Proceedings of the 7th International Conference on the Application of Physical Modelling in Coastal and Port Engineering and Science (Coastlab18)

Santander, Spain, May 22-26, 2018

TOWARDS AN EFFICIENT AND HIGHLY ACCURATE COUPLED NUMERICAL MODELLING APPROACH FOR WAVE INTERACTIONS WITH A DIKE ON A VERY SHALLOW FORESHORE

INE VANDEBEEK[1,2], VINCENT GRUWEZ[1], CORRADO ALTOMARE[3], TOMOHIRO SUZUKI[3], DIETER VANNESTE[3], SIEGLIEN DE ROO[3], ERIK TOORMAN[2], PETER TROCH[1]

1 Dept. of Civil Engineering, Ghent University, Belgium, Ine.Vandebeek@ugent.be

2 Hydraulics Laboratory, Dept. of Civil Engineering, KU Leuven, Belgium, Erik.Toorman@kuleuven.be

3 Flanders Hydraulics Research, Belgium, Corrado.Altomare@mow.vlaanderen.be

ABSTRACT

An accurate prediction of wave overtopping over the crest of coastal structures and wave-induced loading is essential to guarantee coastal safety. Due to the presence of a very shallow foreshore at the Belgian coast, this is not straightforward. Within the CREST project, a numerical model is being developed to accurately predict these wave interactions with structures on a very shallow foreshore. Sediment transport in front of the structure as well as the wave structure interactions are simulated with the open-source CFD software OpenFOAM. Wave propagation over the beach until the wave breaking point is modelled with SWASH, a model based on the non-linear shallow water equations in order to limit the computational cost. A one-way coupling methodology between these two models is proposed. The capability of OpenFOAM as stand-alone model to accurately predict wave loading forces on buildings for a case with a very shallow foreshore is demonstrated by comparing the numerical results to experimental data. The ability of OpenFOAM to simulate scour processes is validated by experimental data of a submerged jet flow over an apron. Furthermore, a first test case with the coupled OpenFOAM-SWASH model presenting the propagation of regular waves is described.

KEWORDS: Numerical modelling, wave-induced loading, beach morphodynamics, shallow foreshores

1 INTRODUCTION

Low-lying countries such as Belgium and the Netherlands have a very typical coastal defence system along most of the coastal urban areas: a mildly sloped and very shallow beach in front of a dike with a promenade and high-rise buildings close to the crest. With the current threat of sea level rise and the expected increase in storminess due to climate change, these coastal defences are susceptible to overtopping wave attacks. To avoid damage and guarantee coastal safety in all circumstances, the sea defences are being reinforced with beach nourishments and storm walls.

For a safe and optimal design of coastal structures, loads exerted by waves on these structures and wave overtopping volumes need to be correctly predicted. However, at present a thorough understanding of the effect of beach nourishments and the presence of the shallow foreshore on wave-induced loading and wave overtopping is lacking. Offshore waves transform considerably before reaching the dike, changing the wave spectra significant from offshore to the toe of the dike, and causing beach erosion in front of the dike. This leads to an important increase in the complexity of the design calculations.

Within the CREST research, a reliable and accurate prediction methodology is being developed to assess wave overtopping and wave-impact forces on coastal defences taking into account the beach topography evolution due to waves and currents. This is done by using advanced numerical models supported by physical model tests and field measurements. This paper focusses on the numerical model developments, more details about the physical model tests can be found in Gruwez *et al.* (2018) and Altomare *et al.* (2018), both associated within the CREST project.

To accurately predict wave-induced loading, overtopping and beach topography two contradictory requirements need to be fulfilled: (1) a very high spatial resolution of the computational grid over the beach and structures and (ii) a large enough computational domain since the processes on the beach are driven by waves originally generated at sea. This is currently impossible to realize with a single model at an acceptable computational cost. Therefore, a chain of models will be applied: a wave transformation model (SWASH) calculates the wave propagation over the beach until the wave breaking point, from this point onwards, the wave transformation and wave structure interaction including changing beach morphodynamics is modelled with the CFD software OpenFOAM.

OpenFOAM resolves the flow over the complete water depth and hence allows modelling of the complex overtopped flow on the dike and promenade. OpenFOAM has proven to provide reliable results for coastal engineering applications and free surface applications (Higuera *et al.*, 2013b).

To model wave transformations over the shallow and mildly sloped foreshore in the non-breaking zone, the use of the computationally intensive model OpenFOAM is not required. Depth-averaged models such as the non-linear shallow water model SWASH (Zijlema *et al*., 2011) are accurate enough for simulating the surface wave and the velocity field and are much more computationally efficient than CFD. However, SWASH is not able to deal with abrupt changes of shape of coastal structures (Altomare *et al.,* 2014).

The presented coupling methodology is a one-way coupling where OpenFOAM receives the information on the wave kinematics from SWASH at a user defined location outside the surf zone. This paper provides a brief overview of the research being done in order to obtain this accurate and efficient (coupled) numerical model. The main goal of the authors is to illustrate the methodologies used within the CREST project, therefore only a limited amount of results of the numerical work are shown.

The capability of OpenFOAM as stand–alone model to accurately predict wave loading forces on buildings is demonstrated in the first test case. The second test case shows the ability to simulate scour processes. A third case consists of a numerical wave tank with regular waves propagating as a first validation of the coupled model.

2 NUMERICAL MODELS

A brief description of the numerical models is reported in this section.

