Coastlab12, 2012, Ghent, Belgium. Book of Abstracts of the Fourth International Conference on the Application of Physical Modelling to Port and Coastal Protection. ISBN 978 90 382 2008 6

ASSESSMENT OF THE 2-D SHALLOW WATER SOLVER IN OPENFOAM FOR APPLICATION IN MODELING OF TIDAL FLOWS

PIETER RAUWOENS [1], JOERI KIEKENS [2] & PETER TROCH [3]

[1] *Ghent University, Department of Civil Engineering, Technologiepark 904, B-9052 Gent, Belgium. pieter.rauwoens@ugent.be*

[2] *Student, Ghent University, Department of Civil Engineering, Technologiepark 904, B-9052 Gent, Belgium. joeri.kiekens@ugent.be*

[3] *Senior lecturer, Ghent University, Department of Civil Engineering, Technologiepark 904, B-9052 Gent, Belgium. peter.troch@ugent.be*

1. Introduction

It is known that the appearance of tides can cause strong currents along the shore. Particularly near ports, these currents can cause problems for ships leaving and entering the port. Our purpose is to create a model that allows to numerically investigate the effect of measurements to locally reduce the current's magnitude. Possible measurements are the creation of breakwaters or artificial islands and the installation of turbines on the bottom of the sea estuary. As a first step in developing such a model, a validation exercise is done, reproducing test cases from literature. Here, OpenFOAM (Silicon Graphics International Corp., 2011) is chosen to numerically solve the tidal flow motion. The model uses a finite volume method on a collocated unstructured grid. In this paper, shallowWaterFoam, solving the two dimensional shallow water equations, is used as a starting point, wich we extended to suit our purpose.

2. OpenFoam

OpenFOAM is a free, open source CFD software package. OpenFOAM has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to solid dynamics and electromagnetics. The program provides a 2-D shallow water solver, that however needs some adjustments. OpenFOAM has the benefits of being open source, that means that the C++ computer language is free for adaptation by the user and thus gives the user the freedom to extend the source code for specific engineering problems. On the other hand, it requires more implementation since the standard solvers in OpenFOAM are nod developed for the specific purpose we have in mind and thus need to be extended with extra source terms: bottom stresses and turbulent stresses (eddy viscosity) are not included and will be added in the source code. The software also develops an unstructured mesh, which is more flexible than a structured mesh and is therefore more convenient for complex geometries.

3. Validation

Implementation of a problem case in the software requires solving of some issues. For instance it is not so convenient to model a tidal wave in an estuary, a special kind of boundary condition is needed that creates a variation of the water level. Other issues concern the grid size, how much elements does a grid need to have in order be accurate enough? Also, for the convective terms a number of discretization schemes are available, but obviously they are not all equivalently accurate. Another issue requires the implementation of a non-reflective boundary so that the internal solution is not disturbed by non-physical wave reflection.

In order to solve this matter, there is a need of reference. In literature different validation cases are available that focus on solving the 2-D shallow water equations. In general simple test cases are used with a somewhat simple geometry or simple boundaries. These cases are rebuilt in OpenFoam and the results are compared with the literature data for different grid sizes, differentiation schemes and boundary conditions. We compare the results for a first order upwind and TVD discretization of the convective fluxes. The level of accuracy for the schems available in OpenFOAM is investigated, using a rigorous grid-convergence study. As such the exact order of accuracy for the problems under study is evaluated. A Richardson extrapolation method serves this purpose.

The first set of test cases does not involve a tidal wave:

- 2-D dam break problem (e.g. Wang & Liu, 2000): the evolution of the water front after dam break is modelled during the time that the water doesn't reflect the wall and the depth-integrated approach remains valid. This case investigates the numerical stability of the scheme for a rapidly varying unsteady flow (Figure 1).
- Flow passed over two circular piers with rectangular bed (Guillou & Nguyen, 1997) or with trapezoidal bed (Hervouet, 1992): In this case an obstacle is placed in a channel creating two vortex streets. The water elevation in the middle of the channel at a certain time and the vortex flow pattern are compared with results from literature.

Other cases include a tidal wave:

- A simple sinusoidal wave passing a rectangular channel (Masson, 1995): this case allows to compare the numerical result with the analytical result.
- A tidal wave entering a square harbor trough a small entrance and creating a large eddy (Daoud *et al.*, 2008).
- Circulation behind a single breakwater parallel to the shoreline (Daoud *et al.*, 2008).

4. Conclusions

In order to model the main problem, the OpenFoam 2-D shallow water solver first hast to be validated. This paper describes the implementation of validation cases known from literature, as to assess the solver's accuracy and suitablility for fidally varying flows near shorelines and ports.

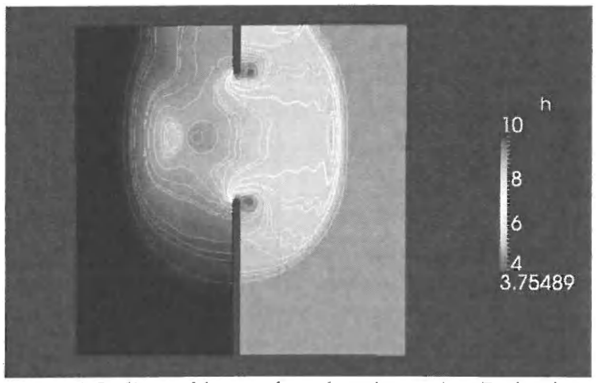

Figure 1. Isolines of free surface elevation at time 7 s for the 2-D dam break case.

References

Daoud, A.H., Rakha, K.A. & Abul-azm, A.G. 2008, 'A two-dimensional finite volume hydrodynamic model for coastal areas: Model development and validation', *Ocean Engineering*, 35, 150–164.

Guillou, S. & Nguyen, K.D. 1999, 'An improved technique for solving two-dimensional shallow water problems', *Internal Journal for Numerical Methods in Fluids*, 29, 465-483.

Hervouet, J.M. 1992, 'Karman vortices behind two cross-flow cylindrical piers', *In:* Telemac-2D validation document, version 6.0, 77 -81.

Masson, A. 1999, 'Propagation of a surface wave in a linear channel', *In:* Telemac-2D validation document, version 6.0, 71 -76.

Silicon Graphics International Corp. 2011, *OpenFOAM, The open source CFD toolbox*. Available: http://www.openfoam.com.

Wang, J.-W. & Liu, R.X. 2000, 'A comparative study of finite volume methods on unstructured meshes for simulation of 2D shallow water wave problems', *Mathematics and Computers in Simulation*, 53, 171–184.