SHIRAZ UNIVERSITY
FACULTY OF ENGINEERING

Ph.D. DISSERTATION IN
CIVIL ENGINEERING

NUMERICAL MODELING OF WAVE FLUME USING SMOOTHED PARTICLE HYDRODYNAMICS

BY
ALI MAHDAVI

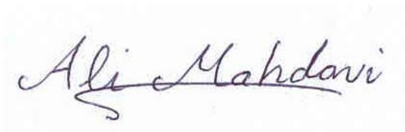
SUPERVISED BY
N. TALEBBEYDOKHTI, Ph.D.

MARCH 2015

Declaration

I, Ali Mahdavi (885455), a Civil Engineering Student from the Faculty of Engineering, declare that this thesis is the result of my research and I have written the exact reference and full indication wherever I used others' source. I also declare that the research and the topic of my thesis are not reduplicative and guarantee that I will not disseminate its accomplishments and not make them accessible to others without the permission of the university. According to the regulations of the mental and spiritual ownership, all rights of this belong to Shiraz University.

Name: Ali Mahdavi
Date and Signature: 2015-06-01



IN THE NAME OF GOD

NUMERICAL MODELING OF WAVE FLUME USING SMOOTHED PARTICLE HYDRODYNAMICS

BY

ALI MAHDAVI

Ph.D. DISSERTATION

SUBMITTED TO THE SCHOOL OF GRADUATE STUDIES IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY (Ph.D.)

IN

CIVIL ENGINEERING
(WATER ENGINEERING)

SHIRAZ UNIVERSITY
SHIRAZ
ISLAMIC REPUBLIC OF IRAN

EVALUATED AND APPROVED BY THE THESIS COMMITTEE AS: EXCELLENT

N. TALEBBEYDOKHTI, Ph.D., PROF. OF CIVIL AND ENVIRONMENTAL ENGINEERING (SUPERVISOR)

G. R. RAKHSHANDEHROO, Ph.D., PROF. OF CIVIL AND ENVIRONMENTAL ENGINEERING (ADVISER)

M. MAZAHERI, Ph.D., ASSISTANT PROF. OF TARBIAT MODARES UNIVERSITY (ADVISER)

A. RANJBARAN, Ph.D., ASSOCIATE PROF. OF CIVIL AND ENVIRONMENTAL ENGINEERING (ADVISER)

P. MALEKZADEH, Ph.D., ASSOCIATE PROF. OF PERSIAN GULF UNIVERSITY (EXTERNAL EXAMINER)

H. EMDAD, Ph.D., ASSOCIATE PROF. OF MECHANICAL ENGINEERING (INTERNAL EXAMINER)

MARCH 2015

Acknowledgments

I would like to take this opportunity to express the warmest thanks to several people:

First and foremost, Prof. Nasser Talebbeydokhti, my supervisor, for your intellectual inspiration and great guidance. I appreciate your assistance and patience which showed me the way to become an independent researcher. One simply could not wish for a better or friendlier supervisor.

Prof. Gholamreza Rakhshandehroo, Dr. Mehdi Mazaheri and Dr. Abdolrasol Ranjbaran for kindly serving on my Ph.D. committee and for valuable discussions of the theoretical and numerical results. I appreciate the time and helpful suggestions given to my investigation.

Dr. Parviz Malekzadeh and Dr. Homayoun Emdad for carefully reading the thesis and giving detailed comments which resulted in improving the quality of the work.

Dr. Mohammad Reza Hashemi and Dr. Zahra Ghadampour for helpful discussions on various aspects of LDQ and SPH.

Dr. Mehdi Moeini, Dr. Seyyed Mehrab Amiri, and Hossein Khorshidi for being great friends, a constant source of support and inspiration.

Dr. Hamid Seyyedian, my mentor, for his endless encouragement, support and advice during my Ph.D. program.

Last but not least, I would like to express my gratitude to my family ─my parents and sisters─ for their love, never-ending support, encouragement and prayers. Without their support this work would not have been possible.

