# Fluid - structure interaction modeling with a coupled 1D - 2D free surface flow solver

S. Erpicum, P. Archambeau, B. Dewals*, S. Detrembleur & M. Pirotton
*Laboratory of Applied Hydrodynamics and Hydraulic Constructions, Civil Engineering Institute, University of Liege, Belgium*
* *FNRS research fellow*

ABSTRACT: Wolf software is a set of coupled numerical tools developed for almost ten years for simulating a wide range of free surface flows and transport phenomena. In the scope of new applications in hydraulic structure design and fluid-structure interaction, additional developments have been implemented in the finite volume 2D model to deal with boundaries not aligned along axis or with moving structures. The cut cell based approach, simpler than other classical methods, is based on the division with a sub-grid of the initial Cartesian meshes. While keeping the advantages of regular grids in terms of simplicity, accuracy and storage capacity, it enables to compute instantaneous effects of a structure movement on the fluid or to deal with complex boundaries inclined compared to mesh axis. Several test cases are presented, as well as an application of the enhanced model, coupled with a 1D one, to the computation of the hydrodynamic fields induced by a lock-door movement.

## 1 INTRODUCTION

Today, hydraulic engineering studies use more and more numerical models to predict or assess flow conditions. Depth-integrated models are very efficient in this field as they represent a good compromise between problem implementation and computation times in comparison with results quality and accuracy. Discretization in two dimensions allows using fine meshes without leading to prohibitive memory requirements, while well-known equations sets and resolution schemes provide a reliable modeling of the most important flow variables for practical engineering applications.

On the other hand, the finite volume method seems to be currently the most suited to solve the hyperbolic partial differential equations system characterizing flow movement (Valiani & al. 2002). As the method is based on the integral form of the conservation equations, the derived numerical scheme is conservative (Hirsch, 1988) and satisfies the shock-capturing property (Toro, 2001).

The application field of such numerical models solving the so-called shallow water equations (SWE) is very broad. Among others, they are used for dam-break (Valiani & al. 2002, Erpicum & al. 2004a) or dam-breaching (Dewals & al. 2002, Froehlich, 2002) simulations, flood inundation mapping even in urban areas (Alcrudo & Mulet, 2005, Archambeau & al. 2004) as well as hydraulic structure design (Erpicum & al. 2004b, Dewals & al. in press) or sedimentation assessment (Dewals & al. 2004).

Generally, grids used for 2D simulation are classified regarding the type of meshes they use. The two main families are the structured and unstructured grids. While unstructured grids allow dealing with complex geometries, they are more computation time consuming than structured ones, even more if the latter are Cartesian. Indeed, to achieve the same global order of precision for the calculation, they need a higher precision of reconstruction of the unknowns (Mouzelard, 2002).

To limit the restriction of the Cartesian grids to fit to complex geometries, methods based on a cut cell approach (Causon & al. 2000) can be used. They can also easily be extended to the simulation of moving bodies (Ingram & al. 2003), and thus reproduce fluid-structure interaction. Such a method is presented in this paper.

Based on a sub grid approach to define complex geometries, the present method is simpler than classical cut cell approaches but is more easily implemented in an existing code. It constitutes thus an appropriate first approach in the domain.

In this paper, the developments realized to simulate fluid-structure interaction by means of mobile boundaries on a fixed Cartesian grid are presented in detail. After its implementation in the free surface flow solver Wolf2D (Erpicum & al. 2004a), the modified SWE model is applied to a bore reflection on a fixed inclined wall and to the simulation of a moving plate in a channel. Finally the modified 2D model, coupled with the one-dimensional free surface flow solver Wolf1D (Erpicum & al. in prep.), is applied to assess the hydrodynamic effects of a ship lock door movement.

## 2 EQUATIONS

### 2.1 *1D approach*

Consider a general 1D conservative law such as:

$$\frac{\partial}{\partial t}s+\frac{\partial}{\partial x}f=S \tag{1}$$

where $s$ = unknown; $f$ = advective term; $S$ = source term; $x$ = space coordinate; and $t$ = time.

Let us solve the equation using a finite volume method with a cell-centered approach (Fig. 1), considering fixed nodes but cell edges moving with velocity $v_{G,x}$ along $x$-axis. Integration of (1) over a finite volume $i$ lead to:

$$\int_{x_{i-1/2}}^{x_{i+1/2}}\frac{\partial}{\partial t}s\,dx+\int_{x_{i-1/2}}^{x_{i+1/2}}f\,dx=\int_{x_{i-1/2}}^{x_{i+1/2}}S\,dx \tag{2}$$

After Leibnitz theorem application, time derivative integral becomes:

$$\frac{\partial}{\partial t}\int_{x_{i-1/2}}^{x_{i+1/2}}s\,dx-s\big|_{i+1/2}\frac{\partial}{\partial t}x_{i+1/2}+s\big|_{i-1/2}\frac{\partial}{\partial t}x_{i-1/2} \tag{3}$$

and is equal to:

$$\frac{\partial}{\partial t}\overline{s}\Delta x_i-s\big|_{i+1/2}v_{G,x}\big|_{i+1/2}+s\big|_{i-1/2}v_{G,x}\big|_{i-1/2} \tag{4}$$

where $\Delta x_i$ = cell $i$ length; and symbol $\overline{a}$ stands for the mean value of $a$ over the finite volume.

