7th International Conference on Hydroinformatics
HIC 2006, Nice, FRANCE

1D AND 2D SOLVERS COUPLING FOR FREE SURFACE FLOW MODELLING

ERPICUM S.
HACH, Civil Engineering Department, University of Liege, Ch. des Chevreuils, 1 B52/3 Liege, 4000, Belgium

ARCHAMBEAU P.
HACH, Civil Engineering Department, University of Liege, Ch. des Chevreuils, 1 B52/3 Liege, 4000, Belgium

DEWALS B. J.
HACH, FNRS Research Fellow, University of Liege, Ch. des Chevreuils, 1 B52/3 Liege, 4000, Belgium

DETREMBLEUR S.
HACH, Civil Engineering Department, University of Liege, Ch. des Chevreuils, 1 B52/3 Liege, 4000, Belgium

PIROTTON M.
HACH, Civil Engineering Department, University of Liege, Ch. des Chevreuils, 1 B52/3 Liege, 4000, Belgium

In the field of hydraulic engineering, the size of the simulations is one of the current challenges to model with precision the large areas involved by practical engineers applications. In this scope, the linking in the same modelization of different solvers, each one used in the part of the simulation where its fundamental characteristics are most suitable, open the door to still unreached modelling possibilities. WOLF software includes a series of numerical tools for simulating a wide range of free surface flows and transport phenomena. This paper presents the coupling of the 1D and 2D multiblock finite volume models of WOLF to simulate in a unified way large river reaches (in 1D) with a very fine computation of local areas (in 2D). An application to the simulation of the propagation of the waves following the filling and emptying operations of a large locks system on a several kilometres long reach of the river Meuse (Belgium) is presented.

INTRODUCTION

Numerical models of great reliability and accuracy are today available for 1D, 2D as well as 3D flow simulations. The choice of a type of model for a specific application depends mainly on the scale of the problem to solve and on the phenomenon to represent. On one side, 1D models are very convenient to assess flood propagation or pollutant transport for example in large applications involving river networks. They can deal with quite few available data and are little computation time or memory capacities consuming. On the other side, 3D models remains generally better fitted to the modelization of small areas but with a very fine representation of local effects such as complex three dimensional flow patterns or turbulence. They need a lot of input data and require more memory capacities and computational resources.

One of the current challenges of hydraulic modelling is in the size of the simulations. Larger and larger areas have to be considered to include the real boundaries of the whole hydrodynamic problem or to deal with the complex practical applications submitted to hydraulic engineers. On the other hand, fine grids have to be used to obtain a suitable representation of the hydrodynamic fields near the points of interest. Even if unstructured meshes or grid refinement allow to decrease the size and the computation time of such simulations, the coupling of different solvers, each one used in the part of the simulation where its fundamental characteristics are most suitable, open the door to still unreached modelling possibilities.

Beside projects such as HarmonIT to create computational environments to link together different existing flow solvers [3], integrated packages of computational hydraulic models allows achieving very easily coupled simulations. In this field, the WOLF software, entirely developed at the University of Liege for almost ten years, includes a series of interconnected numerical tools for simulating a wide range of free surface flows and transport phenomena, from hydrological runoff and river propagation to extreme erosive flows on realistic mobile topography. All the solvers are based on the same finite volume spatial discretization technique, used similar explicit or implicit temporal integration schemes and have been built following the same resolution principles. Thanks to these characteristics, they can easily be linked together [2].

In this paper, the coupling of the 1D and the 2D flow solvers of WOLF is presented. It allows simulating in a unified way large river reaches (in 1D) with a very fine computation of local areas (in 2D). Substantial gains in calculation time and memory requirements can thus be achieved without decreasing model reliability and precision.

After a theoretical test case, an application of the linked model to the simulation of the propagation of the waves following the working operations of a large locks system on a several kilometres long reach of the river Meuse (Belgium) is presented. Suitable and well-positioned boundary conditions for waves reflexion are taken into account in the reach by using Wolf1D while the near field around the lock is accurately modelled in 2D.