Abstract

NUMERICAL MODELING OF WAVE FLUME USING SMOOTHED PARTICLE HYDRODYNAMICS

By:

ALI MAHDAVI

The present research is mainly aimed at developing a numerical wave flume based on smoothed particle hydrodynamics (SPH), a mesh-free particle method. Accordingly, the two-dimensional inviscid Navier–Stokes (Euler) equations are solved in the Lagrangian framework. This numerical tool resembles a prismatic flume of rectangular cross section. In addition to non-breaking wave run-up, the model enables a detailed description of pre- and post-breaking stages of wave motion. The wave flume is equipped with the Scott Russell's wave generator—a falling mass that produces solitary waves while sinking in the water body. It is demonstrated how the wave generator is represented as a collection of "pseudo-fluid particles". Careful attention has been paid to properly include solid walls as well as inflow boundaries. The former allows continuous flow field near the walls while the latter extends the SPH versatility beyond the limit of confined flows.

It is well known that the SPH suffers from the computational costs, making it practically unsuitable for domains of large spatial extent. In contrast, the Eulerian methods relying on depth-averaged conservation laws appear to be computationally efficient when discretized over large areas. Therefore, a hybrid model is attempted that combines the SPH with an Eulerian solver for the Boussinesq equations in a one-way coupling framework. Finally, a computational mapping technique is proposed for a two-way Eulerian-Lagrangian coupling in the context of nonlinear shallow water equations. The performance of proposed schemes is thoroughly assessed by comparing with relevant analytical solutions, experimental data and numerical results found in the published literature.

Keywords: SPH, water wave modeling, Eulerian-Lagrangian coupling, solid boundary treatment, inflow boundary, fluid-rigid body interaction

Table of Contents

Title **Page**

List of Figures

Title **Page**

List of Tables

Title **Page**

CHAPTER 1

INTRODUCTION

Accurate prediction of nearshore waves and their interaction with coastal structures has attracted considerable attention in the coastal engineering community over the last several decades. As such, several complex phenomena are to be correctly understood to draw a detailed picture of nearshore hydrodynamics. These include, but not limited to, wave propagation/transformation, interaction among incident and reflected waves, wave overtopping, wave impact forces on maritime structures and run-up motion. The latter phenomenon is frequently observed by everybody who stands near the border of a large body of water. The run-up motion, apart from its important role in the nearshore hydrodynamics, also exhibits aesthetic aspects, attracting talented poets who do not actually know how to express waves mathematically!

As a long wave approaches nearshore zones, its wavelength and energy undergo a continuous compaction leading to build up in the wave amplitude. Concurrently, the nonlinear effects develop steepening waves, propagating bores or even breaking waves. The latter is a complex process that can be identified in the form of spilling, surging, collapsing, or plunging breaking (Sorensen, 2006).

The mathematical models governing the wave motion have been extensively studied since the development of applied mathematics. Earlier analytical wave models severely suffered from restrictive assumptions and/or over-simplified geometries (e.g., Carrier and Greenspan, 1958; Tuck and Hwang, 1972). It is obvious that such models fail to handle realistic scenarios encountered in a wave environment. Nevertheless, they can still serve as benchmarks which are appreciable to assess the validity of more complicated numerical schemes. Most of the analytical wave run-up models, however, require additional mathematical efforts due to improper integrals involving Bessel functions. Consequently, the evaluation of solution for a given problem has to be accomplished numerically (e.g., Synolakis, 1986, 1987; Li and Raichlen, 2001).

A wave flume is recognized as a laboratory apparatus designed for the physical modeling of different water wave phenomena. This offers valuable insights regarding characteristics of waves and helps understanding, on a sound physical background, how these affect coastal structures, offshore structures, sediment transport and other relevant features.

