Simulation of Free-Surface Flow Using the Smoothed Particle Hydrodynamics (SPH) Method with Radiation Open Boundary Conditions

XINGYE NI
College of Harbor, Coastal and Offshore Engineering, Hohai University, Nanjing, China, and Department of Oceanography, Dalhousie University, Halifax, Nova Scotia, Canada

JINYU SHENG
Department of Oceanography, Dalhousie University, Halifax, Nova Scotia, Canada

WEIBING FENG
College of Harbor, Coastal and Offshore Engineering, Hohai University, Nanjing, China

(Manuscript received 3 September 2015, in final form 30 August 2016)

ABSTRACT

The smoothed particle hydrodynamics (SPH) technique is a mesh-free numerical method that has great potential to be used in the development of the next generation of numerical ocean models. The implementation of open and solid boundary conditions in the SPH method, however, is not as straightforward as the mesh-based numerical methods. Two types of open boundary conditions are considered in this study: the adaptive open boundary condition (AOBC) and Flather’s open boundary condition (FOBC). These two open boundary conditions are implemented in the SPH-based shallow-water equation (SWE) circulation model for simulating sea surface elevations and depth-mean currents over a limited area with open boundaries. The performance of these two open boundaries is assessed in four numerical test cases. In comparison with the conventional characteristic open boundary condition, both the AOBC and the FOBC allow perturbations to propagate out more effectively and are easy to implement with the specification of external flow conditions at the model open boundaries. The model results also demonstrate that the AOBC requires an accurate estimation of the phase speed of perturbations and could lead to a small drift in the mean water level. By comparison, the FOBC is computationally more efficient without any model drift. The SPH-based SWE circulation model is also used in simulating the laboratory observations of the 1993 Okushiri Tsunami. The numerical results in this case demonstrate the feasibility and capability of the SPH-based SWE model for simulating free-surface flows in regions with complicated bathymetry and irregular coastline.

1. Introduction

With the advent of computer technology and numerical methods, considerable progress has been made in the development and application of numerical ocean models in the last four decades. Today, numerical ocean models have widely been used in modeling and predictions of various phenomena in the ocean, ranging from tsunami, storm surges, surface gravity waves, and three-dimensional (3D) currents and hydrography. There are several ocean general circulation and wave models that have achieved a significant level of community management and involvement. Typical examples of these models include the Modular Ocean Model (MOM, www.gfdl.noaa.gov/mom-ocean-model), the Océan Parallélisé (OPA, www.nemo-ocean.eu), the Hybrid Coordinate Ocean Model (HYCOM, hycom.org), the Regional Ocean Modeling System (ROMS, www.myroms.org), the Princeton Ocean Model (POM, http://www.ccpo.odu.edu/POMWEB/), Finite Volume Community Ocean Model (FVCOM, fvcom.smast.umassd.edu), WAVEWATCH III (WW3, polar.ncep.noaa.gov/waves/index2.shtml), and Simulating Waves Nearshore (SWAN, www.swan.tudelft.nl). Among these models the MOM, OPA, HYCOM, POM, ROMS, and FVCOM
are the ocean general ocean circulation models, and WW3 and SWAN are the third-generation wave models. The common feature of the above-mentioned ocean circulation and wave models is that model governing equations are discretized using a set of discrete grid points (or mesh). Each grid point has a fixed number of predefined neighboring points. The connection between neighboring points is used to define mathematical operators to approximate terms such as first- and second-order partial derivatives. These operators are then used to convert the governing differential equations into a set of algebraic equations that can be calculated numerically on an electronic computer. The main advantage of the mesh-based numerical model is its computational efficiency and relative ease in treating solid and open boundaries. The mesh-based numerical models, however, have significant disadvantages due to the use of mesh. The connectivity of the mesh, for example, can be difficult to maintain without introducing numerical errors over regions with deformable boundaries, moving interfaces, and complicated geometries (Liu and Liu 2010). If the mesh becomes tangled or degenerated during simulation, then the operators defined that it may no longer give correct values. Furthermore, the mesh-based model could not be used in simulating large discontinuities that do not coincide with the model grid.

Significant efforts have been made recently in the development of mesh-free numerical methods, which do not require mesh to connect grid points of the model computational domain. The main advantage of the mesh-free numerical model is its ability to simulate discontinuities and deformations in fluid boundaries, which is, otherwise, very difficult using a mesh-based numerical model. The smoothed particle hydrodynamics (SPH) technique is a novel mesh-free numerical method. The SPH method was suggested originally by Gingold and Monaghan (1977) and Lucy (1977) independently for solving astrophysics problems in the 3D open space without any solid boundaries. The SPH method has gained significant popularity in recent years in the field of computational fluid dynamics, due to its pure Lagrangian features and its capability in dealing with complicated hydrodynamic problems with large deformations and discontinuities. We envisage that the SPH method has great potential to be used in the development of the next generation of numerical ocean models. Many important issues, however, remain to be addressed before the theoretical framework of the SPH is used in the future modeling development.

One of outstanding issues with the use of the SPH method is the proper treatment of the open boundary condition (OBC) in simulating hydrodynamics over a limited area. The periodic boundary (Gomez-Gesteira et al. 2012a; Vacondio et al. 2013; Cleary 1998) is one of the commonly used OBCs for the SPH method, in which particles move out of the model domain from the downstream side and come back into the domain from the upstream side. During this out-and-in process, the total number of particles over the model domain remains constant. The periodic boundaries, however, can be used only in very special cases and can also introduce numerical errors by recycling the perturbed (or inaccurate) particle distributions back to the model domain through the upstream OBC. Another common OBC is the sponge layer (Ni and Feng 2013; Molteni et al. 2013), in which the energy of currents (or waves) is significantly absorbed as it approaches the sponge layer. The main disadvantage of the sponge layer OBC is that the external flow conditions cannot be specified at the model open boundary.

Lastiwka et al. (2009) implemented the first general open boundary condition for the SPH method by removing outgoing particles from the outflow (downstream) buffer zones and adding new particles into the inflow (or upstream) buffer zones. The variables of the particles in the buffer zones are specified by the characteristic open boundary condition (COBC). Lastiwka et al. (2009) used this COBC in simulating a two-dimensional (2D) flow around a cylinder using the SPH method. The COBC suggested by Lastiwka et al. (2009), however, could not be used in the free-surface flow, since the free-surface flow requires the total water depth at the outflow open boundary to be prescribed. Federico et al. (2012) used the buffer zone technique in simulating a hydraulic jump and its interaction with artificial structures in an open channel flow, but they neglected the reflection at the OBCs. Similarly, Vacondio et al. (2012b) and Chang and Chang (2013)
considered the COBC in their studies and also ignored the reflection problem. The main objectives of this study, therefore, are to introduce two new open boundary conditions for the SPH method and to assess the performance of these two open boundaries in four standard test cases.

The paper is organized as follows. The basic methodology of the SPH method and its application in simulating 2D circulations are introduced briefly in section 2. A modified dynamic solid boundary treatment (MDSBT) is discussed in section 3. Two numerical test cases of a steady flow over a bump and a subcritical flow in a flat-bottom channel with an initial perturbation are conducted to examine the reflection problem of the COBCs in section 4. In section 5, the adaptive open boundary condition (AOBC) and Flather’s open boundary condition (FOBC) are suggested for the SPH method, and the performance of these two new open boundary conditions is assessed in four numerical test cases. In section 6, the SPH-based circulation model is used in simulating the laboratory observation of the tsunami that occurred in Okushiri, Japan, in 1993. Section 7 provides a summary and conclusions.

2. SPH-based numerical model

a. Principle of the SPH method

The SPH is a mesh-free particle method based on the Lagrangian formulation and has been increasingly used in many fields, such as astrophysics, fluid dynamics, and solid mechanics. The SPH method is based on the theory of integral interpolants using kernels that approximate a delta function. The principle of the SPH method is described briefly as follows.

