ICHE 2014, Hamburg - Lehfeldt & Kopmann (eds) - © 2014 Bundesanstalt für Wasserbau ISBN 978-3-939230-32-8

Modelling the Holtenau Ship Lock with SPH

E. Rustico & T. Brudy-Zippelius
Federal Waterways Engineering and Research Institute, Karlsruhe, Germany

A. Hérault
Conservatoire National des Arts et Métiers (CNAM), Paris, France

G. Bilotta
Istituto Nazionale di Geofisica e Vulcanologia (INGV), Sezione di Catania, Italy

ABSTRACT: We present a proof of concept application of the Smoothed Particle Hydrodynamics (SPH) numerical method to the modelling of a ship lock. The reference geometry we use is that of the Holtenau ship lock which has been recently modelled using the Volume-Of-Fluid (VOF) approach (Thorenz and Anke, 2013). Compared to the VOF method, SPH shows greater flexibility in the handling of the interaction between the water and the ship body, thanks to its mesh-less, Lagrangian nature, but the overall behavior of the flow shows issues related to the modelling of inflow conditions. The results we present are still preliminary, and improvements are expected with the use of semi-analytical boundary conditions (Ferrand et al., 2012), which allows for more accurate treatment of boundary conditions, including the inflow during the filling phase of the ship lock.

Keywords: Ship lock, SPH, CFD, GPU Computing, GPUSPH

1 INTRODUCTION

The large ship locks of the Kiel-Canal will undergo a significant renovation and it is planned to adopt a through-the-gate filling system. The Waterways Engineering and Research Institute (BAW) was commissioned to evaluate the filling and emptying times of the new system and its impact on the forces acting on the ships, which might undermine the safety of the transit. Figure 1 is an aerial view of the lock complex.

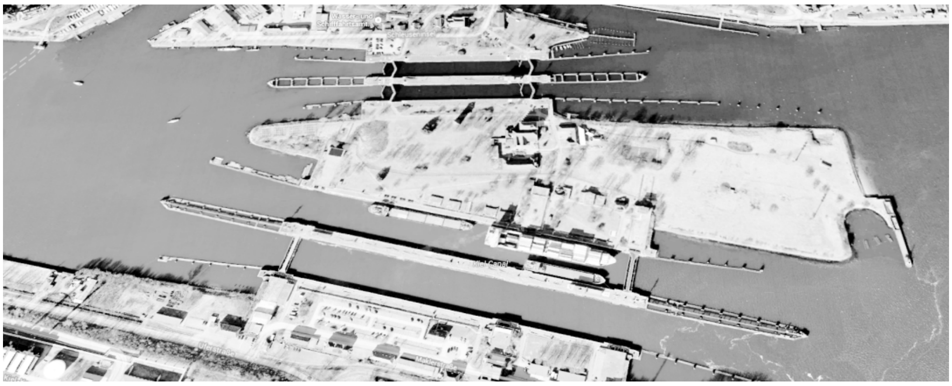

Figure 1. The Kiel Holtenau Lock Complex.

In Thorenz and Anke (2013) a scaled physical model and a purely numerical approach are used to simulate the filling and emptying of the lock. The numerical model used a Volume-Of-Fluid (VOF) Eulerian approach and was in acceptable agreement with the laboratory measurements. The relative movement of the ship with respect to the lock chamber was simulated by grid-morphing or by defining an extra mesh containing the vessel and moving the latter with respect to the background one. The study concluded underlining the relevance of the transversal forces acting on the ships for the planning of the schedule of the valves, especially in extraordinary load conditions. However, it is stated that the most complete way to simulate the behavior of a lock is still a scaled physical model, as the numerical modelling showed several difficulties in respect to the movement of the ship.

In this paper we tackle the same problem with a different numerical method, namely Smoothed Particle Hydrodynamics (SPH). SPH is a Lagrangian meshless model: the fluid is discretized as a set of particles which are free to move with respect of each other. The motion of the particles is driven by the actual equations of motion, in our case by the Navier-Stokes equation. SPH is a highly flexible method and it has been used to model single-fluid problems, multi-fluid, thermal problems and fluid/structure interactions, the latter being particularly of interest in the application to the ship-lock.