Alternatively, the wave evolution process can be well addressed by a numerical wave flume. The accuracy of numerical predictions depends on the assumptions made in developing the mathematical model as well as the numerical scheme implemented for the problem. The major advantages of the numerical wave models, which represent approximate solutions to the governing equations, rely on the fact that they are less demanding compared to physical counterparts. Moreover, most of the restrictive assumptions adhered to analytical models can be relaxed with numerical schemes. Numerical wave models may be regarded as flexible tools to investigate wave evolution under different circumstances, assisting in the understanding and even the detection of previously unknown phenomena. This versatility of application is motivated by the fact that any desired change in the geometrical configurations or on the wave conditions can be imposed by simply redefining the model input data. Besides, resorting to numerical simulation can significantly reduce the amount of time required by experimental methods to evaluate a large number of design alternatives, without suffering from the scale dependency inherent in physical modeling. The abovementioned features clearly highlight the role of numerical studies as a complementary to the physical modeling, particularly when it is difficult to record a flow variable experimentally.

Owing to the recent improvements and developments in the field of numerical methods, computational fluid dynamics (CFD) increasingly manifests itself as a powerful tool to handle coastal engineering problems. They provide a reliable tool to study the wave motion. Formally, a set of partial differential equations in time and space are to be numerically treated with pressure and velocity components being the main state variables. The calculation of flow field at every time is accompanied by properly defined initial and boundary conditions which guarantee a unique solution for the problem. Accordingly, the spatial domain is broken up into a set of computational cells, either structured or unstructured, and time is represented in the discrete form by a finite number of steps. The proportion of spatial and temporal increments is often related to the stability of numerical scheme under consideration. Different numerical techniques can be adopted to convert continuous governing equations into discretized counterparts. Conventional numerical schemes, such as the finite difference method (FDM), the finite volume method (FVM) and the finite element method (FEM), typically embody fixed cells through which the fluid is flowing. The FDM approximates the governing equations on a rectangular grid with truncated expressions from the Taylor series expansion. The method is historically accepted in different branches of engineering and science, thanks to the ease of implementation and accuracy it could offer. However, the FDM requires a quite regular mesh, often rectangular in shape; therefore large deformations in the flow can not be handled properly. Attempts have been recently made to accommodate FDM with irregular-shaped domains via reconstructing the formulation on generalized curvilinear coordinates, rather than traditional Cartesian one. The FVM, probably being the most popular technique in CFD, represents the spatial domain via a number of finite volumes over which the conservation principles are integrated. The Gauss-Green theorem then transforms the volume integration into a surface one; thereby the rate of change of a certain quantity inside a control volume is linked to the fluxes passing through the boundaries. Similarly, the FEM domain is subdivided into a set of elements which share flow data at nodal points. The associated formulation may be established on the variational basis, by minimizing a properly defined error function. The method appears to be very robust in dealing with domains confined by irregular boundaries.

Despite their apparent advantages, the traditional grid-based schemes suffer from inherent drawbacks in many features, with the tendency to restrict the model performance in a variety of situations. One such difficulty is encountered in the case of highly complex free surface flows characterized by large deformation and fragmentation. Supplementary techniques are often required to delineate the free surface as it moves rapidly with time.

Under such circumstances, the smoothed particle hydrodynamics (SPH), as a mesh-free, Lagrangian method, can be preferred to the commonly-used CFD techniques. The efficiency of the SPH has been confirmed even in simulating broken and multi-connected free surface configurations. The SPH principles are founded on finite number of moving interpolation points that can be regarded as particles, each carrying the physical properties of flow. The information gathered from this set of Lagrangian particles fully defines the state of system at discrete level. The equations of motion can be extracted from the Navier–Stokes equations by adopting a kernel interpolation method which converts the integral equations into summations over nearby particles. The partial differential equations are thus simplified to ordinary differential equations that can be integrated in time to evaluate the properties of each particle.