Any arbitrary field function \( f(\mathbf{r}) \) can be expressed mathematically in the following form (Gingold and Monaghan 1977):

\[
\langle f(\mathbf{r}) \rangle = \int_{\Omega} f(\mathbf{r}') \delta(\mathbf{r} - \mathbf{r}') \, d\mathbf{r}',
\]

where \( \Omega \) is the integration domain; \( \mathbf{r} \) is the space vector of an arbitrary point; and \( \delta \) is the Dirac delta function, which is defined as

\[
\delta(\mathbf{r} - \mathbf{r}') = \begin{cases} 
\infty, & \mathbf{r} = \mathbf{r}' \\
0, & \mathbf{r} \neq \mathbf{r}'.
\end{cases}
\]

The above-mentioned delta function is approximated by a weighting function known as the smoothing function or kernel function in the SPH method. Thus, the field function \( f(\mathbf{r}) \) can be written as

\[
\langle f(\mathbf{r}) \rangle = \int_{\Omega} f(\mathbf{r}') W(\mathbf{r} - \mathbf{r}', h) \, d\mathbf{r}',
\]

where \( h \) denotes the smoothing length and the angle brackets represent the kernel approximation. The smoothing function should satisfy certain mathematical requirements, which were discussed in detail by Liu and Liu (2003). A cubic spline smoothing function (Liu and Liu 2003) is used in this paper. By applying the particle approximation to Eq. (3), the integral expression for particle \( i \) can be written as

\[
\langle f(\mathbf{r}_i) \rangle = \sum_{j=1}^{N} \frac{m_i}{\rho_i} f(\mathbf{r}_j) \cdot W_{ij},
\]

The derivative for \( f(\mathbf{r}_i) \) can also be written as

\[
\langle \nabla f(\mathbf{r}_i) \rangle = \sum_{j=1}^{N} \frac{m_i}{\rho_i} [f(\mathbf{r}_i) - f(\mathbf{r}_j)] \cdot \nabla_i W_{ij}. 
\]

Equation (6) is more accurate than Eq. (5) for certain situations (Monaghan 1985). The reader is referred to Danis et al. (2013) for more discussion on mathematical operators for the SPH method.

b. SPH-based SWE model

By neglecting the horizontal eddy mixing terms for simplicity, the Lagrangian form of the shallow-water equation (SWE) can be written as

\[
\frac{Du}{Dt} = -g(\nabla h + \nabla b) + S_C + S_f,
\]

where \( \frac{D}{Dt} \) denotes the total derivative with respect to time; and \( d, b, \) and \( \mathbf{u} \) represents the total water depth, seabed elevation, and depth-mean horizontal velocity vector, respectively (Fig. 1). In Eq. (8), \( S_C \) represents the Coriolis term, \( S_C = (\mathbf{f}_\theta, -\mathbf{f}_u) \), where \( f \) is the Coriolis parameter. For simplicity, the effect of the earth’s rotation on the water movement is neglected in this study.
In Eq. (8) $S_f$ indicates the contribution of the bed friction, expressed as
$$S_f = -g \frac{n^2 u_i |u_i|}{d^{4/3}},$$
where $n$ is the Manning coefficient. By discretizing the gradient terms $\nabla d$ and $\nabla b$ in Eq. (8) using Eqs. (5) and (6), we yield the following expression for the acceleration of particle $i$ (the water column in Fig. 1):
$$\frac{D'u_i}{Dt} = -g \sum_{j=1}^{N} V_j \frac{\nabla W_{ij}}{d} - g \sum_{j=1}^{N} \frac{\nabla W_{ij}}{d} + \frac{n^2 u_i |u_i|}{d^{4/3}} - \frac{\sum_{j=1}^{N} V_j \Pi_{ij} \nabla_i W_{ij}}{d} - g \frac{n^2 u_i |u_i|}{d^{4/3}},$$
where $V_j$ is the volume of neighboring particle $j$. Note that $V_j/d_j$ in the SPH-based SWE model shares a similar meaning with $m_i/\rho_i$ in the SPH-based Reynolds-averaged Navier–Stokes equations (RANSE) model. Vacondio (2010) drew an analogy between $V_j$ and $m_i$, $d_j$, and $\rho_i$ in his SPH-based SWE model. In this paper, however, we use variables from the SWE to prevent any confusion in physical meanings. The artificial viscosity term $\Pi_{ij}$ suggested by Monaghan (1997) is used to eliminate the numerical oscillation as follows:
$$\Pi_{ij} = \left\{ \begin{array}{ll} -\alpha \mu_i (\vec{c}_{ij} - 2\mu_i) \quad & \text{if } \mathbf{u}_{ij} \cdot \mathbf{r}_{ij} < 0 \\ 0 \quad & \text{otherwise} \end{array} \right.,$$
where $\mathbf{u}_{ij}$ and $\mathbf{r}_{ij}$ are the relative velocity vector and relative space vector, respectively; and $\mathbf{u}_{ij} = \mathbf{u}_i - \mathbf{u}_j$ and $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$. In addition, $\bar{c}_i$ and $\bar{d}_i$ denote the average phase speed of the SWE and the average water depth, respectively, which are defined as $\bar{c}_i = (c_i + c_j)/2$ and $\bar{d}_i = (d_i + d_j)/2$. The phase speed is calculated by $c_i = \sqrt{g d_i}$ and $\mu_i = h_{ij} \mathbf{u}_{ij} \cdot \mathbf{r}_{ij} / r_{ij}^2$, where $h_{ij}$ is the average smoothing length defined as $h_{ij} = (h_i + h_j)/2$. In Eq. (11) $\alpha$ represents the artificial viscosity coefficient and is set to 1.0 in this paper.

The total water depth in the SPH-based SWE model is calculated based on
$$d_i = \sum_{j=1}^{N} V_j W_{ij},$$
According to the conservation of mass and the assumption of incompressibility, each particle maintains its volume throughout its lifetime. Thus, the smoothing length reduces when the water depth increases and vice versa. A variable smoothing length is usually adopted,
$$h_i = h_0 \left( \frac{d_0}{d_i} \right)^{1/D_m},$$
where $h_0$ and $d_0$ are the initial smoothing length and the initial total water depth for particle $i$, respectively; and $D_m$ denotes the system dimension. In this paper, $h_0$ is set to 1.2$\Delta x$, where $\Delta x$ is the initial space between the particles. Since the smoothing length $h_i$ is used in the smoothing function $W_{ij}$ for the water depth $d_i$ defined in Eq. (12) and $d_i$ appears on the right-hand side of Eq. (13), an iterative procedure based on these two equations can be used to obtain $d_i$ and $h_i$.

Equation (12) is an unbalanced equation that can cause large errors in the water depth calculation in the presence of steep topography. In the SPH-based SWE
model used in this study, a modified formula suggested by Xia et al. (2013) is used,

$$d_i = \sum_{j=1}^{N} V_j W_j - \sum_{j=1}^{N} \frac{1}{d_j} (b_i - b_j) W_j.$$  \hspace{1cm} (14)

The particle’s bed elevation $b_i$ is calculated using the approach suggested by Vacondio et al. (2012b), in which a set of fixed bed particles are distributed in a Cartesian grid and the bed elevation of particle $i$ is interpolated using the Shepard filter (Shepard 1968; Randles and Libersky 1996) from the bed particles in its supporting domain, defined as

$$b_i = \frac{1}{\kappa_i} \sum_{j=1}^{N_b} \omega^b_j h^b_j W(r_i - r^b_j, h^b_j),$$  \hspace{1cm} (15)

where $N_b$ is the bed particle number in the supporting domain; and $h^b_j, h^b_i$, and $\omega^b_j$ denote the bed elevation, the smoothing length, and the occupied space (area for 2D and length for 1D) of bed particle $j$, respectively. The supporting domain mentioned above, in the 2D case, is an area of a circle with the smoothing radius. Inside the supporting domain, all particles contribute to the integration of the central particle based on the smoothing function (or weighting function). The influence of the outside particles is small and ignored.