For our simulation we rely on GPUSPH (Hérault et al. 2010, Rustico et al., 2014), an implementation of SPH that exploits the high-performance parallel computing power of modern Graphic Processing Units. GPUSPH supports a variety of SPH formulations and can distribute computations across multiple GPUs, on the same as well as on multiple machines in a network, which allows the simulation of large scale problems at high resolutions. Particularly of interest for our application is the support for floating objects, which allows us to model the ship in the basin and its fully coupled interaction with the water during operation.

2 THE SPH NUMERICAL METHOD

2.1 *Mathematical model*

Smoothed Particle Hydrodynamics (SPH) is a meshless Lagrangian numerical method for computational fluid-dynamics (Monaghan, 2005) that has recently seen applications in a number of fields, including geophysics, medicine, hydraulics and a variety of industrial applications. SPH has been used to solved purely dynamic problems for Newtonian flows as well as thermal problems and to model non-Newtonian fluids. Among the benefits of the standard SPH formulations we have the direct computation of all properties of the fluid, implicit surface tracking and the ability to easily manage free-surface problems.

In SPH, the fluid is discretized as a set of particles, each representing a virtual volume of fluid and thus carrying all the relevant properties. The value of any field A at a point $\boldsymbol{r}$ of the domain is determined by convolving the field value at the particle locations with a smoothing kernel $W(\cdot, h)$ with smoothing length h:

$$A(\boldsymbol{r}) \approx \sum_j m_j \frac{A_j}{\rho_j} W(\boldsymbol{r} - \boldsymbol{r}_j, h)$$

where m_j, ρ_j, A_j are respectively the mass, density and value of the field A for particle j, $\boldsymbol{r}$ denotes position and summation is extended to all particles.

Using the properties of convolutions, it can be shown that for gradient computations the expressions can be arranged in such a way that the ∇ operator acts only on the kernel, so that, for example, the continuity equation:

$$\frac{D\rho}{Dt} = -\rho \nabla \cdot \boldsymbol{u}$$

(where D/Dt indicates the material derivative and $\boldsymbol{u}$ is the velocity) takes the SPH form

$$\frac{D\rho_i}{Dt} = \rho_i \sum_j \frac{m_j}{\rho_j} (\boldsymbol{u}_i - \boldsymbol{u}_j) \cdot \nabla_i W(\|\boldsymbol{r}_i - \boldsymbol{r}_j\|, h)$$

(this is actually only one of the possible forms that the continuity equation can take in SPH), showing how the original partial differential equation has been turned into a simple ordinary differential equation in time with a right-hand side that can be directly computed from the mass, density, position and velocity of all the particles.

The smoothing kernel is usually chosen positive (to ensure non-negative density throughout the simulation), symmetric (to ensure first-order accuracy of the method) and with compact support. The latter choice limits the summations to the particles in a small neighborhood of the location, and reduces the computational complexity of the method from $O(N^2)$ to $O(N)$, where N is the total number of particles in the system. It is also convenient to choose a kernel such that the factor

$$F(r, h) = \frac{1}{r} \frac{\partial W(r, h)}{\partial r}$$

can be computed analytically without an actual division by r. This allows us to write the Laplacian of a vector field $\boldsymbol{v}$ in the form

$$\nabla^2 \boldsymbol{v}_i = \sum_j m_j \frac{\rho_i + \rho_j}{\rho_i \rho_j} F_{ij} \boldsymbol{v}_{ij}$$

where $\boldsymbol{v}_{ij} = \boldsymbol{v}_j - \boldsymbol{v}_i$, and $F_{ij} = F(\boldsymbol{r}_{ij}, h)$. The standard SPH formulation models fluid as weakly-compressible, with an equation of state linking pressure P to density ρ. Usually the Tait equation is used in the form

$$P = \rho_0 \frac{c_s^2}{\gamma} \left(\left(\frac{\rho}{\rho_0} \right)^{\gamma} - 1 \right),$$

with ρ the density, ρ_0 the at-rest density of the fluid, γ the polytropic characteristic value and c_s the speed of sound of the fluid.