Green-Gauss theorem application to second term of (2) and source term integration lead to, respectively:

$$f\big|_{i+1/2}-f\big|_{i-1/2}\ \text{ and }\ \overline{S}\Delta x_i \tag{5}$$

Using expressions (4) and (5), equation (2) becomes:

$$\frac{\partial}{\partial t}\overline{s}+\frac{\partial}{\partial x}\Big[\big(f-sv_{G,x}\big)_{i+1/2}-\big(f-sv_{G,x}\big)_{i-1/2}\Big]=\overline{S} \tag{6}$$

Grid movement affects advective terms by mean of additional exchanges through cells faces. For example, regarding mass conservation equation for a 1D flow in a rectangular channel, $s$ is the water height $h$ and $f$ is the unit discharge $q$ while $S$ is zero if no mass exchange occurs with outside. Equation (6) writes:

$$\frac{\partial}{\partial t}h+\frac{\partial}{\partial x}\Big[\big(q-hv_{G,x}\big)_{i+1/2}-\big(q-hv_{G,x}\big)_{i-1/2}\Big]=0 \tag{7}$$

It is the classical form of mass conservation equation for a 1D flow except additional mass transfers $hv_{G,i}$ through cells sides.

If the calculation grid is fixed but a boundary moves through the calculation domain with the velocity $v_{G,x}$, the equation is the same with term $v_{G,x,i}$ equals to zero everywhere except at instantaneous boundary location. For a solid boundary, as there is no mass exchange, fluid velocity $u$ is equal to $v_{G,x}$ and corresponding advective flux in (7) is zero.

On the other hand, the cell length is changing along time. This can be taken into account by an occupancy rate parameter $\theta$ affecting the cell length.

### 2.2 *Generalization to shallow water equations*

Shallow water equations (SWE) can be written under a general form such as:

$$\frac{\partial}{\partial t}\mathbf{s}+\frac{\partial}{\partial x}\mathbf{f}+\frac{\partial}{\partial y}\mathbf{g}=\mathbf{S} \tag{8}$$

where $\mathbf{s}$ = unknowns vector; $\mathbf{f}$ and $\mathbf{g}$ = advective terms vector; and $\mathbf{S}$ = source terms vector. Generalization of previous developments on a moving Cartesian grid provides for the integration over a finite volume of equation (7):

$$\frac{\partial}{\partial t}\overline{\mathbf{s}}\Delta x\Delta y+\oint\big(\mathbf{F}-\mathbf{s}\mathbf{v_G}\big).\mathbf{n}d\Gamma=\overline{\mathbf{S}}\Delta x\Delta y \tag{9}$$

where $\mathbf{F}$ = [$\mathbf{f}$,$\mathbf{g}$]; $\mathbf{v_G}$ = grid velocity vector; $\mathbf{n}$ = normal vector; $\Gamma$ = cell outline; and $\Delta x$ = $\Delta y$ = grid dimensions.

Developments are still the same if the grid is fixed but a boundary moves through the cell with ve locity $\mathbf{v_G}$.

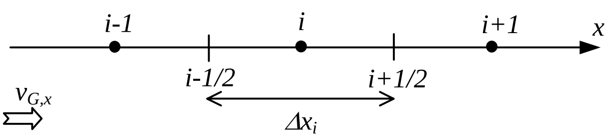


Figure 1. Sketch of 1D grid for cell-centered finite volume method

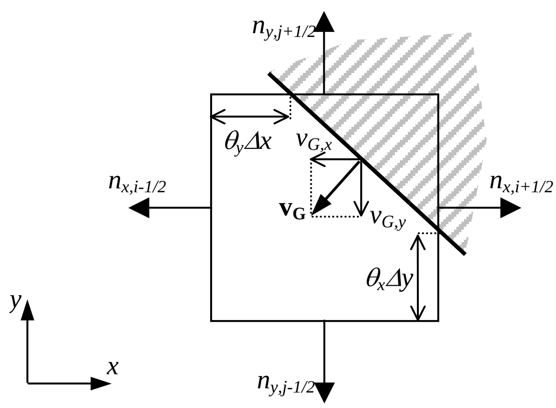


Figure 2. Sketch of the normal vector $\mathbf{n}$ components on a 2D finite volume and solid boundary moving with velocity $\mathbf{v_G}$

Considering Figure 2, outline integral in (9) can so be expressed as:

$$\sum_{k=1}^{N}\int_{\Gamma^k}\left[\left(\mathbf{f}-\mathbf{s}\mathbf{v}_\mathbf{G}\right)n_x+\left(\mathbf{g}-\mathbf{s}\mathbf{v}_\mathbf{G}\right)n_y\right]d\Gamma^k \tag{10}$$

where $N$ = number of cell sides.

For SWE, without taking into account diffusive effects:

$$\mathbf{s}=\begin{Bmatrix} h \\ uh \\ vh \end{Bmatrix};\mathbf{S}=\begin{Bmatrix} 0 \\ -gh\dfrac{\partial z_b}{\partial x}+ghJ_x \\ -gh\dfrac{\partial z_b}{\partial y}+ghJ_y \end{Bmatrix} \tag{11}$$

$$\mathbf{f}=\begin{Bmatrix} uh \\ u^2h+g\dfrac{h^2}{2} \\ uvh \end{Bmatrix};\mathbf{g}=\begin{Bmatrix} vh \\ uvh \\ v^2h+g\dfrac{h^2}{2} \end{Bmatrix} \tag{12}$$

where $u$, $v$ = fluid velocity vector components along $x$ and $y$-axis; $g$ = gravitational acceleration; $z_b$ = bed elevation; and $J_x$, $J_y$ = friction slope components along $x$ and $y$-axis, classically expressed by Manning's formula.