In Eq. (15), $\kappa_i$ is the modification coefficient of the smoothing function, given as

$$\kappa_i = \frac{1}{\sum_{j=1}^{N_b} \omega^b_j W(r_i - r^b_j, h^b_j)}.$$  \hspace{1cm} (16)

The predictor–corrector scheme with second-order accuracy in time iteration is used in this paper, and the variable time step is given by

$$\Delta t = \frac{h_i}{C_{\text{CFL}}} \times \min_{i=1}^{N} \left\{ \frac{h_i}{\max(|u_i|, \sqrt{gd_i})} \right\},$$  \hspace{1cm} (17)

where $C_{\text{CFL}}$ is the Courant–Friedrichs–Lewy number and is set to 0.2–0.3 (Xia et al. 2013).

3. Solid boundary treatment

a. Modified dynamic solid boundary condition

The MDSBT suggested by Ni et al. (2014) is used in this study to prevent the unphysical penetration of fluid particles into solid boundaries. This treatment also improves the accuracy of the solid boundary pressure calculations. The MDSBT requires two levels of solid boundary particles deployed along the solid boundary in the model initialization. The solid boundary particles remain at rest with zero velocities during the integration of the model. These solid boundary particles are, however, involved in the calculation of the fluid particle acceleration and the iteration of the local water depth and smoothing length to avoid the truncation of the smoothing region during the fluid–boundary interaction. The number of solid boundary levels is determined by the smoothing radius of the fluid particle. As mentioned earlier, the smoothing radius in this study is equal to twice that of the smoothing length and the smoothing length is 1.2 times the initial particle space. Thus, two layers of solid boundary particles are thick enough to fulfill the supporting domain with neighboring particles for any fluid particle close to the solid boundary. The water depth of the solid boundary $i$ is estimated using fluid particles in its supporting domain,

$$d_i^{\text{SB}} = \frac{\sum_{j=1}^{N_f} V_j W_j}{\sum_{j=1}^{N_f} d_j W_j}.$$  \hspace{1cm} (18)

where $N_f, V_j$, and $d_j$ denote the number, volume, and depth of fluid particles in the supporting domain of solid particle $i$, respectively. The Shepard filter is also used to keep the depths of solid boundary particles consistent with the depths of the fluid particles nearby.

To prevent fluid particles from penetrating solid boundaries, the velocities of near-boundary fluid particles are modified by a boundary layer scheme after updating the water depth of solid boundary particles based on Eq. (18). Once the fluid particle enters the boundary layer between the solid boundary and the fluid, the particle will be rebounded if its velocity vector points toward the boundary ($u \cdot r_{i\perp} > 0$) (Fig. 2). If the fluid particle moves away from the boundary ($u \cdot r_{i\perp} < 0$), its velocity remains unchanged. The corrected tangential and normal velocities of fluid particle are given as

$$u_{i\parallel}^\text{cor} = \begin{cases} (1 - \alpha_{i\parallel}) \cdot u_{i\parallel}, & \text{if } u_{i\parallel} \cdot r_{i\perp} > 0, \\ u_{i\parallel}, & \text{if } u_{i\parallel} \cdot r_{i\perp} \leq 0 \end{cases},$$  \hspace{1cm} (19)

$$u_{i\perp}^\text{cor} = \begin{cases} -\alpha_{i\perp} \cdot u_{i\perp}, & \text{if } u_{i\perp} \cdot r_{i\parallel} > 0, \\ u_{i\perp}, & \text{if } u_{i\perp} \cdot r_{i\parallel} \leq 0 \end{cases},$$  \hspace{1cm} (20)

where $u_{i\parallel}$ and $u_{i\parallel}^\text{cor}$ denote tangential velocities of fluid particles before and after the rebound, respectively; and $u_{i\perp}$ and $u_{i\perp}^\text{cor}$ represent normal velocities of fluid particles before and after the rebound, respectively. The boundary friction coefficient $\alpha_{i\parallel}$ and the boundary elasticity coefficient $\alpha_{i\perp}$ vary from 0 to 1. The relative position between a fluid particle and the solid boundary is determined by integrating the relative spatial vectors $r_{ij}$.
between this particle and all of the solid boundary particles in its supporting domain, and the unit normal vector \( \mathbf{r}_{i\perp} \) can be written as

\[
\mathbf{r}_{i\perp} = \frac{\sum_{j=1}^{N_{\text{sn}}} \mathbf{r}_{ij} W_{ij}}{\left| \sum_{j=1}^{N_{\text{sn}}} \mathbf{r}_{ij} W_{ij} \right|}.
\]

(b. Solid boundary test case: Still water in a tank with an acute angle)

The performance of the MDSBT is assessed in a test case of still water in a tank suggested by Vacondio et al. (2012a). This simple test case was used previously by Vacondio et al. (2012a) in assessing the zero consistency of the modified virtual boundary particle (MVBP) method. The water tank has a flat bottom, with the horizontal dimension of 1.0 m \( \times \) 1.0 m (Fig. 3a). The tank has a piecewise straight coastline, with an internal angle of 300° at the middle point of the southern coast. The SPH-based SWE model in this test case is initialized with 1544 fluid particles distributed in the horizontal Cartesian grid with a particle space of 2.5 cm. A total of 358 solid boundary particles are arranged in two levels along the solid boundaries. The initial water depths and initial velocities of all particles are set to be 1.0 m and 0 m s\(^{-1}\), respectively. Since the grid-distributed fluid particles do not fit the oblique solid boundary initially, the water body in the tank will adapt to the angle quickly. The distribution of the fluid particles is dynamically rearranged after 1 s of the model integration (Fig. 3b). The free-surface elevation and the velocity magnitude at \( t = 1 \) s are given in Fig. 4a and 4b. The maximum velocity produced by the model is less than 0.9% \( \sqrt{gd_{\text{max}}} \), which is similar to the result obtained...
using the MVBP method (Vacondio et al. 2012a). Figure 4b demonstrates that the maximum velocity occurs near the straight boundary rather than at the acute angle. This indicates that the fluid motion in this test case is mainly caused by the self-adaption of fluid particles, and the high velocity region in Fig. 4b presents the advection residual. Given a better distribution of the fluid particles, the self-adaption motion will be reduced or eliminated. Although the MDSBT has the same order of accuracy as the MVBP method in this test case, the solid boundary particles in the MDSBT are fixed in time. As a result, there is no need to reallocate and calculate corresponding virtual particles at every model time step, which is the main advantage of the MDSBT over the MVBP method.

4. Open boundary treatment

Several open boundary conditions were suggested for the SPH method (Cleary 1998; Lastiwka et al. 2009; Gomez-Gesteira et al. 2012a; Vacondio et al. 2012b; Vacondio et al. 2013; Ni and Feng 2013; Molteni et al. 2013). Among these OBCs, the characteristic open boundary condition was widely used previously in the SPH-based models for simulating general free-surface flow, ranging from subcritical flow to supercritical flow. A brief review of this open boundary condition is given as follows.

a. Characteristic open boundary condition

In the COBC (Vacondio et al. 2012b), buffer zones are specified at the inflow and outflow boundaries (Fig. 5). When an open boundary particle (OBP) from the inflow buffer zone enters into the fluid zone, the particle turns into a normal fluid particle (the open dashed circle in Fig. 5). At the same time, a new OBP (the solid dashed circle in Fig. 5) is inserted into the upstream side of the inflow buffer zone to ensure enough OBPs over the inflow buffer zone. When a normal fluid particle enters the outflow buffer zone, the particle turns into an OBP. When an OBP goes into the outside of the computational domain, it will be deleted from the memory. During their lifetime, the OBPs are used in the calculation of normal fluid particles’ acceleration and the iterative process in computing the local water depth and smoothing length, in order to pass dynamic information into the fluid domain and to avoid the truncation of the smoothing regions. The total water depths and velocities of OBPs are updated according to the strategies defined by Eqs. (A1)–(A5) in the appendix.

b. Open boundary test case A: Steady flow over a bump

The performance of the COBC is assessed in the case of a steady flow over a bump in a channel with inflow and outflow open boundaries (Table 1). For simplicity, the bottom friction term in the SWE is ignored. A one-dimensional (1D) frictionless channel with the length of 10 m is used, in which the bed elevation is specified as

\[ b(x) = \begin{cases} b_0[1 - 0.25(x - 5)^2] & x \in [3, 7] \text{ m} \\ 0 & \text{elsewhere} \end{cases} \]

where the maximum height of the bump \( b_0 \) is set to 0.2 m. This is a classic benchmark test case for the SWE models (Greenberg and Leroux 1996; Aureli et al. 2008; Chang and Chang 2013).

