Coupling Large-Scale and Detailed Site Flooding Simulations

Niels Montanari[a], Ramprasad Sampath[a],
Donna Calhoun[b], Steven Prescott[c], Curtis Smith[c]
[a] Centroid LAB, Inc., Mar Vista, CA, USA
[b] Boise State University, Boise, ID, USA
[c] Idaho National Laboratory, Idaho Falls, ID, USA

Abstract: When simulating large-scale flooding scenarios, 2.5D solution methods are often used to give a good estimation of flood levels for a specified site. Although computationally more involved, 3D simulation methods are able to more accurately characterize the flood flows. We designed a methodology for spatially coupling both 2.5D and 3D simulation methods based on, respectively, a shallow water model and a Navier-Stokes model discretized with smoothed-particle hydrodynamics. A demonstration of this coupled-simulation method is done with an upstream dam failure scenario which then floods a nuclear facility, impacting flood protection equipment and affecting external components. The computational efficiency of the 2.5D model is leveraged for simulating the large area between the dam location to the vicinity of the nuclear facility, whereas the 3D model is used near the nuclear facility, where the risk assessment would greatly benefit from a more detailed characterization of the flow and a more advanced analysis.

Keywords: External Flooding, Model Coupling, Risk Assessment, SPH.

1. INTRODUCTION

1.1. External Flooding Risk Assessment

External flooding may occur from various phenomena [1]. The most common cause of flood is heavy precipitation, producing runoff over the ground surface and into rivers. Riverine flooding may be induced by dam breaches and failures initiated by seismic events, storm surges, structural failures and so on. In coastal areas, tsunamis are a major cause of flooding, and strong storm winds may give rise to a storm surge. In bay and lake areas, overtopping and subsequent flooding may also occur due to standing wave phenomena known as seiches.

Flooding hazards may threaten the integrity of systems, structures, and components (SSC) which are critical for the safety of a nuclear or chemical facility. Plant operators, plant manufacturers, and policy makers may assess the risks based on a bounding analysis and/or a probabilistic analysis. In the case of a bounding assessment, theorized worst-case scenarios are being considered and conservative assumptions, inputs, and methods are used. Conversely, a probabilistic risk assessment (PRA) also estimates the frequency of occurrence of flooding hazards and probabilities of SSC failures. In such case, deterministic simulations as well as Monte-Carlo simulations are performed. Dynamic PRA methods make extensive use of physical simulations and, to be practical, require both computational speed and a detailed representation.

1.2. Fluid Modeling

1.2.1. The 3D Navier-Stokes (NS) Equations

In most cases, water and its dynamics are accurately modeled as a Newtonian, incompressible fluid and can be described using the 3D Navier-Stokes (NS) equations for isothermal flows. When dealing with flooding, the surface tension effects and the influence of air on the water dynamics are usually neglected. The corresponding NS equations read, in their usual, Eulerian velocity-pressure formulation, as:

$$\begin{cases} \frac{\partial \mathbf{v}}{\partial \mathrm{t}} + (\mathbf{v}.\boldsymbol{\nabla})\mathbf{v} = -\frac{1}{\rho}\boldsymbol{\nabla}\mathrm{p} + \nu\boldsymbol{\nabla}^2\mathbf{v} + \mathbf{g} \\ \boldsymbol{\nabla}.\mathbf{v} = 0 \end{cases} \quad (1)$$

where $\mathbf{v}$, p, ρ, ν and $\mathbf{g}$ stand, respectively, for the velocity, the pressure, the fluid density, the kinematic viscosity of the fluid, and the gravitational acceleration. The system of equations must, of course, be complemented with initial and boundary conditions. Solving for the 3D NS equations is computationally very involved, restricting its practical use to simulation domains of small-to-moderate scale is desirable.

1.2.2. 2.5D Models of Reduced Complexity and the Shallow-Water (SW) Equations

For a large class of flooding problems, some fluid models of reduced complexity are good alternatives to solving the 3D NS system [2]. Hence, the physical models conventionally in use in flooding risk assessment contain a significantly higher level of assumptions and approximations [1]. These models are derived from the NS equations with additional assumptions from the following sub-set: irrotational flow; shallow-water (SW) approximation, in which the horizontal length scale is assumed much larger than the vertical one; and hydrostatic approximation, in which the vertical acceleration is assumed negligible compared to the gravity effects. These lead to strong restrictions on the velocity field and to a “stratification” of the flow: the vertical speed is zero, while the horizontal components follow a prescribed vertical distribution. As such, we will employ the term “2.5D” rather than “3D” for qualifying these models and simulations; note that the term “2D” is often found in the literature, although it is somewhat inaccurate as it implies that the vertical dimension is not modeled at all. Furthermore, some models also ignore all dissipation effects, by neglecting the fluid viscosity and the ground friction. Due to all these assumptions and approximations, complex flows, such as those involving breaking waves, 3D turbulent effects, or interaction with structures, must be considered as outside of the validity domain of these models. However, from a computational point of view, the usage of a 2.5D model can decrease the computational burden by one or even several orders of magnitude.