Considering sliding condition against boundary:

$$\mathbf{u}.\mathbf{n}=\mathbf{v}_\mathbf{G}.\mathbf{n} \tag{13}$$

where $\mathbf{u}$ = fluid velocity vector.

Formulation for SWE, taking into account a solid boundary moving across grid, expresses finally using (11), (12) and (13) in (9) and (10) as:

$$\begin{aligned}
&\frac{\partial}{\partial t}(\theta h)+\frac{\partial}{\partial x}(\theta_x uh)+\frac{\partial}{\partial y}(\theta_y vh)=0\\
&\frac{\partial}{\partial t}(\theta uh)+\frac{\partial}{\partial x}\left(\theta_x u^2h+g\frac{h^2}{2}\right)+\frac{\partial}{\partial y}(\theta_y uvh)=\\
&\qquad\qquad -gh\theta\left(\frac{\partial}{\partial x}z_b-J_x\right)\\
&\frac{\partial}{\partial t}(\theta vh)+\frac{\partial}{\partial x}(\theta_x uvh)+\frac{\partial}{\partial y}\left(\theta_y v^2h+g\frac{h^2}{2}\right)=\\
&\qquad\qquad -gh\theta\left(\frac{\partial}{\partial y}z_b-J_y\right)
\end{aligned} \tag{14}$$

where $\theta$ = occupancy rate of the cell by water, i.e. percentage of the cell surface not occupied by solid; $\theta_x$, $\theta_y$ = free portion of cell sides (Fig. 2), i.e. part of cell sides where mass and momentum exchanges can occur.

Pressure terms are not affected by a moving solid boundary, but other advective terms involving mass transfers are directly modified as the length of the cell side they can pass trough is reduced. Terms involving mean unknown value over the cell are also modified as the cell area decreases by solid intrusion.

## 3 NUMERICAL ASPECTS

### 3.1 *Wolf package*

Wolf software 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. It has been entirely developed at the University of Liege for almost ten years and is still continuously enhanced (Pirotton & al. 2003).

In all the solvers, the space discretization of the conservative equations is performed by a finite volume method. Flux treatment is based on an original flux-vector splitting technique developed for Wolf. The hydrodynamic fluxes are split and evaluated partly downstream and partly upstream according to the requirements of a Von Neumann stability analysis (Mouzelard, 2002, Dewals, 2006). Optimal agreement with non-conservative and source terms as well as low computational cost are the main advantages of this original scheme. Constant or linear reconstruction of the variables is used to achieve a global first or second order of precision.

As we are mostly interested in transient flows and flood waves, explicit Runge-Kutta schemes are used for time integration.

Equations (14) have been implemented in Wolf2D, specifically devoted to the resolution of the SWE. It includes an efficient mesh generator and deals with multiblock Cartesian grids (Erpicum & al. 2004a). The algorithm is also designed to deal automatically with any moving boundary between wet and dry cells.

A spatially distributed hydrological solver, WolfHydro, a one-dimensional free surface flow solver, Wolf1D, and an optimization tool based on genetic algorithms technique, WolfAG, complete the Wolf software package (Pirotton & al. 2003).

### 3.2 *Coupling of models*

As an extension of the multiblock feature of Wolf2D, and thanks to the use of the same discretization and calculation techniques for all the solvers of Wolf software, a coupling between the 1D and the 2D models has been achieved. It allows simulating in a unified way large river reaches (in 1D) with a very fine computation of local areas (in 2D) without leading to prohibitive computation times (Erpicum & al. in prep.)

### 3.3 *Sub grid approach*

In order to deal with mobile structures or boundaries inclined along mesh axis, a simple approach based on a regular sub-grid has been applied.

To calculate a cell water occupancy rate and the free length of its sides as a function of the solid boundary instantaneous position, the mesh is subdivided by a *step* factor. The position of the sub-grid cells gravity center automatically defines whether they are under water or not (Fig. 3).

The main advantage of this simple technique is that the cell water occupancy rate is never lower than one *step*, and thus calculation stability criterion is easily evaluated. In addition, updating the boundary position and cells occupancy rates is very easy and rapid.

If *step* equals one, boundary discretization is as rough as the initial Cartesian grid. When *step* increases, the representation of inclined boundary becomes better.

