Ocean Engineering 131 (2017) 57–67

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Validation of a piston type wave-maker using Numerical Wave Tank

Deepak Divashkar Prasad[a], Mohammed Rafiuddin Ahmed[a,*], Young-Ho Lee[b], Rajnish N. Sharma[c]

[a] Division of Mechanical Engineering The University of the South Pacific Suva, Fiji Islands
[b] Division of Mechanical and Energy System Engineering College of Engineering Korea Maritime and Ocean University, 1 Dongsam-dong Youngdo-ku, Busan 606-791, South Korea
[c] Department of Mechanical Engineering The University of Auckland Auckland, New Zealand

ABSTRACT

Commercial Computational Fluid Dynamics (CFD) code ANSYS – CFX which solves the Reynolds Averaged Navier-Stokes Equations (RANSE) is used to simulate waves in a 3D Numerical Wave Tank (NWT). The free surface is captured using Volume of Fluid (VOF) method. The numerical code to generate waves in NWT is first validated against experimental data to check the accuracy and robustness of the code. The experimental setup included a turbine and as such the initial numerical code was validated by comparing the wave height and the performance of the turbine against the experimental data. The initial numerical results show good agreement with the experimental data. Later an actual NWT is constructed to generate desired wave climate based on field measurements at a site in Fiji. The CFD results show very good agreement with field measured wave height of 1.23 m and field measured wave power of 9.81 kW/m. The CFD results were also verified with analytical solutions. The difference in the results is within 3% highlighting the validity of the CFD code which can further be extended to investigate other phenomena.

1. Introduction

A wave tank is characterized as a long and narrow enclosure with a wave-maker at one end (Mikkola, 2007). Waves in the wave tank are generated through the movement of a paddle (also known as a wave-maker) that is located at one of the ends of the wave flume (Oliveira et al., 2009). The most common of these wave-makers are piston, flap and wedge type. The difference amongst these wave-makers solely lies in their motion. Wave tanks or wave flumes have been used for decades to conduct tests and research. These tests have provided many valuable results that have helped in design of devices, structures and even help setup codes. The sizes of these wave tanks vary from as small tanks used for educational purposes to as large scale tanks as used in advanced facilities for product testing and development. It is apparent that to acquire wave tanks is an expensive task. It also demands large enough space to house the wave tank which can also add to the cost. In addition to this, prototype construction and testing takes a lot of time. Furthermore, the cost associated with construction of a model at its infant stage is colossal and there is no guarantee that the design would work properly in the first instance. To compound to these expenses is the cost associated with redesign and re-testing.

The need to find an alternative without compromising the integrity of the results has led to the development of Numerical Wave Tanks (NWT). NWT in simple is the numerical representation of the physical wave tank. The development and advancements in computer processing power have paved the way to the use of Computational Fluid Dynamics (CFD) codes that are used to accomplish this task. The CFD code solves the Reynolds Averaged Navier-Stokes Equations (RANSE). The main governing equations used in the CFD solver are: conservation of mass (continuity), conservation of momentum (Newton's second law of motion) and conservation of energy (first law of thermodynamics). Researchers have proposed many different varieties of NWT based on specific application. Generally, they can be divided into two groups, one which is based on Non Linear Shallow Water (NLSW) equations and the other based on Navier-Stokes (NS) equations. NWT based on N-S equations are generally controlled by either Volume of Fluid (VOF) technique or Smooth Particle Hydrodynamics (SPH) technique. Liu et al. (2008) used commercial CFD code FLUENT to study the performance of Oscillating Water Column (OWC). The authors employed VOF technique to capture the free surface. Repalle (2007) also employed VOF method for NWT application to model wave run up around a spar cylinder. Papers by Horko (2007) and Lemos (1990) also highlighted the use of this model. On the other hand, Dalrymple and Rogers (2006) employed the SPH model in their simulations to study

---

* Corresponding author.
E-mail addresses: prasad_dp@usp.ac.fj (D.D. Prasad), ahmed_r@usp.ac.fj (M.R. Ahmed), lyh@kmou.ac.kr (Y.-H. Lee), r.sharma@auckland.ac.nz (R.N. Sharma).

http://dx.doi.org/10.1016/j.oceaneng.2016.12.031
Received 14 November 2014; Received in revised form 30 December 2016; Accepted 31 December 2016

plunging type wave breaking. This model was also used by Shao et al. (2006) to investigate overtopping in coastal structures.