The class of 2.5D fluid models of reduced complexity include: the hydrostatic NS models, the Boussinesq-type models, and the SW models such as those represented by the Saint-Venant system. Although the methodology presented in the present paper is largely applicable to most 2.5D models, the focus here is more specifically on the SW sub-class for which the vertical distribution of the velocity is uniform and depth-averaged. We consider the following SW system, formulated in its conservative, Eulerian form as:

$$\begin{cases} \frac{\partial(\mathrm{h}\mathbf{v}_{\boldsymbol{h}})}{\partial \mathrm{t}} + \boldsymbol{\nabla}_{\boldsymbol{h}}.\left(\mathrm{h}\mathbf{v}_{\boldsymbol{h}}{}^2 + \frac{1}{2}\mathrm{g}\mathrm{h}^2\mathbf{I}_h\right) = \mathbf{0} \\ \frac{\partial \mathrm{h}}{\partial \mathrm{t}} + \boldsymbol{\nabla}_{\boldsymbol{h}}.(\mathrm{h}\mathbf{v}_{\boldsymbol{h}}) = 0 \end{cases} \quad (2)$$

where the subscript h denotes that fact that only the horizontal components are considered, and $\mathbf{I}_h$ represents the 2D identity matrix. The quantities $\mathbf{v}_{\boldsymbol{h}}$, h, and g stand, respectively, for the depth-averaged horizontal velocity, the fluid depth, and the magnitude of the gravitational acceleration.

1.2.3. Smoothed-Particle Hydrodynamics (SPH)

To obtain an approximate solution of the full 3D NS equations, a numerical method for discretizing the spatial differential operators (i.e., the gradient, the divergence, the Laplacian, and other such operators) needs to be employed. This work considers the case of the smoothed-particle hydrodynamics (SPH) method [3, 4]. In comparison to more conventional methods in computational fluid dynamics (e.g., the usual finite-difference, finite-volume and finite-element methods), SPH does not use any explicit mesh. Instead, the fluid is represented as a set of particles with a given mass, position, velocity, and potentially other properties, fixed or evolving in time. SPH is most commonly applied in a Lagrangian setting, in which the fluid particles are not fixed at a given position but move with the flow. This Lagrangian, mesh-free nature makes SPH particularly well suited for simulating violent flows, for handling complex geometries, and for modeling spray, splashes, and impact forces often present in flooding scenarios.

1.3. Coupling Between 2.5D and 3D Fluid Models

Modeling and simulating a riverine flood, in particular, brings unique issues. Not only is the expanse of land area concerned by the flood very large, but also the duration of the event may be long. These make it a challenge to produce a detailed analysis on the entire domain and for the duration of the flooding event. To balance between accuracy and computational cost in an external flooding risk analysis, a natural idea is to spatially couple a 3D NS model and a 2.5D model of reduced complexity. The former would be used where a detailed characterization of the flood flow is desire, whereas the computational efficiency of the latter would be leveraged for simulating the large-scale domain elsewhere.

In practice, a flood flow has a clear flow direction and does not feature any backflow except at few locations. Rather than designing a two-way coupling scheme, we consider in this work the one-way coupling between the 2D external flow and the 3D flow near inundated structures. Since in most cases a detailed description of the flow is desired downstream at the site and SCC locations, the focus is more specifically on the 2.5D-to-3D model coupling. We designed and implemented a method to spatially couple a SW model and a SPH-discretized NS model. In Section 2, we present a high-level description of how our method works. In Section 3, we showcase its usage on a dam-failure flooding scenario.

2. COUPLING METHODOLOGY

2.1. Input 2.5D Data

The data used or output by the 2.5D simulation model and also relevant to the 3D SPH simulation can be split into the following categories:

- Topography data, comprising the elevation of the ground surface or sea floor and sampled at a sufficient number of locations;
- Gauges, representing the positions where to transmit the SW data and defining the spatial boundary between the 2.5D and 3D simulation domains;
- SW data, consisting of the water height and horizontal velocity fields sampled at the gauge locations and at each time step of the 2.5D simulation.