In most applications, explicit integration methods are used, so that the timestep is limited by the speed of sound. Hence, rather than using the physical speed of sound, a fictitious speed of sound is used, which is at least one order of magnitude higher than the maximum velocity of the fluid in the given problem: this ensures that density fluctuations are kept small (less than 1%) while allowing for larger timesteps.

The Navier-Stokes equation of motion

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{\nabla P}{\rho} + \nu \nabla^2 \boldsymbol{u} + \boldsymbol{g}$$

is then discretized in SPH in the form

$$\frac{D\boldsymbol{u}_i}{Dt} = -\sum_j m_j \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} \right) \nabla_i W_{ij} + \sum_j m_j \frac{4\nu}{\rho_i + \rho_j} F_{ij} \boldsymbol{u}_{ij} + \boldsymbol{g}$$

where ν is the kinematic viscosity coefficient and $\boldsymbol{g}$ represents external forces, in our case gravity.

2.2 *SPH on GPU*

The SPH method exhibits a high degree of parallelization since the state of each particle at any step depends on the state of the same particle and of a relatively small number of neighbors at the previous step. This makes the model ideal for implementation on high-performance parallel computing hardware such as Graphic Processing Units (GPUs).

In the application shown here, we rely on GPUSPH, an implementation of SPH that runs entirely on GPU using the CUDA architecture (Hérault et al., 2010) initially developed at the Sezione di Catania of the Istituto Nazionale di Geofisica e Vulcanologia (INGV-CT) in cooperation with the Department of Civil Engineering of the Johns Hopkins University and recently released as an open-source project. The multi-GPU version of GPUSPH was initially developed at INGV and later perfected in collaboration with the Bundesanstalt für Wasserbau in Karlsruhe, by adding the possibility of exploiting GPU devices distributed across different nodes of a network (Rustico et al., 2014). The ability to run a simulation across multiple GPUs is extremely important, as it allows to model large problems and/or use a very high resolution.

The ship lock model presents a very high ratio between the domain size and the smallest geometry details, ratio that imposes a lower limit for the resolution used to discretize the fluid and thus for the number of particles being simulated. With a domain size of ~330m and minimum meaningful resolution of ~20cm, the number of particles to be simulated is about 30 millions. The simulation of the ship lock model presented in this paper was made possible by exploiting a cluster of 16 GPU devices in 8 nodes at BAW.

3 SPH MODEL OF THE SHIP LOCK

3.1 *Size and resolution*

The ship lock has been modeled in 1:1 proportion. Figure 2 shows the ground plan of the lock while Figure 3 is an overview of the SPH model. The main chamber is 330m long and 45m wide. The smallest geometrical size is the height of the small gate channels (1.3m), along which at least 5 particles must fit; we used a linear particle size of 0.2m. The initial amount of water is 12.5m x 45m x 330m = 185,625m³, which equals to about 23M particles. The total amount of fluid particles does not overcome 30M including the fluid being added in the filling process.

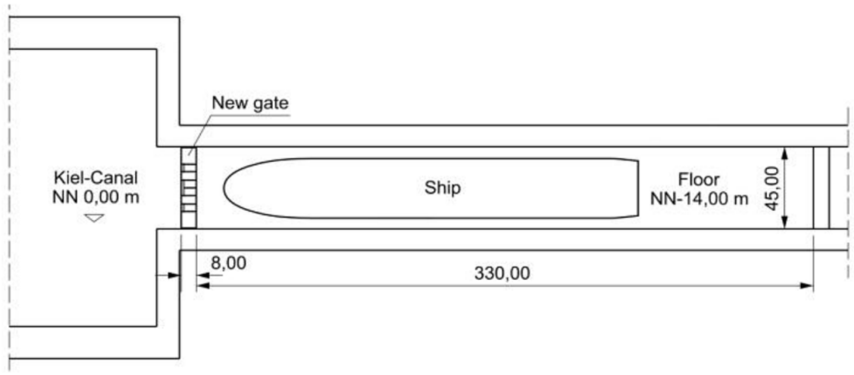

Figure 2. Ground plan of the lock complex.