2.1 Wave propagation models – SWASH

SWASH is chosen as wave propagation model as it already has proven to provide reliable results for both wave transformation and overtopping for shallow foreshores in 1D calculations with very little computational resources due to the depth-averaged assumption and parallel computation capabilities (Suzuki *et al.*, 2011). It is a time domain model for simulating non-hydrostatic, free surface and rotational flow. Governing equations are the shallow water equations including a non-hydrostatic pressure term.

$$\frac{\partial \zeta}{\partial t} + \frac{\partial hu}{\partial x} = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + g\frac{\partial \zeta}{\partial x} + \frac{1}{2}\frac{\partial q_b}{\partial x} + \frac{1}{2}\frac{q_b}{h}\frac{\partial(\zeta - d)}{\partial x} + c_f\frac{u|u|}{h} = \frac{1}{h}\frac{\partial}{\partial x}\left(h\nu_t\frac{\partial u}{\partial x}\right) \tag{2}$$

$$\frac{\partial w_s}{\partial t} = \frac{2q_b}{h} - \frac{\partial w_b}{\partial t}, w_b = -u\frac{\partial d}{\partial x} \tag{3}$$

$$\frac{\partial u}{\partial x} + \frac{w_s - w_b}{h} = 0 \tag{4}$$

Where t is the time, x the horizontal coordinate, u the depth-averaged velocity in *x*-direction, w_s and w_b the velocity in *z*-direction at the surface and at the bottom respectively. ζ is the free surface elevation from still water level, d the still water depth and *h* the total depth. q_b is the non-hydrostatic pressure at the bottom, *g* the gravitational acceleration, c_f the dimensionless bottom friction coefficient and $\nu_t$the eddy viscosity. A full description of the numerical model, boundary conditions, numerical schemes and applications can be found in (Zijlema *et al*., 2011).

2.2 Wave structure interaction model - OpenFOAM

In the vicinity of the coastal structures, the open-source software OpenFOAM is chosen. OpenFOAM (Weller and Tabor, 1998) provides solvers for several types of physics problems, including compressible and incompressible fluid dynamics, single and multiphase flow dynamics, solid mechanics and electromagnetics. The two-phase solver interFoam is selected for the present study. Furthermore, OpenFOAM has capabilities for different turbulence and rheological modelling and powerful meshing and parallel computations. Discretization is based on the finite volume method.

The governing equations are given by the RANS (Reynolds-Averaged Navier Stokes) equations and the continuity equation for incompressible flow.

$$\frac{\partial \rho u}{\partial t} + \nabla.[\rho u u^T] = -\nabla p^* - g.x\nabla\rho + \nabla.[\mu\nabla u + \rho\tau] + \sigma_T\kappa_\gamma\nabla\gamma \tag{5}$$

$$\nabla.u = 0 \tag{6}$$

With u (u,v,w) the velocity field, $p*$ the pressure in excess of the hydrostatic pressure, ρ the density, g the gravitational acceleration, *x* the position vector and μ the dynamic viscosity. τ is the specific Reynolds stress tensor. The last term of equation 5 represents the effect of surface tension with σ_T the surface tension coefficient and κ_γ the surface curvature.

Equations 5 and 6 are solved for two immiscible fluids simultaneously where the fluids and the interface between the two phases is modelled with a volume of fluid (VOF) method. With the VOF method, the volume fraction α is defined as the fraction of volume occupied by the liquid in each cell (i.e. α is 0 for air and 1 for water). The distribution of α is modelled by an advection equation.

$$\frac{\partial \alpha}{\partial t} + \nabla.[u\alpha] + \nabla.[u_r\alpha(1-\alpha)] = 0 \tag{7}$$

The last term in equation 8 is a compression term, which limits the smearing of the interface, and u_r is the vector of relative velocity between the two fluids.

For the generation and active absorption of waves, OlaFlow, an evolution of IHFoam (Higuera *et al.*, 2013a), is employed. OlaFlow provides the generation of several wave types including solitary, linear and non-linear waves based on Stokes, cnoidal and stream function theories. No relaxation zone is necessary due to the active absorption capability of OlaFlow.

2.3 Sediment transport model

To simulate sediment transport processes and to take into account its effect on wave overtopping and wave loading forces, following equations are implemented and solved in OpenFOAM. The sediment module consists of three components: the sediment transport model including bed load and suspended load, a sand slide model and the sediment continuity equation. The bed load transport rate is calculated according to the formula proposed by Engelund and Fredsoe (1976).

$$q_{bl} = 18.74(\theta - \theta_c)\left[\theta^{\frac{1}{2}} - \theta_{cr}^{\frac{1}{2}}\right]\sqrt{(s-1)gd}d \tag{9}$$

With q_{bl} the bed load transport rate per unit width, θ the Shields number, *g* gravitational acceleration, *d* the median grain diameter and *s* the relative density of sediment. θ_{cr} is the critical shields number which is calculated according to the Soulsby (1997) formula:

$$\theta_{cr} = \theta_{cr0}\left(cos\beta\sqrt{1 - \frac{sin^2\phi tan^2\beta}{\mu_s^2}} - \frac{cos\phi sin\beta}{\mu_s}\right) \tag{10}$$

$$\theta_{cr0} = \frac{0.3}{1+1.2d_*} + 0.055\,(1 - \exp(-0.02d_*)) \tag{11}$$

$$d_* = \left[\frac{g(S-1)}{\nu^2}\right]^{\frac{1}{3}} d \tag{12}$$

Where θ_{cr0} is the critical shields number for a flat bed, ϕ the angle between the steepest slope of the bed and the bed shear stress direction, β the angle between the bed and the horizontal plane, μ_s the static friction coefficient and d_* the dimensionless grain size.