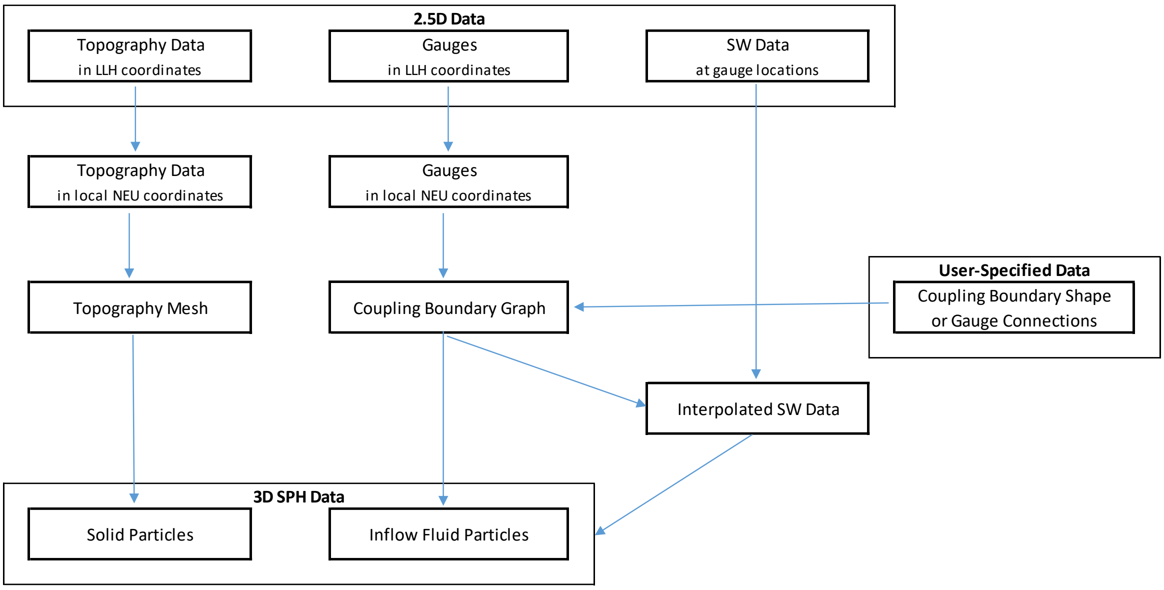
The 3D SPH model cannot use these pieces of data directly but must translate them into a particle-based representation meaningful in the SPH framework for modelling appropriate inflow and wall boundaries. The sub-sections that follow describe the translation procedure. Figure 1 shows a diagram illustrating the overall coupling method.

2.2. Geographic Coordinate System Conversion

In most cases, the topography data (ground surface elevation) and 2.5D SW data (water height and horizontal velocity) are sampled at locations specified in latitude-longitude-height (LLH) coordinates. These coordinates are defined for a given geodetic system, which approximates the Earth's surface as a certain sphere or ellipsoid. Some widely used geodetic systems are: the world geodetic system (WGS) 84, defined with respect to the average of stations all over the world and used by the global positioning system (GPS); and the North American datum (NAD) 83, designed to be particularly accurate for points on the North American Plate.

Conversely, the much smaller scale of the domain simulated in 3D is more simply handled using 3D Cartesian geographic coordinates. One convenient choice of such coordinate systems is, for instance, the local north-east-up (NEU) coordinate system, which is defined on a local plane tangent to the Earth's

Figure 1: Diagram summarizing the 2.5D-to-3D model coupling method.



surface. Therefore, making use of the topography and SW data in the 3D simulation may require converting the locations from a LLH coordinate system to a local Cartesian coordinate system. In what follows, we will use the local NEU coordinate system for the local, 3D SPH simulations.

The conversion of the geographic coordinates system cannot be directly used. It also requires: 1) an intermediate conversion to the Earth-centered, Earth-fixed (ECEF) geocentric coordinates, which correspond to a Cartesian system that is geocentric instead of being local; and 2) a reference location to define the local NEU coordinate system. The overall conversion process consists in the following steps:

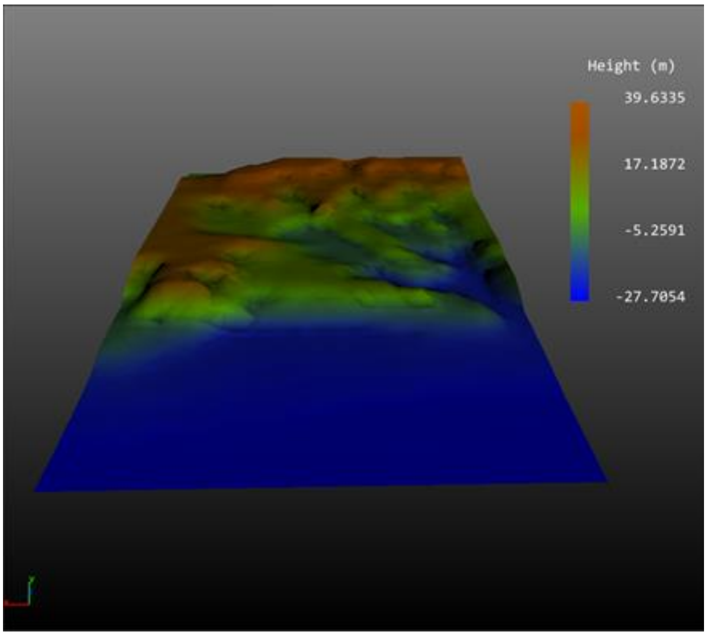
1. A reference location expressed in LLH coordinates is calculated. A natural choice is to consider the average location of the topography data and/or the gauges. A key-point here is that a common reference location must be for the conversion of both the topography data and gauges.
2. The reference location is converted from LLH coordinates to ECEF coordinates.
3. The coordinates of all topography data and gauges are converted from the LLH system to the ECEF system.
4. Using the reference location, the coordinates of all topography data and gauges are converted from the ECEF system to the local NEU system.