3.2 *Floating ship*

Thanks to the mesh-less, Lagrangian nature of SPH, fluid/solid interaction and thus the modeling of floating objects is quite straightforward in SPH, compared to e.g. VOF or other mesh-based methods. In our model we use the thin-shell approach to floating objects: the object is modeled as a thin layer of boundary particles. The interaction of these particles with the fluid particles is used to compute the total force and torque acting on the object, which is then used in the integration phase to determine the motion of the object (as a rigid body), and consequently correct the position and velocities of the particles that model its boundary. In GPUSPH, the actual motion of the rigid body is delegated to the open-source Object Dynamics Engine library (Smith, 2002), which can also model impacts between solid objects or impose a variety of constraints on the object motion.

The simulator supports a variety of primitive objects and generic triangular meshes loaded from file. The meshes should feature equilateral triangles only and have a resolution comparable with the one of the fluid. We are in the process of remeshing the original ship model to fit the resolution requirements of the simulator; the tests here reported have been performed by approximating the ship with a cylinder with the diameter equal to the ship width.

The movement of the ship has been constrained to resemble the tests described in Thorenz and Anke (2013): it is free to move only along the Z axis and to rotate only around X and Y axes. The force and torque contributions applied by the fluid to the remaining axes are ignored.

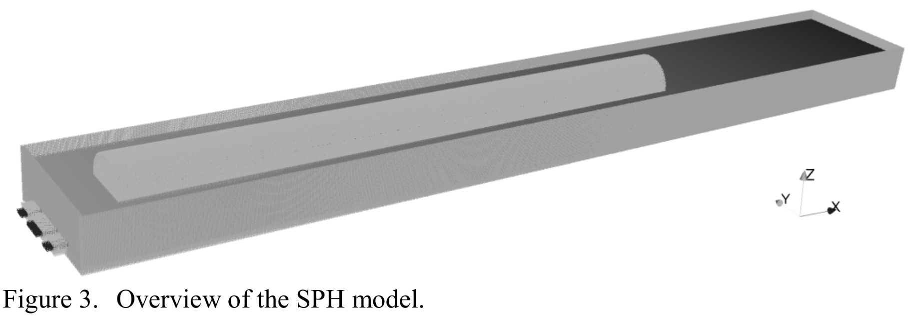

Figure 3. Overview of the SPH model.

3.3 *Boundary conditions*

Three different boundary models have been used for different parts of the ship lock:

- the lateral sides, the floor and the back wall have been modeled with infinite planes applying a Lennard-Jones repulsive force (Hérault et al., 2010);
- the inlet channels and the front wall which contains them has been modeled with Lennard-Jones particles; the behavior is not physical but their influence is small on the overall simulation and they have reduced computational and memory requirements;
- since Lennard-Jones boundary particles do not allow us to compute physically correct forces in the case of slow dynamics, for the cylinder we have used the computationally more expensive dynamic boundary approach of Crespo et al. (2007); the cylinder is filled with three layers of dynamic boundary particles, since we use a Wendland kernel of radius 2 and smoothing coefficient 1.3; the standard SPH discretization of the Navier-Stokes equations is used to compute the forces acting on these particles, and thus ultimately the force and torque acting on the ship, whose motion then prescribes the motion of the boundary particles, as mentioned.

3.4 *Inlet with time-dependent velocity*

Instead of modelling and allocating a tank of water behind the channels, three fluid inlets capable of particle generation have been placed inside the three channels. This allows for halving the number of particles to be simulated and thus decreases the memory requirements and the simulation times.