The suspended load is calculated by a traditional convection-diffusion equation as presented in equation 12.

$$\frac{\partial c}{\partial t} + \nabla.(u - v_s)c = \nabla.(\nu_t \nabla c) \tag{13}$$

With *c* the volumetric sediment concentration, *u* the velocity (u,v,w), $v_s$the sediment fall velocity in still water and ν_t the diffusivity of sediment, which equals the turbulent eddy viscosity in the present study.

A geometric sand slide routine is implemented following the approach proposed by Marieu *et al.* (2008).

The morphological change of the bed is calculated by the Exner equation (sediment continuity equation). This equation involves four terms: the rate at which the bed elevation changes, the spatial variation of the bed load and two terms for erosion and deposition. The deposition rate is calculated according to equation 14. The entrainment rate is calculated with the formula of Van Rijn (1987) which is a reference height based approach.

$$\frac{\partial h}{\partial t} = \frac{1}{1-n}(-\nabla q_{bl} + D - E) \tag{14}$$

h represents the bed elevation, *n* the porosity, q_{bl} the bed load transport rate, *D* the deposition rate and *E* the entrainment rate, respectively calculated as:

$$D = c_B v_s \tag{15}$$

$$E = c_E v_s \tag{16}$$

c_B denotes the sediment concentration in the first layer of grid cells above the bed, c_E the equilibrium concentration of suspended load at a reference level and v_s the sediment fall velocity in still water.

Van Rijn (1987) proposes following formula to calculate the equilibrium concentration c_E with Δ_d the reference height.

$$c_E = 0.015 \frac{d_{50}\left(\frac{\theta-\theta_c}{\theta_c}\right)^{1.5}}{\Delta_d d_*^{\frac{1}{3}}} \tag{17}$$

The bed is updated with dynamic meshing capabilities to adjust to the changes in bed topography. The change of bed height is determined by equation 13. OpenFOAM provides a mesh smoothing method which smooths the mesh by solving the Laplace equation on the whole mesh domain.

$$\nabla.(\gamma \nabla v_1) = 0 \tag{18}$$

With v_1 the displacement field of the mesh points and γ the diffusion coefficient. An inverse distance diffusion coefficient is adopted in this work.

2.4 Coupling methodology

OpenFOAM provides reliable results for wave loading forces and simulating sediment transport (as will be discussed in paragraph 3.1 and 3.2). However, these calculations are computationally intensive. Therefore, a coupled model is being developed where wave transformation over the beach up until the wave breaking point is done by a more efficient wave propagation model, SWASH. The presented coupling methodology comprises of a one-way coupling for a 1D SWASH with a 2D OpenFOAM model. The coupling is done outside the surf zone, as wave breaking turbulence is an important factor in sediment transport and hydrodynamics in the zone near the dike. The presented coupling methodology is tested on a case with regular wave propagation. The waves are generated and propagated by SWASH over a sufficiently large distance. SWASH supplies both the surface elevation and the horizontal wave kinematics over the full bathymetry. At the SWASH-OpenFOAM interface this information is transferred between the two models and the received velocity is imposed at the OpenFOAM boundary up until the water surface elevation level.

The coupling is implemented using the language independent communication protocol MPI that has the advantage that limited adaptations have to be made to the source codes of the various programs, similar to the coupling methodology developed by Verbrugghe (2017). Both OpenFOAM and SWASH already use MPI for communication between parallel processes, which makes it easy to add an additional communication process. Three processes are run simultaneously, while the computations are performed by one program (continuous line), the other program is waiting (dotted line), as illustrated in Figure 1.

- Process 0: SWASH runs for one time step and calculates the free surface elevation and the depth-averaged velocity. These parameters are sent to a Fortran script with the MPI_SEND command.
- Process 1: The Fortran script receives the depth-averaged velocity and the calculated surface elevation from SWASH. As the OpenFOAM grid, in most cases, has more cells in the vertical direction than the number of layers in the SWASH calculations, the velocity is interpolated on the OpenFOAM grid. Subsequently, a uniform velocity profile over the full depth and the surface elevation is sent to OpenFOAM.
- Process 2: OpenFOAM receives the velocity profile and surface elevation from the Fortran script with the MPI_RECEIVE command. This data is imposed at the wave generation boundary of the OpenFOAM domain. For every time step in SWASH, OpenFOAM runs with a smaller or equal time step. When the OpenFOAM simulation time equals the time step of SWASH, the process is repeated.

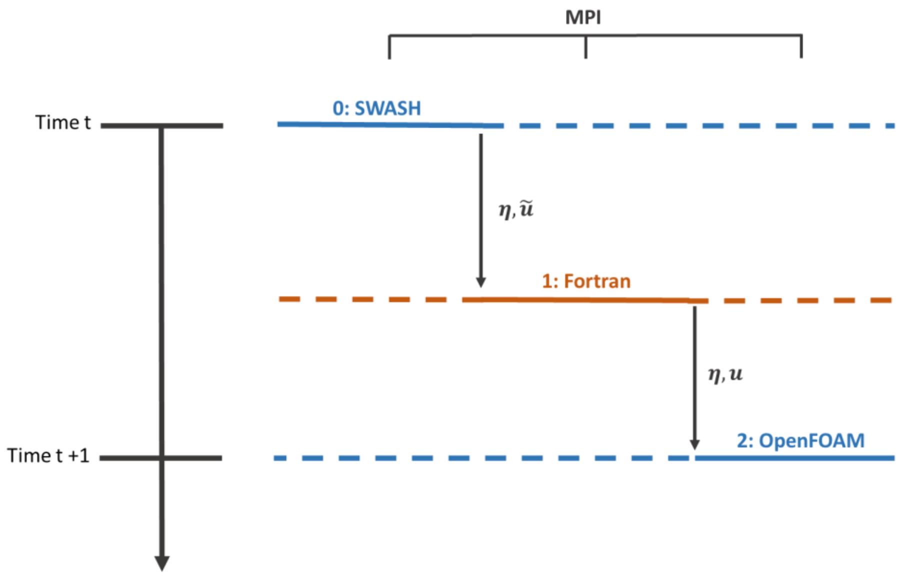

Figure 1. Coupling algorithm of coupling between SWASH and OpenFOAM.