1-1 Objectives

This thesis mainly focuses on developing a numerical wave flume based on smoothed particle hydrodynamics, a mesh-free particle method. Accordingly, the two-dimensional Euler equations are dealt with in the Lagrangian framework, reproducing a prismatic flume of rectangular cross section. In addition to non-breaking wave run-up, the model enables a detailed description of pre- and post-breaking stages of wave motion. The wave flume is equipped with the Scott Russell's wave generator—a falling mass that produces solitary waves while sinking in the water body.

It can be concluded from the published literature that the SPH is highly demanding with regard to computational cost, making it practically unsuitable for domains of large spatial extent. On the contrary, the Eulerian methods relying on depth-averaged conservation laws (e.g., nonlinear shallow water equations and Boussinesq equations) appear to be computationally efficient when discretized over large areas. Therefore, a hybrid model will be attempted that simultaneously incorporates the SPH and an Eulerian solver for the Boussinesq equations in a one-way coupling framework. Finally, a computational mapping technique will be proposed for a two-way Eulerian-Lagrangian coupling in the context of nonlinear shallow water equations.

The performance of proposed schemes will be thoroughly assessed by comparing with relevant analytical solutions, experimental data and numerical results found in the published literature.

1-2 Novelties

Although widespread attention has been given to SPH technique in recent years, there are still important drawbacks because the method is comparatively new in the field of free-surface flows. Therefore the aim of present study is to develop numerical techniques for free-surface flows interacting with rigid structures. Listed below are the new aspects of the numerical methods developed in this study:

- *Solid wall modeling*: A hybrid technique is proposed for solid boundary treatment within SPH context to ensure accurate computation of wave impact pressures on the wall. The basic concept is to fill an impervious region with some layers of dummy particles for improving the solution accuracy and a single layer of repulsive particles for imposing no-penetration condition along the solid–fluid interface.
- *Inflow boundary condition*: The application of SPH is often limited to flows occurring in a confined flume with a free surface and the system is completely isolated from the surroundings by solid walls. Introducing inflow conditions in the wave flume can broaden its versatility to handle open-channel flows, as well. It is accomplished by developing a particle injection algorithm to mimic the desired inlet boundaries. It will be shown, through numerical tests, that the proposed algorithm works equally well for both the steady and unsteady inflows. The latter may be thought of as an alternative way for generating solitary waves at the SPH inlet. Besides its simplicity, the algorithm can potentially extend the application of SPH for the cases where the flow is driven by arbitrary velocity profiles at the boundaries.
- *Fluid-solid interaction (Rigid body motion)*: A Scott Russell's wave generator is also modeled in the wave flume. Just like the fluid continuum, this type of moving boundary is represented by a set of particles, referred to as "pseudo-fluid particles". Their positions obey the rigid body dynamics in a fully coupled framework with fluid pressure forces acting as external excitations for the object. The computation of density and pressure for a pseudo-fluid particle follows the same rule as that of a real fluid particle —by satisfying the equation of state and mass conservation law. The proposed algorithm allows treating the actual rigid body as hollow shape represented by few layers of pseudo-fluid particles, the mass of which is being introduced as an input data, rather than computed by summing up the contribution from individual particles. This decreases the computational cost by reducing the number of particles to be involved in the simulation.
- *The two-step discretization scheme for the Boussinesq equations*: The present algorithm comprises a predictor-corrector discretization scheme for the Boussinesq equations which implicitly calculates both the free surface elevation and the flow velocity. However, given the nonlinearity of the governing equations, the procedure requires a small number of iterations per time step.
- *The one-way coupling algorithm*: The algorithm proposed herein utilizes an embedded domain, where a 1D finite difference Boussinesq method solves the flow field over large areas. This Eulerian sub-model is then coupled to a 2D Lagrangian SPH method which can potentially delineate local flow features in regions of interest. The modeler therefore enjoys the speed of the Eulerian solver as well as the detailed flow features inherent in the SPH method. The Eulerian solver is first applied over the entire computational domain. The flow data are extracted from the Boussinesq solver and then submitted to the SPH. The data transformation