Four numerical runs are carried out in this test case, including two transcritical flows, one supercritical flow, and one subcritical flow. The inflow and outflow conditions in each run are listed in Table 2. The flow conditions and analytical solutions for the first three runs (runs A1–A3) are taken from the paper of Vacondio et al. (2012b). However, Vacondio et al. did not present...
their results in the fourth run (run A4) of subcritical flow in their paper. As a result, the flow conditions in this test case are set separately and its analytical solution (Delestre et al. 2013) is calculated based on

\[
d(x)^3 + \left[ b(x) - \frac{g_0}{2gd_L} - d_L \right] d(x)^2 + \frac{g_0^2}{2g} = 0, \quad (23)
\]

where \( x \in [0, L] \), \( d_L = d(x = L) \), with \( L = 10 \) m, \( d_L = 0.4 \) m, and flow discharge \( q_0 = 0.12 \) m\(^2\) s\(^{-1}\). The initial particle space in all four runs is set to 2 cm.

The numerical results of the free-surface elevation, the depth-mean horizontal velocity, and the discharge produced with the use of COBC are compared with the corresponding analytical solutions in the four runs (Fig. 6). In the first three runs (runs A1–A3), the simulated flows by the SPH-based SWE model with the COBC reach a steady state very quickly. The SPH-based SWE model with the COBC, however, fails to reach a steady state of the subcritical flow in the fourth run (run A4). The initial velocity and the initial free-surface elevation in run d are set to be 0.3 m s\(^{-1}\) and 0.4 m, respectively. The sea surface is still in an oscillation state even after running for a long time (\( t = 600 \) s).

To quantify the performance of the COBC, the normalized model error \( L_2(\Phi) \) is computed using

\[
L_2(\Phi) = \frac{1}{N} \sum_{i=1}^{N} \left( \frac{\Phi_i - \Phi_i^e}{\Phi_i^{ND}} \right)^2,
\]

where \( \Phi \) denotes the depth-mean current \( (u) \) or the water depth \( (d) \); \( \Phi_i \) and \( \Phi_i^e \) denote the numerical and the analytical solutions of the \( i \)th particle, respectively; and \( \Phi_i^{ND} \) presents the normalization factor, defined as \( \Phi_i^{ND} = d_i^e \) for \( L_2(d) \) and \( \Phi_i^{ND} = \sqrt{gd_i^e} \) for \( L_2(u) \). Figure 7 presents time series of normalized errors for the total water depth and depth-mean currents using the COBC in the run of subcritical flow (run A4). Because of the bump in the middle of the channel, some differences

<table>
<thead>
<tr>
<th>Case</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Steady flow over a bump in a channel with two open ends</td>
</tr>
<tr>
<td>B</td>
<td>Propagation of sea level perturbations in a flat-bottom channel with two open ends. There is a steady subcritical flow in the channel.</td>
</tr>
<tr>
<td>C</td>
<td>Propagation of sea level perturbations in a flat-bottom wave tank with one open end. There is no steady flow in the channel.</td>
</tr>
<tr>
<td>D</td>
<td>Vortex shedding behind a cylinder in a channel with two open ends. There is an inflow entering the channel from one open end and outflow exiting from the other open end.</td>
</tr>
</tbody>
</table>
occur between the initial condition and the steady state. These differences result in initial (physical or computational) perturbations in the sea surface and the depth-mean current. The use of the COBC does not allow the energy of model-generated perturbations to propagate out through the model open boundaries in run A4. As a result, the perturbations reflect between the inflow and outflow zones and are accumulated inside the system.

Table 2. Inflow and outflow boundary conditions used in four runs for the steady flow over a bump in test case A.

<table>
<thead>
<tr>
<th>Run</th>
<th>Inflow boundary conditions</th>
<th>Outflow boundary conditions</th>
<th>Flow type</th>
</tr>
</thead>
<tbody>
<tr>
<td>A1</td>
<td>$u_B = 0.435 \text{ m s}^{-1}$</td>
<td>$d_B = 0.330 \text{ m}$</td>
<td>Transcritical</td>
</tr>
<tr>
<td>A2</td>
<td>$u_B = 0.435 \text{ m s}^{-1}$</td>
<td>$d_B = 0.100 \text{ m}$</td>
<td>Transcritical</td>
</tr>
<tr>
<td>A3</td>
<td>$u_B = 4.000 \text{ m s}^{-1}$</td>
<td>$d_B = 0.100 \text{ m}$</td>
<td>Supercritical</td>
</tr>
<tr>
<td>A4</td>
<td>$u_B = 0.300 \text{ m s}^{-1}$, $d_B = 0.400 \text{ m}$</td>
<td>$u_B = 0.300 \text{ m s}^{-1}$, $d_B = 0.400 \text{ m}$</td>
<td>Subcritical</td>
</tr>
</tbody>
</table>

Fig. 6. Comparisons between numerical results produced by the SPH-based SWE model and analytical solutions for a steady flow over a bump (test case A, Table 1) in four model runs (from top to bottom): (A1) transcritical flow with a hydraulic jump, (A2) transcritical flow without a hydraulic jump, (A3) supercritical flow, and (A4) subcritical flow. The panels in each row present validations of the (left) free-surface elevation, (center) depth-mean velocity, and (right) discharge, respectively. Numerical results are produced by the SWE-SPH method with the conventional COBC in runs A1–A3 and the AOBC in run A4. The model results with the conventional COBC fail to converge in run A4.
leading to the final failure of the simulation using the COBC.

Note that the model-generated perturbations also occur in runs A1–A3. The outflow conditions in runs A2 and A3 are supercritical outflows, where the model-generated perturbations cannot propagate upstream. Instead, the perturbations are flushed out from the downstream open boundaries immediately and the numerical simulation converges to a steady flow. In run A1, there is a hydraulic jump in the downstream of the bump that keeps dissipating kinetic energy. The initial and subsequent perturbations are dissipated when passing through this area. Thus, the convergence is achieved eventually. In run A4, where the COBCs is replaced by the radiation open boundary condition to be described in section 5, the computational perturbations pass through the upstream and downstream open boundaries as freely as possible.

c. Open boundary test case B: Propagation of sea level perturbation in a flat-bottom channel with two open ends

To further demonstrate the reflection problem of the COBC, we consider a test case of propagations of sea level perturbations in a 1D flat-bottom channel along the $x$ axis (i.e., test case B in Table 1). The 1D channel is 10 m long ($L$) with open boundaries at two ends ($x = 0$ and $x = L$). The initial water depth $d_0$ is 0.5 m. There is a steady subcritical flow in the channel. The initial (physical) perturbation in the sea level is set to 0.2 m above mean sea level at the middle point of the channel ($x = L/2$, Fig. 8a). The initial velocity $u_0 = 1.0 \text{ m s}^{-1}$ and the initial particle space for the SPH method is set to 0.02 m. Physically, the initial sea level perturbation in this flat-bottom channel separates into two external gravity waves after the initialization, with one wave propagating upstream (in the negative direction of the $x$ axis) and another propagating downstream (in the positive direction). Figures 8b and 8c present the model-calculated sea surface elevations in the channel at three different times after the model initialization. The SPH-based SWE model with the COBC reproduces reasonably well two oppositely propagating waves in the sea level after the initialization and before the downstream wave reaches the outflow open boundary (Fig. 8b). The model also reproduces very well the faster propagation of the downstream-propagating wave and the slower propagation of the upstream-propagating wave due to the effect of steady currents in the channel. Because of the use of the COBC, however, a wave is reflected once the downstream-propagating wave reaches the outflow open boundary and then starts to propagate upstream (Fig. 8c). Another reflected wave also appears when the initial upstream-propagating wave arrives at the inflow boundary on the left (Fig. 8d). These two reflected waves are significantly different in shape and size, because of different treatments for the subcritical inflow and outflow in the COBC.