3 CASE STUDIES

3.1 OpenFOAM – wave loading forces

A first case is presented to show the ability of OpenFOAM to accurately predict wave loading forces. Validation is performed by experimental test data of regular wave propagation over a fixed shallow foreshore and regular wave impact on a building on top of a dike with a promenade. The experimental set-up is in detail described in Chen (2016). The case is a typical example of the Belgian coastline with a building constructed on top of the dike. The foreshore and the dike have a slope of 1:35 and 1:3 respectively. Physical model tests were carried out in the wave flume with a length of 70.0 m, a depth of 1.4 m and a width of 4.0 m at Flanders Hydraulics Research. Experimental measurements consist of forces on the vertical wall (i.e. building), layer thicknesses and velocities of overtopping flow. Tests were performed both with regular and bichromatic waves, however only a regular wave case is discussed here.

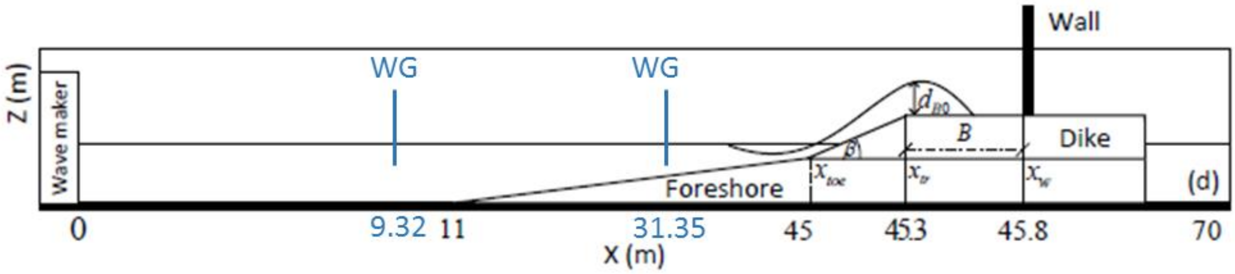

Figure 2. Experimental set-up physical model tests (Chen, 2016).

The computational domain includes the complete flume. The upper boundary is located at height z = 2.0 m to avoid water leaving the domain due to the impact of waves against vertical wall. At the inlet, the experimental wave paddle displacement is imposed. No active absorption is included as it was not included in the experimental paddle motion. The computational grid has a vertical resolution of 0.02 m (10 cells per wave height) and a horizontal resolution of 0.05 m around the expected location of the free surface. In the air phase the vertical and horizontal grid size are doubled. Additional refinement is performed in the shoaling zone (vertical grid size of 0.01 m) and in the breaking zone (vertical grid size of 0.005 m) to accurately resolve wave breaking. The highest mesh resolution is found on top of the dike slope and the promenade (vertical grid size = 0.0025 m) to resolve better the small flow depths. The time step is adjusted according to the Courant number which is limited to 0.1 as suggested by Roenby *et al*. (2017). The k-ω-SST turbulence model is used as this was found to be the best for wave breaking by Brown *et al.* (2016).

3.2 OpenFOAM – sediment transport

A necessary first step is to apply the implemented sediment module in simple flow situations for which analytical solutions or laboratory measurements exist. In order to validate the sediment module, run 2 of the experiment of Chatterjee *et al.* (1994) is simulated. Chatterjee investigated the effect of a submerged wall jet on the flow field and the sediment transport. A schematic view of the experiment and the geometric properties are given in Figure 3. The jet is formed by a small opening under a gate and flows over a rigid bottom and an erodible sand bed. The median grain diameter is 0.76 mm, the porosity 0.43 and the submerged specific gravity 1.65.

In the experimental tests, the jet is caused by a head difference between the part upstream and downstream of the gate. However, in the numerical model only the part downstream of the gate is modelled. At the inlet, a volumetric flowrate of 0.0204 m³/s/m is imposed, at the outlet a velocity is imposed to maintain the same flowrate and to keep the water level fixed at z = 0.291m. Turbulence is modelled with the $k-\epsilon$ model. A uniform mesh is used with a mesh size of 0.01 m in both the vertical as horizontal direction.