The current implementation of the inlets is experimental and can impose a time-dependent velocity instead of a pressure value. The inflow rate computed in Thorenz and Anke (2013) has been used to compute the velocity over time; it raises linearly up to a maximum of 74 m^3/s, reached at 228s, and then decreases linearly again to 0 m^3/s in about 392s. Before the inlets start to generate fluid particles, 60 seconds of simulation are run as settling time; this has been estimated as a period sufficient to let the Z force of the ship stabilize.

4 RESULTS

The overall simulation took 72.5 hours for 680s of simulated time (60s settling plus 620s of fluid injection). In this preliminary test we focused our attention on the shape of the injected stream and on the longitudinal forces acting on the ship.

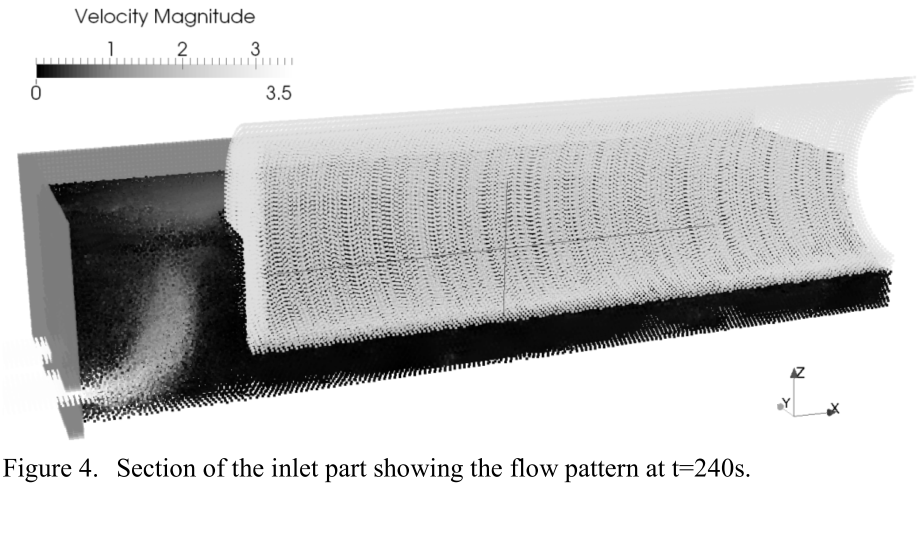

Figure 4. Section of the inlet part showing the flow pattern at t=240s.

4.1 *Flow pattern*

In a first phase, the injected particles tend to move towards the fluid surface soon after being injected (Figure 4). This causes a vertical recirculation in front of the cylinder and, after the peak velocity is reached, the fluid starts to flow under it. We are still investigating the phenomenon to find out the cause of this kind of motion. The same behavior is experienced when the boundaries are modelled with the more physical dynamic particles, thus we can exclude a boundary effect as cause of the phenomenon. The anomalous stream shape can be attributed to the experimental nature of the inlets, where we can impose velocity but not pressure, and the non-hydrodynamic shape of the cylindrical “ship”, which opposes a significant resistance to the inlet stream. We have currently no specific measurements to assess if a similar phenomenon is present also in laboratory experiments.

4.2 *Longitudinal forces on the ship*

The main difference with respect to the values measured in the laboratory tests is the trend of the longitudinal force acting on the ship. The comparison plot is shown in Figure 5.

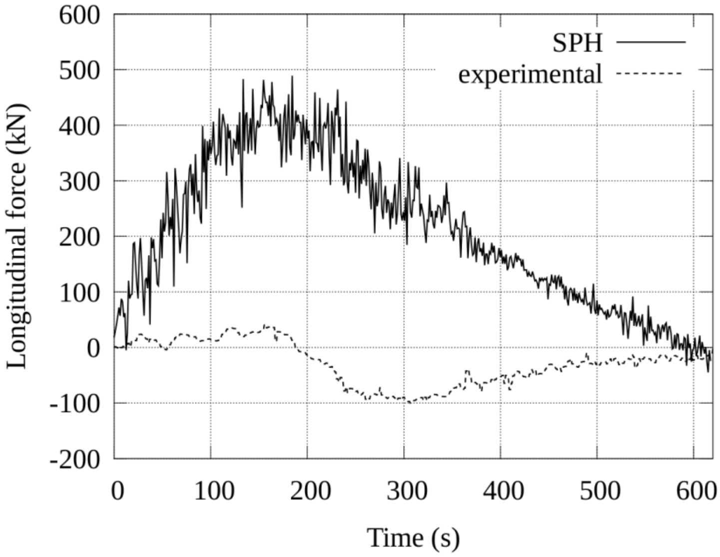

Figure 5. Longitudinal forces in the model and in the laboratory experiments. A positive longitudinal force, by convention, pushes the ship away from the inlet channels.