FREE SURFACE FLOW MODELS

The Wolf package

Entirely developed by the HACH, the WOLF free surface flows computation package provides in the same development environment the resolution of 1D and 2D flow equations as well as a physically based hydrological model and powerful optimization capabilities based on Genetic Algorithms. The interactive and unique user-interface, with high performance pre- and post-processing, allows monitoring 3-D large-scale runs graphically while they proceed. Each code handles general multiblock meshes, dealing with natural topography and mobile bed simultaneously, for any unsteady situation with mixed regimes and moving hydraulic jumps.

WOLF components allow the package to deal with all free surface hydraulic phenomena, from hydrological runoff and river propagation to extreme erosive flows on realistic mobile topography, such as gradual dam breaching processes. It has proved its efficiency and reliability for years by numerous real applications [1, 2].

1D model

Wolf1D, a quasi-bidimensional model, has been developed in order to better manage floods in complete river networks. As common methods based on conveyance considerations lead to substantial errors, Wolf1D takes explicitly into account the flows in compound channels [2]. The complete set of equations solved in Wolf1D for each flow bed is expressed as follows:

$$\frac{\partial \omega}{\partial t} + \frac{\partial Q}{\partial x} = 0 \tag{1}$$

$$\frac{\partial Q}{\partial t} + \frac{\partial uQ}{\partial x} + g\cos\theta \frac{\partial p_\omega}{\partial x} = -g\omega\cos\theta \frac{\partial z_b}{\partial x} + g\omega\cos\theta J + g\cos\theta\, p_x + g\omega\sin\theta \tag{2}$$

where ω is the cross section, Q the discharge, u the mean velocity, J a global term for bottom roughness and shear fluid effect, θ the channel bottom slope, g the gravity acceleration and $p_\omega(h)$, $p_x(h)$ are pressure terms.

The spatial discretization of the equations is performed by a widely used finite volume method. Flux treatment is based on an original flux-vector splitting technique developed for WOLF. Variable reconstruction can be selected to gain first or second order accuracy on regular grids. Besides, an explicit Runge-Kutta scheme or an implicit algorithm (based on the GMRES) is applied to solve the ordinary differential equation operator, and an original treatment of the confluences based on Lagrange multipliers allows the modelization in a single way of large rivers networks.

2D model

The 2D multiblock flow solver Wolf2D is based on the classical form of the so-called shallow water equations:

$$\frac{\partial h}{\partial t} + \frac{\partial uh}{\partial x} + \frac{\partial vh}{\partial y} = 0 \tag{5}$$

$$\frac{\partial}{\partial t}(hu) + \frac{\partial}{\partial x}\left(hu^2 + \frac{gh^2}{2}\right) + \frac{\partial}{\partial y}(huv) = -gh\frac{\partial z_b}{\partial x} + ghJ_x \tag{6}$$

$$\frac{\partial}{\partial t}(hv)+\frac{\partial}{\partial y}\left(hv^2+\frac{gh^2}{2}\right)+\frac{\partial}{\partial x}(huv)=-gh\frac{\partial z_b}{\partial y}+ghJ_y \tag{7}$$

where u and v are the velocity components, h the water height, l the channel width z_b the bottom elevation and J_x and J_y the components along axis of roughness. Additional terms or sets of equations are available to model turbulence or moving structures effects as well as air or sediments transport.

As for Wolf1D, the spatial discretization of the 2D conservative equations is performed by a finite volume method and flux treatment is based on the original flux-vector splitting technique developed for WOLF. An accurate and non-dissipative Runge-Kutta explicit temporal scheme has been chosen for the time integration.

Wolf2D includes an efficient mesh generator and deals with multiblock structured grids. These features increase the size of potential problems to be solved and allow mesh refinement close to interesting areas without leading to prohibitive CPU times [1].

LINKING STRATEGY

The same finite volume method is used for the spatial discretization in the two solvers. Moreover, temporal integration schemes can be both explicit. Thanks to these common features, the linking of the models is easily achieved by working at each time step on the control volumes boundaries at the domains extremities (Fig. 1).