2.2. Topography Mesh & Solid Particle Generation

There are various approaches for modelling wall boundaries in SPH [4]. Some are purely mesh-based; others rely on regenerating at each time step solid particles at positions mirroring the neighbouring fluid particles; and a last usual class of techniques is to uniformly sample the ground surface with solid particles at the initial time only. We restrict the discussion here to this last strategy.

To algorithmically sample the ground surface with particles, it is convenient to first produce a mesh representing the topography. Starting from the local NEU coordinates of the topography data, a 2D Delaunay triangulation is performed. A Delaunay triangulation yields a regular mesh with desirable properties, and many efficient algorithms exist (e.g., the Bowyer-Watson algorithm [5,6]). Note that only the north and east components are used when triangulating. The up component is then used to adjust the elevation of the mesh vertices. An example of such resulting topography mesh is shown in Figure 2.

Figure 2: Visualization of a topography mesh generated after geographic coordinate conversion of the topography data and Delaunay triangulation. The color represents the relative elevation of the ground surface and sea floor in comparison to the average elevation of the topography.



In SPH, the presence of solid particles is needed to produce appropriate pressure forces and prevent the neighbouring fluid particles from penetrating and leaking through the ground. The accuracy of these pressure forces is dependent on how many layers of solid particles were created; for most SPH schemes, two to six layers are required for full accuracy, although from our experience a single layer may already be enough prevent penetration unless the time step is excessively large. To sample the ground surface with particles, we propose filling each triangle of the topography mesh with regularly spaced particles. Each additional layer of solid particles can be generated simply by copying the particles of the previous layer and offsetting their vertical positions by the particle size.

2.3. Coupling Boundary Modeling

The vertical boundary separating the spatial domains of the 2.5D and 3D simulation models is modeled as a graph and defined using the gauge locations. The gauges serve as the graph vertices, and an edge is used to represent each pair of adjacent gauges. In the case of a boundary shaped according to a simple analytical shape (e.g., a linear segment or a portion of a rectangle), the edges connecting the gauges can be implicitly determined; conversely, if the boundary has an arbitrary shape, the connections have to be explicitly specified by the user. Note that the graph is treated as simple and directed, may have several connected components, and each component may feature a cycle.

2.4. Inflow Particle Generation

At each time step, new fluid particles have to be appropriately generated at the coupling boundary to account for the inflow. The number of fluid particles is determined by the water heights predicted by the SW simulation and the length of the coupling boundary. To minimize the deviation from mass conservation while avoiding any significant spurious pressures, the fluid particles are generated along the coupling boundary at regularly spaced intervals. Their velocities are prescribed according to the horizontal velocities of the SW data.

Because many inflow particles are created between gauges, which are the only locations for which SW data are available, linear interpolation in space for each gauge pair is performed to better sample the SW data along the coupling boundary. Similarly, the SW data are time series that, in practice, do not correspond to the much smaller time steps of the 3D simulation. Linear interpolation in time is thus performed to improve the SW data sampling along the time dimension. To summarize, the number of inflow particles along the vertical direction and their velocities are locally determined using a water height and a horizontal velocity coming from linear interpolation in both space and time of the SW data of the closest pair of gauges. The graph modeling the coupling boundary is traversed to perform all these interpolations and to generate all new fluid particles.

3. DAM-FAILURE FLOODING SIMULATION

We demonstrate the 2.5D-3D model coupling approach by simulating the Teton Idaho dam failure of June 5th, 1976. A fictional nuclear power plant (NPP) facility is added at a location near the Teton river, downstream from the dam failure, as depicted in Figure 3. The flood from the dam location to the vicinity of the NPP site is simulated with a 2.5D model, whereas at the NPP site itself a 3D model is used. The 2.5D simulation domain is as large as a rectangular area of $5\text{x}25\ \text{km}^2$; the domain of the NPP site is defined by a squared area of $300\text{x}300\ \text{m}^2$. Along the coupling boundary, 80 gauges uniformly spaced every $15\ \text{m}$ are set to transmit the SW data when coupling the models and performing the 3D simulation.

