Comparison of mathematical and CFD models of overwash of a step

David Skene[1,†], **Luke Bennetts**[1], **Michael Meylan**[2], **Michael Wright**[3] **and Kevin Maki**[3]

[1]School of Mathematical Sciences, University of Adelaide, SA 5005, Australia

[2]School of Mathematical and Physical Sciences, University of Newcastle, NSW 2308 Australia

[3]Department of Naval Architecture and Marine Engineering, University of Michigan, MI 48109, USA

[†]david.skene@adelaide.edu.au

Introduction

Overwash is the process in which waves force water onto the surface of a body that is otherwise not fully submerged. It occurs in wave interactions with thin floating plates due to their small freeboards. For example, when experimentally investigating water wave transmission by an array of thin floating plates, Bennetts & Williams (2015) reported overwash in laboratory experiments of regular incident waves interacting with an array of thin floating wooden disks. They noted shallow overwash for relatively moderate incident amplitudes, and deeper and more energetic overwash for larger incident amplitudes. Moreover, they showed that the occurrence of strong overwash was correlated to the array transmitting significantly less wave energy than predicted by linear potential flow theory.

Skene *et al.* (2015) proposed a 2D (one horizontal dimension and one depth dimension) mathematical model of overwash of a thin floating plate. They separated the overwash domain from the surrounding water and plate domain; applying linear potential flow theory to the latter domain, and the nonlinear shallow water equations to the former. The potential flow model prediction of the wave elevation above the plate and velocity there was used to force the shallow water equations.

Further, they compared the model predictions to measurements made during laboratory experiments. In these experiments, thin floating plastic plates were subjected to regular incident waves with steepnesses ranging from $ka = 0.04$ to 0.15 and wavelength to plate length ratios from 0.56 to 1.51. A depth gauge placed at the centre of the upper surface of the plate was used to measure the overwash depth. The motion of the plate was also measured, and Meylan *et al.* (2015) had earlier shown that linear potential flow theory accurately predicts this motion.

Skene *et al.* (2015)'s model accurately predicted the overwash depth signals for relatively low incident steepnesses and short wavelengths, for which the mathematical model predicted a mean overwash depth of less than approximately 2 mm. However, for higher incident steepnesses and longer incident wavelengths, the model became inaccurate and overpredicted the overwash depth by a factor of up to three. The likely causes of the loss of accuracy were hypothesised to be that: (i) the shallow water equations neglect turbulence, which was visible in the experiments for the largest incident amplitudes; (ii) the model lacks back-coupling from the overwash to the plate and surrounding water; and (iii) the shallow water equations do not model the wave breaking that occurred in the experiments when bores created at the leading and trailing plate edges collided.

Here, the model is applied to the problem of overwash of a step that has the height of the undisturbed water depth. The simplified geometry removes any errors created by (iii), and thus provides a direct means to test the impacts of (i) and (ii). The model predictions are compared to data from computational fluid dynamics (CFD) models of the problem.

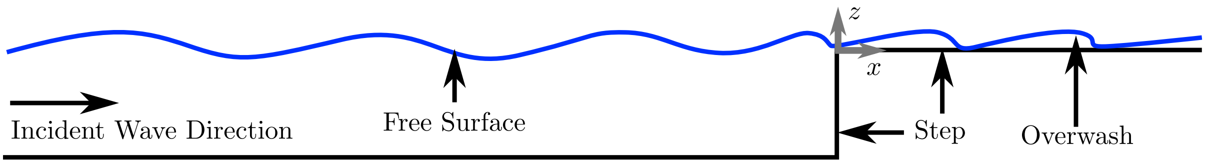

Figure 1: Schematic of overwash step problem (not to scale).

Mathematical model

Figure 1 shows a schematic of the problem. The geometry is two dimensional, with locations defined by the Cartesian coordinate system (x, z). Here, x is the horizontal coordinate, and z is the upwards pointing vertical coordinate. The origin is located at the upper corner of the step. Initially, water occupies the domain $x < 0$ and $-H < z < 0$. The vertical face of the step is located at $x = 0$ and $-H < z < 0$. Its upper surface is $x \geq 0$ and $z = 0$. Overwash is forced by a monochromatic incident wave.