To solve the problem of a mesh leaving the calculation domain between to time steps while respecting the mass conservation principle, the following operation is performed. If the cell *i* leaving the calculation domain in time *t*:

$$0 < \theta_i^{t-\Delta t} \leq 1 \text{ and } \theta_i^t = 0 \tag{15}$$

Calculation in the entire domain is realized in time *t* for cells containing water, i.e. without cell *i*. After time step evaluation, water heights in the four cells adjacent to cell *i* is increased as a function of their respective occupancy rate:

$$h_l \rightarrow h_l + \frac{\theta_i h_i}{\sum_{k=1}^{4} \theta_k} \theta_l \text{ with } l = 1,\ldots,4 \tag{16}$$

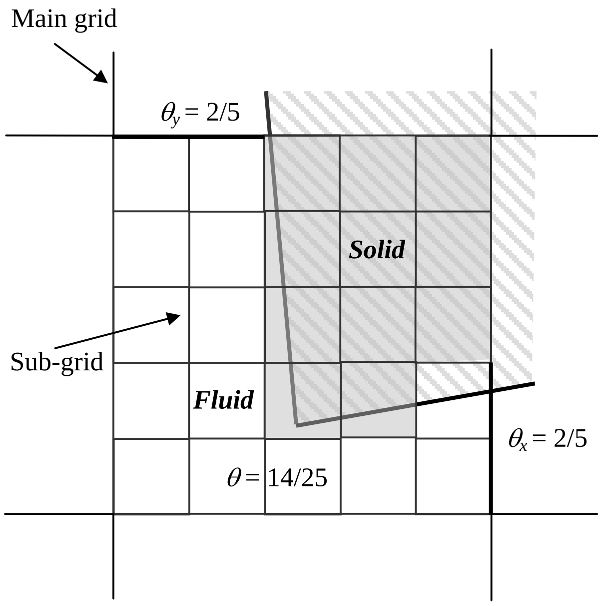


Figure 3. Example of application of the sub-grid to evaluate boundary location and occupancy rate in a mesh - *step* factor = 5

On the contrary, when a cell enters the calculation domain, its hydrodynamic variables are initialized as a function of those of its neighbors containing water. The water height of all the adjacent meshes containing water and of the new mesh is leveled:

$$h_l = \frac{\sum_{k=1}^{M+1} \theta_k}{\sum_{k=1}^{M} \theta_k h_k} \text{ with l = 1,\ldots,M+1} \tag{17}$$

where $M$ = number of cells adjacent to cell *i* with $\theta$ higher than zero.

For the specific discharges, they are initialized to the mean value of the corresponding variables in the two adjacent cells containing water on the same axis.

### 3.4 *Stability criterion*

As explicit time integration scheme is used in the solver, the calculation stability criterion remains classically in the form of a Courant-Friedrichs-Lewy number *CFL* maximal value (Hirsch, 1988). For a Runge-Kutta scheme RK*PQ*, where *P* is the number of sub-steps to calculate one time step and *Q* is the order of precision of the method, *CFL* condition writes (Dewals, 2006):

$$CFL = \frac{\left(\|\mathbf{u}\| + c\right)\Delta t}{\Delta x} \leq \frac{R_0}{2} \tag{18}$$

where $c$ = wave celerity; $\Delta t$ = time step; $R_0$ = factor depending on the Runge-Kutta scheme.

In practice, *CFL* number is evaluated for each cell of the calculation domain to identify the local stable time step $\Delta t_i$, and the most restrictive one is chosen as the effective calculation time step $\Delta t$.

Because of the sub-grid, the effective size of the cells near an inclined boundary can be smaller than Cartesian steps $\Delta x$ or $\Delta y$. A good approximation is the smallest length of the free sides of the cell (Fig. 3), and condition (18) becomes for cells near a mobile boundary:

$$CFL = \frac{\left(\|\mathbf{u}\| + c\right)\Delta t}{\min(\theta_y \Delta x, \theta_x \Delta y)} \leq \frac{R_0}{2} \tag{19}$$

## 4 VALIDATION AND CASE STUDY

### 4.1 *Bore reflection*

The example of a bore reflection on an inclined bank in shallow water has first been investigated. A linear incident bore wave moves from right to left and encounters a solid boundary inclined at a fixed angle