For the 3D simulation, we used Neutrino [7], a platform for particle-based simulation and visualization, and its implicit incompressible SPH (IISPH) solver. Other kinds of detailed flooding simulations have been performed with Neutrino and its IISPH solver, such as a tsunami impact causing flooding at a coastal NPP site [8] and an internal flooding induced by a pipe break [9, 10]. A relatively coarse particle size of $1\ \text{m}$ is chosen, to ensure fast simulations. Yet, given the sizes of the 3D domain, water levels, and structures, it is already sufficient to yield a significant amount of details. The 1-min simulation run took about an hour on a 12-core CPU machine. More than 100,000 fluid particles have been generated throughout the run.

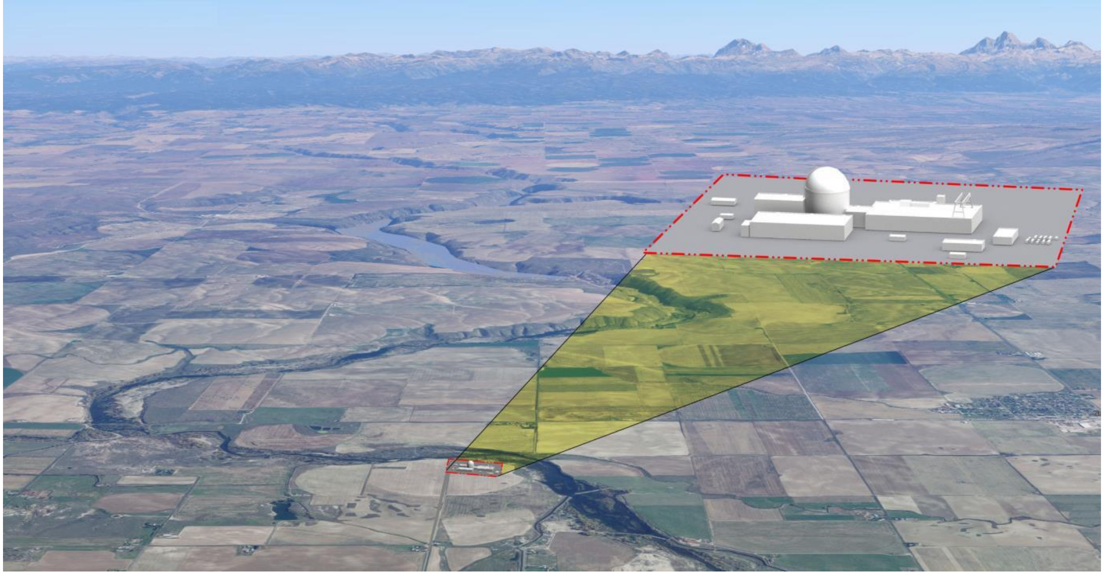
For the 2.5D simulation, we use GeoClaw, an Eulerian, Cartesian grid SW simulation code based on a finite-volume scheme with adaptive mesh refinement [11]. It has been benchmarked against the Malpassat dam-break flood and showed very good agreement with historical records [12]. It was also benchmarked at the National Tsunami Hazard Mitigation Program (NTHMP) benchmarking workshop [13]. Some important features of this numerical method upon which GeoClaw is based are that: 1) conservation of mass and momentum is ensured (up to machine precision); and 2) the cell size is locally and dynamically refined in flooded areas, while in dry areas only a very coarse mesh is needed. A base cell size of $54\text{x}19\ \text{m}^2$ and four levels of refinement, with factors 2, 4, 4, 4, were used. The simulation runtime was in the order of a few hours.

Figure 4 shows the topography and water levels computed by the 2.5D simulation. At the initial time, the water height behind the dam is set to $115\ \text{m}$. The dam is assumed to fail instantaneously, releasing a sudden wall of water (top sub-figure). Note that a more realistic initiation would consider a gradual rupture of the dam and model the erosion process. The flood quickly flows downstream throughout the riverbed and onto the flood plain. Around $20\ \text{min}$ after failure, the flood reaches the NPP site (middle sub-figure). At the end of the simulation, $45\ \text{min}$ after failure, most of the simulated land further downstream of the NPP site is flooded (bottom sub-figure).

Figure 5 illustrates the 2.5D-3D model coupling and shows how the flood propagates inside the NPP site. The time scale is much shorter, as less than a minute of 3D physical simulation is needed. The inflow starts at the upper-right corner of the square boundary, with the flood moving first towards the lower-left corner (top sub-figure). Quickly, the first structure of the NPP is impacted by the flood (middle sub-figure), then is overtopped at the time the flood reaches the buildings of the second reactor (bottom sub-figure).