For $x < 0$, it is assumed that the water is irrotational, incompressible and inviscid. As the waves tested are small perturbations about the equilibrium depth, the free surface is linearised. Thus, this water is modelled with linear potential theory using a velocity potential, $\phi(x, z, t)$, where t is time and the gradient of ϕ represents the water velocity field. The motion in this region is assumed to be time-harmonic. Hence, the velocity potential is expressed as $\phi(x, z, t) = \mathrm{Re}\{\bar{\phi}(x, z)\mathrm{e}^{-\mathrm{i}\omega t}\}$, where ω is some prescribed frequency, and $\bar{\phi}(x, z)$ is the velocity potential in the frequency domain. The velocity potential satisfies the standard governing equations for surface gravity water waves. These are

$$\nabla^2\bar{\phi} = 0 \quad \text{in} \quad -H < z < 0 \quad \text{and} \quad x < 0, \tag{1}$$

$$\partial_z\bar{\phi} = 0 \quad \text{on} \quad z = -H, \quad \text{and} \quad \partial_z\bar{\phi} - \omega^2 g^{-1}\bar{\phi} = 0 \quad \text{on} \quad z = 0, \tag{2}$$

where $g \approx 9.81\,\mathrm{m\,s^{-1}}$ is the acceleration due to gravity. The vertical face of the step bounds the right hand-end of the potential flow domain. A no-penetration condition is applied at this boundary, i.e. $\partial_x\bar{\phi} = 0$ on $-H < z < 0$ and $x = 0$.

The velocity potential is the sum of an incident wave potential, $\bar{\phi}_I$, where the incident wave travels in the positive x-direction (towards the step), and a reflected wave potential, $\bar{\phi}_R$, where the reflected wave travels in the negative x-direction (away from the step). The incident wave potential is

$$\bar{\phi}_I = \frac{ag\mathrm{e}^{\mathrm{i}kx}\cosh\{k(z + H)\}}{\mathrm{i}\omega\cosh(kH)}, \tag{3}$$

where a is the incident amplitude and the wave number k is the real positive root to the dispersion relation $k\tanh(kH) = \omega^2/g$. The reflected wave potential can be calculated explicitly (details omitted).

At the free surface, the wave elevation and horizontal velocity are given by $\eta(x, t) = -\partial_t\phi/g$ and $u_x(x, z, t) = \partial_x\phi$, respectively, where η is the free surface height and u_x is horizontal the velocity at $z = 0$. At the interface with the overwash, they are $\eta(x = 0, t) = 2a\cos(\omega t)$, and $u_x(x = 0, z = 0, t) = 0$.

Neglecting turbulence and viscosity, the nonlinear shallow water equations are used to model the overwash, which are

$$\partial_t(h) + \partial_x(uh) = 0 \quad \text{and} \quad \partial_t(uh) + \partial_x\left(u^2 h + 0.5gh^2\right) = 0, \tag{4}$$

where $h(x,t)$ is the height of the overwash water and $u(x,t)$ is its depth averaged velocity in the x-direction. The shallow water equations require numerical solution. The spatial derivative is explicitly discretized using the finite volume method of Kurganov & Tadmor (2000), which is presented for the shallow water equations in Skene *et al.* (2015). The solution is time stepped using the RK2 method.

At the interface between the shallow water and the potential flow, $x = 0$, the height and depth averaged horizontal velocity of the overwash are set to be the wave elevation and horizontal velocity predicted by the potential flow, i.e. $h(x = 0, t) = 2a \cos(\omega t)$, and $u(x = 0, t) = 0$. Note that, although the potential flow model predicts zero velocity here as the effect of overwash is assumed to be negligible upon it, the shallow water equations still allow water to flow onto and off the step here. Downstream in the overwash, the conditions $h(x = 2\,\mathrm{m}, t) = 0$ and $u(x = 2\,\mathrm{m}, t) = 0$ are used to approximate water running off towards infinity.

CFD models

Two CFD models are created using the open-source software OpenFOAM. Each test case has the initial condition of still water with its free surface located along the line made by $z = 0$. An upstream wave maker region is used to generate the incident wave according to Stokes fifth-order approximation. The solution to the two-phase Navier-Stokes equations are computed on a two-dimensional mesh with initial grid spacing of 0.1 mm using a customized version of the interFoam solver. The solver uses the finite-volume method to solve the governing equations on unstructured meshes. The volume-of-fluid method is used to track the air-water interface.

The CFD model was tested for a slip and no-slip boundary condition at the water basin's boundaries. The slip condition is consistent with the mathematical model. The no-slip condition tests the effects of a boundary layer.

Results

The models were run for the incident wave frequency $\omega = 2.5\pi\,\mathrm{rad\,s^{-1}}$, which corresponds to the wavelength $2\pi/k \equiv \lambda = 1.0\,\mathrm{m}$, and steepnesses $ka = 0.031$, 0.074, 0.063, and 0.094. Both models are analysed once the overwash becomes quasi-time-harmonic. Time offsets are set such that $t = 0$ corresponds the minimum depth at $x = 0_+$, i.e. the interface viewed from the edge of the shallow water domain.