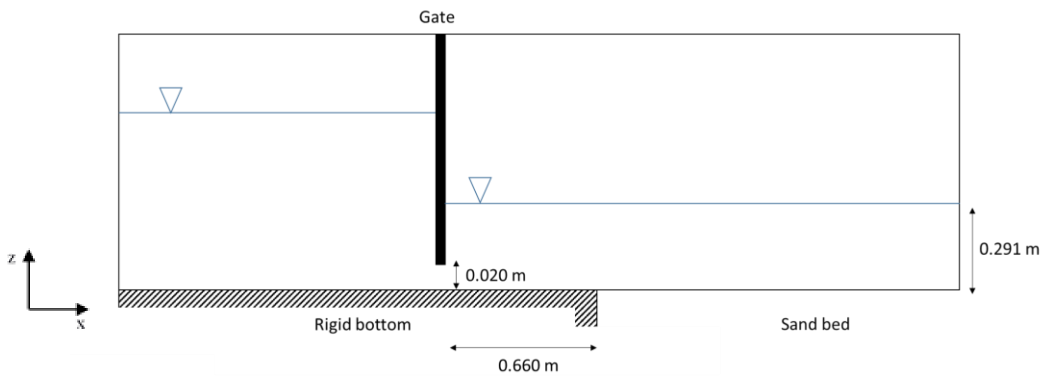

Figure 3. Schematic view of the experiment of (Chatterjee *et al.*, 1994).

3.3 OpenFOAM – SWASH coupling

The accuracy of the coupling of OpenFOAM – SWASH is assessed based on the propagation of regular waves over a flat fixed bottom. The set-up is illustrated in Figure 4. The SWASH and OpenFOAM domain have a length of 18 m and 12 m respectively. Regular linear waves are generated at the SWASH boundary with a wave period of 2.0 seconds and a wave height of 0.02 m. The still water depth is equal to 0.4 m. An equal time step, Δt = 0.005 seconds, is used in both models. The OpenFOAM domain has a spatial resolution of 0.002 m in the vertical direction (10 cells per wave height) and 0.02 m in the horizontal direction. In the SWASH model a horizontal resolution of 0.05 m is used. No mesh study to find the most optimal spatial resolution is done for this case.

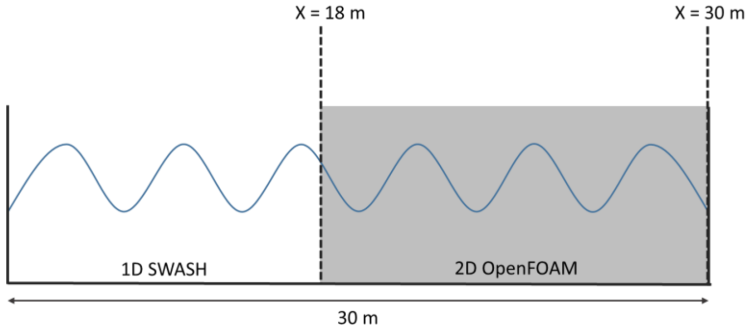

Figure 4. Set-up of the test case with OpenFOAM – SWASH coupling.

4 RESULTS AND DISCUSSION

4.1 OpenFOAM – wave loading forces

Figure 5 shows the water surface elevations for two wave gauge positions, as shown in Figure 2, and the wave-induced forces on the vertical wall. The model provides a good approximation of both water surface elevation and overtopped wave-impact force on a vertical wall. The numerical model does not as accurately reproduce the wave trough as the wave crest. Since first order wave generation was used, free super-harmonics are generated as well, which are mostly visible in the wave troughs. Possibly the resolution of the numerical model is not able to resolve these short waves sufficiently causing inaccuracies in their propagation. A small difference is present in the wave loading forces. This can be explained by the highly turbulent and stochastic nature of overtopping wave impact.