Waves in NWT can be generated either by inlet velocity profile method or by wave-maker method. In inlet velocity profile method, the water particle velocity together with the wave elevation at the inlet is specified. A suitable wave theory is used as input and the wave is generated by specifying the wave elevation at the inlet of the domain by volume fraction. On the other hand, the wave-maker method which resembles a physical wave flume incorporates a moving wall. The moving wall is normally referred to as a flap hence this method is also known as flap method. Dynamic mesh technique is used which allows for controlling the flap displacement and hence the waves generated (Gomes et al., 2009). It is necessary to know the wave period and the wave height in order to generate waves. Using these parameters, the oscillation of the flap is achieved by a transfer function which relates the displacement of the flap to the wave height. The free surface is captured using the VOF method. In physical wave tanks there is always the issue of reflected waves, which, if not dealt properly will affect the results. Normally wave tanks would be equipped with tunable beaches or simply have a slope at the back wall. Reflection is also present in NWT and few methods exist to deal with this. One such method is the periodic boundary condition where the solution is assumed to be periodic in space hence the values of variables on one vertical boundary can be equal to those on the other vertical boundary (Maguire and Ingram, 2009). Another technique to deal with reflected waves is by incorporating an artificial damping zone or sometimes referred to as sponge layer at the back wall of NWT. The technique involves applying a dissipative term to the equations near the boundary of the truncated domain. Damping terms are added to the dynamic and kinematic free surface boundary conditions to give artificial damping effect to free surface. This technique was employed by Cointe et al. (1990) and Tanizawa (1996) in their studies. The idea of absorbing beach to deal with reflected waves was introduced by Larsen and Dancy (1983). The method involved applying an artificial counteracting pressure over a given distance in the dynamic free surface condition which created a negative work against incident waves. Similar method to deal with reflected waves in NWT were employed by Guerber et al. (2012), Grilli and Horrillo (1997) and Clement (1996). Orlanski (1976) provided a new approach by imposing Sommerfeld radiation condition on the boundary. The proposed condition showed to be free of reflection for single wave propagation. In addition to the above mentioned techniques, damping of reflected waves from the back wall of NWT can be controlled by grid size in this region. Coarse grid size in the back wall region provides damping and this is an easier option as it requires less simulation time compared to other methods. Park et al. (2004) employed artificial beach and increased mesh size gradually in the horizontal direction away from the wave-maker to provide additional numerical damping. Baudic et al. (2001) numerically simulated fully nonlinear transient waves in a semi-infinite 2-D NWT. They used a damping region and radiation condition to prevent wave refection. The results showed good agreement with previous published works of the authors.

Use of NWT has gained vast interest over the past few years. In a paper by Finnegan and Goggins (2012), 2-D NWT was used to simulate linear deep water waves and linear waves for finite depth. The authors also compared turbulence model, laminar and k-ε in order to investigate the effect of viscosity. They found no difference in the generated wave elevation between the two models and hence the turbulence model is not a factor in the generation of waves using wave-maker. Furthermore, they studied wave-structure interaction in which a floating truncated vertical cylinder was used. Li and Lin (2010, 2012) studied nonlinear wave-body interaction for a stationary floating structure under regular and irregular waves at various water depths, wave heights and periods in a 2-D NWT. The same authors (Li and Lin, 2012) in another paper investigated the fully nonlinear wave-body interaction for a surface piercing body. They employed a 2-D NWT which was mainly based on the spatially averaged N-S equations. For modeling of the turbulence of flow, k-ε model was incorporated. Hydrodynamics and turbulence on wave propagation over coarse grained sloping beach experimentally and numerically was investigated by Lai et al. (2010). The numerical results were in good agreement with the experimental data. The results showed maximum turbulent kinematic energy and turbulent dissipation rate occurring around the surface of the spherical ball layer near the surf zone. They also highlighted that coarse grained porous slope recorded lower wave breaking and run-up compared to impermeable slope. The wave fluctuation on porous bed was small due to the bottom friction and influence of porosity.

NWT have also been employed in the field of wave energy to design new Wave Energy Converters (WEC), studying performance of WEC or optimizing WEC. El Marjani et al. (2006) using commercial CFD code FLUENT predicted the air flow behaviour inside the chamber of an OWC wave converter. The authors only considered monochromatic sinusoidal excitations with varying frequencies and fixed amplitudes. They looked at flow characteristics, velocity, pressure, flow rate and power. They used a porous medium to model the turbine and incorporated a linear law relating the pressure drop to the mass flow rate in order to take into account the presence of the turbine. Senturk and Ozdamar (2011) using FLUENT investigated the interaction between regular waves and the OWC geometry. To conduct 3D analysis they adopted a piston like pumping flow which used the free surface elevation data from their previous 2-D simulation. The inner water level recorded in the OWC during 2-D simulation was in good agreement to their theoretical model. Liu et al. (2012) using NWT based on VOF simulated the water column oscillation in the chamber of OWC and compared their numerical results with experimental results. They reviewed state of the art in interaction among wave elevation inside the chamber and air flow rate in the duct which considered the turbine effect. The study also encompassed investigation into the effect of incident wave conditions and shape parameters on the performance of OWC. The pressure drop effects induced by impulse turbines was successfully modeled using orifice device and implemented numerically. The method showed great promise for future studies. Turbine – chamber coupling in an OWC was studied experimentally and numerically by Lopez et al. (2012). Authors stated that it is important to study these two together because the damping caused by the air turbine affects the primary energy conversion. This ultimately affects the air flow driving the turbine. For numerical work they used 2-D NWT which solved 2-D RANSE for incompressible fluid and free surface was captured by VOF method. There was no turbine included however; the influence of the turbine was achieved by setting a variable slot which simulated different damping conditions.

Gomes et al. (2012) presented 2-D numerical study on the geometric optimization of WEC which principally worked on OWC concept. The height to length ratio of the OWC were varied while keeping the height to length ratio of the chimney constant. The OWC chamber area and the total OWC area were fixed as well. The objective of the study was to optimize the geometry of the device such that it absorbed maximum power when subjected to a defined wave climate. The paper highlighted the applicability of constructional design to optimize the OWC. They highlighted that the model with height to length ratio of the OWC equal to 0.84 performed the best and when compared to the worst case the improvement in the efficiency was 10 fold. Grimmler et al. (2012) conducted similar work on 3D OWC WEC. Winchester et al. (2011) conducted CFD analysis of a novel multi axis WEC called Pelican. The shape of the Pelican was obtained using genetic algorithm. The point absorber was designed for regular waves with period ranging from 5 to 14 s. The paper highlighted effect of vorticity around WEC. A novel design of high efficiency impulse turbine for OWC use was proposed by Natanzi et al. (2011). The design showed 75% total static efficiency for the full scale turbine. The authors used a Varying Radius Turbine (VRT) and their optimized design minimized