unstable hydraulic jump forms at $x=-0.5$ m. More interestingly, in Figure 3.21 a strong dynamic pressure emerges from the bottom around $x=-0.5$ m at the location where the first hydraulic jump occurs. Here, the change in directions and the magnitude of the vertical velocity are much more intense, which reflects strong cross-shore pressure gradients. As we will discuss in more detail next, the flow here is associated with a formation of flow separation.

Considerable magnitude of turbulence intensity starts appearing in Figure 3.22 in both the bottom separation region and the unstable surface. The bottom and the surface turbulence intensity disperse upward and downward, respectively. They ultimately converge into one turbulent region as shown in Figure 3.23(c). Also, the unstable jump collapses at this moment. The maximum turbulence intensity becomes greater than 0.5 m/s even though the figure shows only up to 0.1 m/s. The first separation region is now advected further offshore (at around $x=-0.7$ m) while according to dynamics pressure shown in Figure 3.23(d), the second flow separation starts to be initiated at $x=-0.5$ m. In Figure 3.24, the flow from the slope eventually becomes very shallow, and the velocity reaches as high as 2 m/s. The water surface crumbled in Figure 3.23 stabilizes, and the much organized and

large hydraulic jump simultaneously emerges at $x=-0.7$ m. The entire region near the hydraulic jump is of very high turbulence level.

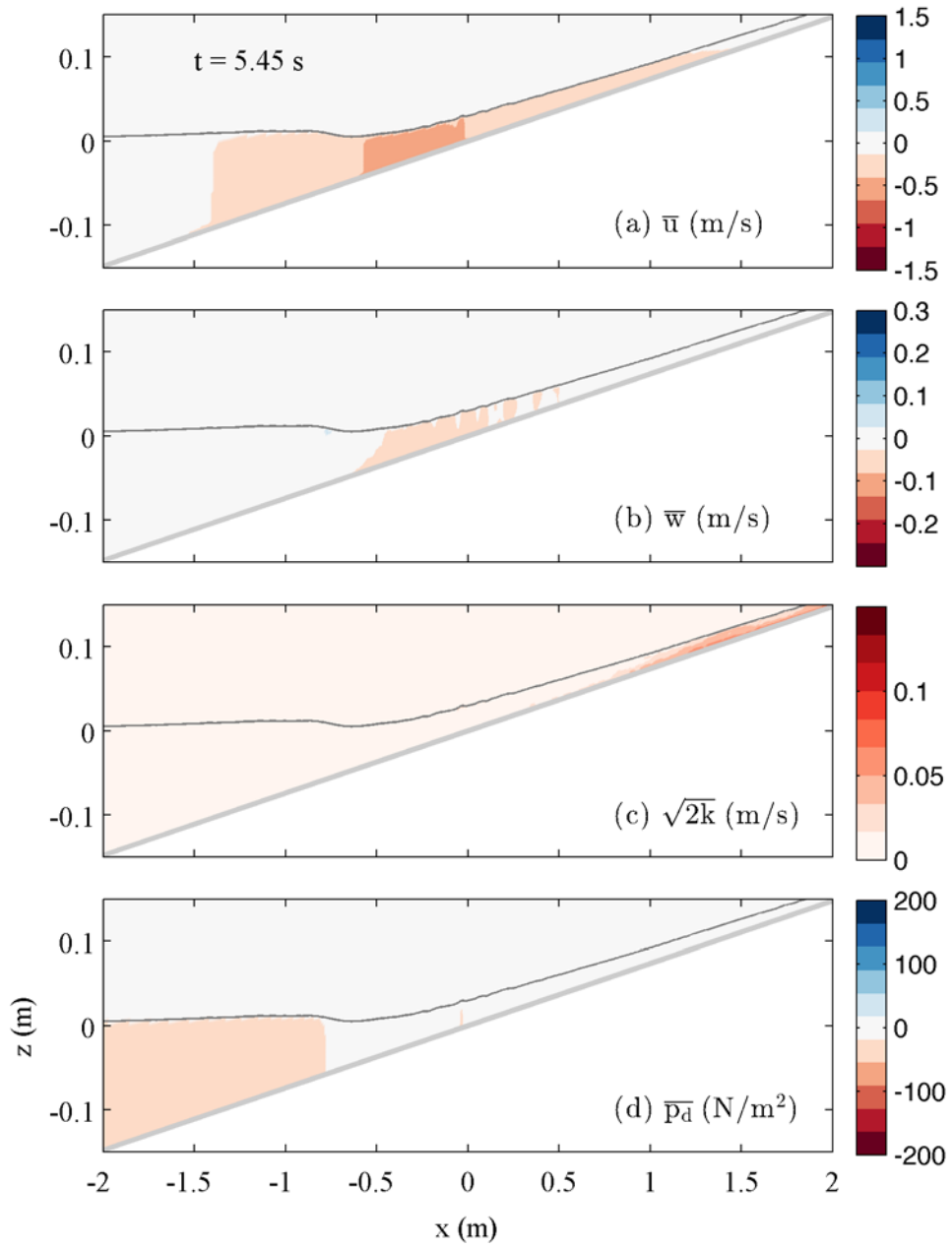


Figure 3.19: Snapshots of the contours of span-wise averaged (a) cross-shore velocity, $\bar{u}$; (b) vertical velocity, $\bar{w}$; (c) turbulence intensity, $\sqrt{2k}$; and (d) dynamic pressure $\bar{p}_d$ at $t = 5.45$ s