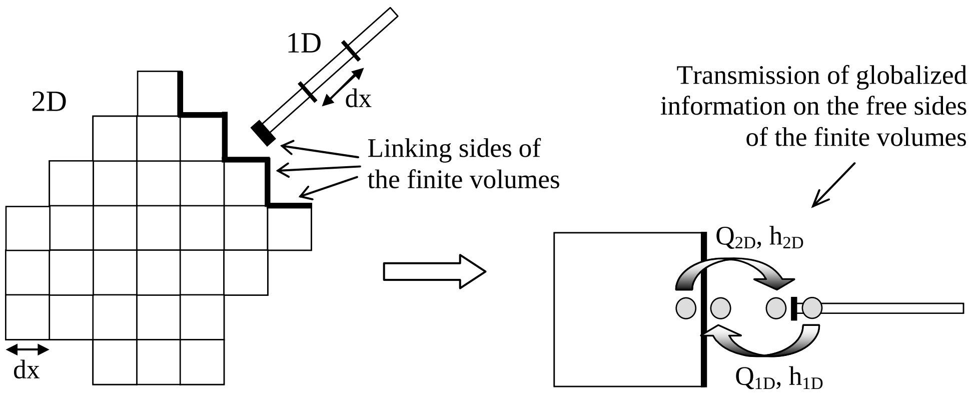

Figure 1. Principle of the model linking

Information in the cells at the 2D domain extremities are used to evaluate unknown boundary conditions for the 1D model and vice-versa. This exchange of information is performed in order to conserve the discharge as well as the momentum direction and the free surface level through linking boundaries.

For the 1D model, no degree of freedom exists and the linking conditions are simply the discharge through the 2D boundary and the 2D mean free surface elevation.

For the 2D model, the direction θ of the flow from the 1D model, defined by the modeller, can be taken into account (Fig. 2). The 2D discharge components $Q_{x,2D}$ and $Q_{y,2D}$ along x and y axis are first evaluated to keep the 1D discharge Q_{1D} direction:

$$Q_{x,2D} = Q_{1D} \frac{1}{1+\tan\theta} \text{ and } Q_{y,2D} = Q_{1D} \frac{\tan\theta}{1+\tan\theta} \tag{8}$$

In a second time, they are used to compute the specific discharges $q_{x,i}$ and $q_{y,i}$ for each 2D boundary side i by keeping a constant flow velocity and the 1D free surface elevation. The latter is used to compute the 2D flow section Ω_{2D} and the local height h_i.

$$q_{j,i} = V_{j,2D} h_i \text{ with } V_{j,2D} = \frac{Q_{j,2D}}{\Omega_{j,2D}} \text{ and } j = x, y \tag{9}$$

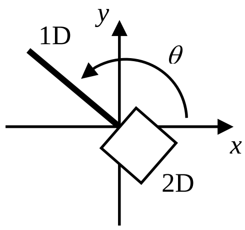

Figure 2. Definition of the 1D reach angle θ compared to the 2D axis

From a computational point of view, the two solvers are written in Fortran. Codes are executed in parallel in the same application using different threads. To each time iteration, time step is evaluated for the whole simulation, i.e. the smallest allowable one regarding local stability criterion is chosen for the whole computation domain. Time evolution is then performed for each model separately, but waiting commands allow aligning of the two codes when exchanges of information are necessary.

VALIDATION

To validate the approach, the following test case has been realized. A winding river reach has been modelled in two dimensions with the solver Wolf2D, and the simulation of a flood propagation has been performed as a referenced configuration. In a second time, the upstream and downstream parts of the reach have been modelled with Wolf1D. Simulation in this new model of the propagation of the flood provides reliable validation by comparison of the hydrodynamic parameters evolution in the central area.

Meshes 1 m in side have been used for 1D and 2D discretizations. The reach was 10 m wide and 280 m long, without bottom slope and had a rectangular cross-section. The length of the 1D reaches was 90 m. (Fig. 3)

The initial condition corresponded to the steady flow of 10 m³/s in the reach with a water height of 1 m downstream. Manning's roughness coefficient was .04 for 2D domain, and .054 for 1D one. The latter value has been calibrated to have the same free surface level in the coupled model towards the full 2D one. The downstream boundary condition remained constant along time during all the simulations.