Figure 9 presents the time evolution of model-calculated free-surface elevations in this flat-bottom channel produced by the SPH-based SWE model with the COBC. The model results exhibit clearly the separation of two oppositely propagating waves after the initialization, and multiple reflections of waves from the inflow and downstream open boundaries. The multiple wave reflections from the open boundaries indicate that the COBC performs poorly in this simple test case.

5. New radiation open boundary conditions for SPH-based model

Several open boundary conditions were developed for the mesh-based ocean circulation models (Raymond and Kuo 1984; Chapman 1985; Marchesiello et al. 2001). Among them the radiation open boundary condition
(ROBC) based on the Sommerfeld equation (Sommerfeld 1949) is the most popular. The main idea of the Sommerfeld radiation condition is that perturbations generated inside the model interior are allowed to propagate through the open boundaries in the form of advective waves. Inspired by the success of the Sommerfeld radiation open boundary conditions for the mesh-based models, we introduce two new open boundary conditions for the SPH-based model: the AOBC and the FOBC.

a. AOBC

For simplicity, we consider only the 1D radiation equation, written as (Chapman 1985)

$$\frac{\partial \Phi}{\partial t} + c \frac{\partial \Phi}{\partial x} = 0,$$

where $\Phi$ denotes characteristic variables, such as the depth-mean current $u$ or the water depth $d$; and $c$ is the phase speed of the characteristic variable. The explicit updating equation for the characteristic variables of the OBPs in the SPH models can be written as

$$\Phi_B^{k+1} = \Phi_B^k - \frac{c \Delta t}{\Delta x} (\Phi_B^k - \Phi_I^k),$$

where $\Phi_B^k$ and $\Phi_B^{k+1}$ denote characteristic variables of OBPs (marked as subscript $B$) at model time steps $k$ and $(k + 1)$, respectively; $\Phi_I^k$ is the characteristic variable of an internal fluid particle near the open boundary at the $k$th time step and can be interpolated using Eq. (21); $\Delta t$ is the time step; and $\Delta x$ is the distance between the OBP and the interpolation point of internal fluid particle $I$.

It should be noted that the open boundary condition described in Eq. (26) is a pure radiation (or passive) open boundary condition, through which the external information cannot be specified at the model open boundary.
Marchesiello et al. (2001) suggested an adaptive radiation equation with a nudging term added to the right-hand side (rhs) of Eq. (25),
\[ \frac{\partial \Phi}{\partial t} + c \frac{\partial \Phi}{\partial x} = -\frac{1}{\tau} (\Phi - \Phi^{\text{ext}}), \] (27)
where \( \tau = \tau_{\text{out}} \) when the external information propagates toward the open boundary, \( \tau = \tau_{\text{in}} \) when the external information propagates away from the open boundary, and \( \tau_{\text{out}} \gg \tau_{\text{in}} \). By adjusting the relaxing parameter \( \tau \) according to the direction of the phase speed, the state of activity (transmitting external information into the model domain) and passivity (transmitting internal perturbation to the outside) can be switched smoothly. The explicit updating equation for Eq. (27) can be expressed as
\[ \Phi_{B}^{k+1} = \Phi_{B}^{k} - \frac{c}{\Delta x} (\Phi_{B}^{k} - \Phi_{I}^{k}) - \frac{\Delta t}{\tau} (\Phi_{B}^{k} - \Phi^{\text{ext}}). \] (28)
As indicated in Marchesiello et al. (2001), the success of the above-mentioned AOBC depends on the accurate estimation of the phase speed, which is calculated from characteristic variables of fluid particles close to the open boundary based on
\[ c = -\frac{\partial \Phi/\partial t}{\partial \Phi/\partial x}. \] (29)

The 1D adaptive open boundary condition discussed above can be extended to the 2D or 3D cases (Raymond and Kuo 1984).

b. FOBC

Flather (1976) suggested a different type of open boundary condition based on the 1D external gravity wave,
\[ \mathbf{n}_{n} = \mathbf{n}_{n}^{\text{ext}} \pm \sqrt{g} \mathbf{d} (\eta - \eta^{\text{ext}}), \] (30)
where superscript “ext” denotes external information and the subscript \( n \) denotes the normal component. Equation (30) contains the external input terms of the free-surface elevation and the velocity, and it can achieve activity and passivity similar to the above-mentioned AOBC. The FOBC is very easy to implement and is computationally more efficient than the AOBC. Also note that the FOBC uses the phase speed of the external gravity wave. Thus, the FOBC works well on the condition that the gravity wave is dominant. Moreover, both the external information \( u^{\text{ext}} \) and \( d^{\text{ext}} \) need to be prescribed in the FOBC, and the velocity should be normal to the open boundary. The inflow and outflow conditions for subcritical flow are given as
\[ \begin{cases} u_{B,n} = u^{\text{ext}} \pm \sqrt{g} (d_i - d^{\text{ext}}), & u_{B,t} = 0, \\ d_{B} = d^{\text{ext}}. \end{cases} \] (31)
c. Open boundary test case C: Wave propagation in a channel with one open end

The performance of these three open boundary treatments—COBC, AOBC, and FOBC—is assessed in this benchmark test case for wave propagation and absorption in a flat-bottom wave tank (Clément 1999) suggested by the International Society of Offshore and Polar Engineers (ISOPE; test case C in Table 1). The initial setup of the wave tank (or channel) in this test case is shown in Fig. 10a, with the water depth \( h = 0.4 \) m and the length of the main channel \( L = 10h = 4 \) m. A sponge layer, which is also to be tested, appears on the right side of the channel. The initial condition of this test case is zero currents everywhere, with a free-surface bump on the left side of the channel, defined as
\[ \eta = 0.4 \exp \left[ -\left( \frac{x}{h} \right)^{4} \right]. \] (32)
This initial sea level bump has a broadband spectrum, resulting in waves of continuously varying frequencies at the output (Clément 1999). After the model initialization, some of the potential energy of the bump is transformed into the kinetic energy and the bump propagates to the right in terms of waves. Both the potential energy and kinetic energy of waves are significantly
dissipated as waves start to enter the sponge layer (Fig. 11a). In the various absorption tests reported by Clément (1999), it took more than $T = 20.0$ to reduce the total energy to zero, where $T$ is the normalized time defined as $T = t/\sqrt{h/g}$ and $t$ is the model time.

For numerical experiments using the SPH-based SWE model with three open boundary conditions, the sponge layer is replaced by an open boundary at the right edge of the main channel (Fig. 10b). The MDSBT for the SPH method suggested by Ni et al. (2014) is used for the solid boundary on the left of the tank. Figure 11 presents the time evolution of nondimensional kinetic energy $E_k$, potential energy $E_p$, and total energy $E_T$ calculated from model results produced by the SPH-based SWE model using three types of open boundary conditions (COBC, AOBC, and FOBC). The model results with the use of the sponge layer (Fig. 11a) are compared with model results using the above-mentioned three open boundary conditions.