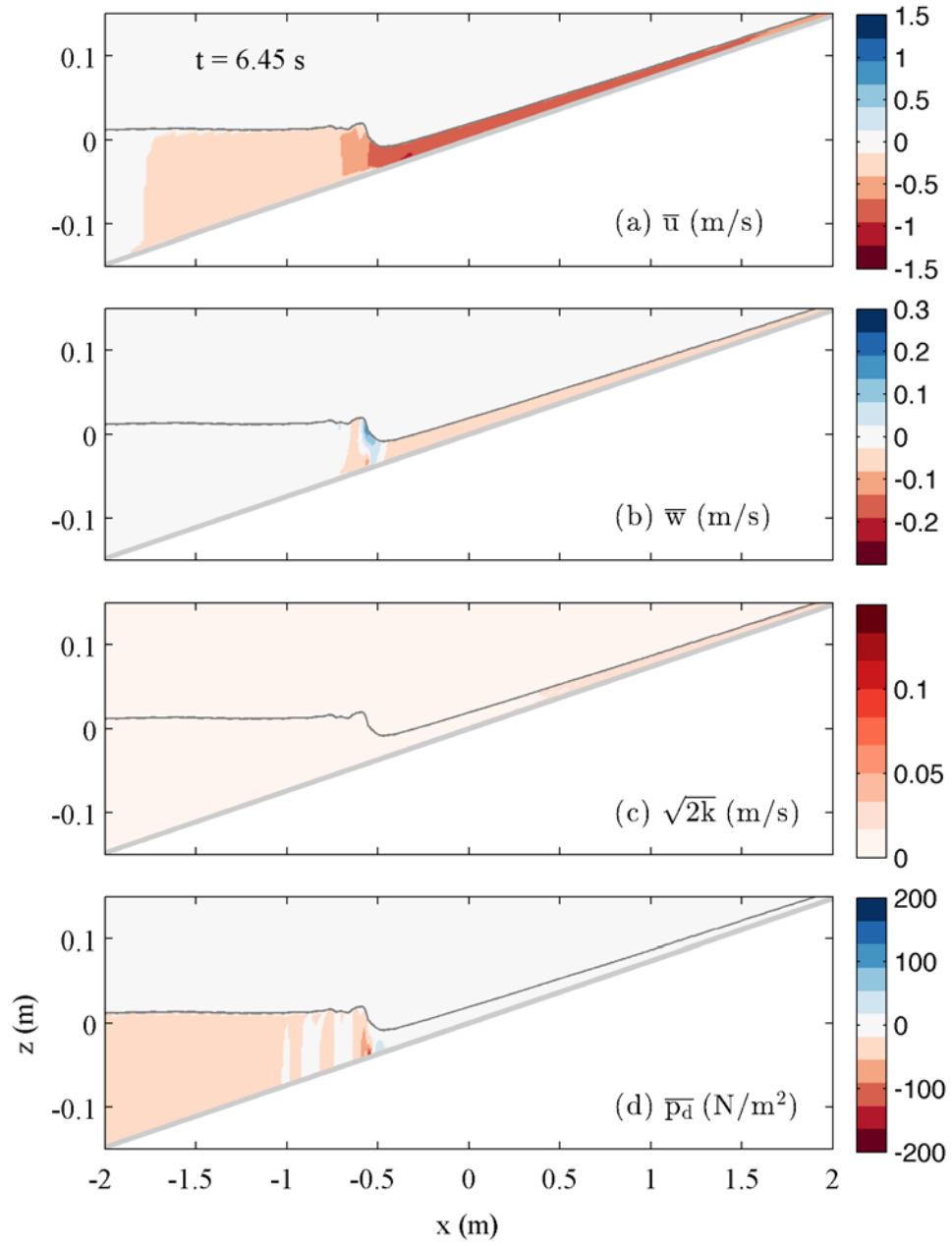


Figure 3.20: Same as Figure 3.19 except at $t = 6.45$ s

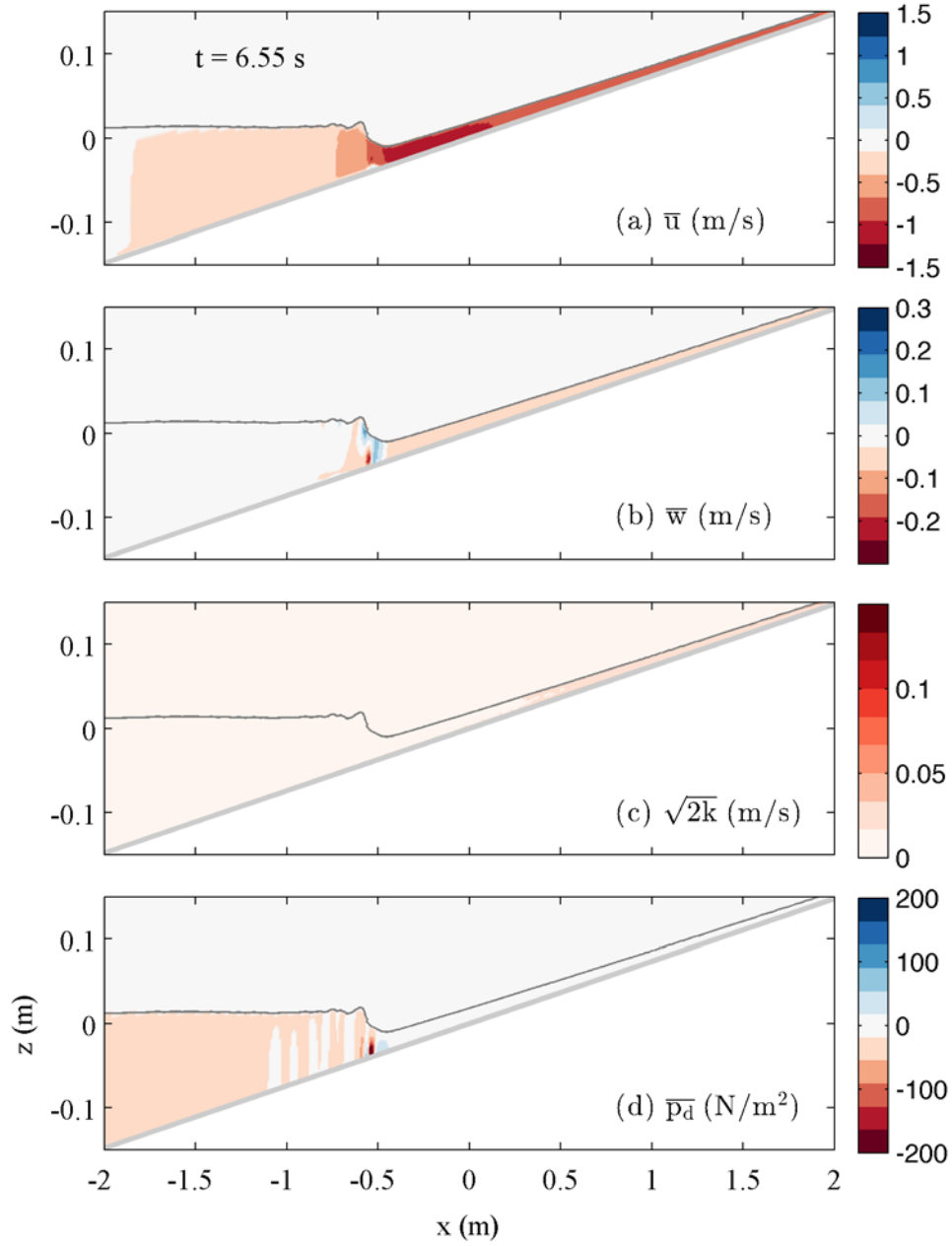


Figure 3.21: Same as Figure 3.20 except at $t = 6.55$ s

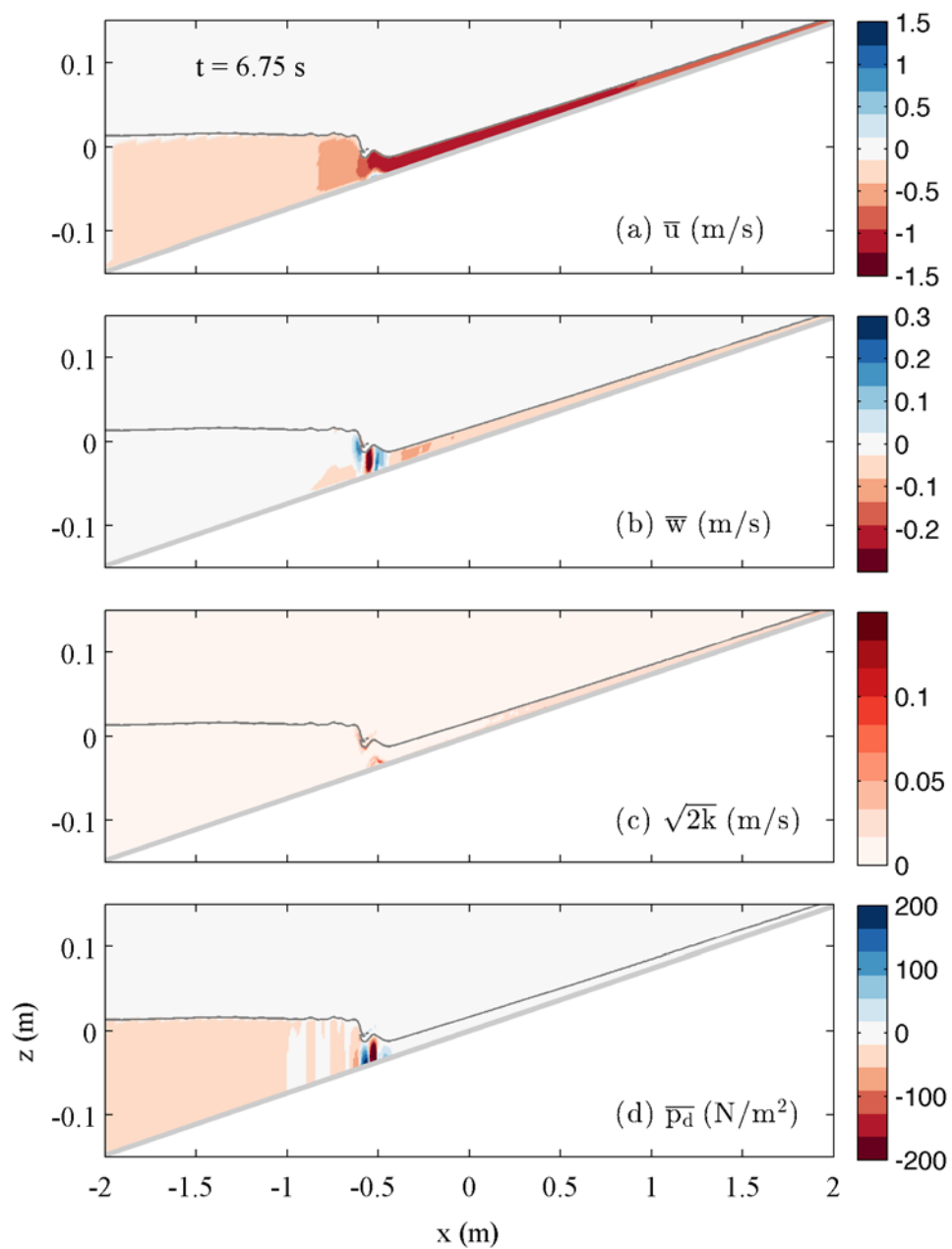


Figure 3.22: Same as Figure 3.21 except at $t = 6.75$ s

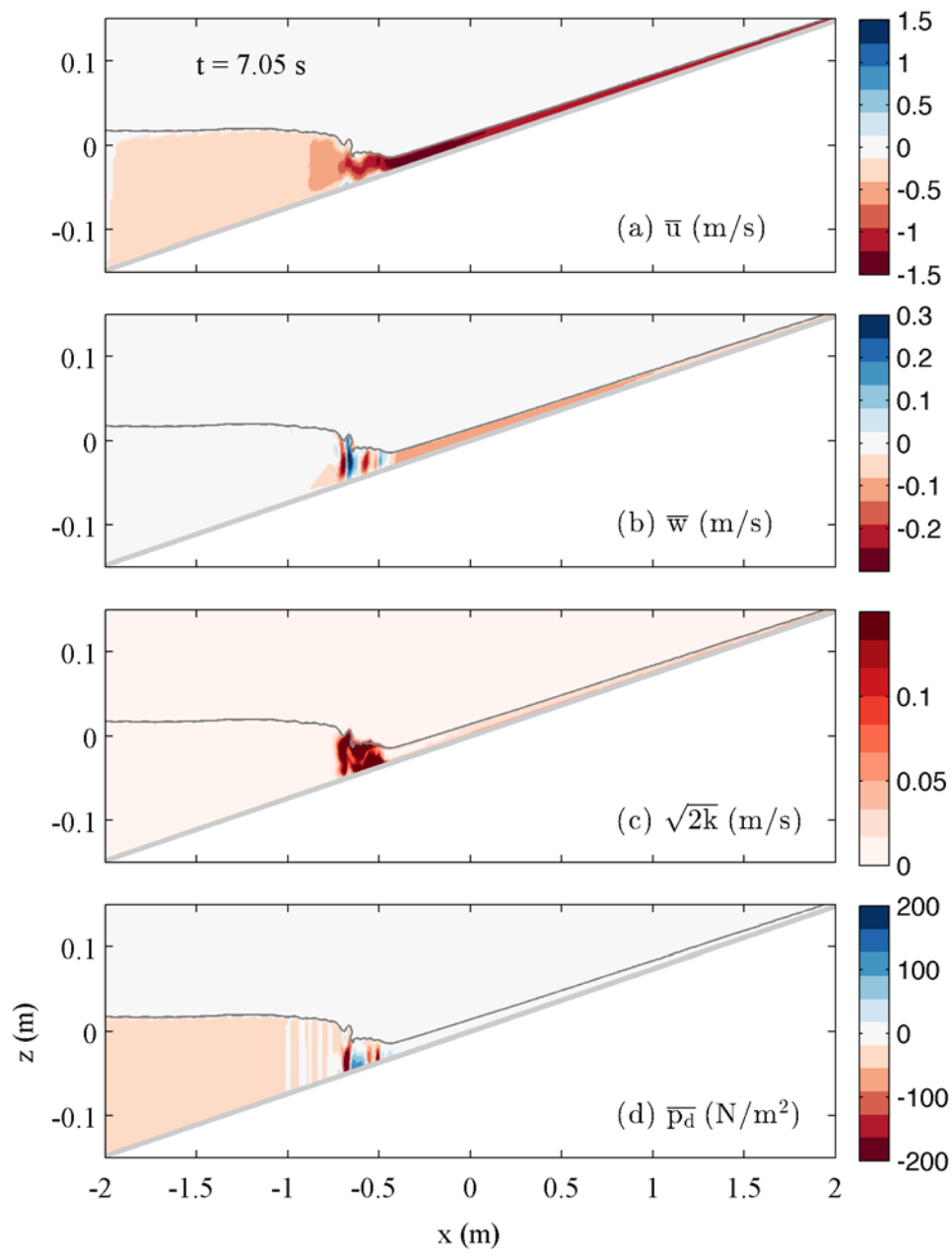


Figure 3.23: Same as Figure 3.22 except at $t = 7.05$ s

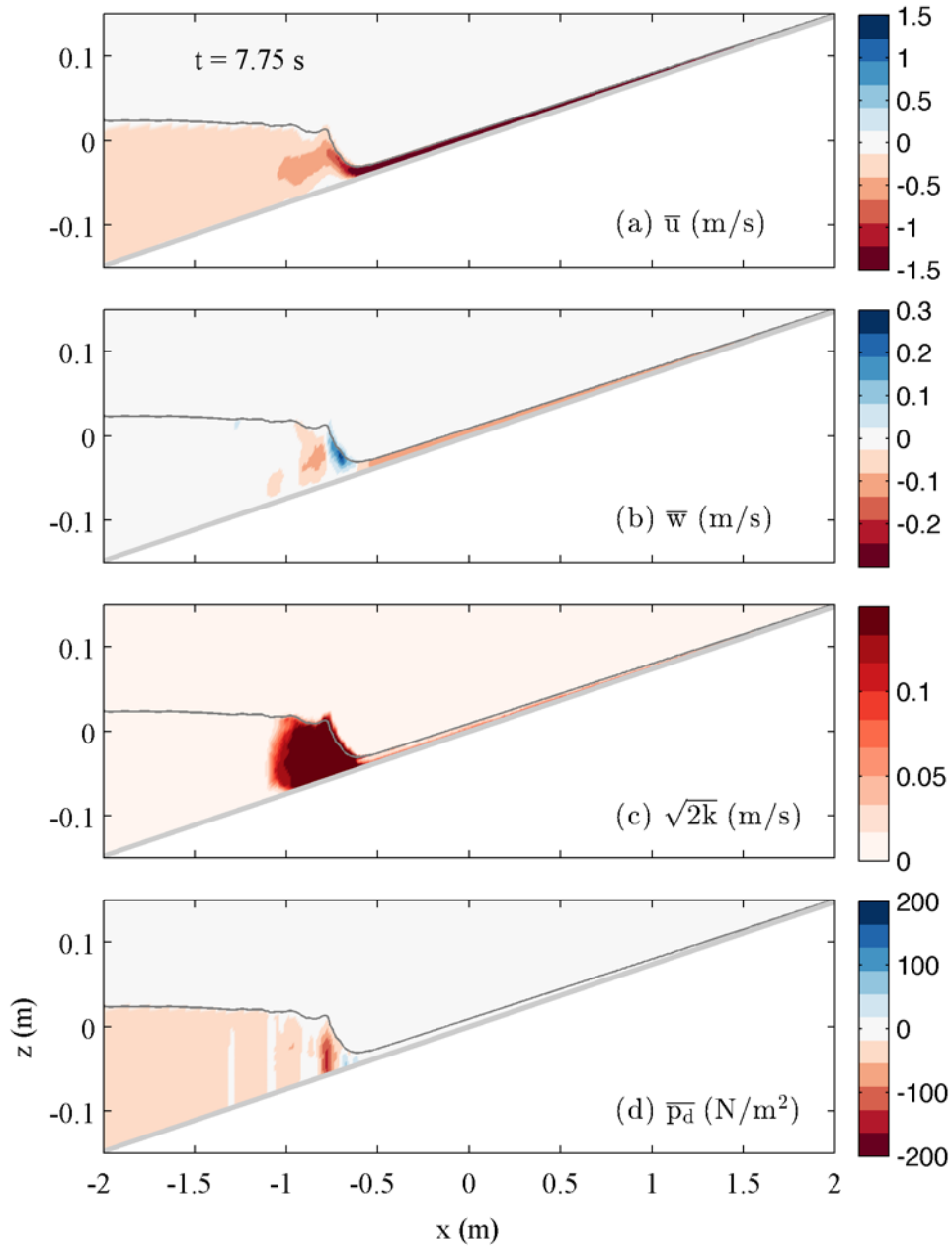


Figure 3.24: Same as Figure 3.23 except at $t = 7.75$ s

In Figure 3.25, the λ -2 contour is used to visualize the coherent structure evolution during the drawdown process. In Figure 3.25(a), a flat panel-like roller builds up from the bottom at $x=-0.5$ m. Only 0.1 sec later (see Figure 3.25(b)), the 2-D roller develops into more mature form at $x=-0.55$ m right under the hydraulic jump. This roller is associated with boundary layer flow separation mentioned in Figure 3.21. In Figure 3.25(c), the hydraulic jump becomes unstable, and a smaller 2-D roller forms just under the oscillating jump surface. Hence, at this instant ($t=6.65$ s), two rollers coexist, one near the solid bottom due to boundary layer separation, and the other near the surface due to free shear flow mechanism. The roller near the surface start to degenerating into smaller 3D coherent structures in at $t=6.75$ s (Figure 3.25(d)). In Figure 3.25(e), as the unstable jump collapses, the surface roller completely breaks down to 3-D structures, while the bottom coherent structure also starts to deform into 3D. This evolution into 3D is also reflected in the considerably elevated TKE shown in Figure 3.23(c). The surface roller completely crumbles into small, irregular shaped turbulent coherent structures in Figure 3.25(f).

The formation of the rollers coincides with the adverse dynamic pressure gradient, and the breaking of the rollers corresponds to the generation of the turbulence, as discussed earlier. *Sou and Yeh*, [2011] report similar flow features in their studies in surf and swash zone flow interactions under periodic waves. They show that the generation of strong vorticity is closely related to the cross-shore

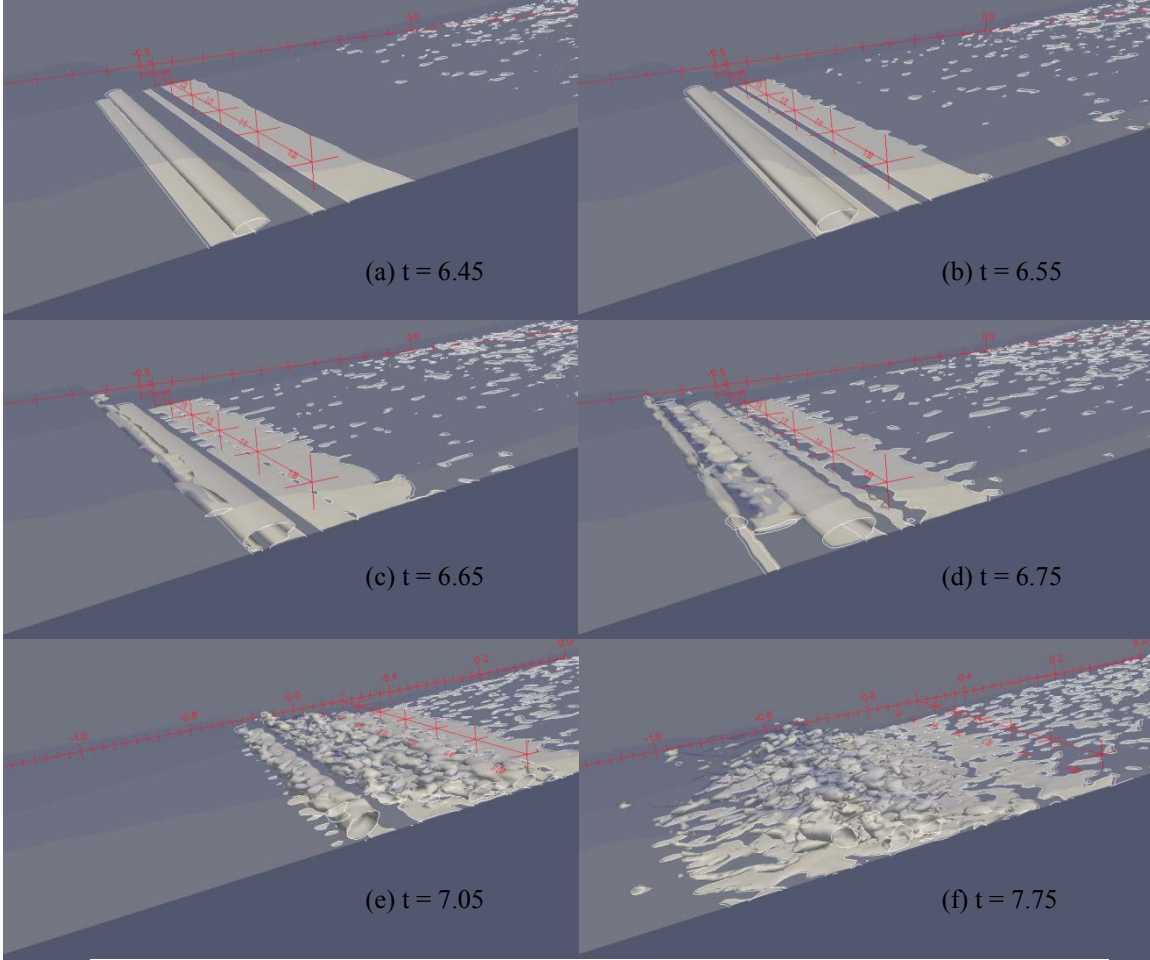


Figure 3.25: Turbulent coherent structures under the hydraulic jumps. (a) $t = 6.45$ s; (b) $t = 6.55$ s; (c) $t = 6.65$ s; (d) $t = 6.75$ s; (e) $t = 7.05$ s; (f) $t = 7.75$ s.

pressure gradient near the solid boundary based on simplified (boundary layer approximation) momentum equation. Similarly, in this numerical investigation, the sudden change in pressure along the cross-shore axis appears to lead the genera-

tion of 2-D rollers at the bottom under the hydraulic jump, which ultimately leads to the generation of turbulent kinetic energy.

