2D-3D Coupling of Shallow Water Equations and Navier-Stokes Equations

Name: Florian Mintgen
E-Mail: f.mintgen@tum.de
Supervisor: Prof. Michael Manhart
Chair of Hydromechanics
Started: 07/2011
Status: ☑ ongoing □ finalized:

Introduction
Detailed modeling and prediction of flood events is of substantial interest in nearly all parts of the world. Within the framework of Computational Fluid Dynamics (CFD), current 2D Shallow Water Equations (SWE) solvers are capable of predicting the overall behavior of flood events with reasonably good accuracy on large domains at acceptable computational cost. Problems arise when a more detailed understanding of local processes becomes necessary. The commonly used hydrostatic SWE are not capable of reproducing secondary flow structures, which are responsible for phenomena like the interaction between flow and structures or local sediment transport. Such phenomena can be covered by using a 3D Navier-Stokes Equations (NSE) solver with free surface, but due to computational cost this is restricted to small parts of the domain only.

In (Qi, 2004) a 2D-3D coupling for calculation of wave forces on ships has been described, but only with potential flow in the 2D region. A coupling between full SWE and NSE has been done in (Kilanehei, 2011), but here the 2D solution is taken as constant initial and boundary condition of the 3D domain, so only steady state problems can be covered.

In this PhD thesis a full bi-directional coupling between the SWE and the NSE with free surface is developed, based on the framework of the Finite Volume Open Source code OpenFOAM. By this full
coupling the advantages of both approaches are combined:

- Speed of computation with the 2D SWE on the major part of the domain
- Detailed modeling of complex 3D flow phenomena in chosen areas of interest

Main Concept
In our work we utilize two separate solvers:

- interFoam for the Navier Stokes equations, which is one of the standard solvers in the OpenFOAM framework.
- shallowFoam for the shallow water equations, which is a solver developed at the Chair of Hydromechanics.

These two solvers are included in one single executable, and coupled via the boundary conditions, where the values at the boundaries are obtained from the respective cells in the opposite domain.

As a test case, the flow around a quadratic obstacle, e.g. a house, has been simulated. Results are shown in fig. 1 at different instances of time. The region adjacent to the obstacle, indicated by the white rectangle, has been simulated in 3D, whereas the outer part of the domain has been simulated in 2D.

Further validation studies have shown that instationary phenomena like waves can cross the interface without significant distortion.

Figure 1: Coupled simulation of the flow around an obstacle

References