Figures 11c and 11d demonstrates that the right edge of the sea surface bump reaches the open boundary at the nondimensional time $T \approx 5.5$ and that the total energy decreases with time very quickly and reaches zero after $T > 7$ with the use of the AOBC and FOBC. By comparison, only 22% of the total energy is transmitted with the use of the COBC (Fig. 11b) and the rest is reflected back to the model interior (the left part of the wave tank). Figure 11a shows that the sponge layer has difficulty in dealing with long waves. By comparison, both the AOBC (Fig. 11c) and the FOBC (Fig. 11d) perform much better than the COBC and sponge layer. The total energy decreases gradually with time for $T < 5$, indicating the model is numerically dissipative, which is one of outstanding issues for the SPH method (Liu and Liu 2010; De Leffe et al. 2010).

d. Performance of AOBC and FOBC in the open boundary test case A

We first assess the performance of two new open boundaries (AOBC and FOBC) in test case A, which is a steady flow over a bump in a channel with two open ends (Table 1). The unique feature of this test case is the presence of external information input. As discussed in section 4b, the SPH-based SWE model with the use of the COBC performs reasonably well in experiments of supercritical and transcritical flow, but it fails to converge in the experiment of subcritical flow. By comparison, both the AOBC (Fig. 6d) and FOBC (not shown in Fig. 6) work reasonably well. Figure 12 presents time series of the nondimensional errors $L_2(d)$ and $L_2(u)$ in five different numerical experiments with different inflow and outflow boundary conditions based on the COBC, AOBC, and FOBC. Among these five runs, the model run using the FOBC as its inflow and outflow conditions (green curve) converges much faster than other runs. By comparison, the model run using the AOBC as its outflow
boundary condition (blue curve) has the largest errors—about 10 times of the other cases. The main reason is that the Sommerfeld radiation equation does not guarantee the volume conservation, which may lead to a gradual model drift in the mean sea level (Perkins et al. 1997).

Note that the model run using the COBC for both the inflow and outflow conditions fails to converge. In addition, the three runs using the AOBC as their inflow conditions also fail to converge. The main reason is that the phase speed of characteristic variables calculated from fluid particles for the AOBC is not accurate enough to manage the switching time for active and passive open boundaries, resulting in a conflict between the external input and the internal characteristic variables. The oscillation at the inflow open boundary grows larger and larger until the model blows up. However, the rest of the five model runs converge to a steady state in a very short time with satisfactory agreement with the analytical solutions. The convergence values of $L_2(d)$ and $L_2(u)$ in different model runs are listed in Tables 3 and 4.
e. Performance of AOBC and FOBC in the open boundary test case B

We also assess the performance of the two new open boundary conditions in test case B discussed in section 4c. This test case has a steady subcritical flow in a flat-bottom channel with two open ends (Table 1). An initial sea level perturbation of 0.2 m occurs at the middle point of the channel. Figure 13 presents the simulated sea surface elevations produced by the SPH-based SWE model with the use of the AOBC and FOBC as the inflow and outflow open boundary conditions. In comparison with the model results with the COBC shown in Fig. 10, the reflections by the inflow and outflow open boundaries are negligible for both the FOBC and AOBC. At the outflow boundary, the AOBC gives a slightly better result than the FOBC. The reason is that the ratio between the steady flow velocity and the phase speed of the external gravity wave in the still water \( u_0/\sqrt{gd_0} \approx 0.45 \), indicating that the free-surface gravity wave is no longer dominant in this case. But in the governing equation for the FOBC, the phase speed of the external gravity wave \( \sqrt{gd} \) is used to replace the actual phase speed of the perturbation \( \sqrt{gd \pm u_0} \). Nevertheless, the performance of the FOBC is still satisfactory.

f. Open boundary test case D: Vortex shedding behind a cylinder in a free-surface flow

Vortex shedding behind a cylinder is a classical hydrodynamic problem with many practical applications (Williamson 1996). There is renewed research interest in simulating vortex shedding using the SPH model (Lastiwka et al. 2009; Vacondio et al. 2013; Colagrossi et al. 2014) and the Godunov-type finite volume method

---

**Table 3.** The convergence values of \( L_2(d) \) in test case A of steady subcritical flow over a bump using various OBC combinations.

<table>
<thead>
<tr>
<th>Outflow</th>
<th>Inflow</th>
<th>COBC</th>
<th>FOBC</th>
<th>AOBC</th>
</tr>
</thead>
<tbody>
<tr>
<td>COBC</td>
<td>Nonconvergence</td>
<td>1.94 \times 10^{-3}</td>
<td>1.53 \times 10^{-3}</td>
<td></td>
</tr>
<tr>
<td>FOBC</td>
<td>2.07 \times 10^{-3}</td>
<td>1.77 \times 10^{-3}</td>
<td>4.10 \times 10^{-3}</td>
<td></td>
</tr>
<tr>
<td>AOBC</td>
<td>Blow up</td>
<td>Blow up</td>
<td>Blow up</td>
<td></td>
</tr>
</tbody>
</table>

**Table 4.** The convergence values of \( L_2(u) \) in test case A of steady subcritical flow over a bump using various OBC combinations.

<table>
<thead>
<tr>
<th>Outflow</th>
<th>Inflow</th>
<th>COBC</th>
<th>FOBC</th>
<th>AOBC</th>
</tr>
</thead>
<tbody>
<tr>
<td>COBC</td>
<td>Nonconvergence</td>
<td>6.12 \times 10^{-3}</td>
<td>5.83 \times 10^{-4}</td>
<td></td>
</tr>
<tr>
<td>FOBC</td>
<td>1.60 \times 10^{-3}</td>
<td>1.07 \times 10^{-3}</td>
<td>3.49 \times 10^{-3}</td>
<td></td>
</tr>
<tr>
<td>AOBC</td>
<td>Blow up</td>
<td>Blow up</td>
<td>Blow up</td>
<td></td>
</tr>
</tbody>
</table>
In this section we assess the performance of the SPH-based SWE model with the use of the AOBC or FOBC in simulating vortex shedding behind a cylinder in a free-surface flow using the configuration suggested by Fujihara and Borthwick (2000). In this configuration, a cylinder with a diameter \(D = 2.0\) m is placed at \(5D\) (10 m) from the upstream boundary at the longitudinal axis of a straight and flat-bottom channel (Fig. 14; test case D in Table 1). The channel is \(8D\) wide and \(20D\) long, with a depth-mean flow coming in from the left (inflow) open boundary and going out from the right (outflow) open boundary. At the inflow and outflow open boundaries, the water depth and depth-mean flow are set to \(d_0 = 0.10\) m and \(u_0 = 0.5\) m s\(^{-1}\), respectively. The kinematic viscosity coefficient \(\nu\) is set to 0.005 m\(^2\) s\(^{-1}\), and the Reynolds number \(Re = u_0D/\nu = 200\). The MDSBT is used to set a slip boundary for the lateral boundaries of the channel and a nonslip boundary for the cylinder. The fluid particles are initially distributed in a Cartesian grid with the particle space of 0.1 m. The initial water depth and velocity equal 0.10 m and 0.5 m s\(^{-1}\), respectively.

The SPH-based SWE model is integrated first using the COBC at the inflow and outflow open boundaries. After the flow reaches a regular oscillatory pattern, the simulated free surface and the velocity near the cylinder evolve periodically, which are consistent with previous findings (Fujihara and Borthwick 2000). The vortexes shed the cylinder and propagate downstream sequentially as shown in Fig. 15. Because of the pure Lagrangian feature of the SPH method, the positions of particles at any time are readily available without any additional particle tracking calculation. Figure 15a presents evolutions of fluid particles during the model integration period between 158 and 174.5 s. These particles are marked by red and blue alternatively when they are in the upstream side of the cylinder. During this period, a vortex is generated to the right of the cylinder and propagates downstream after detaching from the cylinder to form a Kármán vortex street.