3.4.3 Separation of Boundary Layers during Hydraulic Jump

The most intriguing feature of the hydraulic jump observed in this simulation is the separation of boundary layer flow underneath it. Figure 3.26 shows the vector plot of the velocity under the hydraulic jumps between $x=-1$ m and $x=-0.3$ m. In Figure 3.26(a)~(c), an inflection point appears at the bottom of the velocity profile and ascends with the evolution of at $x=-0.55$ m. The sudden changes of the flow directions along the water depth indicate the separation of the boundary layer and the creation of a 2-D roller, which are observed in the coherent structures in Figure 3.25(a)~(d). While the first hydraulic jump is collapsing in Figure 3.26(c), another inflection point appears in the velocity profile at $x=-0.65$ m, signaling the formation of the second hydraulic jump. This inflection point moves upward like the previous one, and then migrates downstream to $x=-0.7$ m.

Figure 3.27 (a-1)~(f-1) shows the evolution of the Froude number along the cross-shore axis where the hydraulic jumps occur. Clearly, landward of $x=-0.5$ m, the downrush flow can be categorized as supercritical ($Fr > 10.0$), while seaward of $x=-0.5$ m, the Froude number is no more than 4. The more than two-fold difference between the landward and seaward Froude numbers at $x=-0.5$ m is consistent with the formation of hydraulic jump feature (or a sharp transitional re-

gion). In the open-channel flow, the sharp change of Froude number can indicate the formation of the hydraulic jump (*Chow*, [1973]). Notice that the seaward Froude number is never smaller than 1, suggesting that the hydraulic jump has to migrate seaward and hence the process is more complicated than the classic hydraulic jump in open-channel flow. Indeed, numerical results show that there exists evolution of the first (smaller) and secondary (larger) jump while the hydraulic jump migrates from $x=-0.5$ m to $x=-0.7$ m within 1.5 sec.

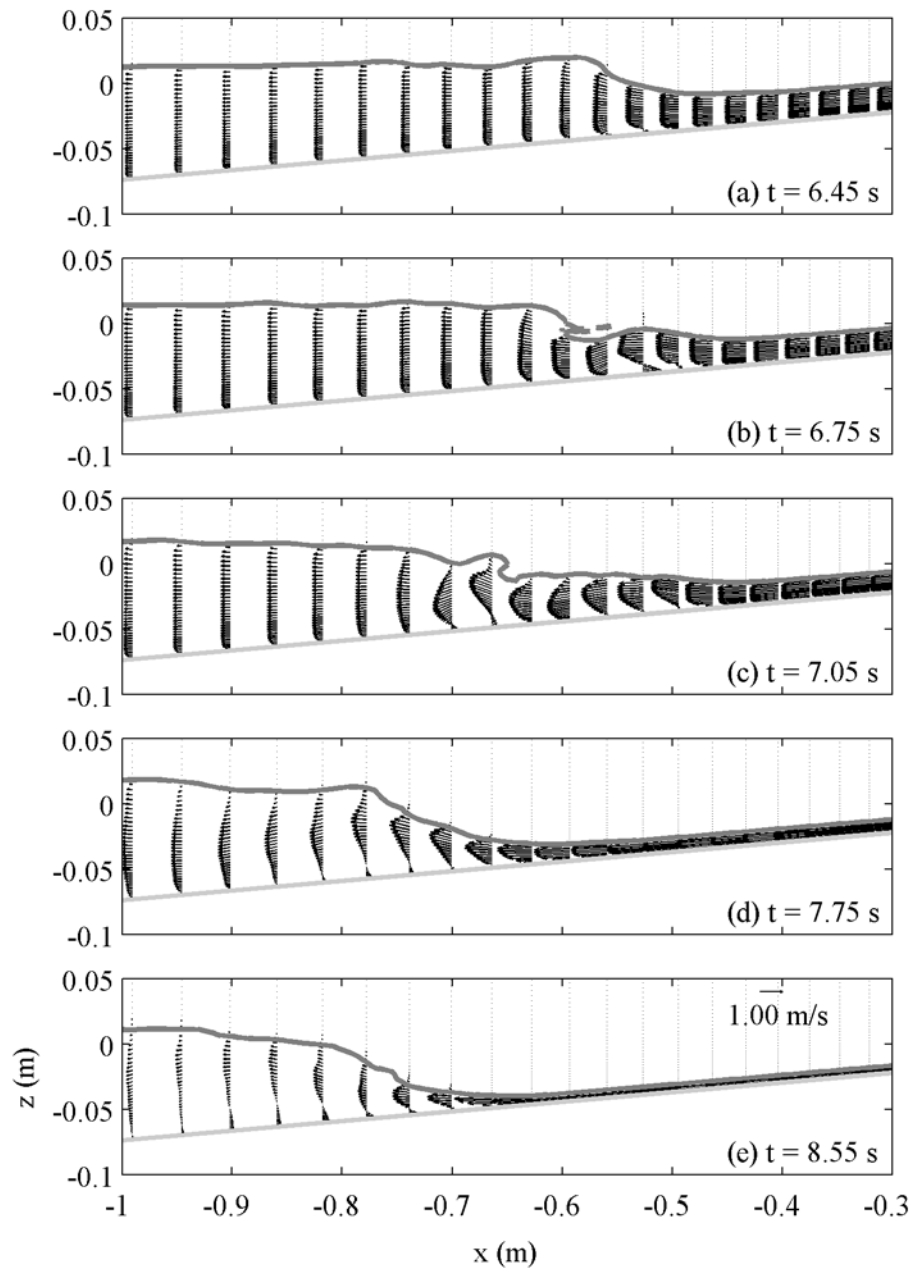


Figure 3.26: The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 6.45$ s; (b) $t = 6.75$ s; (c) $t = 7.05$ s; (d) $t = 7.75$ s; (e) $t = 8.55$ s.

The hydraulic jump in this simulation is very similar to the D-jump described in *Ohtsu and Yasua* [1990]. *Ohtsu and Yasuda* [1990] categorized the hydraulic jumps in sloping channels depending on the slope, θ , and the subcritical and supercritical depth ratio, h_1/h_2 in which h_1 and h_2 are the upstream and downstream water depths, respectively. With sufficiently high depth ratio and mild slope ($\theta < 19^\circ$), the flow on the sloping channel forms a D-jump, a hydraulic jump that occurs on the slope without affecting the downstream flow on the flat bottom. The flows on steep slopes, however, were observed to be incapable of creating a hydraulic jump or any visible jump-like shape in the water surface; the flows rather resembled classical wall jets. The conditions such as the slope and the depth ratios in this simulation safely satisfy those for the D-jump category. The beach slope is around 4.1 degree, and the depth ratio is sufficiently large throughout the run-down process (above 40).

The right panels of Figure 3.27 show the corresponding cross-shore distribution of bottom friction velocity. At the early stage of the drawdown process (see Figure 3.1. 20(a-2)), friction velocity is offshore directed (negative) with larger magnitude in the landward location and smaller magnitude in the seaward direction. In Figure 3.27(b-2), the friction velocity at around $x=-0.6$ m reaches 0 m/s. This is the same location where boundary layer separation occurs at slightly later time (see Figure 3.20). According to boundary layer theory, adverse pressure gradient can counteract with bottom stress and the resulting location where bot-

tom stress magnitude becomes zero signifies the onset of flow separation. The corresponding Fr also shows drastic change at this location (see Figure 3.27 (a-2)). In Figure 3.27(c-2), the range of positive friction velocity becomes wider, signifies the expansion of flow separation. At later time (see Figure 3.27(d-2) and (e-2)), we can identify two separation region according to the variation of bottom friction velocity (two peaks of friction velocity exceed positive value). The positive friction velocity peak coincides with the negative dynamic pressure; the negative friction velocity, with the positive dynamic pressure. The magnitude of the fluctuations decreases as the hydraulic jump further develops, but the cross-shore distribution of the peaks expands seaward (see Figure 3.27(f-2)).

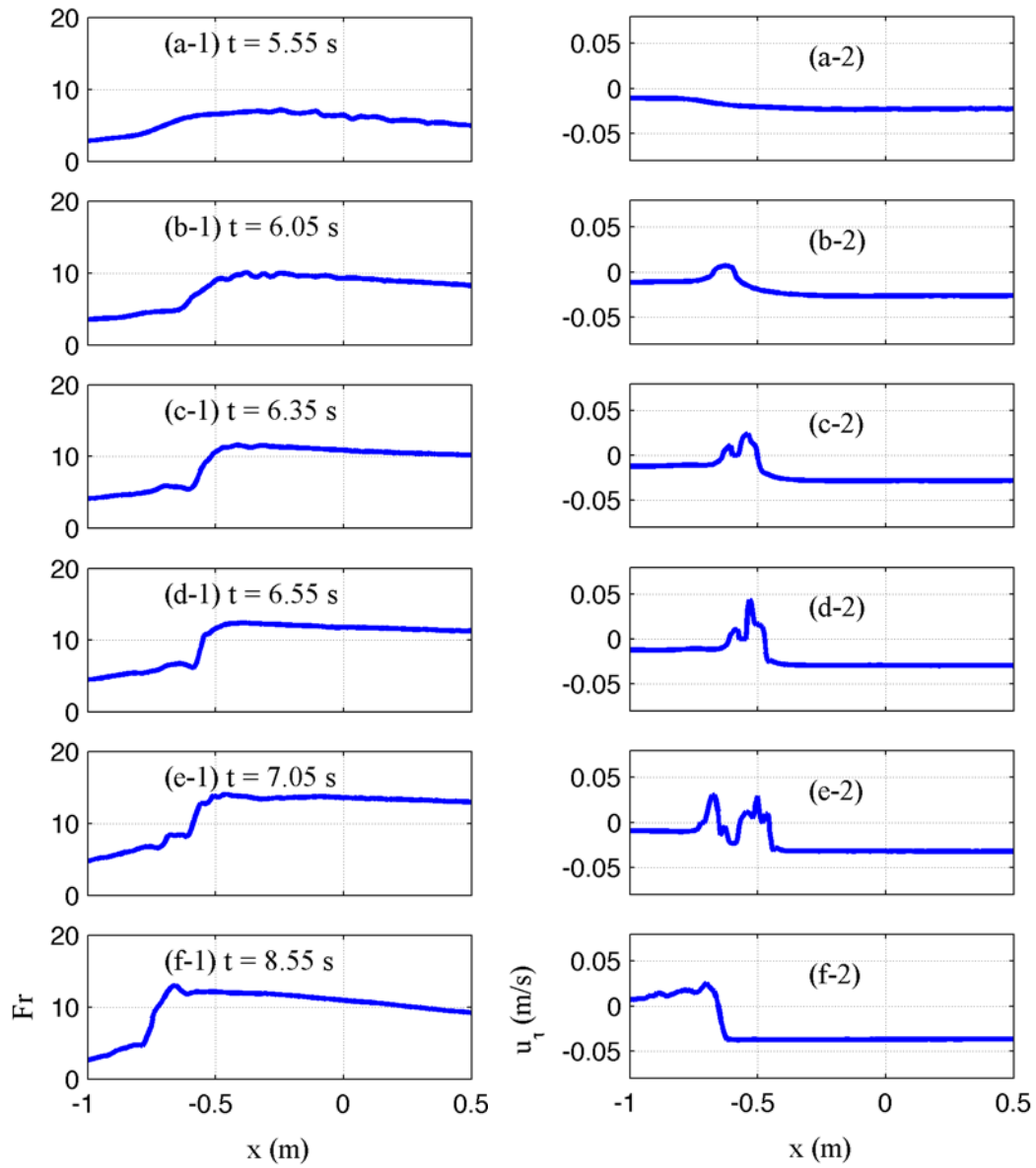


Figure 3.27: Froude number (first column) and friction velocity (second column) under the hydraulic jumps. (a) $t = 5.55$ s; (b) $t = 6.05$ s; (c) $t = 6.35$ s; (d) $t = 6.55$ s; (e) $t = 7.05$ s; (f) $t = 8.55$ s.

3.4.4 Fluctuation Component of BSS and turbulent intensity

The spanwise-averaged mean and fluctuating components of the bed friction velocity are plotted in the bird-eye view at different instants (see Figure 3.28~3.32). Qualitatively, the fluctuating component of bed friction velocity also represents the turbulent intensity very close to the bed. While large-magnitude of mean bed friction velocity is expected to be associated with the passage of the wave front, intense fluctuating bottom stress is also expected to be mainly due to surface generated turbulence and flow separations.

At $t=2.65$ s (see Figure 3.28), the steep wave front causes intense mean bed friction velocity exceeding 0.05 m/s at around 20 cm seaward of the initial shoreline. At this instant, the fluctuating component of friction velocity is rather weak (magnitude much below 0.01 cm/s). Only 0.1 sec later, the fluctuating component of friction velocity becomes intense (peak value exceed 0.02 m/s) after the second impingements hit the dry part of the slope at 2.75 s (see Figure 3.29), while the mean component of friction velocity also increase slightly to 0.07 m/s. Toward the end of run-up, the fluctuating component of bed friction velocity weakens with the maximum value generally below 0.02 m/s (see Figure 3.30). We also observe strip features in the streamwise direction which is associated with the shape of turbulent coherent structures.

During the run-down, the increase in the fluctuating component of bed friction velocity on the slope is evident in comparison between $t = 6.85$ s and 7.75

s (see and Figure 3.31 and Figure 3.32). It is also evident that at around $x=-0.5$ m, intense fluctuating component of bed friction velocity, well exceeding 0.02 m/s, is observed. This location is associated with the occurrence of hydraulic jumps and the corresponding boundary layer separation discussed previously. It is also remarkable that the mean component of bed friction velocity in this region is quite low (no more than 0.02 m/s). Hence, considering the ratio of fluctuating contribution to the mean contribution, this ratio is about 1 under the flow separation and hydraulic jump generated during the run-down process. On the other hand, during the impingement of the plunging breaker at the beginning of the run-up, mean component of the bed friction velocity is about 3 times larger than the fluctuating component.