The force measured in laboratory is positive for the first ~180s of the experiment, with a peak value of about 45kN, and drops to almost −100kN in the next 100s. The sign change is due to the main stream passing under the hull of the ship and pushing from the back of the chamber. Forces in the SPH simulation, instead, exhibit an always positive trend, with a peak force of about 400kN achieved near the peak positive force of the experimental data; it then slowly decreases to zero without going negative.

The difference in magnitude and sign between the laboratory and SPH force measurements is explained by the difference in the stream of inlet fluid and the less hydrodynamic profile of the cylinder with respect to the actual shape of the ship, which plays a key role in hindering the transit of the main stream to the back of the chamber. The measured force, both in the physical experiment and in the SPH simulation, is the sum of all the forces acting on the ship. However, while in the experimental set-up the stream passes around and below the ship and the forces partially cancel out, in SPH, where the stream shape is different, the forces are focused on the front of the ship and do not cancel, resulting in a larger overall forces, always with a positive sign. This difference is currently under examination and is expected to disappear with the implementation of better inflow conditions and the use of the actual ship model.

5 CONCLUSIONS

The tests here described represent, to the best of our knowledge, the first complete three-dimensional SPH model of a ship lock. Although initially planned as simple feasibility tests, the simulations showed interesting results in terms of the shape of the stream and the forces applied to the ship. The discrepancies of the results with respect to laboratory measurements require further investigation.

The future work will focus on loading the mesh of the ship, after the necessary remeshing process, and using the new semi-analytical boundary model (Ferrand et al., 2012) recently implemented in GPUSPH (Bilotta et al. 2014), which allows for more physical boundary interaction and for pressure-driven inlets.

REFERENCES

Bilotta, G., Vorobyev, A., Hérault, A., Mayrhofer, A., Violeau, D. (2014). Modelling real-life flows in hydraulic waterworks with GPUSPH. Proceedins of the 9th International SPHERIC Workshop.

Crespo, A. J. C., Gómez-Gesteira, M., Dalrymple, R. A. (2007). Boundary conditions generated by dynamic particles in SPH methods. CMC-TECH SCIENCE PRESS.

Ferrand, M., Laurence, D., Rogers, B.D., Violeau, D., Kassiotis, C. (2012). Unified semi-analytical wall bounary conditions for inviscid, laminar or turbulent flows in the meshless SPH method. Int. J. Num. Meth. Fluids, Vol 71, No. 4, pp.446-472.

Hérault, A., Bilotta, G., Dalrymple, R.A. (2010). SPH on GPU with CUDA. J. Hydr. Res. 48:74–79.

Hérault, A., Bilotta, G., Vicari, A., Rustico, E., Del Negro, C. (2011). Numerical simulation of lava flow using a GPU SPH model. Annals of Geophysics, Vol. 54, No. 5

Monaghan, J.J. (2005). Smoothed Particle Hydrodynamics. Reports on Progress in Physics, Vol. 68, No. 5, 1703.

Rustico, E., Hérault, A., Bilotta, G., Jankowski J., Del Negro, C. (2014) Multi-GPU, multi-node SPH implementation with arbitrary domain decomposition. 9th SPHERIC International Workshop, pp. 127-133.

Smith, R. (2002). Open Dynamics Engine. IGC'02.

Thorenz, C., Anke, J. (2013). Evaluation of ship forces for a through-the-gate filling system. Proc. Smart Rivers 2013, Liege (BE), Maastricht (NL).