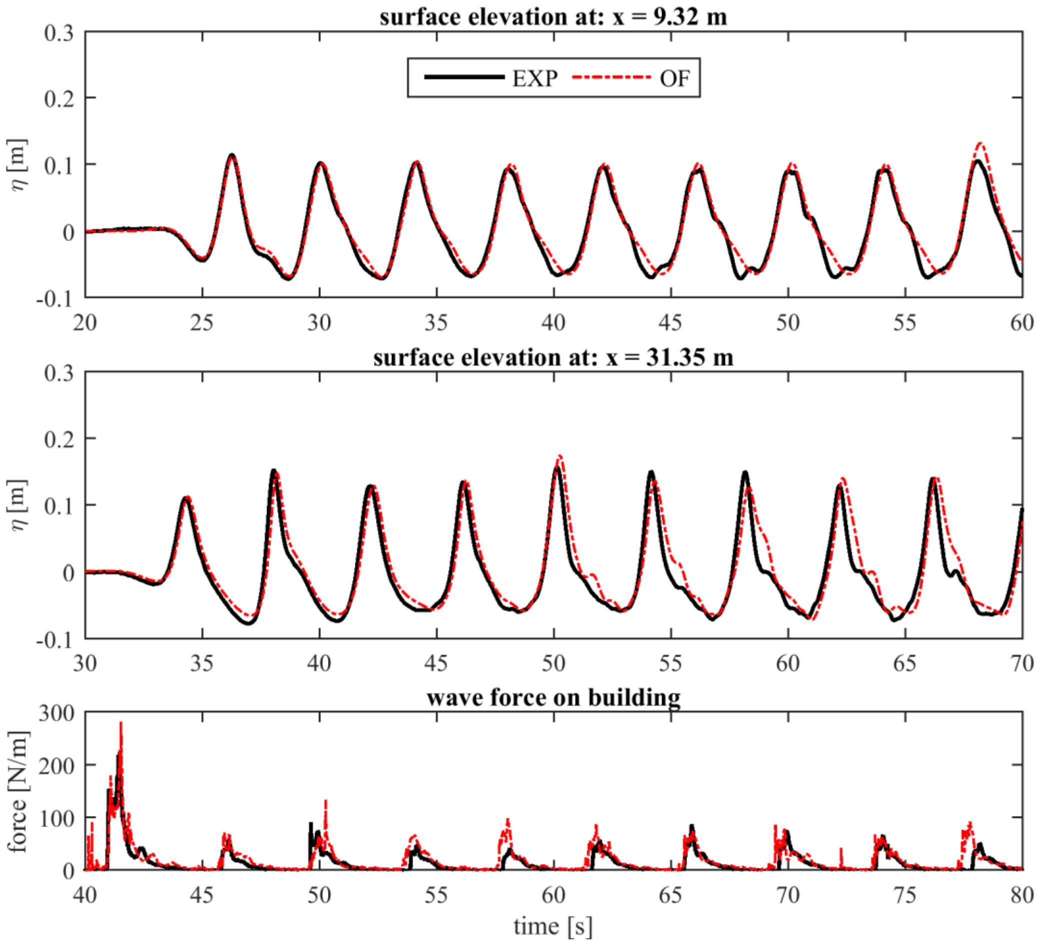

Figure 5. Comparison between experimental (EXP) and numerical model OpenFOAM (OF) of the surface elevation in front of the wave paddle (upper figure), on the foreshore (middle figure) and of the wave loading forces on the building (lower figure) (Gruwez *et al.*, 2017).

4.2 OpenFOAM – sediment transport

Figure 6 shows the volume fraction field and the development of the scour hole at equilibrium stage. The simulated scour profile agrees fairly well with the observed profile in the experiments.

Figure 7 shows the evolution of the scour hole depth and the dune height in function of time. The values are compared with measured values at several time instances: t = 60, 180, 300, 480, 720, 1200 seconds. The results agree quite well. An overestimation of the deposition dune can be observed. As previously discussed by Zhou (2017), this can be explained by horizontal forces acting on the dune caused by the pressure difference between the upstream and downstream faces of the dune. This force can push some of the sediment down the slope of the dune in the experiments. In the numerical model this horizontal force is not taken into account.

Figure 8 shows the modelled and measured velocity field at the equilibrium stage. Two simulations are performed, one with a static mesh (no scour) and one with a dynamic mesh (scour) to study the influence of the scour on the flow field. The x-axis represents the dimensionless velocity where U_{max} represents the maximal horizontal flow velocity in a specific section on the rigid bed (x = 0.6 m). The y-axis represents the non-dimensional distance where z_{max} represents the height above the bed where the maximal horizontal velocity U_{max} occurs. The results show a good correspondence between simulated and experimental results. Not much difference is observed with or without scour. The calculated velocities in the simulation with scour are a little bit lower than the measured values and become negative. This does not occur in the experiments but can be explained by the difference in set-up: in the experimental tests, the jet is created by a head difference while in the numerical simulation the jet is created by a uniform inlet velocity.

Figure 6. Numerical simulation results of flow field at equilibrium state – volume fraction indicator field.

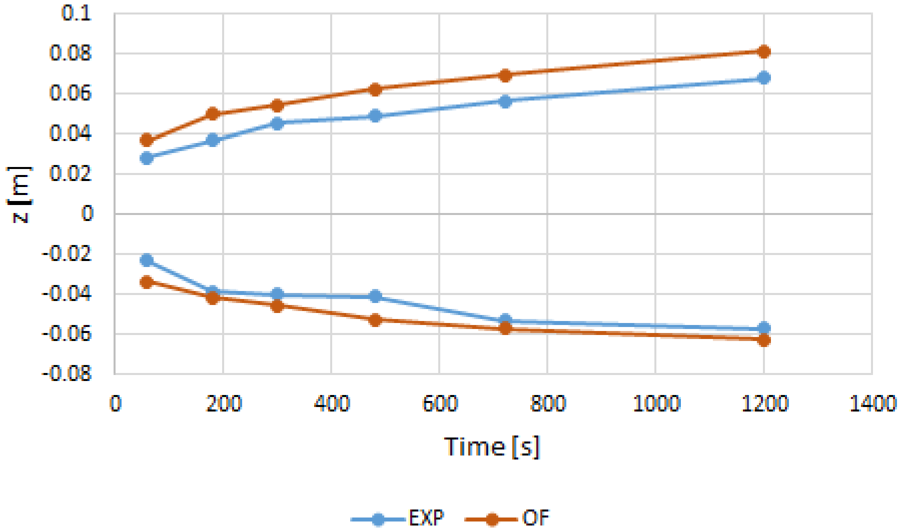

Figure 7. Comparison of the simulated (OF) and measured (EXP) scour depth and dune height at t =60, 180, 300, 480, 720 and 1200 seconds.

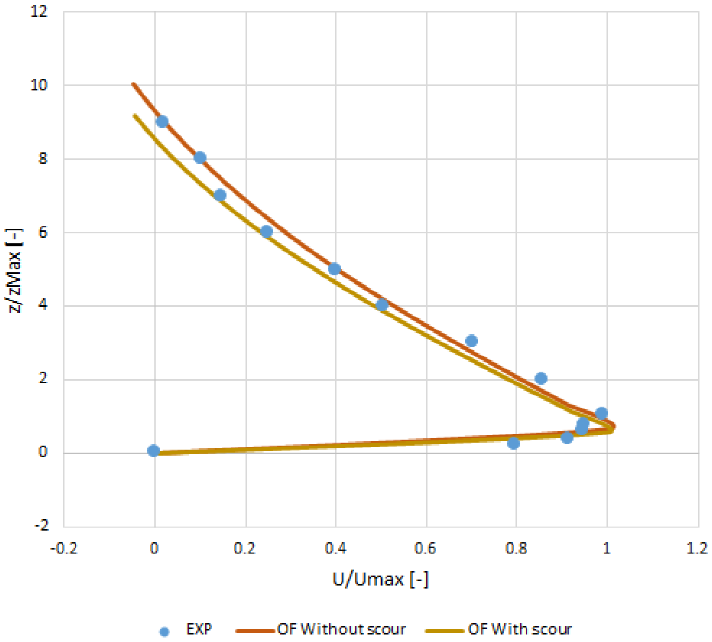

Figure 8. Comparison between simulated (OF) and measured velocity (EXP) profile in the vertical direction (z-axis) at x = 0.6 m located on the rigid bed.