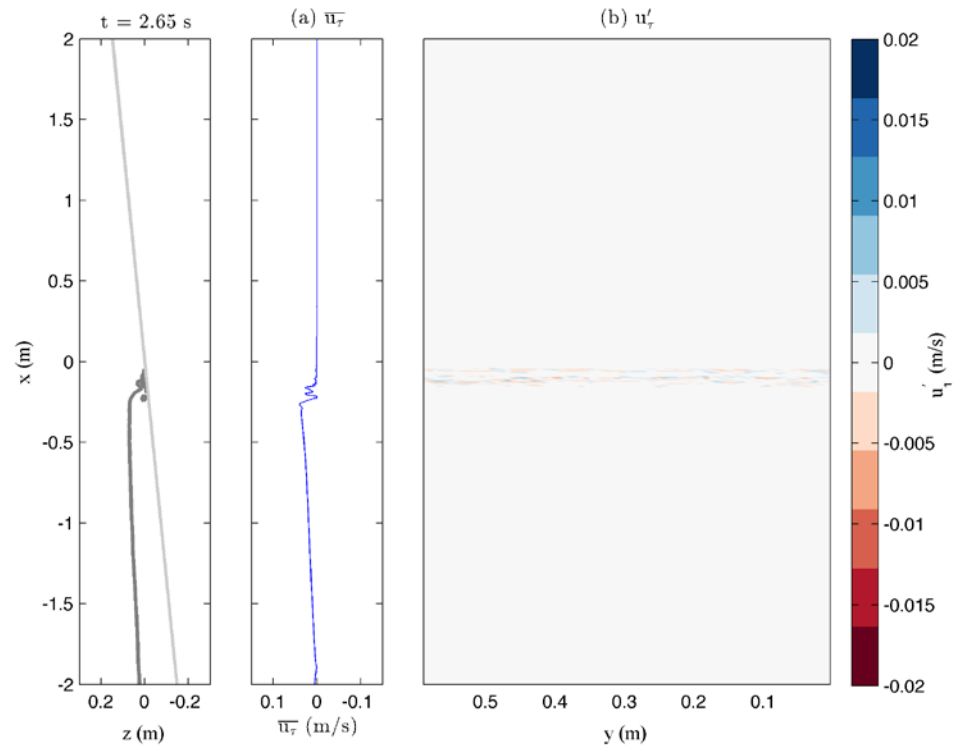


Figure 3.28: BSS plot at $t = 2.65$ s. (a) span-wise averaged value; (b) fluctuating components

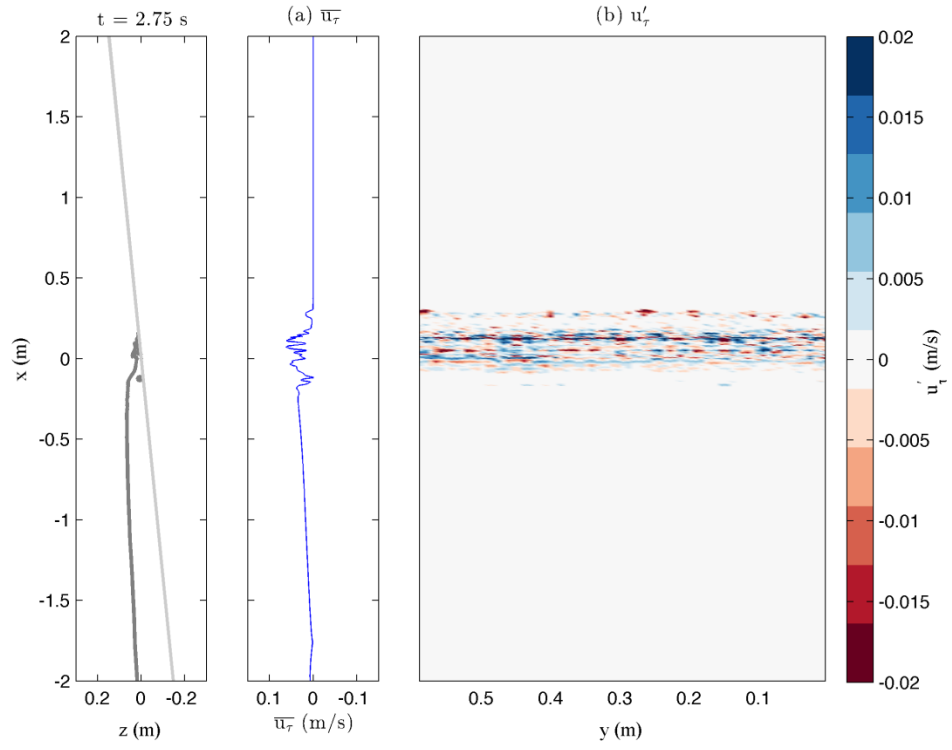


Figure 3.29: Same as Figure 3.28 except at $t = 2.75$ s.

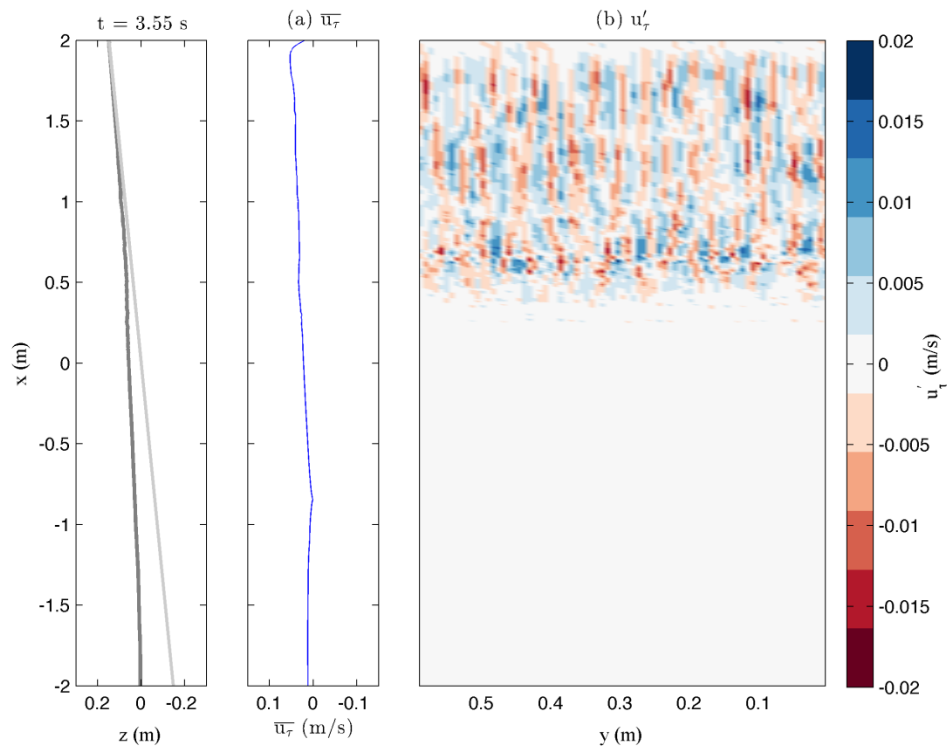


Figure 3.30: Same as Figure 3.29 except at $t = 3.55$ s

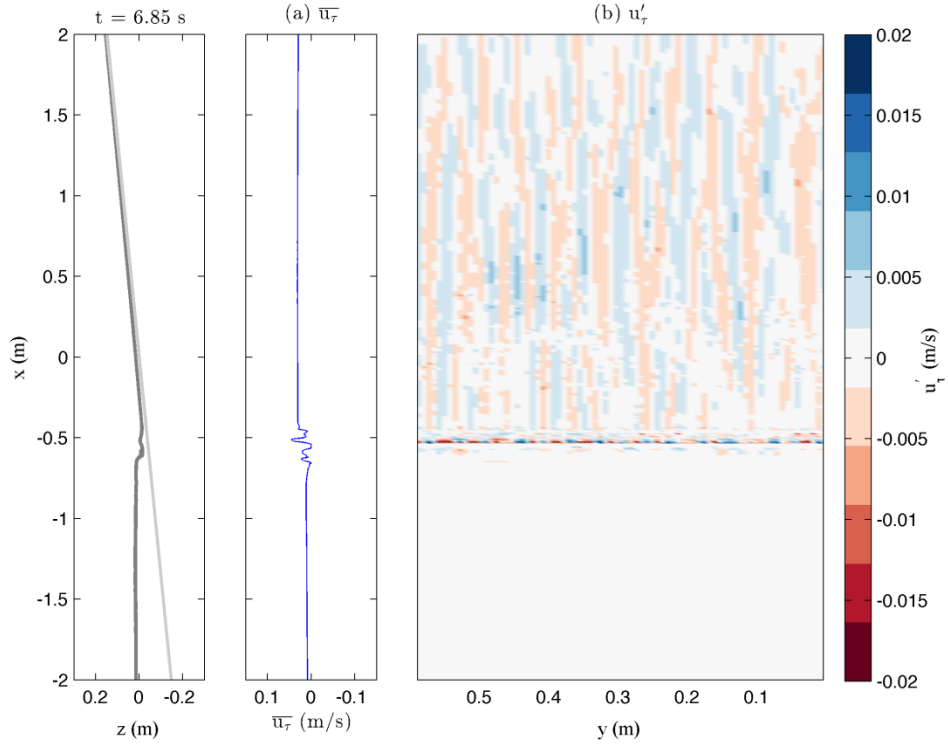


Figure 3.31: Same as Figure 3.30 except at $t = 6.85 \text{ s}$

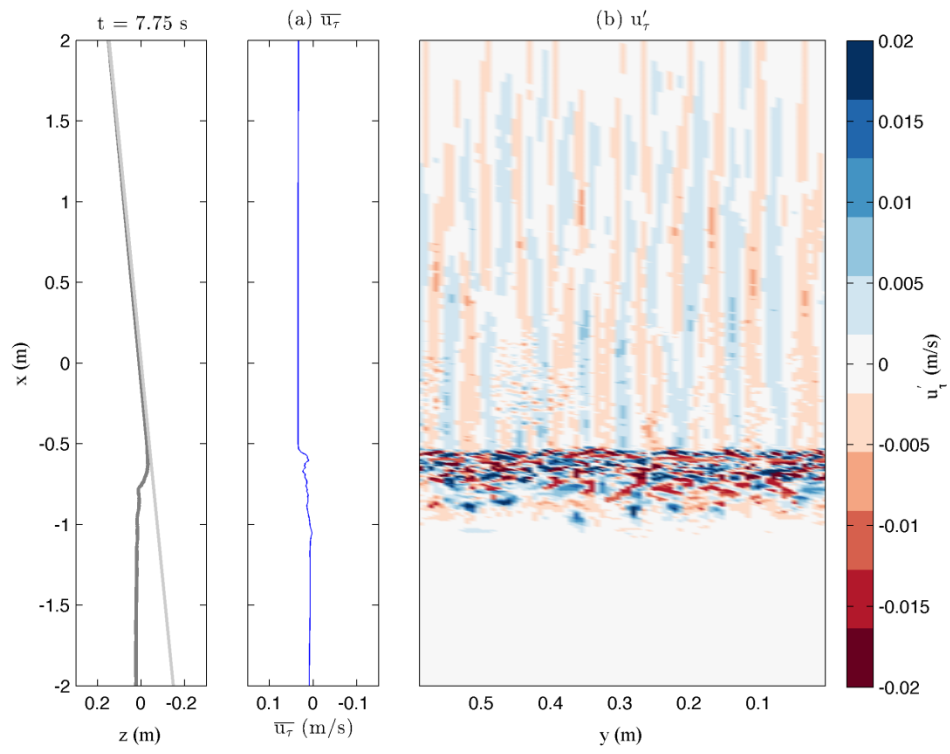


Figure 3.32: Same as Figure 3.31 except at $t = 7.75$ s

Chapter 4

DISCUSSION

4.1 Numerical Simulation of Experiment of *Synolakis*, [1986]

4.1.1 Numerical Model Set-up

This numerical simulation is performed in order to examine the boundary separation under the hydraulic jump with a different beach slope and wave condition. The numerical wave tank is modified here from the one shown in Chapter 3 (see Figure 3.1) based on the dimensions reported in *Synolakis* [1986]. The total cross-shore length is of 9.94 m, the total tank height is of 0.5 m and the width is of 0.39 m. More specifically, the cross-shore length of Section 1 (between the wave maker and toe) is 2 m, and those of both Section 2 (between the toe and the initial shoreline) and Section 3 (between the initial shoreline and the rear end) are 3.97 m. The initial water depth in the flat bottom region (Section 1) is 0.2 m. The beach slope is 1/19.85. The wave height of the solitary wave is set to be 0.056 m.

A total number of 1,000,000 grids are used for the simulation, which includes 400 in the x direction; 50 in both y and z directions. Each section is provided with different number of cross-shore grids. Section 1 is given with 50 grid points, Section 2 is of 200 grid points and Section 3 is of 150 grid points. In all

sections, the vertical grids are non-uniformly distributed by 1:20 ratio to allow dense grids in the bottom, but the span-wise grids are uniformly distributed. Both Section 2 and Section 3 have non-uniform cross-shore grid distributions with 1:10 ratio converging around the initial shoreline.

4.1.2 Numerical Model Validation

In this simulation, only the snapshots of surface elevation and the time series of the wave gauge data at one location ($x = -3.19$ m) are available for comparison. The wave gauge data in the numerical model results and in the experimental data matches very well (Figure 4.1). Similar to that discussed in the numerical simulation of *Sumer et al*, [2011], the numerical wave maker add water mass into the domain and a slight increase in the mean water level is observed at later time. .

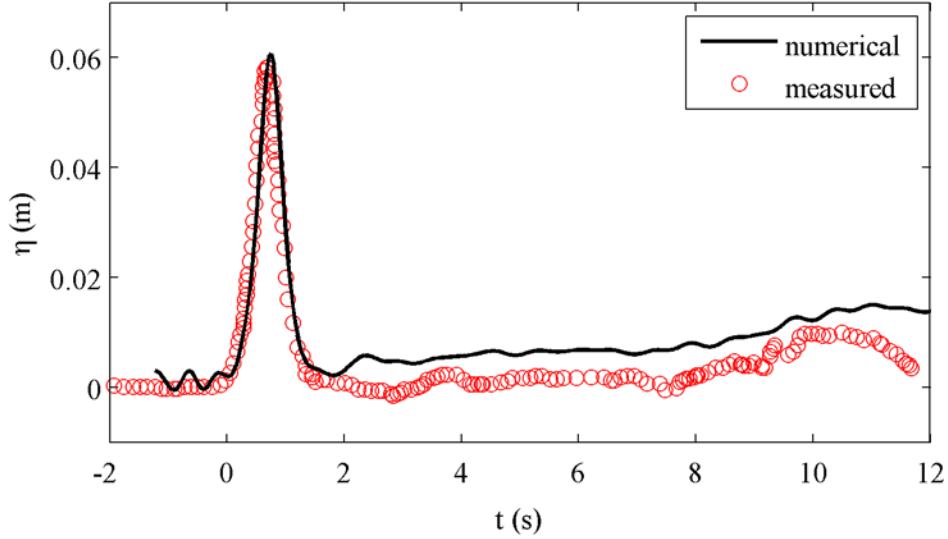


Figure 4.1: The comparison between the experimental wave gauge data at $x = -3.82$ m (red circles) and the numerical data at -3.84 m (black lines)

The simulated surface elevation (Figure 4.2 and 4.3) also shows a good agreement with the experimental data. The simulation predicts well the surface elevation in both the progressing stage and the breaking stage (Figure 4.2). But, slight discrepancy appears in the run-down stage (See Figure 4.3). The predicted run-down flow appears slower than that in the experiment. The simulation also overestimates the maximum run-up distance. In addition, the hydraulic jump in the simulation takes place about 0.2 m closer to the initial shoreline. These discrepancies may be due to the relative coarse numerical resolution used in the present simulation which is coarser than that presented in Chapter 3.

