Assessment of uncertainty due to wave reflections in experiments via numerical flow simulations

Robinson Perić, Moustafa Abdel-Maksoud
Institute of Fluid Dynamics and Ship Theory, Technical University Hamburg-Harburg (TUHH), Hamburg, Germany

ABSTRACT

Reflections at the wave-maker and tank boundaries can be a severe problem in wave tank experiments. Often, such reflections cannot be avoided. The present paper proposes an approach to quantify this influence via numerical flow simulations. It is straightforward to redo the actual experiment “virtually” in a flow simulation, featuring full reflections as present in the wave tank experiment. It is shown that reflections at the wave-maker and domain boundaries can be nearly eliminated by applying appropriate boundary conditions in combination with domain-inertial wave generation approaches. This is demonstrated for the case of a boat subjected to regular deep water waves from starboard side. Flow simulations based on the Euler equations are performed for a realistic model scale setup, in a similar manner as experiments in a narrow wave tank would be performed. Subsequently, the simulation is repeated in such a way, that undesired wave reflections at both the domain boundaries and the wave-maker are minimized. From the comparison, it is possible to separately assess the influence of wave reflections at the wave-maker as well as at the wave tank boundaries.

KEY WORDS: Wave generation; wave reflection; Euler equations; mass sources.

INTRODUCTION

In experiments, wave reflections at the wave-maker can pose a severe problem. This is typically the case when investigating the flow around floating or fixed bodies in waves, which is relevant for many ocean engineering applications. In wave-tank experiments, reflections at the wave-maker and the tank boundaries cannot be completely avoided. Depending on the facility, some damping of undesired wave reflections can be obtained by beaches or active control of the wave-maker (Cruz (2008); Schäffer and Klopman (2000)). However, especially for the traditional long narrow towing tank, reflections at the tank boundaries cannot be avoided. The combined influence of all reflections on the experimental results has so far been difficult to quantify. An approach to assess the uncertainty in such experiments via numerical flow simulations is proposed in the following. Established wave damping techniques are used in combination with numerical wave-makers, which can be placed inside the solution domain and let waves propagate through them with negligible reflection. These methods are discussed in the following sections and validated by 2D flow simulations of a standing wave. The actual experiment can thus be virtually repeated with as well as without undesired reflections in flow simulations. This is demonstrated via four 3D simulations of the flow around a tug boat in regular waves from starboard, which resemble an actual experimental wave tank configuration.

GOVERNING EQUATIONS AND SIMULATION SETUP

The incompressible Euler equations are chosen as governing equations, since viscous effects can be expected to be negligible in the flow problems considered in this work. The mass and momentum conservation equations are thus

\[
d \frac{d}{dt} \int_{V} \rho \, dV + \int_{\partial V} \rho \mathbf{v} \cdot \mathbf{n} \, dS = \int_{\partial V} \rho \mathbf{q} \cdot \mathbf{n} \, dS, \quad (1)
\]

\[
d \frac{d}{dt} \int_{V} \rho \mathbf{u} \, dV + \int_{\partial V} \rho \mathbf{u} \mathbf{v} \cdot \mathbf{n} \, dS = \int_{\partial V} -\mathbf{p} \mathbf{i} \cdot \mathbf{n} \, dS + \int_{\partial V} \rho \mathbf{g} \mathbf{i} \, dV + \int_{\partial V} \rho \mathbf{f} \, dV \quad (2)
\]

Here \(V\) is the control volume (CV) bounded by the closed surface \(S\), \(\mathbf{v}\) is the velocity vector of the fluid with Cartesian components \(\mathbf{u}\), \(\mathbf{n}\) is the unit vector normal to \(S\) and pointing outwards, \(t\) is time, \(\rho\) is the fluid density, \(\mathbf{p}\) is the pressure, \(\mathbf{i}\) is the unit vector in direction \(x_i\), \(\mathbf{g}\) represents the body force due to gravitational acceleration \(\mathbf{g} = (0,0,9.81) \text{ m/s}^2\), \(q_s\) is the mass source term and \(q_l\) is the momentum source term. The commercial flow solver STAR-CCM+ by CD-adapco, extended by user programming, was used for the simulations in this study. In all simulations, the coordinate system has its origin at the free surface, located at or above the wave-maker, depending on the wave generation mechanism, as depicted in Figs. 2 and 3. The \(x\)-axis points in the desired wave propagation direction, the \(z\)-axis points upwards. To account for the two phases, the volume of fluid (VOF) method is used. Thus the density is described as

\[
\rho = \rho_{\text{air}} \alpha_{\text{air}} + \rho_{\text{water}} \alpha_{\text{water}}, \quad (3)
\]

with densities of air \(\rho_{\text{air}} = 1.2 \text{ kg/m}^3\) and water \(\rho_{\text{water}} = 1000 \text{ kg/m}^3\), and volume fractions of air \(\alpha_{\text{air}}\) and water \(\alpha_{\text{water}}\). The sum of \(\alpha_{\text{water}}\) and \(\alpha_{\text{air}}\) equals 1 for each cell, while \(0 \leq \alpha_i \leq 1\) for each phase \(i\). The transport equation for the volume fractions is