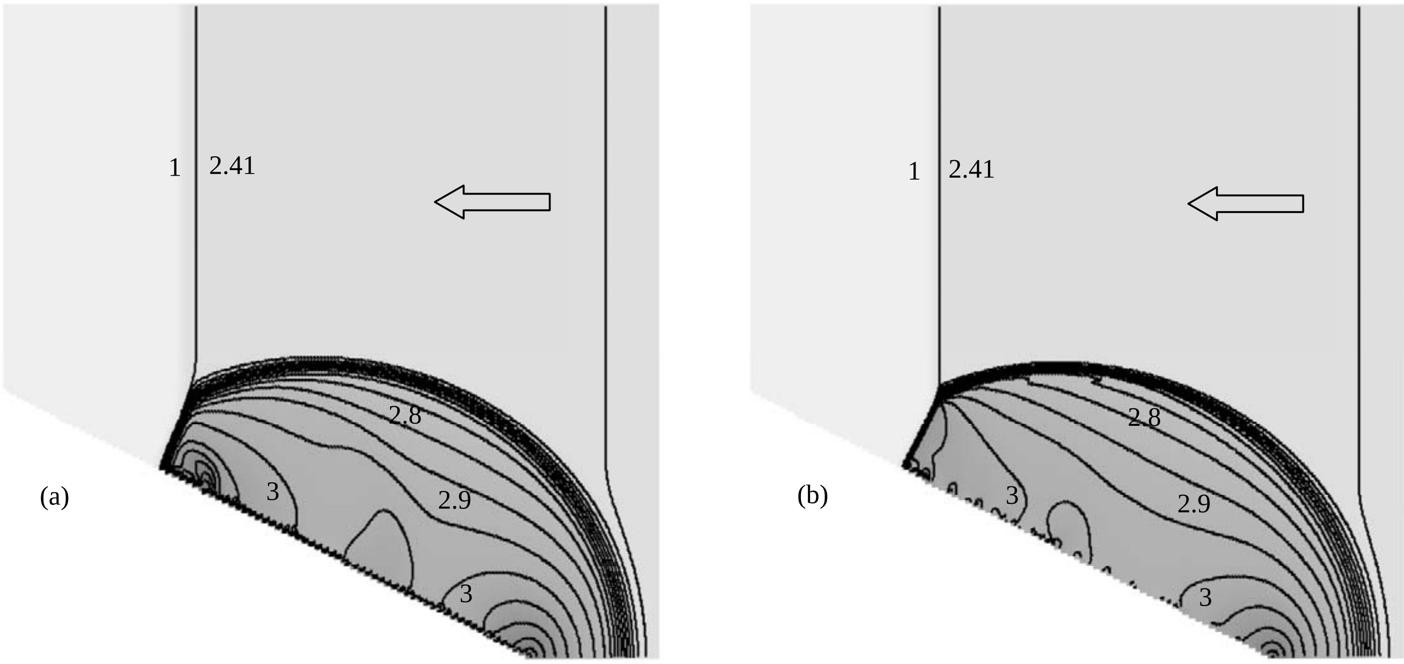


Figure 4. Single Mach reflection of a bore on an inclined solid wall. No sub grid (step = 1 - (a)) and sub grid with step = 5 (b) - Representation of water heights (m)

into the flow direction. The bore is therefore reflected (Causon & al. 2000) .

If the incident bore is defined by a Froude number $F_s$ equal to 2.03 and the boundary is inclined of an angle $\alpha_w$ of 27°, a so called single Mach reflection occurs (Causon & al. 2000).

To assess the influence of the developments proposed in this paper, the simulations have been realized with the new model with and without a sub-grid (*step* = 5).

A grid with 200 x 200 meshes 5 cm long has been used. Initial conditions (Causon & al. 2000) were water at rest with a height of 1 m everywhere in front of the bore, and water height of 2.41 m with a specific discharge from right to left of 8.965 m²/s behind the discontinuity. No roughness effects have been considered.

Results are presented on figure 4. Even if the two calculations show physically meaningful results, less spurious effects near the inclined boundary are observed with the new model. Reflected shock is also sharper, and so the results are globally better.

### 4.2 *Moving plate*

Consider a plate moving with constant velocity $v_P$ in a frictionless horizontal rectangular channel containing water initially at rest with height $h_0$ (Fig. 5). Under the assumption that fluid velocity near the plate is equal to the piston velocity, the analytical solution of the waves propagation in the channel can be calculated, both behind and in front of the plate.

Indeed, from mass and momentum conservation equations applied on both sides of the flow discontinuities, Rankine-Hugoniot shock propagation principle provides two relations linking the two unknowns: front velocity $V$ and water height of the wave $h_d$:

$$V = \underbrace{\frac{h_d v_P}{h_d - h_0}}_{mass} = \underbrace{\frac{v_P^2 h_d + \frac{g}{2}\left(h_d^2 - h_0^2\right)}{v_P h_d}}_{momentum} \tag{20}$$

For example, if $h_0$ = 1 m and $v_P$ = 0.2 m/s, $h_{d,d}$ = 1.065 m and $V_d$ = 3.28 m/s downstream of the plate, while $h_{d,u}$ = 0.937 m and $V_u$ = 2.98 m/s upstream.

The numerical simulation of this test case has been realized.

A channel 1 km long and 1 m wide has been modeled using 1 to 1 m regular meshes. A plate 2 m thick divided it into to reaches 499 m long. No bed slope nor friction effects have been considered. All the simulation limits were assumed to be impermeable.

Hydraulic initial condition was water at rest with height $h_0$ = 1 m everywhere in the reaches.

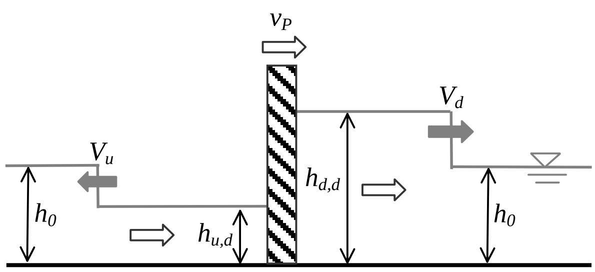


Figure 5. Sketch of shocks propagating around a plate moving with constant velocity $v_P$ in a channel where water is at rest with height $h_0$.

At time t = 0 s, the plate instantaneously began to move from left to right with a constant velocity $v_P$ = 0.2 m/s. Sub grid 0.01 m (*step* = 100) has been used for plate movement.

Numerical results are illustrated in Figure 6. Water heights and propagation times are in satisfactory agreement with the analytical solution.

The shock propagation in front of the plate is well represented, with a sharp front moving in accordance with the analytical celerity. Behind the plate, the wave is smoother. This is in agreement with the physics of the problem and can be explained by the characteristics theory for shallow flows.