It should be noted, however, that the Strouhal number \(St = f_vD/u_0 = 0.242\) in model results using the COBC is about 25% larger than the typical values discussed in the literature (\(St = 0.190\), Roshko 1961; \(St = 0.194\), Franke et al. 1990; \(St = 0.197\), Williamson 1996; \(St = 0.192\), Liu et al. 1998), where \(f_v\) is the vortex shedding frequency.
This indicates that the numerical results using the COBC overestimate the vortex shedding frequency. An examination of model results in this case indicates that both the simulated water depths close to the inflow open boundary and the flow velocity behind the cylinder are about 10% larger than the reference values. This indicates that the flow condition created by the conventional COBCs has a relatively larger Reynolds number than expected. In addition, the use of the COBC introduces significant reflections from the inflow and outflow open boundaries. Figure 16a presents the time-space distribution of simulated water depths and currents along the along-channel transect at $y = 3D$ in the case of the COBC, which undoubtedly affects the vortex shedding frequency and Kármán vortex street. This transect is away from the longitudinal axis of the channel and that the vortex shedding has limited influence on the circulation along this transect. Thus, the water depth and the velocity at this transect $y = 3D$ should be close to the theoretical reference values, that is, $d_0 = 0.10$ m and $u_0 = 0.5$ m s$^{-1}$, respectively.

In the next two additional experiments, the FOBC and AOBC are used for the inflow and outflow boundary conditions, respectively, in the first experiment (run FOBC–AOBC) and the FOBC is used for both the inflow and outflow open boundaries in the second experiment.
(FOBC–FOBC). The patterns of the Kármán vortex street are also well represented in both the FOBC–AOBC and FOBC–FOBC cases and are almost the same as that in the COBC–COBC case. Thus, the fluid particle evolutions of the two additional cases are not given in this paper. Figures 16b and 16c presents the time–space distributions of simulated water depths and currents at $y = 3D$ in these two experiments. The model results in the first experiment (run FOBC–AOBC) demonstrate that the use of the FOBC at the inflow open boundary reduces significantly the model-generated perturbations and keeps the water depth and the velocity close to the reference values. The use of the AOBC at the outflow open boundary also allows perturbations to propagate out, but there is a gradual drift in the mean water level in run FOBC–AOBC. The water depth is about 5% smaller than the reference value, and the velocity is 5% larger than the reference value in this experiment. The Strouhal number for the experiment FOBC–AOBC is about 0.225, which is still significantly larger than the typical values appearing in the literature.

The model performance in run FOBC–FOBC is the best among these three experiments in simulating the vortex shedding and the Kármán vortex street behind the cylinder, with very minor reflections from the inflow and outflow open boundaries (Fig. 16c). In this experiment, the Strouhal number $St = 0.204$, which is very close to the typical value in the literature.

6. Okushiri Tsunami simulation

To further demonstrate the feasibility and capability of the SPH-based SWE model with new open boundaries for simulating free-surface flows in more
complicated bathymetry than in the above four test cases, the SPH-based SWE model is applied to simulate the laboratory observations associated with the Okushiri Tsunami in 1993. This tsunami was triggered by a large earthquake that occurred at 1317:12 UTC 12 July 1993 in the Sea of Japan near the island of Hokkaido. The destructive tsunami hit the island of Okushiri with the extreme runup height of about 32 m measured near the

**FIG. 17.** Bathymetry of the laboratory experiment for simulating the 1993 Okushiri tsunami. Red dots mark the locations of three gauges.

**FIG. 18.** (a) Time series of water level elevations specified at the western open boundary ($x = 0.0\,\text{m}$), and (b)–(d) time series of observed and simulated water level elevations at three gauges located at (4.521 m, 1.196 m), (4.521 m, 1.696 m), and (4.521 m, 2.196 m), respectively.
village of Monai and caused heavy casualties. The Central Research Institute of Electric Power Industry later reproduced the Monai run-up using a 1/400-scale laboratory experiment. The experiment data are available at the website of the Third International Workshop on Long-Wave Runup Models (http://isec.nacse.org/workshop/2004_cornell/bmark2.html), including the bathymetry (Fig. 17), the water level elevation of the incoming wave at $x = 0.0$ m (Fig. 18a), and the water level elevations of the three gauges at (4.521, 1.196), (4.521, 1.696), and (4.521, 2.196). In this benchmark test, the tsunami wave propagates from left to right and rushes over a small island and a shoal. After running up on the coast, the wave is reflected back to the sea side. Because of the complex bathymetry and irregular coastline in this case, only a few numerical studies were made in the past in simulating this tsunami process using the Eulerian mesh-based SWE models (LeVeque and George 2008;
Delis et al. (2008) or the SPH–SWE model (Vacondio et al. 2012b). This is because a numerical model is required to handle the open and solid boundaries properly and track the wetting and drying fronts accurately in order to simulate successfully the laboratory observations of this tsunami.

A total of 85,299 fluid particles are initially arranged over a Cartesian grid with a separation distance of 1.4 cm. The MDSBT is applied at the eastern (x = 5.448 m), southern (y = 0.0 m), and northern (y = 3.402 m) closed boundary. The COBC (or FOBC) is applied at the western open boundary (x = 0.0 m) for the specification of tsunami waves (Fig. 18a). The Manning coefficient is set to 0.025 s m$^{-1/3}$ based on suggestions made in the literature. The numerical results using the SPH-based SWE model with the COBC and FOBC are compared with the observations made in the laboratory experiment.

Figures 18b–d present observed and simulated water level elevations at three gauges off the western coast of Okushiri. The model results produced by Delis et al. (2008) using state-of-the-art finite volume scheme are
also presented in the figure for comparison. The results shown in Fig. 18 indicate that the SPH-based SWE model is capable of simulating the tsunami process over the complex topography. As shown in Fig. 19, the SPH-based model reproduces the drawdown of water levels between the island and the coast before the arrival of the tsunami \((t = 10 \sim 12 \text{ s})\), the shockwave induced by the topography \((t = 12 \sim 16 \text{ s})\), and the run-up and the reflection of the tsunami wave \((t = 16 \sim 20 \text{ s})\). The maximum water level elevations at the gauges produced by our model are smaller than those measured from the laboratory experiment (Figs. 18b–d). Similar model results were also reported by Vaccondo et al. (2012b), which might be caused by the numerical dissipation of the SPH method. Nevertheless, considering that tedious algorithms had to be used in the mesh-based model to handle the shockwave propagation and to track the wetting and drying fronts in this case (Delis et al. 2008) and also considering that reasonable results are obtained by our SPH-based model without any special treatment, it can be argued that the SPH-based model has significant advantages over conventional mesh-based models when solving hydrodynamics in regions with complicated bathymetry and highly irregular coastline.

Model results shown in Figs. 18 and 19 are those produced by the SPH-based model using the FOBC before reflective waves reached the eastern open boundary \((at \ x = 0.0 \text{ m})\). During this period, the model results using the COBC are the same as those using the FOBC, as expected. Different characters between the COBC and FOBC, however, start to generate different effects on model results when the reflective waves reach the eastern open boundary. We extend the run time to 40 s and examine the model results at three transects at \(y = 1.0, 1.7, \text{ and } 2.5 \text{ m}\) to investigate the influence of different open boundary conditions on the numerical model results. Figure 20 shows that waves reflected from the island, the coast, and the lateral boundaries (arrows with solid lines) arrive at the eastern open boundary successively after \(t = 20 \text{ s}\). Multiple reflections (arrows with dashed lines) are observed at the open boundary in the COBC case (Fig. 20a), which is not consistent with the actual situations in the ocean. Figure 20b indicates that the FOBC allows the reflective waves to propagate out effectively and guarantees the stability and accuracy of numerical model during long-time simulation.