An important source of uncertainty and potential inaccuracy of the simulation is that both the GeoClaw model and the SPH model use the Manning coefficient to characterize the surface roughness. The Manning coefficient being largely phenomenological, it is difficult to set it an accurate value. Furthermore, the time-dependent behavior of the dam rupture is not modeled. A more realistic initiation would consider a gradual rupture of the dam and model the erosion process.

Figure 3: View of the Teton river with the addition of a fictional NPP facility.



4. CONCLUSION

We have designed a method for spatially coupling both 2.5D and 3D simulation methods based on, respectively, a shallow water wave equation (SW) model and a SPH-discretized Navier Stokes (NS) model. We showed how the fluid and topography data used by the shallow water code can be translated into a particle-based representation for modeling open and wall boundaries that is meaningful in the SPH framework. Using Neutrino and GeoClaw, we performed a complete flooding simulation from source of the dam failure to site of the nuclear power plant (NPP). A demonstration of this coupled-simulation method is done with an upstream dam failure scenario which then floods a nuclear facility, impacting flood protection equipment and affecting external components. The computational efficiency of the SW model is leveraged for simulating the large area lying from the dam location to the vicinity of the nuclear facility, whereas the SPH-NS model is used further on, where the risk assessment would greatly benefit from a more detailed characterization of the flow and a more advanced analysis.

For progressing from external to internal flooding risk analysis, the penetration of the flood from the exterior into the NPP infrastructure and its propagation inside, could be modeled using a Torricelli's law based model and the coupling methodology described in [10]. The presence of a draining system could also be accounted for in the same manner.

Although the 2.5D-3D model coupling method is demonstrated for a dam-break induced riverine flooding scenario, it could also be applied, without any modification needed, to tsunami impact and flooding. In [8], the tsunami waves in the surf zone were modeled as a solitary wave and generated using a piston-like technique. GeoClaw could be leveraged to provide more realistic and accurate wave profiles and boundary conditions at the inlet.

The 3D SPH representation at the site makes it possible to account for the complex interactions between water and solid bodies, including rigid objects, elastic structures, and debris. Similarly, large deformations and fragmentation of the ground could be modeled. The fluid-structure two-way coupling could be done either in a unified way within the SPH framework [14] or by coupling with a solid model based on the finite-element method.

For future development work specifically related to the coupling methodology, we can mention the following avenues:

Figure 4: Topography (in green) and water surface elevations (in blue) in meters as predicted by the GeoClaw simulation when: the dam fails (top), at time = 0; the NPP site becomes flooded (middle), at time = 20 min; the flood has mostly settled down (bottom), at time = 45 min. The black dot represents the location of the fictitious NPP site.