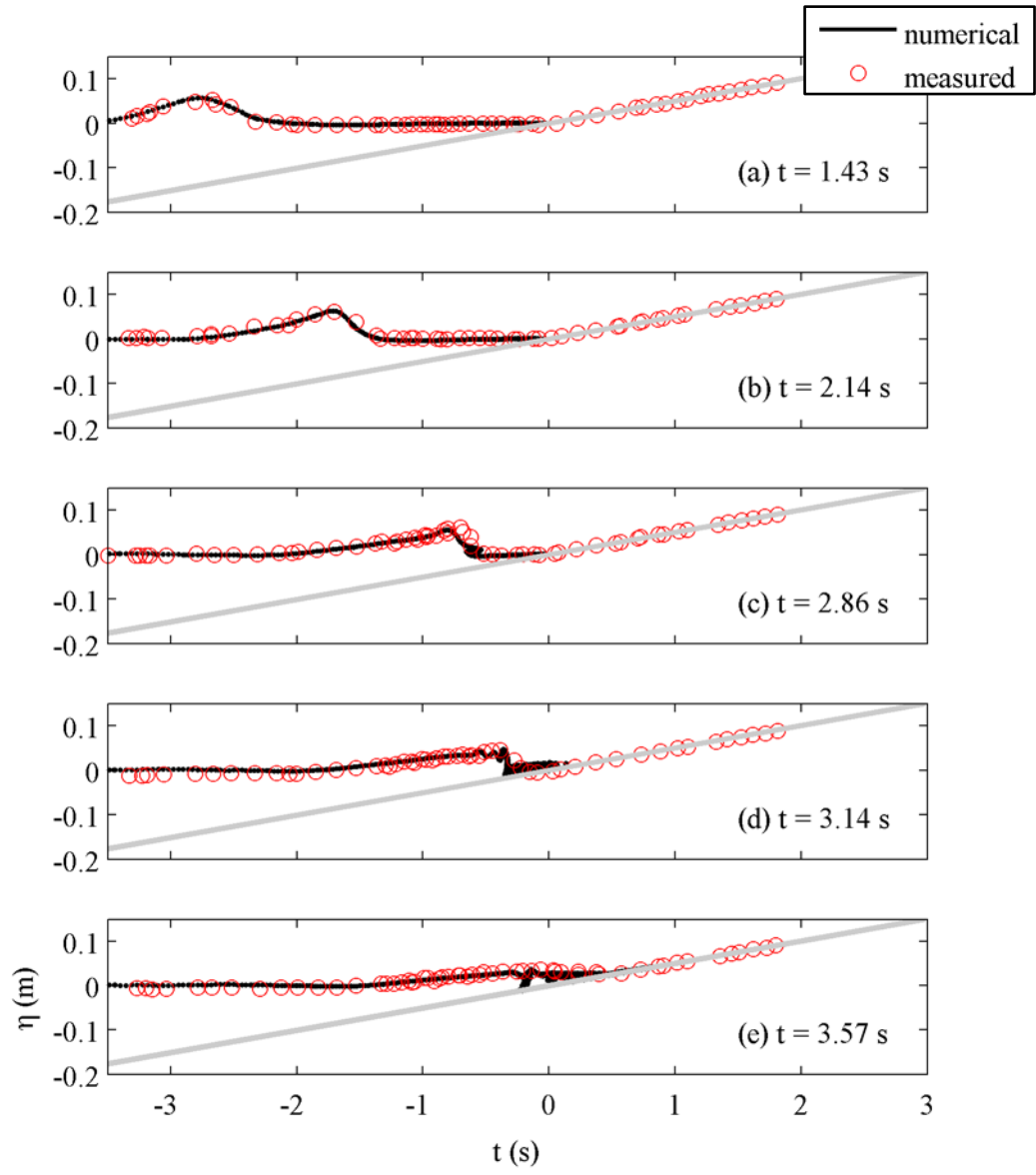


Figure 4.2: The surface elevation comparison during the run-up and wave breaking phase at (a) $t = 1.43$ s; (b) $t = 2.14$ s; (c) $t = 2.86$ s; (d) $t = 3.14$ s; (e) $t = 3.57$ s.

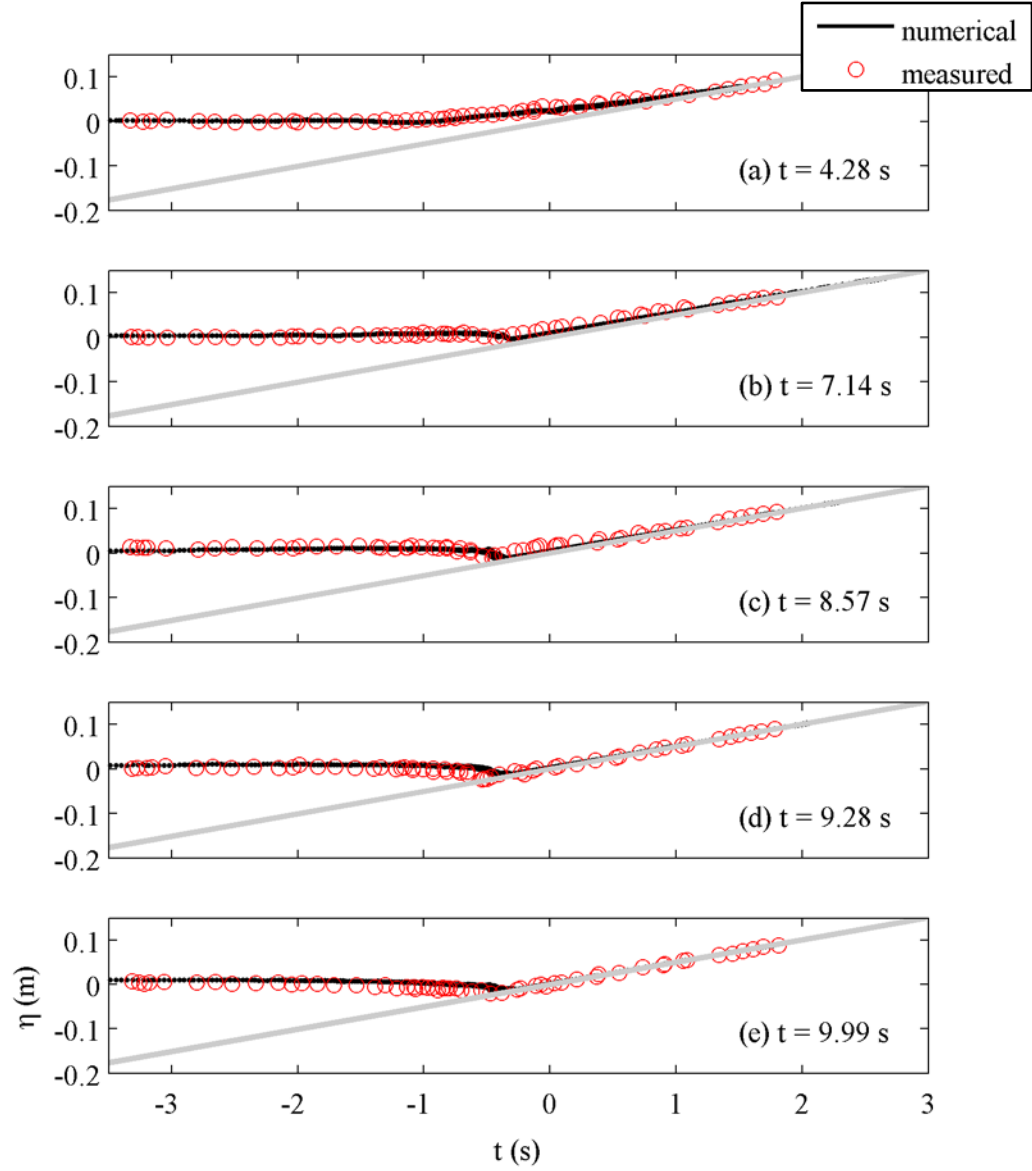


Figure 4.3: Same as Figure 4.2 except during the run-down phase and hydraulic jump at (a) $t = 4.28$ s; (b) $t = 7.14$ s; (c) $t = 8.57$ s; (d) $t = 9.28$ s; (e) $t = 9.99$ s.

4.1.3 Hydraulic Jump and Boundary Separation in *Synolakis* [1986]

Despite of the different wave height, beach slope, and grid resolution, the numerical simulation of *Synolakis*, [1986] shows the boundary separation under the hydraulic jump as does that of *Sumer et al*, [2011]. Figure 4.4 and Figure 4.5 shows the vector plot of the velocity between -0.6 m and 0 m throughout the evolution of the hydraulic jump. In Figure 4.4(c), an inflection point appears at the bottom of the velocity profile and ascends at -0.4 m. In Figure 4.4(e), a second inflection point appears in the velocity profile at -0.48 m. As the upstream jet weakens and gets shallower, the first inflection point migrates downstream, and the hydraulic jump resettles near the location of the second inflection point (Figure 4.5(c) and (d)).

Figure 4.6 shows the generation of turbulence intensity under the hydraulic jump. The turbulence intensity is first observed in Figure 4.6 (b) from both surface and bottom. This instant coincides with Figure 4.4(c), in which the boundary separation is observed. In Figure 4.6(c), while the turbulence further intensifies, another turbulent spot evolves slightly downstream from the first one. In Figure 4.6(d), the turbulence intensity spots from different origins merge together.

In summary, the generation of hydraulic jump and flow separation during the drawdown stage of plunging solitary wave over a slope may be quite common. Here, we show that flow separation under hydraulic jump can occur in two different experiments of plunging solitary wave over a slope ranges between $1/19.85$

(*Synolakis*, [1986]) and $1/14$ (*Sumer et al.*, [2011]). Hence, the implication of the vortical flow field and elevated turbulence level during the drawdown stage on the mixing and transport of sediment warrants more complete future work.

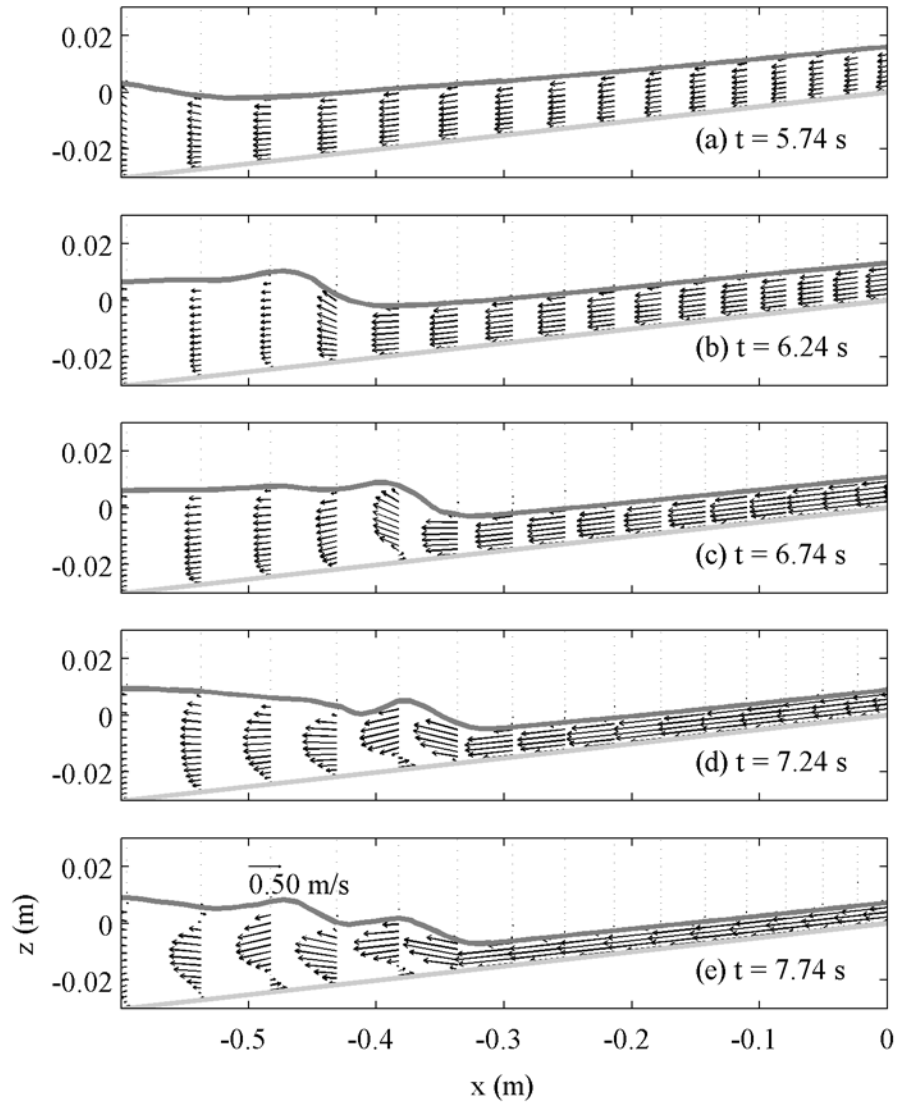


Figure 4.4: The velocity profiles during the transition between the run-up and run-down phases. (a) $t = 5.74$ s; (b) $t = 6.24$ s; (c) $t = 6.74$ s; (d) $t = 7.24$ s; (e) $t = 7.74$ s.