Figure 2 shows the depth averaged velocity and depth signals over three wave periods at $x = 0_+$ and $x = 100\,\mathrm{mm}$ for a wave of amplitude $a = 10\,\mathrm{mm}$. The results for this amplitude are representative of the other wave amplitudes tested. The three models have similar depth and velocity signal shapes at each position, however, the CFD models have a slightly skewed velocity at $x = 0_+$ and show some roughness due to turbulence downstream. The slip and mathematical model share similar vertical offsets, however, the no-slip model's signals are vertically shifted to be slower and deeper at $x = 100\,\mathrm{mm}$. All three signals share similar wave amplitudes (maximum minus minimum values) and periods (peak to peak time differences).

Figure 2 also shows the mean depth over three wave periods for each incident wavelength at $x = 0_+$ and $x = 100\,\mathrm{mm}$. This figure shows close agreement across all wave amplitudes at the interface. Downstream however, while there is strong agreement between slip and mathematical model, the no-slip model differs systematically by approximately 1.5 mm in depth. Although not shown, there is near perfect agreement for velocity at $x = 0_+$ but a systematic error of $0.07\,\mathrm{m\,s^{-1}}$ downstream between the no-slip and other models.

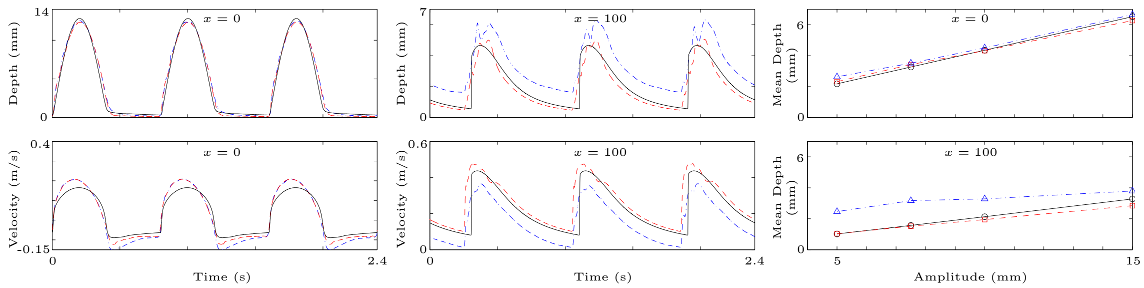

Figure 2: Depth (top left and centre) and depth averaged velocity (bottom left and centre) signal comparisons of slip (dashed), no-slip (dot-dashed), and mathematical model (solid) at $x = 0_+$ (left) and $x = 100\,\text{mm}$ (centre) for $a = 10\,\text{mm}$. Also, mean depth for varying amplitude comparisons at $x = 0_+$ (top right) and $x = 100\,\text{mm}$ (bottom right)

The source of these systematic errors is shown in Figure 3. In this figure the horizontal velocity of the slip, no-slip, and mathematical model are plotted against their depth for a wave of amplitude $a = 5\,\text{mm}$ at $t = 0.2\,\text{s}$. The plot shows that while the slip model has an approximately uniform velocity profile, like that of the mathematical model, the no-slip boundary condition creates a boundary layer approximately 2.5 mm thick.

Conclusions

- The one way coupling captures the coupling in the fully coupled CFD model well.
- Turbulence in the overwash does not appear to create large differences between models in terms of the mean depth and horizontal velocity up to 100 mm downstream.
- The models without boundary layers systematically predict smaller overwash depths and larger mean horizontal velocities than the model with a boundary layer.

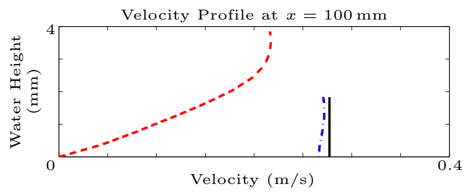

Figure 3: Horizontal velocity profiles of slip (dashed), no-slip (dot-dashed), and mathematical model (solid) at $x = 100\,\text{mm}$ for $a = 5\,\text{mm}$.

Acknowledgements The Australian Antarctic Science Program funds DS and LB. The Australian Research Council funds LB. The US Office of Naval Research funds MM and KM.

References

Bennetts, L. G. & Williams, T. D. 2015 Water wave transmission by an array of floating disks. *Proc. R. Soc. Lond. A* **471** (2014069).

Kurganov, A. & Tadmor, E. 2000 New high-resolution central schemes for nonlinear conservation laws and convection–diffusion equations. *J. Comput. Phys.* **160** (1), 241–282.

Meylan, M. H., Bennetts, L. G., Alberello, A., Cavaliere, C. & Toffoli, A. 2015 Experimental and theoretical models of wave-induced flexure of a sea ice floe. *Phys. Fluids* **27**, 041704.

Skene, D. M., Bennetts, L. G., Meylan, M. H., & Toffoli A., 2015 Modelling water wave overwash of a thin floating plate. *J. Fluid Mech.* **777**, R3.