The celerity $c$ of a wave in water with height $h$ is:

$$c = \sqrt{gh} \tag{21}$$

In the channel, water height is $h_0$ = 1 m and thus waves celerity $c_0$ is 3.13 m/s.

In front of the plate, front velocity $V_d$ is higher than waves celerity $c_0$ and thus a shock occurs. Beside the plate, front velocity $V_u$ is a bit smaller than waves celerity $c_0$ and the front is smoothed as information propagates faster in the fluid than the wave.

Considering more precisely the characteristics curves behind the plate (Pirotton, 1994), only two lines exist. The faster one has a celerity $-c_0$, as explained above, and the second one has a celerity equal to:

$$\frac{3}{2} v_P - c_0 = -2.83\, m/s \tag{22}$$

Between the moving plate and this second characteristic, nothing else occurs and water height is constant.

For each time, the wave upstream the plate is bounded between two instantaneous positions defined by these two characteristic lines. For example, for $t$ = 60 s, $x$-abscissa related to the faster characteristic is equal to 187.8 m and those related to the second one is equal to 169.8 (Fig. 6).

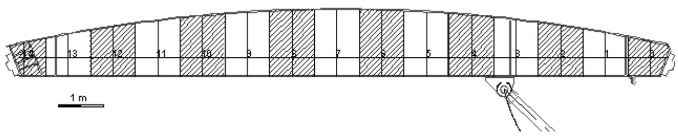


Figure 7. Shape of a downstream gate of the new Lanaye lock.

4.3 *Lock door movement*

In the scope of the studies of a new 225 x 25 x 14 m lock in Lanaye on the river Meuse in Belgium, numerical simulations of the hydrodynamic effects of the downstream mitering gates movements have been realized. Both opening and closure operations have been modeled.

The door is made of two gates, symmetrical regarding lock longitudinal axis and vertically articulated on lock walls. Plane dimensions of a gate are 14.3 m long and 0.9 to 1.5 m wide (Fig. 7). Its height is 17.5 m.

Water height in the lock and the channel, during door movement, is 4.8 m. Water is initially at rest.

By symmetry, only a gate and half a lock have been simulated. Cartesian meshes from 1 x 1 m to 0.2 x 0.2 m have been used. The coarser grid was for lock and downstream channel, while the finer one was for door location.

The lock layout has been modeled in 2D, with the upstream lock wall represented as an impermeable boundary. Downstream, the channel is going to the Meuse river. A suitable boundary condition has to be used to allow waves propagation without introducing spurious numerical reflections. 2D simulations domain has so been completed with a reach

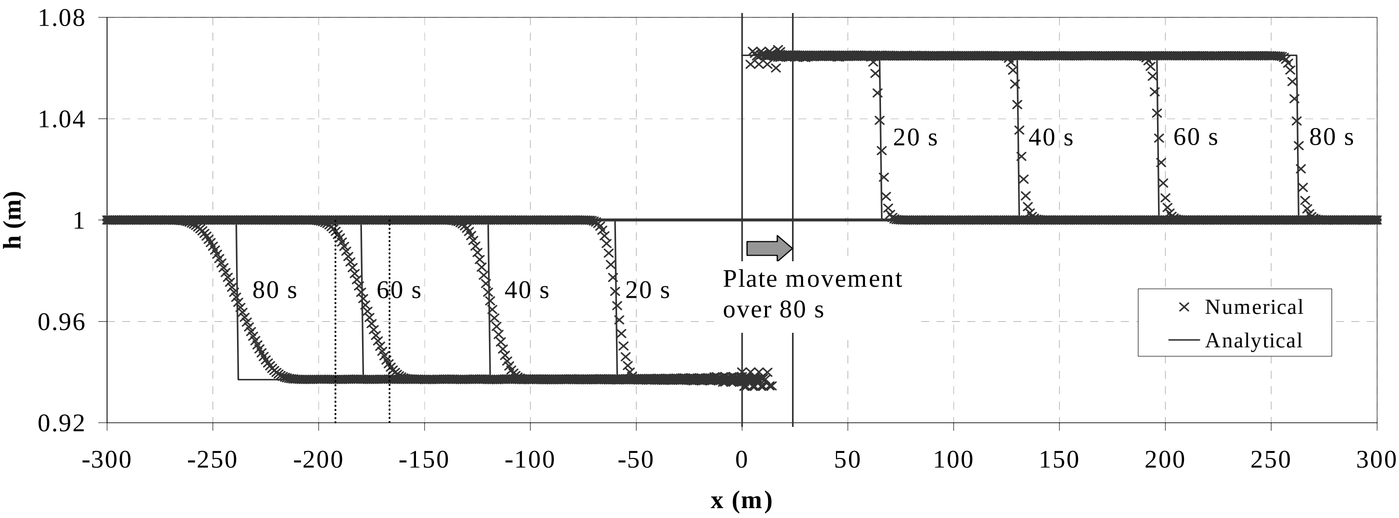


Figure 6. Waves propagation in a rectangular channel induces by the movement with a constant velocity $v_P$ = 0.2 m/s of a vertical plate - Comparison between numerical results and analytical solution at times 20, 40, 60 and 80 s. Water level initially at rest at height $h_0$ = 1 m.

modeled in 1D. Thanks to this original approach, wave propagation is very well represented, without any numerical reflection at the boundary between 1- and 2D models (Erpicum & al. in prep).