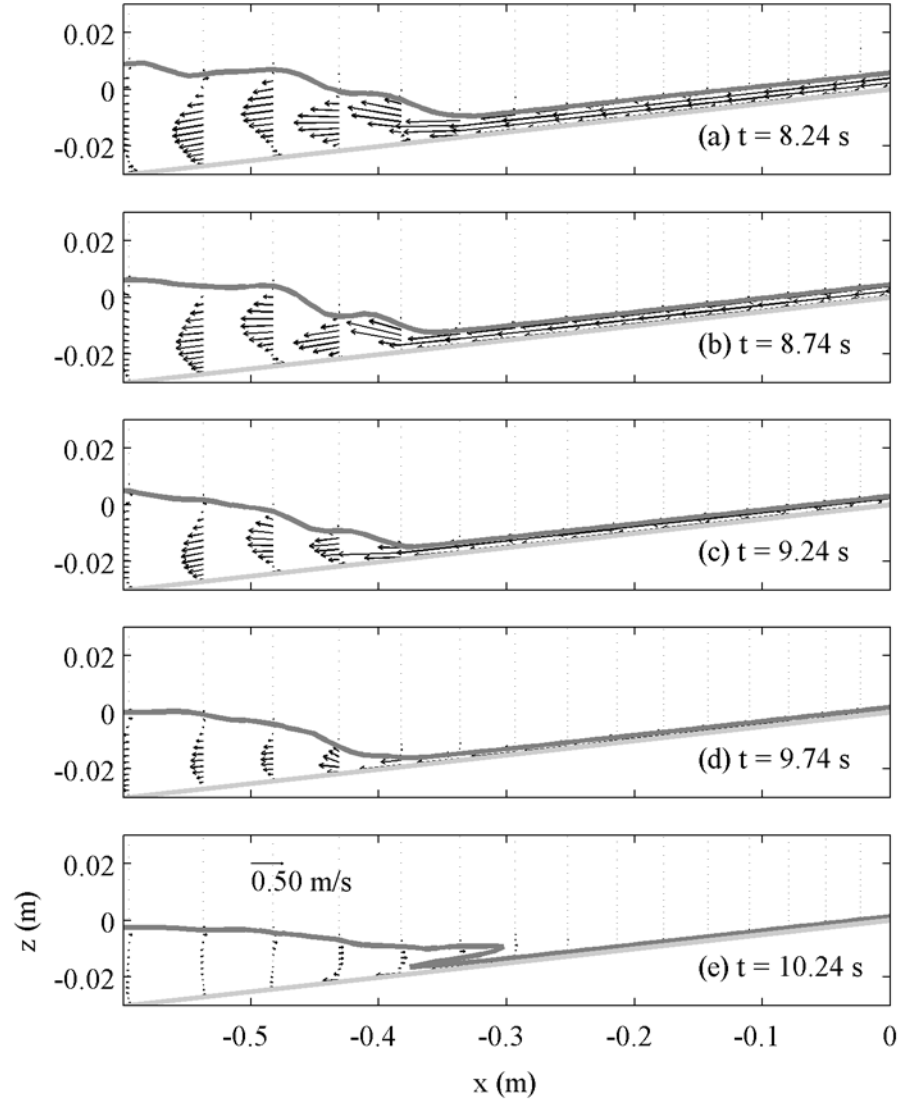


Figure 4.5: Same as Figure 4.4 except at (a) $t = 8.24$ s; (b) $t = 8.74$ s; (c) $t = 9.24$ s; (d) $t = 9.74$ s; (e) $t = 10.24$ s.

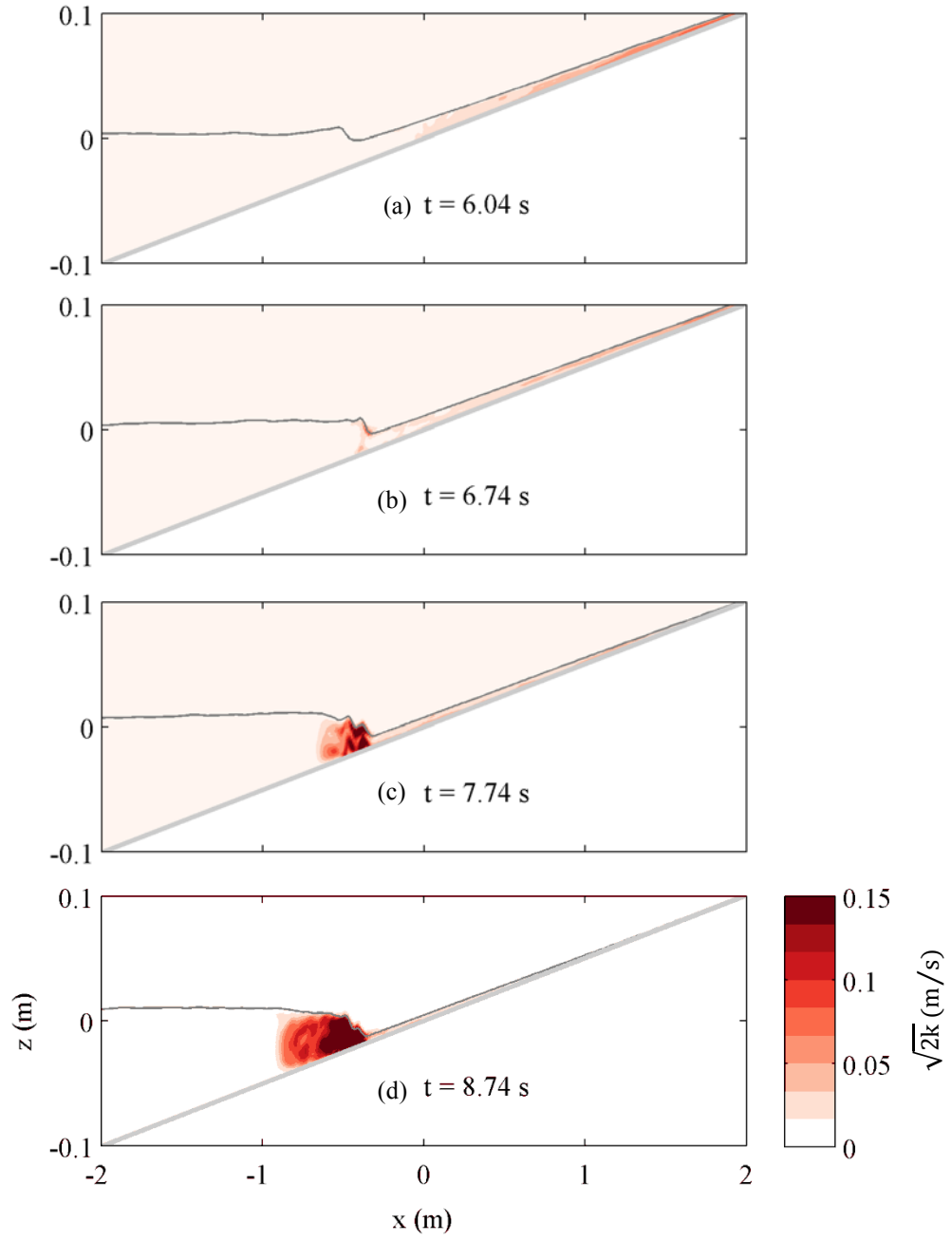


Figure 4.6: The contour snapshots of the span-wise averaged turbulence intensity at (a) $t = 6.04$ s; (b) $t = 6.74$ s; (c) $t = 7.74$ s; (d) $t = 8.74$ s

4.2 Bottom Flow Acceleration in *Sumer et al.*, [2011]

Conventional concept of sediment transport is described by bed friction velocity (non-dimensionalized as the Shield parameter). Hence, the mean and fluctuating components of bed friction velocity are presented in Chapter 3.11. The bed-shear-stress based sediment transport concept is appropriate when flow acceleration is weak. *Sleath*, [1999] measures the occurrence of plug flow in a laboratory U-tube using light particles. Analytical solution of simple model shows that the occurrence of plug flow is associated with flow acceleration.

A nondimensional parameter, later called the Sleath parameter is defined as

$$S = \frac{\partial u_b / \partial t}{(s - 1)g} \quad (4.2.1)$$

where $\frac{\partial u_b}{\partial t}$ is the local acceleration of the bottom (near-bed) flow, s is the specific gravity of sediment ($s=2.65$ for sand) and g is the gravitational acceleration. Based on his experimental data using artificial light particles, *Sleath*, [1999] reports the threshold value for the occurrence of plug flow is around $S=0.3$. Later, *Foster et al.*, [2006] observe field evidence of plug flow for realistic sand bed and they report the threshold value to be as low as 0.1. Physically, the threshold value for the occurrence of plug flow should depends on the packing and also the Shields parameter. While more study is required to study the occurrence of plug flow, we will investigate the magnitude and temporal evolution of the near-bed local acceleration field of the present simulation. By taking the threshold value to

be $S=0.3$ and consider typical sand grain, the threshold acceleration for the occurrence of plug flow is around 5 m/s^2 .

The bottom flow (local) acceleration $\frac{\partial u_b}{\partial t}$ is estimated by the following equations

$$\frac{\partial u_b}{\partial t} \approx \frac{\Delta u_b}{\Delta t} = \frac{u_{i+1} - u_i}{t_{i+1} - t_i} \quad (4.8)$$

in which the index, i stands for the time step for the instantaneous velocity fields. The local acceleration is calculated based on the output of instantaneous velocity field at every 0.1 sec. More detailed snapshots of acceleration field will be discussed next. However, to provide a sense on the magnitude and timing of high acceleration, peak value acceleration at (in magnitude) a given instant is plotted as a function of time throughout the entire simulation (see Figure 4.7). During run-up, the acceleration peaks to 30 m/s^2 around $t = 2.7\text{s}$ in which the wave impinges. Then the acceleration decreases, stays below 5 m/s^2 during the run-up and run-down phases (between $t = 4\text{s}$ and 6.5s). After $t = 6.5 \text{ sec}$, the peak acceleration remains above 5 m/s^2 and eventually increases again to 10 m/s^2 past $t = 8.5 \text{ s}$. This later uprising of peak acceleration during the run-down process is clearly associated with flow separation and hydraulic jump.

Figure 4.8(a)~(e) present the top view of the bottom flow acceleration. At the first impingement of the plunging wave (see Figure 4.8(a)), large mean flow acceleration of 10 m/s^2 is generated at the plunging point just seaward of the initial shoreline. Notice that at this instant, the fluctuating component of acceleration is nearly zero because strong turbulence is not yet generated in the simulation (see Figure 3.11 and Figure 3.12) until $t=2.65 \text{ s}$. However, the acceleration appears to peaks at $t=2.75 \text{ s}$ (see Figure 4.8(b)) when the wave collapses and impinges around the initial shoreline. The mean value exceeds 20 m/s^2 just landward of the initial shoreline while the peak fluctuating component of flow acceleration also approaches 10 m/s^2 . Afterward, the flow acceleration decreases as the run-up pro-

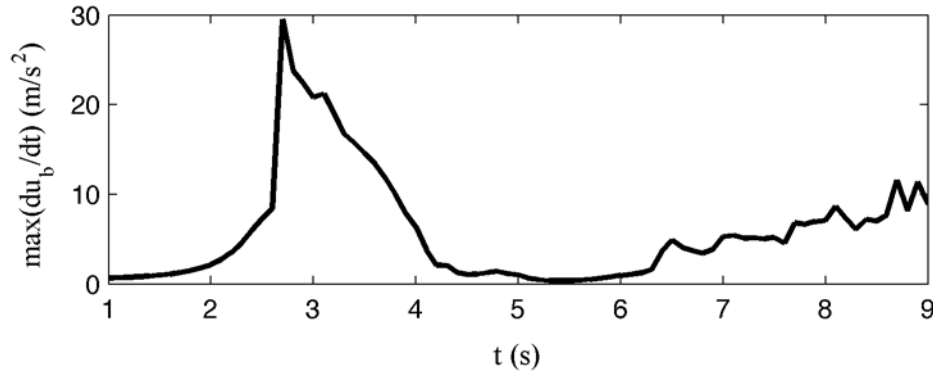


Figure 4.7. The maximum bottom flow acceleration $\frac{\partial u_b}{\partial t}$ over the entire tank versus time.

ceeds. Even toward the later stage of the run-up, fairly strong acceleration (around 15 m/s^2) is observed at the front (see Figure 4.8(c)) before it falls below 5 m/s^2 at about $t=4.0 \text{ s}$ and stays around 0 m/s^2 during the transition between the run-up and run-down phases (between $t=4.2 \text{ s}$ to 6.2 sec). Hence, during the early stage of the run-down, the flow acceleration is quite small. However, at $t=6.5 \text{ sec}$ (see Figure 4.8(d)), we start to observe notable mean acceleration approaching 5 m/s^2 at the location coincide with the hydraulic jump. At $t=7.1 \text{ sec}$ (Figure 4.8(e)), the mean acceleration decays, however the fluctuating component starts to become dominant with the total value clearly exceeds 5 m/s^2 . Between $x=-0.7 \text{ m}$ and -0.5 m , large fluctuating component of acceleration is persistent and in fact approaching 10 m/s^2 at $t=8.0 \text{ sec}$ (see Figure 4.8(f)). This large fluctuating component of flow acceleration is associated with flow turbulence generated via flow separation under the hydraulic jump. In summary, flow large acceleration is observed under a plunging solitary wave and according to *Sleath*, [1999], plug flow is very likely to occur even at this laboratory scale study. The largest flow acceleration occurs during the impingement of plunging breaker near the initial still water shoreline. However, during the run-down stage, notable flow acceleration can also be expected due to flow separation and hydraulic jump.

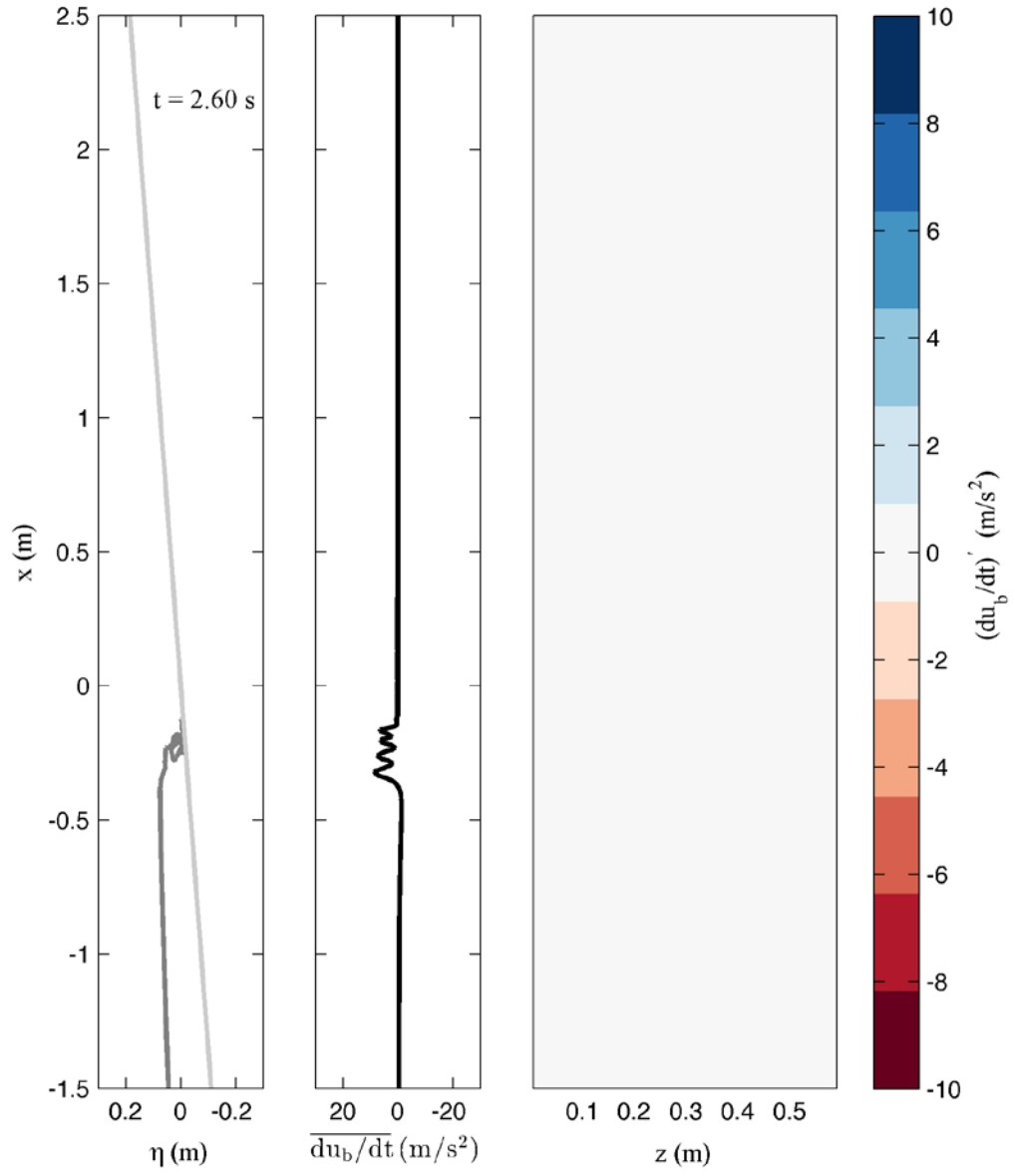


Figure 4.8(a): The surface elevation (left), the span-wise averaged bottom flow acceleration (middle), and the top view of the contour of bottom flow acceleration at $t = 2.60$ s