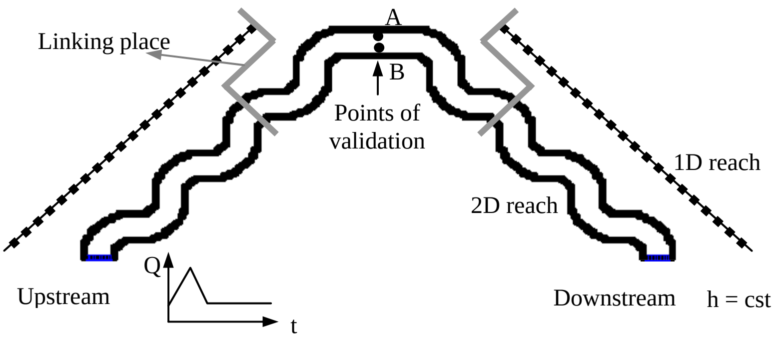

Figure 3. Sketch of the simulation for purely 2D and coupled 1D-2D computations

Points A and B (fig. 3) have been chosen for results comparison. They were respectively located outside and inside the central bend. The evolution along time of the water height and the absolute specific discharge at these points is presented on figure 4.

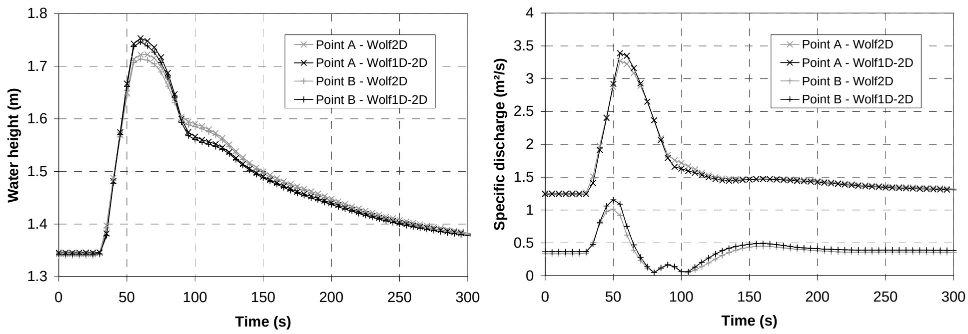

Figure 4. Water height and specific discharge evolution to points A and B - Comparison of the two models

Time evolution of the flow variables is similar. Only a small difference in the amplitude of the peaks can be observed, but it doesn't undermine the approach. No spurious effect of reflection or waves generation appears in the linking zone. 2D hydrodynamic effects are very well reproduced, as shown by the discharge evolution to point B. Moreover, the number of calculation meshes is 2860 for totally 2D model against 1346 for the coupled one. Thanks to this, calculation time to simulate 600 s of real time is 195.32 s in the first case and 84.67 s in the second one.

The coupled approach is thus very efficient to decrease memory requirements and computation time of 2D simulations without damaging the results near interesting places.

APPLICATION

Finally, the coupled model has been applied to the simulation of the propagation of the waves following the working operations of a large locks system. The goal of the study was to assess the hydrodynamic effects of the building of a fourth 225 x 25 m lock near the existing three locks of Lanaye on the river Meuse in Belgium.

Downstream of the plant, a channel links the locks to a 14 km long reach of the river Meuse with a mobile dam at each extremity. An inundated area is located on the right side of the river, in front of the confluence with the lock channel (Fig. 5). This complex layout leads to the propagation and the reflection of multiple waves when emptying the locks, whose study is of great interest to assess ship navigation conditions, mobile dam management instructions as well as hydraulic structures design.

The field near the locks and the channel – river confluence have to be modelled in 2D to correctly represent flow patterns and waves propagation. On the other hand, it is necessary to model the whole river reach to take into account the waves reflection against mobile dams. Indeed, their travelling time in the domain has the same order of magnitude as the time to empty a lock. The use of the new coupled 1D-2D model took thus on its all meaning to allow setting up a usable but still relevant numerical model.