4.3 OpenFOAM – SWASH COUPLING

In this section the results of the one-way coupled model for a case with propagating waves over a fixed bathymetry are given. The results of the wave propagation are shown in Figure 9. The surface elevation is compared to the theoretical solution and a stand-alone SWASH simulation at a distance of 6.0 m from the coupling location. The simulations show very good agreement. Both wave crest and wave trough are close to theoretical results.

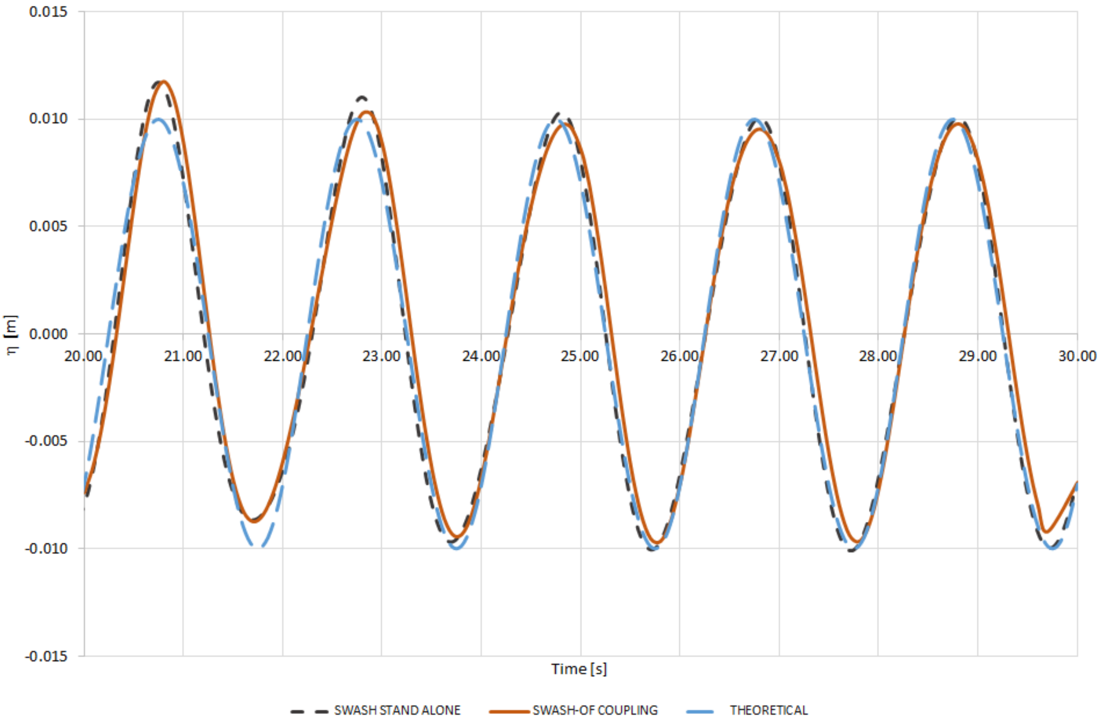

Figure 9. Comparison of the surface elevation between the theoretical profile, a stand-alone SWASH run, and the one-way coupled model run.

5 CONCLUSIONS AND FUTURE WORK

Due to the risk of sea level rise and increasing storminess, the Belgian coast may be at risk for overtopping wave events. Therefore, the Belgian coast is being adapted to guarantee enough coastal safety. This is done by beach nourishing as well as the placement of storm walls. However, the design of coastal structures such as for example storm walls is not straightforward due to the lack of knowledge on the effect of the shallow foreshore on wave loading forces and wave overtopping.

This paper describes a first step to a prediction methodology to accurately assess wave loading forces on structures and buildings on the dike for the case of a very shallow foreshore. The numerical model is being developed in OpenFOAM and SWASH.

The capability of the CFD software OpenFOAM to accurately predict wave loading forces on a vertical wall for the case of regular waves propagating on a very shallow foreshore has been demonstrated. Simulated surface wave elevations show a very good agreement with the experimental data as well as the wave loading forces.

The implementation of the sediment transport module in OpenFOAM for simulating local scour with free surface and automatic mesh deformation is discussed. The free surface is modelled by the volume of fluid method, while scour is modelled by a moving mesh method. Numerical simulations for a submerged wall jet are carried out and compared with experimental data. Good results have been obtained. The velocity profile compares well to the measurements. The scour depth can be estimated fairly well, the maximum dune height is however a little overestimated. Further research and validation of the sediment transport module is necessary and will be carried out in the CREST project.