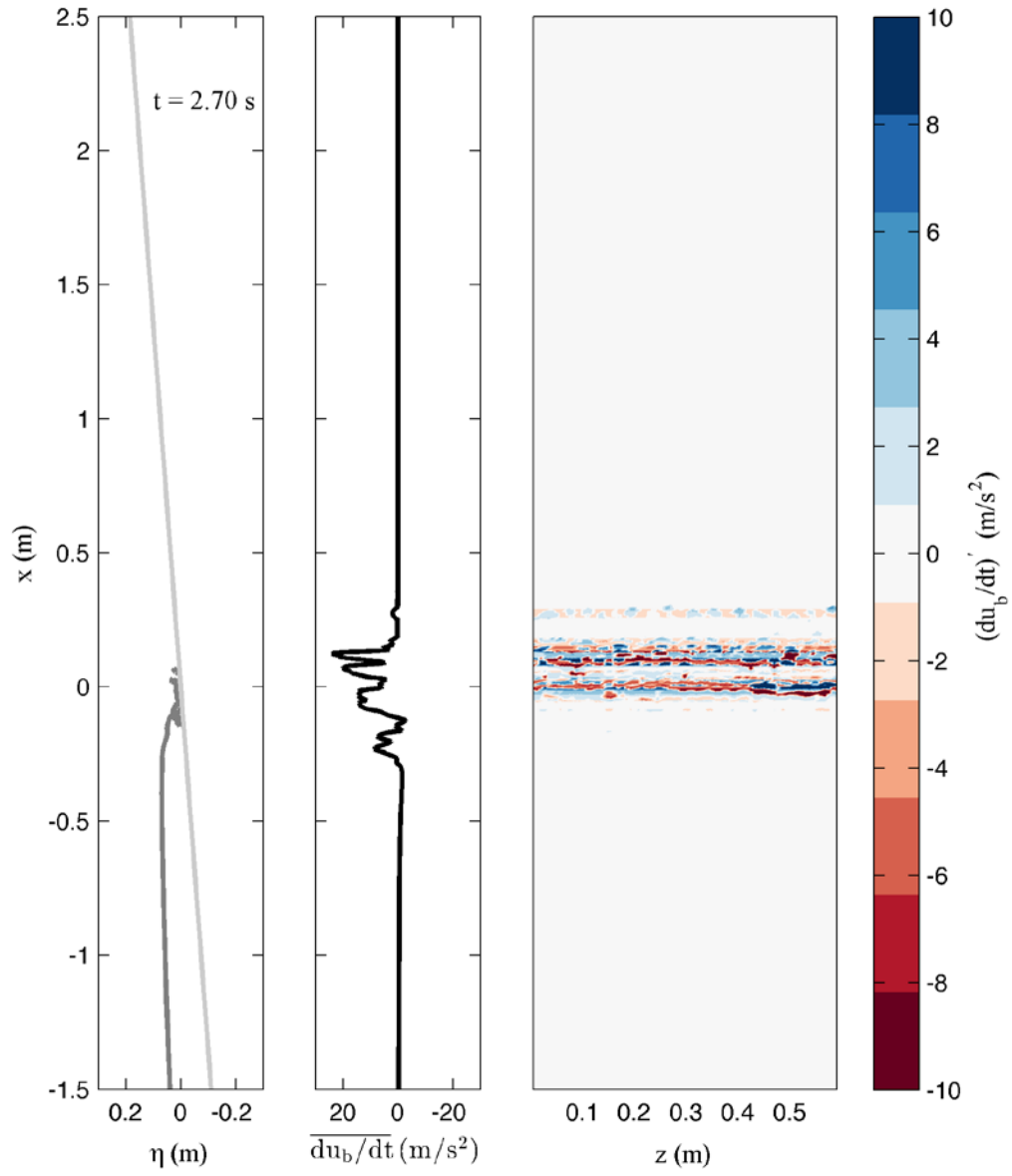


Figure 4.8(b): Same as Figure 4.8(a) except at $t = 2.70$ s

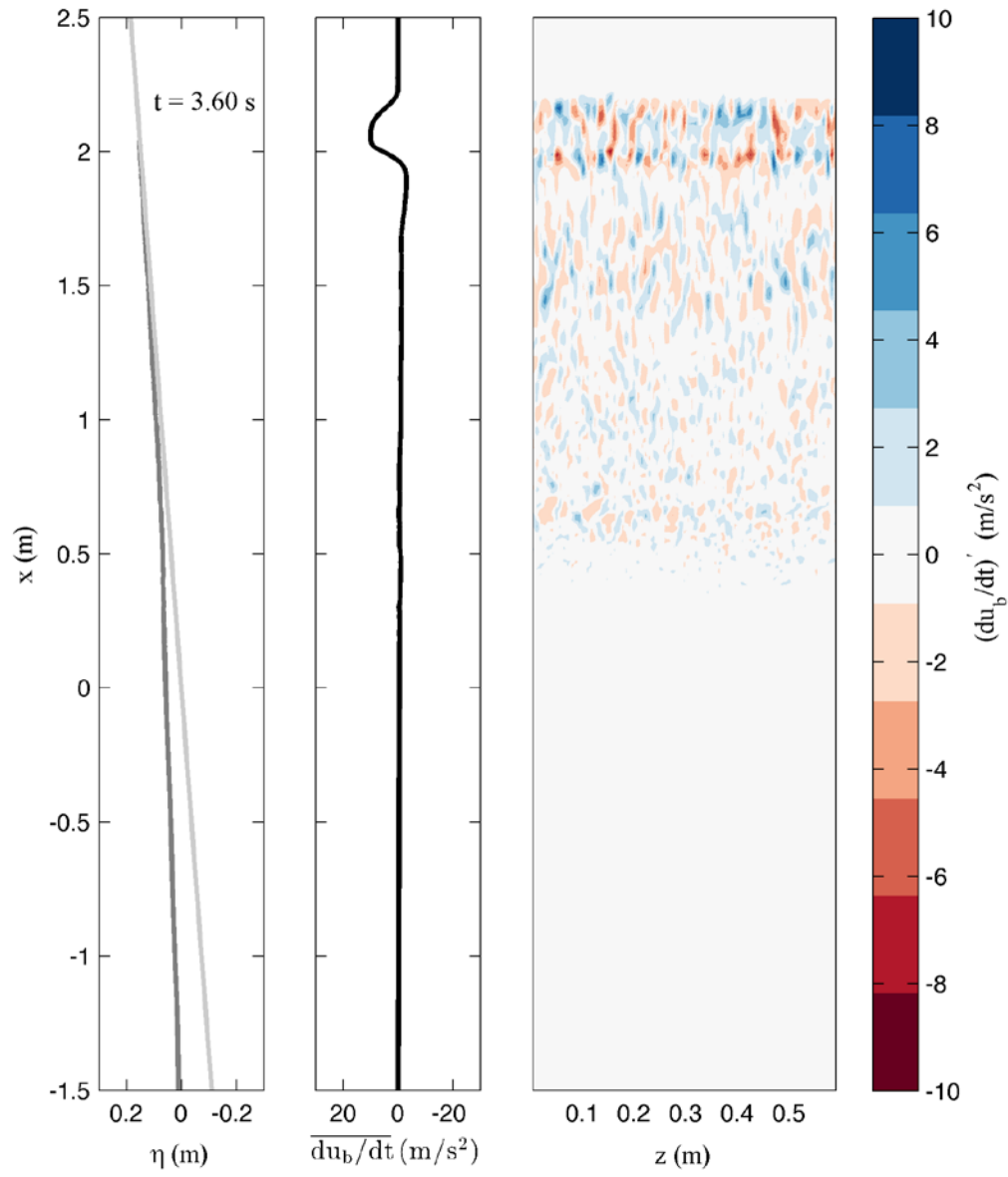


Figure 4.8(c): Same as Figure 4.8(b) except at $t = 3.60$ s

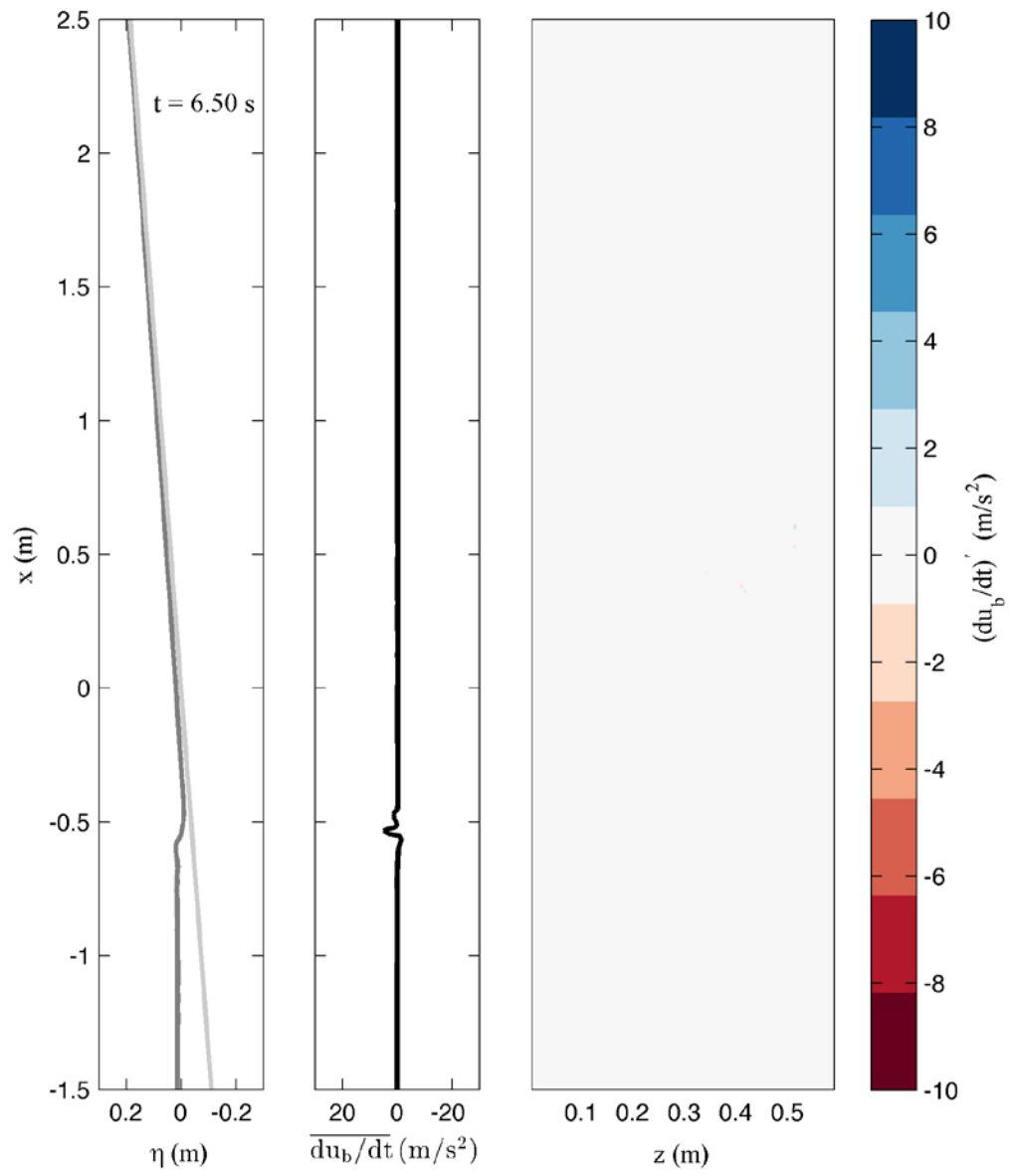


Figure 4.8(d): Same as Figure 4.8(c) except at $t = 6.50$ s

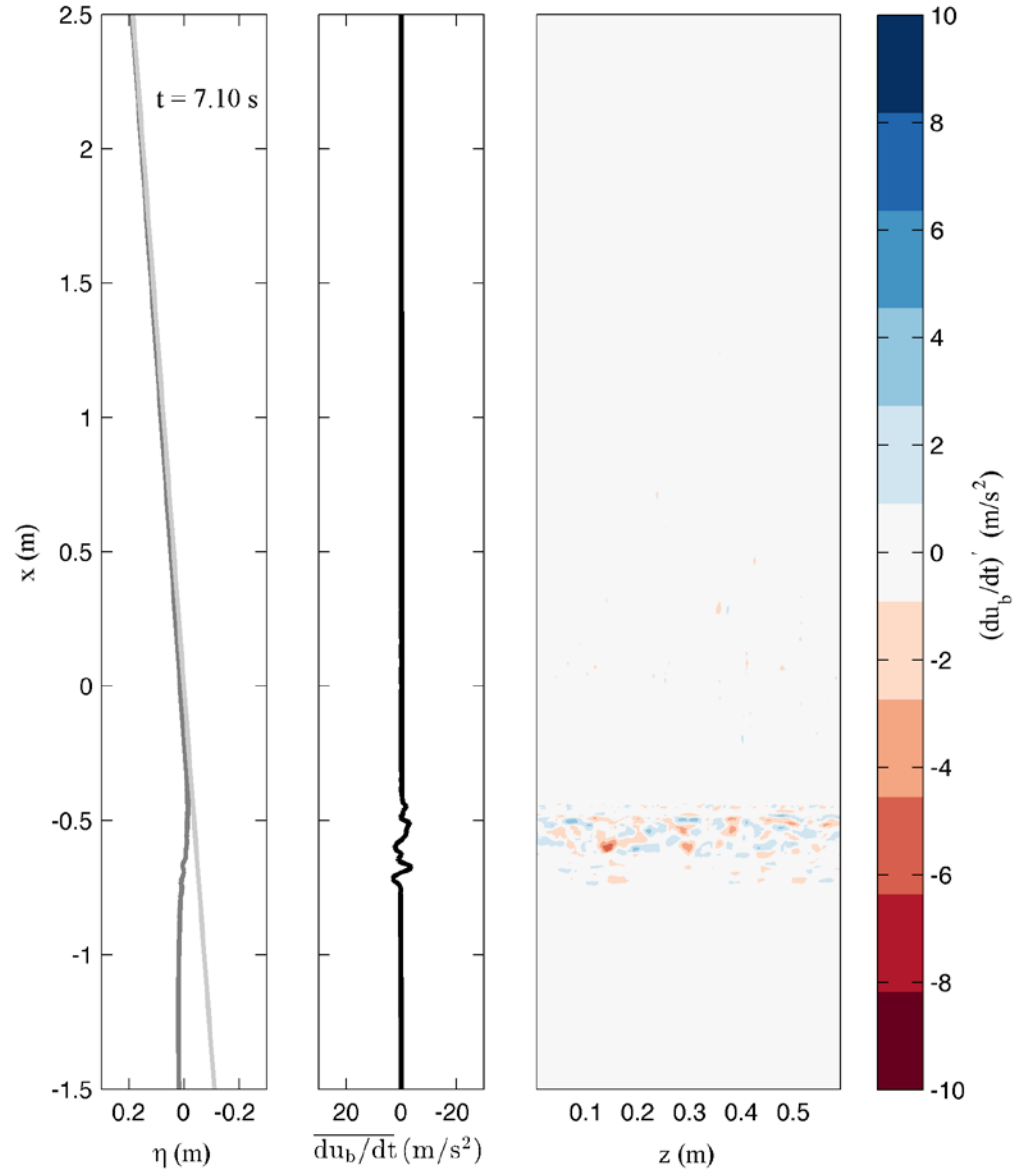


Figure 4.8(e): Same as Figure 4.8(d) except at $t = 7.10$ s

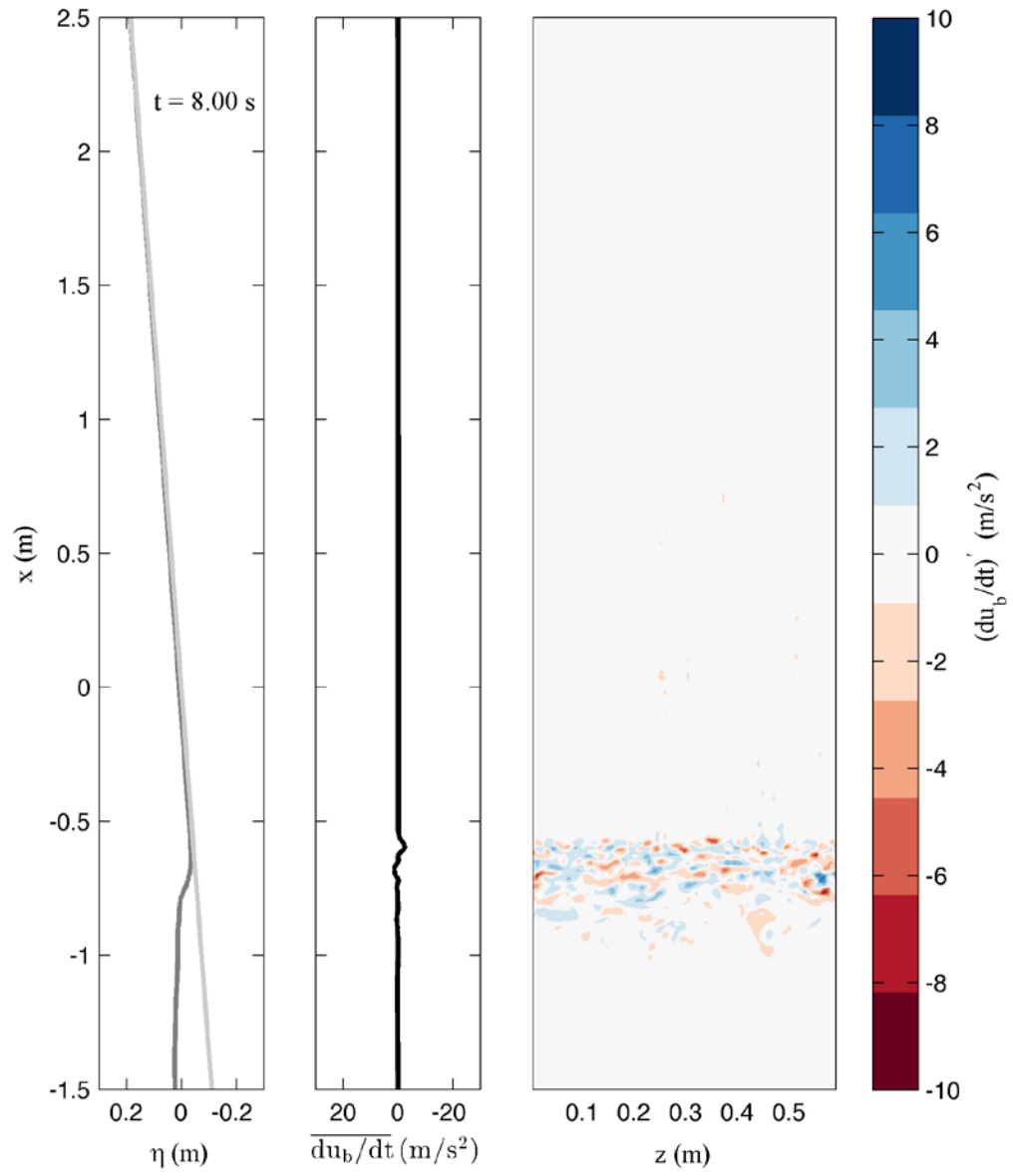


Figure 4.8(f): Same as Figure 4.8(e) except at $t = 8.00$ s

Chapter 5

CONCLUSION

The hydrodynamic processes beneath plunging solitary waves are investigated via 3-D large-eddy simulation (LES) based on the experiments of *Sumer et al.*, [2011] and *Synolakis*, [1986]. Most of the analysis and discussions are based on the simulation of *Sumer et al.*, [2011]. The simulation of *Synolakis*, [1986] is performed to confirm the occurrence of hydraulic jump and boundary separation in other plunging solitary wave condition with a milder slope.

During run-up, wave breaking turbulence is generated when the plunging breaker impinges onto the bed. The λ -2 iso-surfaces confirm that the plunging breaker produced several 2D rollers. Through nonlinear process, these rollers can further evolve in 3D structures and degenerated into smaller eddies. During the drawdown stage, the boundary layer processes come as dominant processes. Some short-lived inflection points in the velocity profiles were observed in the landward flow. Further into the rundown process, a moving hydraulic jump is observed slightly offshore from the initial shoreline. Beneath the jumps, the separation of boundary layer is observed. This separation coincides with the 2-D coherent struc-

ture, the location of strong adverse pressure gradient, reversal of bottom shear stress, and intensive turbulence generation.

The bottom flow acceleration shows that the impinging wave breaker causes the greatest bottom acceleration as high as 30 m/s^2 at the initial shoreline. The hydraulic jump and flow acceleration during the drawdown stage also cause significant bed acceleration as large as 10 m/s^2 . The high flow acceleration may trigger the occurrence of plug flows, and could contribute to transporting sediment during the rundown process.

The presence of the boundary separation and of the considerably high bed flow acceleration beneath the hydraulic jump provides some insights for the future study. The numerical investigations of both *Sumer, et al.*, [2011] and *Synolakis*, [1986] indicate that the hydraulic jump induces strong turbulence, bed shear stress, and bed flow acceleration. The future study should investigate whether the occurrence of a hydraulic jump is a common phenomenon of every plunging solitary wave and analyze how varied beach slopes, wave conditions, and initial still water levels affect the evolution of the hydraulic jump. In addition, the future numerical simulation should include a layer of sediment on the bottom of the wave tank in order to study the sediment transport process associated with the inundation and retreat of a solitary wave.

REFERENCES

- Atwater, B. A., (1987). Evidence for great Holocene earthquakes along the outer coast of Washington State. *Science*, 236, pp 942-944
- Baldock, T. E., Cox, D., Maddux, T., Killian, J., Fayler, L., (2009). Kinematics of breaking tsunami wavefronts: A data set from large scale laboratory experiments. *Coastal Engineering*, 56, pp 506-516
- Baldock, T. E., Peiris, D., Hogg, J., (2011). Overtopping of solitary waves and solitary bores on a plane beach. *Proceedings of the Royal Society A*,
- Chow, V. T., (1973). Open channel hydraulics. New York, USA: McGraw-Hill International
- Danielle, K., Crowley, K., Frye, A., (2011). Reinsurers decline as Japan Quake. Tsunami may cause \$10 billion in claims. Bloomberg. Retrieved from <http://www.bloomberg.com/news/2011-03-11/european-reinsurers-fall-leading-u-s-carriers-lower-on-quake.html>
- Dawson, A. G., (1994). Geomorphological effects of tsunami run-up and backwash. *Geomorphology*. 10, pp 83-94
- Dawson, A. G., Stewart, I., (2007). Tsunami deposits in the geological record. *Sedimentary Geology*. 200, pp 166-183
- Foster, D. L., Bowen, A. J., Holman, R. A., Natoo, P., (2006). Field evidence of pressure gradient induced incipient motion. *Journal of Geophysical Research*, 111, C05004
- Grilli, S. T., Svendsen, A., (1997). Breaking criterion and characteristics for solitary waves on slopes. *Journal of Waterway, Port, Coastal, and Ocean Engineering*. 123(3), pp 102-112