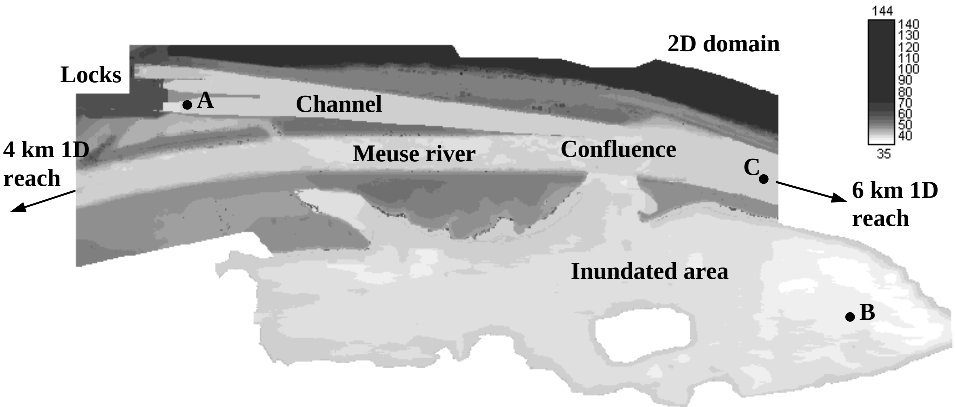

Figure 5. Sketch and 2D topography of the simulation domain to study the area downstream of the locks

An area 5 km long and 1 km wide has been modelled in 2D with +/- 60 000 square meshes from 2 x 2 m up to 16 x 16 m. It included the lock and the downstream channel, 3 km of the Meuse river reach and the inundated area. The rest of the Meuse reach has been modelled in 1D with 2500 finite volumes 4 m long (Fig. 5).

Several simulations have been performed, with different scenarios for locks functioning and discharge in the river… For each case, the time evolution of pertinent flow variables at strategic points has been analyzed (fig. 6) to assess the effects of the new lock. 2D flow patterns have also been used to assess ship navigation conditions, especially at the confluence, and hydraulic structures design has been validated.

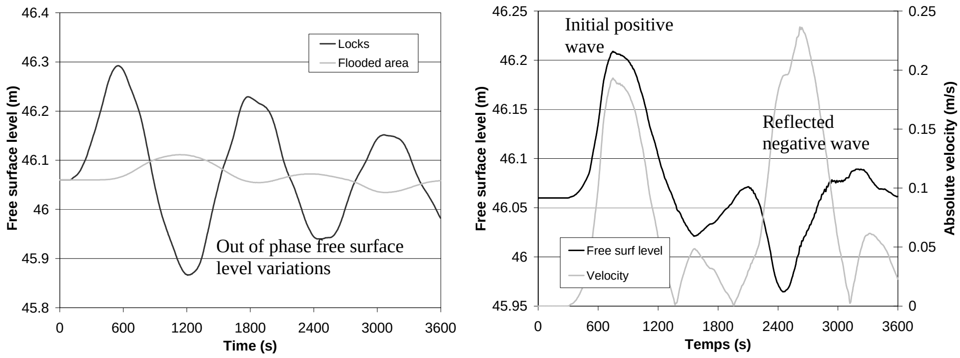

Figure 6. Example of results: mass oscillations between the lock channel - point A - and the flooded area - point B - (left) and wave reflection against downstream mobile dam (right) to point C (Fig. 5)

CONCLUSIONS

Development of complementary free surface flow solvers in the same environment and using the same numerical techniques, such as in WOLF software, allows an easy and very efficient linking of different models, each of ones being used where its features are the best fitted. Very extended or complex practical hydraulic applications can thus been considered, without leading to prohibitive computation time or huge memory requirements, but still guaranteeing the results representativeness and reliability.

REFERENCES

[1] Erpicum, S., Archambeau P., Dewals B., Detrembleur S., Pirotton M., “Computation of the Malpasset dam break with a 2D conservative flow solver on a multiblock structured grid”, *Proc. of 6th Int. Conf. on Hydroinformatics*, Singapore, (2004).

[2] Erpicum, S., Archambeau P., Dewals B., Detrembleur S., Pirotton M., “Optimization of hydroelectric power stations operations with WOLF package”, *Proc. Hydropower '05 - The backbone of sustainable energy supply*, Stavanger, Norway, (2005)

[3] Moore R., Tindall I., D. Fortune, “Update on the HarmonIT project, the OpenMI standard for model linking”, *Proc. 6th Int. Conf. on Hydroinformatics*, Singapore, (2004).