Finally a one-way coupling methodology between a non-linear shallow water equation model SWASH and the CFD model OpenFOAM is proposed to be able to simulate the wave transformation over the beach until the coastal structures in order to reduce computational costs without losing accuracy. The proposed coupling methodology uses the MPI message passing protocol to send the surface elevation and the horizontal wave kinematics from SWASH to OpenFOAM at the interface between the two domains. At the inlet boundary of the OpenFOAM domain the velocity and surface elevation obtained from the SWASH calculation are imposed. The coupled model is programmed with the MPI environment, where three processes (OpenFOAM, SWASH and Fortran) are transferring and processing data. The capability of the coupled model to propagate regular waves over a fixed flat bottom is demonstrated in the paper. The obtained results from the coupled model are in good correspondence with the theoretical solution and the stand-alone SWASH results. This approach seems promising to be able to simulate more complex wave states in the future. Future work will also include more validation of the proposed model and simulation of real test cases.

ACKNOWLEDGEMENT

This research is part of the fundamental strategic research project CREST – Climate Resilient coast project (VLIZ, 2018), funded by the Flemish Agency for Innovation by Science and Technology.

REFERENCES

Altomare, C., Suzuki, T., Dominguez, , J.M., Crespo, A.J.C., Gomez-Gesteira, M., Caceres, I., 2014. A Hybrid Numerical Model for Coastal Engineering Problems. *Proceedings of the 34th International Conference on Coastal Engineering (ICCE2014),* Seoul, Korea, June 2014.

Altomare, C., 2018., Multi-Directional Wave Basin for Shallow Foreshore Applications. *Proceedings of the 7th International Conference on the Application of Physical Modelling in Coastal and Port Engineering and Science (Coastlab18)*. Santander, Spain, May 22-26, 2018 (present edition).

Brown, S. A., Greaves, D. M., Magar, V.,. Conley, D.C., 2016. Evaluation of Turbulence Closure Models under Spilling and Plunging Breakers in the Surf Zone. *Coastal Engineering*, 114, 177–93.

Chatterjee, S.S., Ghosh, S.N., Chatterjee, M., 1994. Local Scour Due to Submerged Horizontal Jet, *Journal of Hydraulic Engineering.*, ASCE, 120(8), 973-992.

Chen, X., 2016. Impacts of Overtopping Waves on Buildings on Coastal Dikes, doctoral thesis, *Delft University of Technology*.

Engelund, F., Fredsoe, J., 1976. A Sediment Transport Model for Straight Alluvial Channels. *Nordic Hydrology,* 7, 293–306.

Gruwez, V., Vandebeek, I., Altomare, C., Suzuki, T., Vanneste, D., De Roo, S., Troch, P., Kortenhaus, A., Verwaest, T., Mertens, T., Monbaliu, J., 2017. CREST-Climate resilient coast. Advanced modelling of overtopping risks in coastal urban areas - Development and validation of the coupled model chain, Project report, *Ghent University, University of Leuven, Flanders Hydraulics Research*, Ghent, Belgium.

Gruwez, V., Vandebeek, I., Kisacik, D., Streicher, M., Altomare, C., Suzuki, T., Verwaest, T., Kortenhaus, A., Troch, P., 2018. 2D Experiments of Wave-Interactions with a Dike on a Very Shallow and Mildly Sloping Foreshore. *Proceedings of the 7th International Conference on the Application of Physical Modelling in Coastal and Port Engineering and Science (Coastlab18)*, Santander, Spain, May 22-26, 2018 (present edition).

Higuera, P., Lara, J. L., Losada, I. J., 2013a. Realistic Wave Generation and Active Wave Absorption for Navier-Stokes Model: Application to OpenFOAM, *Coastal Engineering,* 71, 102–118.

Higuera, P., Lara, J. L., Losada, I. J., 2013b. Simulating Coastal Engineering Processes with OpenFOAM®, *Coastal Engineering*, 71, 119-134.

Marieu, V., Bonneton, P., Foster, D., Ardhuin, F., 2008. Modeling of vortex ripple morphodynamics, *Journal of Geophysical Research: Oceans*, 113 (C9).

Roenby, J., Larsen, B.E.,, Bredmose, H., Jasak, H., 2017. A New Volume-Of-Fluid Method In OpenFOAM, *Proceedings of the VII International Conference on Computational Methods in Marine Engineering (Marine2017)*, Nantes, France, May15-17, 2071.

Soulsby, R., 1997. Dynamics of Marine Sands : A Manual for Practical Applications, *Telford,* London.

Suzuki, T., Verwaest, T., Hassan, W., 2011. The Applicability of SWASH Model for Wave Transformation and Wave Overtopping: A Case Study for the Flemish Coast. *Proceedings of the 5th International Conference on Advanced Computational Methods (Acomen2011),* Liege, Belgium, November 14–17, 2011.

Van Rijn, L.C., 1987. Mathematical Modeling Of Suspended Sediment In Nonuniform Flows, *Journal of Hydraulic Engineering,* ASCE, 112(6), 433-455.

Verbrugghe, T., Kortenhaus, A., Troch, P., Dominguez, J., 2017. A non-linear 2-way coupling between DualSPHysics and a wave propagation model, *Proceedings of Spheric2017,* Ourense, Spain, June 13 - 15, 2017.

Weller, H. G., Tabor, G., 1998. A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques, *Computers in Physics,* 12 (6), 620-631.

Zhou, L., 2017. "Numerical Modelling of Scour in Steady Flow, Doctoral thesis, *Université de Lyon.*

Zijlema, M., Stelling, G., Smit, P., 2011. SWASH: An Operational Public Domain Code for Simulating Wave Fields and Rapidly Varied Flows in Coastal Waters, *Coastal Engineering*, 58, 992–1012.

VLIZ (Flanders Marine Institute), 2018. The CREST project webpage. http://www.crestproject.be/en.