The area for the 2D simulation is 470 m long and 30 m wide, with 27,000 meshes. The 1D reach is 8 km long with finite volume cells of 1 m.

A sub-grid with *step* equal to 10 has been used. A representation of the door with regular meshes of 0.02 x 0.02 m is thus performed.

Electromechanical engineering designers have provided the kinetics of the door movement. Time for a gate operation is 105 s. The course angle is 68.1°. Maximal angular velocity is 0.8 °/s.

Examples of instantaneous results for the gate closure are shown on figure 8 in terms of flow velocity fields. The simulations allowed assessing water levels variation along the gate faces, and thus evolution of the hydrostatic resultant along time to be counterbalanced by the door mechanism.

To complete this numerical approach, experiments are currently carried out at the Laboratory of Hydraulic Constructions of the University of Liège.

## 5 CONCLUSIONS

2D finite volumes free surface flow solvers are of great interest in most fields of hydraulic engineering. Among them, those based on Cartesian grids, such as Wolf2D, are simple, accurate and require less storage capacities than models using unstructured meshes.

Even if Wolf2D multiblock or coupling potentialities allow mesh refinement near interesting areas without increasing dramatically the number of cells and thus computation time, the main problem of structured grids remains their inability to model boundaries inclined compared to Cartesian axis, and thus complex geometries. Fluid-structure interaction modeling has to face the same problem to accurately take into account the movement of a solid on the simulation domain.

A first approach has been developed and presented in this paper to deal with boundaries inclined compared to axis or moving structures on Cartesian grid in Wolf2D. Based on the cut cell approach, the method is simpler than classical ones as it uses a sub-grid of the initial Cartesian meshes to assess structure instantaneous position. It allows keeping the advantages of regular grids in terms of simplicity, accuracy and storage capacity and is weekly restrictive on solver stability criterion.

The model showed its potentialities through several test cases and an application to the computation of the hydrodynamic fields induced by a lock-door movement. In this last case, 2D model has been coupled with a 1D one to extend the simulation domain without increasing too much computation time.

Improvements of the model should be considered to increase the precision of fluxes evaluation near moving boundaries and to refine the representation of inclined ones. For example, more precise calculation of the structure occupation rate in a cell should be realized.

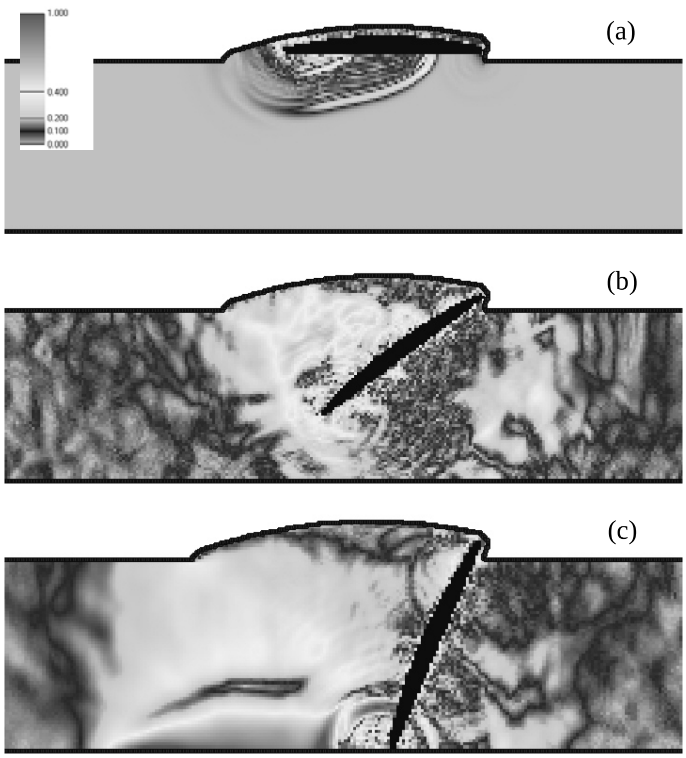


Figure 8. Instantaneous velocity fields (m/s) near the gate during closure operation to time t = 2 s (a), 50 s (b) and 100 s (c)

## NOTATIONS

$c$ = wave celerity
$CFL$ = Courant-Friedrichs-Lewy number
$\mathbf{F}$ = advective terms vector = $[\mathbf{f},\mathbf{g}]$
$\mathbf{f}$ = x-component of advective terms vector
$f$ = generic advective term
$\mathbf{g}$ = y-component of advective terms vector
$g$ = gravitational acceleration
$h$ = water height
$J_x$ = friction slope component along x-axis
$J_y$ = friction slope component along y-axis
$M$ = number of calculation adjacent cells
$N$ = number of cell sides
$\mathbf{n}$ = normal vector
$n_x$ = normal vector component along x-axis
$n_y$ = normal vector component along y-axis
$q$ = specific discharge
$\mathbf{s}$ = unknowns vector
$s$ = generic unknown
$\mathbf{S}$ = source terms vector
$S$ = source term
$t$ = time
$\mathbf{u}$ = fluid velocity vector
$u$ = average fluid velocity along x-direction
$v$ = average fluid velocity along y-direction
$V$ = front velocity