7. Summary and conclusions

The mesh-based numerical ocean models have increasingly been used in simulating various hydrodynamic phenomena in the ocean. The mesh-based numerical models, however, have certain disadvantages in comparison with the mesh-free models. The mesh-based model could not be used in simulating large discontinuities that do not coincide with the model grid or situations with the breakage of fluid into a large number of fragments. The smoothed particle hydrodynamics (SPH) technique is a mesh-free numerical method that has great potential to be used in the next generation of numerical ocean models. The implementation of nonreflected open boundary conditions in the SPH method, however, is not as straightforward. Inspired by the success of several open boundary conditions widely used in the mesh-based ocean circulation models, two radiation open boundary treatments were considered in this study for the SPH model: the adaptive open boundary condition (AOBC) and Flather’s open boundary condition (FOBC). The performance of these two open boundary conditions for the SPH method was assessed using the SPH-based shallow-water equation (SWE) model in several test cases. These test cases include 1) a steady flow over a bump in a channel with two open ends, 2) propagation of an initial sea level perturbations in a flat-bottom channel with two open ends and a steady flow, 3) wave propagation in a channel with one open end, and 4) vortex shedding behind a cylinder in a channel with two open ends. The results of the AOBC and FOBC are compared with the conventional characteristic open boundary condition (COBC) implemented in the SPH-based SWE model. An analysis of model results demonstrated that both the AOBC and FOBC are capable of allowing perturbations generated inside the computational domain to propagate out effectively while simultaneously giving specification of external flow information at the model open boundary. It was found that the AOBC needs an accurate calculation of the phase speed of perturbations from the fluid particle close to the open boundary, and it could lead to a gradual drift of the mean water level after a long run. By comparison, the FOBC is more concise in the equation form, more efficient in computation, and more stable in practical cases.

The SPH-based SWE model was also applied to simulate the laboratory observations of the Okushiri tsunami. In comparison with results produced by a state-of-the-art mesh-based model, the SPH-based model can obtain reasonable results without any special treatment in simulating the free-surface flow in regions with complex topography and wetting and drying fronts. It was shown that the FOBC allows the reflective waves to propagate out effectively through the model open
boundary and guarantees the stability and accuracy of long-term numerical simulations.

We demonstrated in this paper that the FOBC performs best in comparison with other OBCs in all the test cases presented in this paper. It is important to point out that, although the two new open boundary treatments (AOBC and FOBC) were implemented in the framework of the SPH-based SWE model, these two open boundary conditions can be implemented in the SPH-based three-dimensional (3D) Reynolds-averaged Navier–Stokes equations (RANSE) models. Before the SPH method is used in simulating the general circulations in the ocean, however, more studies are needed, such as the use of the SPH method in solving the 3D RANSE and the development of a methodology for specifying the momentum flux and the net heat/freshwater fluxes at the sea surface.

Acknowledgments. This work is financially supported by the China Scholarship Council (201406710003). JS is also supported by NSERC and the Lloyd’s Register Foundation (LRF). The LRF helps to protect life and property by supporting engineering-related education, public engagement, and the application of research. WF is supported by National Natural Science Foundation of China (51279055). The authors wish to thank three anonymous reviewers for their constructive comments.
APPENDIX

List of Three Open Boundary Conditions

a. COBC

1) INFLOW: PRESCRIBE $\mathbf{u}_B$ AND $d_B$

(i) Subcritical inflow ($\mathbf{u}_B < \sqrt{gd_B}$)

$$
\begin{align*}
\mathbf{u}_B &= (u_{B,i}, u_{B,n}) \\
B_d &= \left[ \frac{1}{2\sqrt{g}} (u_{B,n} - u_{l,n}) + \sqrt{d_l} \right]^2. 
\end{align*}
$$  \tag{A1}

(ii) Supercritical inflow ($\mathbf{u}_B \geq \sqrt{gd_B}$)

$$
\begin{align*}
\mathbf{u}_B &= (u_{B,i}, u_{B,n}) \\
B_d &= d_B. 
\end{align*}
$$  \tag{A2}

2) OUTFLOW: PRESCRIBE $d_B$

(i) Subcritical outflow ($u_{l,n} < \sqrt{gd_B}$)

$$
\begin{align*}
u_{B,n} &= u_{l,n} + 2\sqrt{g}(\sqrt{d_l} - \sqrt{d_B}) \\
B_d &= d_B. 
\end{align*}
$$  \tag{A3}

(ii) Supercritical outflow ($u_{l,n} \geq \sqrt{gd_B}$)

$$
\begin{align*}
u_{B,n} &= u_{l,n} \\
B_d &= d_B. 
\end{align*}
$$  \tag{A4}

In Eqs. (A1)–(A4), subscripts $i$ and $n$ denote the tangential component and the normal component of the characteristic variables, respectively; subscript $B$ represents the characteristic variables of OBPs; and subscript $I$ represents the characteristic variables inside the fluid domain. The characteristic Riemann invariants $\Phi_I$ are interpolated from normal fluid particles near the open boundaries based on

$$
\Phi_I = \sum_{j=1}^{N_I} \frac{V_{f,j}^{I}}{d_{l,j}^{I}} \Phi_{f,j}^{I} W_{j} / \sum_{j=1}^{N_I} \frac{V_{f,j}^{I}}{d_{l,j}^{I}} W_{j},
$$  \tag{A5}

where $\Phi$ denotes either the velocity vector $\mathbf{u}$ or the local water depth $d$, and the superscript $f$ denotes the fluid particles.

b. AOBC

INFLOW OR OUTFLOW: PRESCRIBE $\mathbf{u}^{ext}$ AND $d^{ext}$

(i) Subcritical flow ($\mathbf{u}^{ext} < \sqrt{gd^{ext}}$)

$$
\begin{align*}
u_{B,n} &= \mathbf{u}_{B}^{ext} \pm \sqrt{g}(d_{l}^{ext} - d^{ext}) \\
B_d &= d^{ext}. 
\end{align*}
$$  \tag{A6}

(ii) Supercritical outflow ($\mathbf{u}^{ext} \equiv \sqrt{gd^{ext}}$)

$$
\begin{align*}
u_{B,n} &= \mathbf{u}_{B}^{ext} \\
B_d &= d^{ext}. 
\end{align*}
$$  \tag{A7}

In Eqs. (A6) and (A7), $\Phi_B^k$ and $\Phi_B^{k+1}$ denote the characteristic variable of OBPs at model time steps $k$ and $(k + 1)$, respectively; $\Phi_B^k$ is the characteristic variable of internal fluid particle near the open boundary at the $k$th time step; $\Delta t$ is the time step; and $\Delta x$ is the distance between the OBP $B$ and the interpolation point of internal fluid $I$. The term $c$ is the phase speed of the characteristic variable. The relaxing parameter $\tau$: $\tau = \tau_{out}$ when the external information propagates toward the open boundary; $\tau = \tau_{in}$ when the external information propagates away from the open boundary. $\tau_{out} \gg \tau_{in}$.

c. FOBC

INFLOW OR OUTFLOW: PRESCRIBE $\mathbf{u}^{ext}$ AND $d^{ext}$

(i) Subcritical flow ($\mathbf{u}^{ext} < \sqrt{gd^{ext}}$)

$$
\begin{align*}
u_{B,n} &= \mathbf{u}_{B}^{ext} \pm \sqrt{g}(d_{l}^{ext} - d^{ext}) \\
B_d &= d^{ext}. 
\end{align*}
$$  \tag{A8}

(ii) Supercritical outflow ($\mathbf{u}^{ext} \equiv \sqrt{gd^{ext}}$)

$$
\begin{align*}
u_{B,n} &= \mathbf{u}_{B}^{ext} \\
B_d &= d^{ext}. 
\end{align*}
$$  \tag{A9}

REFERENCES