- Gschaider, B., (2009). Contrib groovyBC. *OpenFOAM Wiki*. January, 2012, http://openfoamwiki.net/index.php/Contrib_groovyBC

- Hall, J. V., Watts, J. W., (1953). Laboratory investigation of the vertical rise of solitary waves on impermeable slopes. Technical Memorandum 33. US Army Corps of Engineers, Beach Erosion Board
- Hsiao, S.-C., Hsu, T.-W., Lin, T.-C., Chang, Y.-H., (2008) On the evolution and run-up of breaking solitary wave on a mild sloping beach. *Coastal Engineering*. 55, pp 975-988
- Jeong, J., Hussain, F. (1995). On the identification of a vortex. *Journal of Fluid Mechanics*. 285, pp 69-94
- Klostermann, J, Schaake, K., Schwarze, R., (2012). Numerical simulation of a single rising bubble by VOF with surface compression. *International Journal for Numerical Methods in Fluids*. 71(8), pp 960-982
- Lee, J.-J., Skjelbreia, J. E., Raichlen, F., (1982). Measurement of velocities in solitary waves. *Journal of the Waterway, Port, Coastal, and Ocean Division*. 108 (2), pp 200-218
- Li, Y., Raichlen, F., (2002). Non-breaking and breaking solitary wave run-up. *Journal of Fluid Mechanics*. 456, pp 295-318
- Lilly, D.K., (1992). A proposed modification of the Germano subgrid-scale closure method. *Physics of Fluids A*. 4, pp 633-634
- Lin, P., Chang, K.-A., Liu, P. L.-F., (1999). Runup and rundown of solitary waves on sloping beaches. *Journal of Waterway, Port, Coastal, and Ocean Engineering*. 125(5), pp 247-255
- Liu, P. L.-F., Park, Y. S., Cowen, E. A., (2007). Boundary layer flow and bed shear stress under a solitary wave. *Journal of Fluid Mechanics*. 574, pp 440-463
- Mori, N., Takahashi, T., The 2011 Tohoku Earthquake Tsunami Joint Survey Group, (2012). Nationwide post event survey and analysis of the 2011 Tohoku Earthquake Tsunami. *Coastal Engineering Journal*. 54(1) , pp 1-27
- OpenCFD Limited, (2011). OpenFOAM User Guide, Version 2.0.0. Technical Report
- Ohtsu, I., Yasuda, Y., (1991). Hydraulic jump in sloping channels. *Journal of Hydraulic Engineering*. 117(7), pp 905-921

- Park, Y.S., Liu, P.L.-F., Chen, I.-C., (2012). Contact line dynamics and boundary layer flow during reflection of a solitary wave. *Journal of Fluid Mechanics*. 707, pp 307-330
- Pope, S. B., (2000). Turbulent flows. Cambridge, UK: Cambridge University Press
- Ross, S. L., Boore, D. M., Fisher, M. A., Frankel, A. D., Geist, E. L., Hudnut, K. W., Kayen, R. E., Lee, H. J., Normark, W. R., Wong, F. L., (2004). Comments on potential geologic and seismic hazards affecting coastal Ventura County, California. Open-file Report 2004-1286
- Sangermano, J. J., (2013). A numerical study of wave-breaking turbulence beneath solitary waves using large eddy simulation. Master Thesis, University of Delaware
- Sleath, J. F. A., (1999). Conditions for plug formation in oscillatory flow, *Continental Shelf Research*. 19, pp 1643-1664
- Son, S., Lynett, P. J., Kim, D.-H., (2011). Nested and multi-physics modeling of tsunami evolution from generation to inundation. *Ocean Modelling*. 38, 96-113
- Sou, I. M., Yeh, H.H., (2011). Laboratory study of the cross-shore flow structure in the surf and swash zones. *Journal of Geophysical Research*. 116, C03002
- Stover, C. W., Coffman, J. L., (1993). Seismicity of the United States, 1568-1989. U.S. Geological Survey Professional Paper 1527. Washington, D.C.: United States Government Printing Office
- Sumer, C.B., Sen, M.B., Karagali, I., Ceren, B., Fredsoe, J., Sottile, M., Zilioli, L., Fuhrman, D. R., (2011). Flow and sediment transport induced by a plunging solitary wave. *Journal of Geophysical Research*. 116, C01008
- Synolakis, C.E., (1986). The runup of long waves, Ph.D. thesis, California Institute of Technology, Pasadena, California. pp 228
- Ting, F.C.K., (2006). Large-scale turbulence under a solitary wave. *Coastal Engineering*. 53, pp 441-462
- Ting, F.C.K., (2008) . Large-scale turbulence under a solitary wave: Part 2; Forms and evolution of coherent structures. *Coastal Engineering*. 55, pp 522-536

- Ting, F.C.K., Kirby, J.T., (1996). Dynamics of surf-zone turbulence in a spilling breaker. *Coastal Engineering*. 27, pp 131-160
- Ting, F.C.K., Nelson, J.R., (2011). Laboratory measurements of large-scale near-bed turbulent flow structures under spilling regular waves. *Coastal Engineering*. 58, pp 151-172
- Yeh, H. H., Ghazali, A., (1988). On bore collapse. *Journal of Geophysical Research*. 93(C6), pp 6930-6936
- Yeh, H. H., Ghazali, A., Marton, I., (1989). Experimental study of bore run-up. *Journal of Fluid Mechanics*. 206, pp 563-578
- Zhang, Q., Liu, P. L.-F., (2008). A numerical study of swash flows generated by bores. *Coastal Engineering*. 55, pp 1113-1134