$\mathbf{v_G}$ = grid velocity vector
$v_{G,x}$ = grid velocity along x-direction
$v_{G,y}$ = grid velocity along y-direction
$v_P$ = plate velocity
$x$ = x Cartesian coordinate
$y$ = y Cartesian coordinate
$z_b$ = bed elevation
$\Delta x$, $\Delta y$ = cell dimensions
$\Delta t$ = time step
$\theta$ = cell occupancy rate parameter
$\theta_x$ = x-side occupancy rate parameter
$\theta_y$ = y-cell occupancy rate parameter
$\Gamma$ = cell outline

## REFERENCES


Alcrudo, F., Mulet, J. 2005. Urban inundation models based on the shallow water equations. Numerical and practical issues. In Benkhaldoun, Ouazar, Raghay (ed.), *Finite volumes for complex applications, Proc. 4th intern. symp., Marrakech, July 2005*. London, England: ISTE

Archambeau, P., Dewals, B., Erpicum, S., Detrembleur, S., Pirotton, M. 2004. New trends in flood risk analysis: working with 2D flow models, laser DEM and a GIS environment. In Greco, Carravetta & Della Morte (Ed.), *River Flow 2004, Proc. 2nd intern. conf. on fluvial hydraulics, Napoli, 23-25 June 2004*. Rotterdam, the Netherland: Balkema.

Causon, D., Ingram, D., Mingham, C., Yang, G., Pearson, R. 2000. Calculation of shallow water flows using a Cartesian cut cell approach. *Advances in Water Resources*. Vol. 23: 545-562

Dewals, B., Archambeau, P., Erpicum, S., Mouzelard, T., Pirotton, M. 2002. Coupled computations of highly erosive flows with WOLF software. In Holz, Kawahara & Wang (ed.), *Proc. 5th inter. conf. on Hydro-Science & -Engineering, Warsaw, 18-21 September 2002*.

Dewals, B., Erpicum, S., Archambeau, P., Detrembleur, S., Fraikin, C., Pirotton, M. 2004. Large scale 2D numerical modelling of reservoirs sedimentation and flushing operations. In Hu & Tan (ed.), *Proc. of the 9th inter. symp. on River Sedimentation, Yichang, 18-21 October 2004*.

Dewals, B., Erpicum, S., Archambeau, P., Detrembleur, S., Pirotton, M. In press. Depth-integrated flow modelling taking into account bottom curvature. *Journal of Hydraulic Engineering*. accepted for publication.

Dewals, B. 2006. *Une approche unifiée pour la modélisation d'écoulements à surface libre, de leur effet érosif sur une structure et de leur interaction avec divers constituants*. PhD Thesis. Liège, Belgium: University of Liège

Erpicum, S., Archambeau, P., Dewals, B., Detrembleur, S., Pirotton, M. 2004a. Computation of the Malpasset dam break with a 2D conservative flow solver on a multiblock structured grid. In Liong, Phoon & Babovic (ed.), *Proc. of the 6th inter. conf. on Hydroinformatics, Singapore, 21-24 June 2004*. Singapore: World Scientific Publishing

Erpicum, S., Dewals, B., Archambeau, P., Detrembleur, S., Fraikin, C., Pirotton, M. 2004b. Global design of hydraulic structures optimized with physically based flow solvers on multiblock structured grids. In Hu & Tan (ed.), *Proc. of the 9th inter. symp. on River Sedimentation, Yichang, 18-21 October 2004*.

Erpicum, S., Archambeau, P., Dewals, B., Detrembleur, S., Pirotton, M. In prep. 1D and 2D solvers coupling for free surface flows modeling. in *Proc. of the 7th inter. conf. on Hydroinformatics, Nice, 4-8 September 2006*

Froehlich, D. 2002. Project Field Tests 1 & 2: "Blind Simulation by Davef", in *2nd IMPACT Project Workshop, Mo-i-Rana, 12-13 September 2002*.

Hirsch, C. 1988. *Numerical computation of internal and external flows. Vols. I-II*. Chichester, England: Wiley.

Ingram, D., Causon, D., Mingham, C. 2003. Developments in Cartesian cut cell methods. *Mathematics and Computers in Simulation*, Vol. 61: 561-572

Mouzelard, T. 2002. *Contribution à la modélisation des écoulements quasi tridimensionnels instationnaires à surface libre*. PhD Thesis. Liège, Belgium: University of Liège

Pirotton, M. 1994. *Modélisation des discontinuités en écoulement instationnaire à surface libre. Du ruissellement hydrologique en fine lame à la propagation d'ondes consécutives aux ruptures de barrages*. PhD Thesis. Liège, Belgium: University of Liège

Pirotton, M., Lejeune, A., Archambeau, P., Erpicum, S., Dewals, B. 2003. Numerical - experimental interaction in hydrodynamics: an integrated approach for the optimal management of hydraulic structures and hydrographic basins. In Lu, Lamas, Li & Mok (Ed.), *Computational methods in engineering and science*. Rotterdam, the Netherland: Balkema.

Toro, E. 2001. *Shock-capturing methods for free-surface shallow flows*. Chichester, England: Wiley.

Valiani, A., Caleffi, V., Zanni, A. 2002. Case study: Malpasset dam-break simulation using a two dimensional finite volume method, *Journal of Hydraulic Engineering*, Vol. 128, No. 5: 460-472.