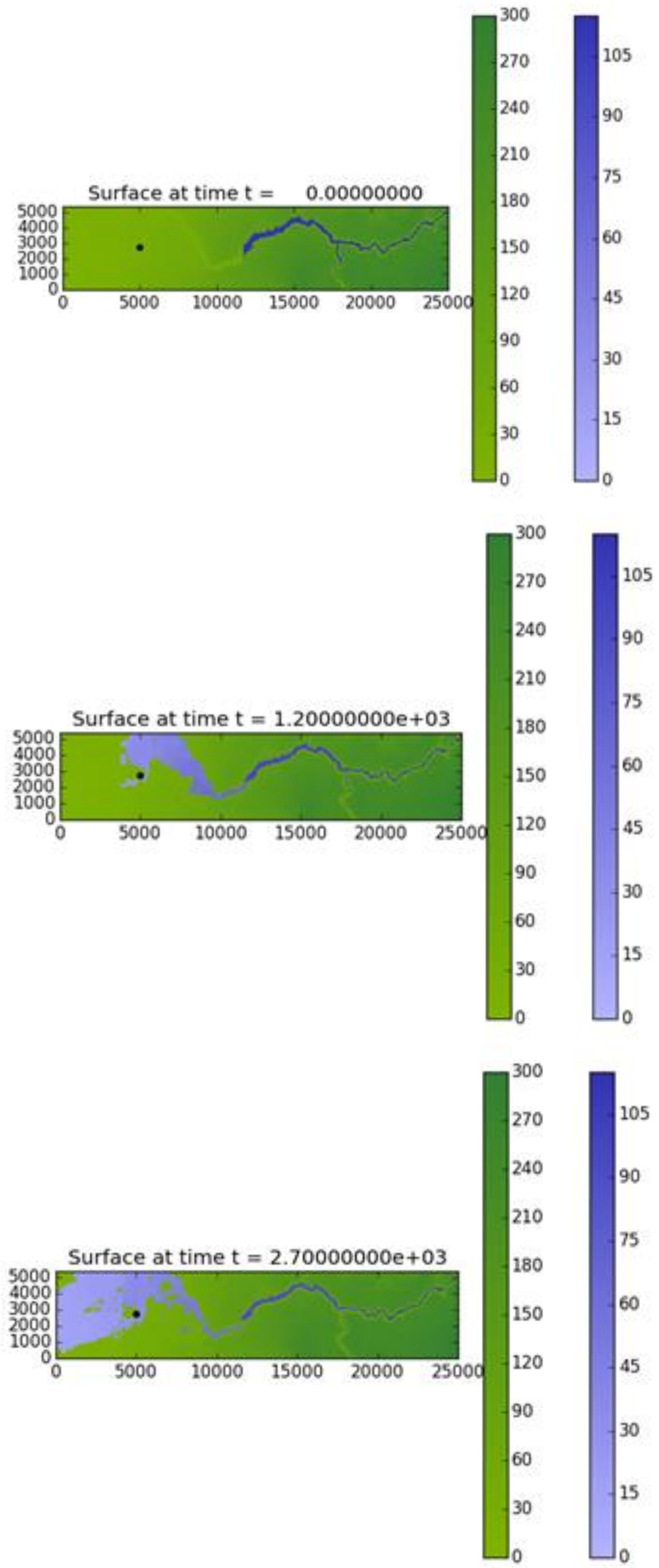
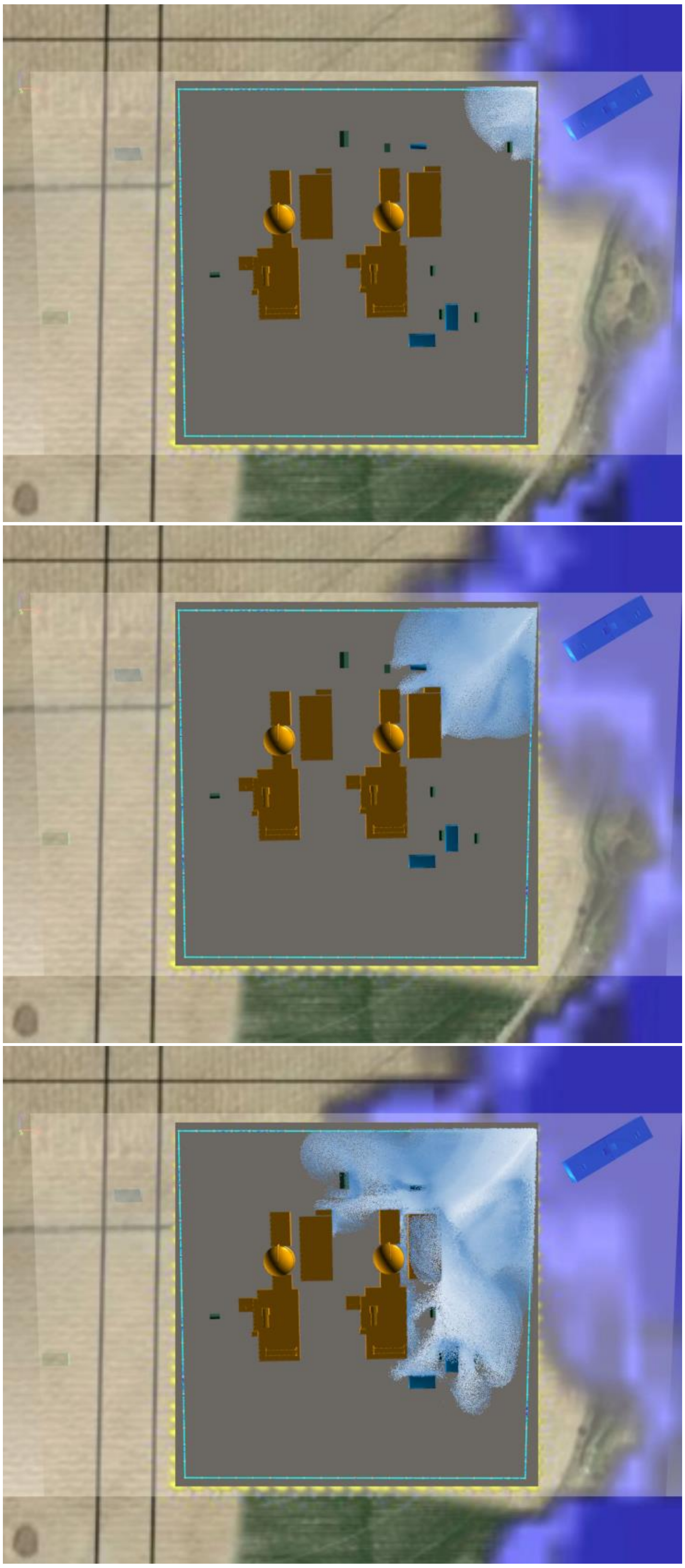


Figure 5: Flooding simulation output at the fictitious NPP site for both Neutrino IISPH (inner domain) and GeoClaw (outer domain) simulations at three different times: approximately 5 s (top), 15 s (middle), and 25 s (bottom) after the start of the inflow to the 3D domain. The coupling boundary is a square represented in cyan. The fluid particles in the inner domain are color-coded according to their speed, from dark blue (zero speed) to white (maximum speed).



- Extensive validation must be performed on test cases such as wave propagation or dam break problems. Numerical diffusion, gain/loss of mass, volume and energy, and possible instabilities at the transition from 2.5D to 3D should be quantified for different regimes.
- Coupling from 3D to 2.5D could be implemented. It should be relatively straightforward, requiring to replace the inflow boundary in the SPH simulation by an outflow boundary and to interpolate the fluid quantities of the SPH particles at the outlet and transmit them to the SW code.
- Full two-way coupling could be realized by applying a Schwarz method with appropriate interface conditions [15] and by using a more general open boundary able to treat the occurrence of backflow. This would, however, certainly require a considerable amount of work.
- A one-way coupling 2.5D-to-3D not only in space but also in time could be implemented. This would enable us to start the 3D SPH simulation with areas already wet at the beginning of the simulation.

Acknowledgements

This work was funded by the U.S. Department of Energy under the Light Water Reactor Sustainability Program.

References

[1] "*Evaluation of Deterministic Approaches to Characterizing Flood Hazards*", EPRI, Palo Alto, CA, USA, 2016, 3002008113.
[2] J. Sainte-Marie, "*Models and Numerical Schemes for Free Surface Flows. Beyond the Saint-Venant System*", Habilitation Thesis, Université Pierre et Marie Curie – Paris VI, 2010.
[3] D. Violeau, "*Fluid Mechanics and the SPH Method: Theory and Applications*", Oxford University Press, 2012, Oxford, United Kingdom.
[4] D. Violeau and B. D. Rogers, "*Smoothed Particle Hydrodynamics (SPH) for Free-Surface Flows: Past, Present and Future*", Journal of Hydraulic Research, volume 54, issue 1, pp. 1-26, (2016).
[5] A. Bowyer, "*Computing Dirichlet Tesselations*", The Computer Journal, volume 24, issue 2, pp. 162-166, (1981).
[6] D. F. Watson, "*Computing the n-Dimensional Delaunay Tesselation with Application to Voronoi Polytopes*", The Computer Journal, volume 24, issue 2, pp. 167-172, (1981).
[7] Neutrino website, *neutrinodynamics.com.*
[8] R. Sampath et al., "*Large-Scale Solitary Wave Simulation with Implicit Incompressible SPH*", Journal of Ocean Engineering and Marine Energy, volume 2, issue 3, pp. 313-329, (2016).
[9] "*Investigation into the Use of Three-Dimensional Modeling Techniques to Assess Internal Flooding Scenarios*", EPRI, Palo Alto, CA, USA, 2017, 3002010673.
[10] N. Montanari et al., "*Coupling Smoothed-Particle Hydrodynamics and Torricelli's Law-Based Hydraulic Models for Flooding Risk Analysis*", Proc. of International Topical Meeting on Probabilistic Safety Assessment and Analysis, (2017).
[11] M. J. Berger et al., "*The GeoClaw Software for Depth-Averaged Flows with Adaptive Mesh Refinement*", Advances in Water Resources, volume 34, issue 9, pp. 1195-1206, (2011).
[12] D. L. George, "*Adaptive finite volume methods with well-balanced Riemann solvers for modeling floods in rugged terrain: Application to the Malpasset dam-break flood (France, 1959)*", International Journal for Numerical Methods in Fluid, volume 66, issue 8, pp. 1000-1018, (2011).
[13] J. Horrillo et al., "*Performance Benchmarking Tsunami Models for NTHMP's Inundation Mapping Activities*", Pure and Applied Geophysics, volume 172, issue 3-4, pp. 869-884, (2015).
[14] A. Khayyer et al., "An enhanced ISPH–SPH coupled method for simulation of incompressible fluid–elastic structure interactions", Computer Physics Communication, In Press, (2018).
[15] E. Blayo and A. Rousseau, "*About Interface Conditions for Coupling Hydrostatic and Nonhydrostatic Navier-Stokes Flows*", Discrete and Continuous Dynamical Systems – Series S, American Institute of Mathematical Sciences, volume 9, issue 5, pp. 1565-1574, (2016).