IMPLICIT FORMULATION FOR 1D AND 2D ST VENANT EQUATIONS – PRESENTATION OF THE METHOD, VALIDATION AND APPLICATIONS

Thierry Lepelletier[1], Quentin Araud[2]

[1]Hydratec – Immeuble Central Seine - 42 quai de la Râpée – 75012 Paris - lepelletier@ hydra.setec.fr

[2]Hydratec – 1 rue de la Course – 67000 Strasbourg – araud@hydra.setec.fr

KEY WORDS

St Venant equations, Implicit formulation, solving procedure, shock capture, benchmarking.

ABSTRACT

St Venant equations form the theoretical basis to hydraulic modeling in the field of river flow and urban hydraulics. Numerical schemes may prove difficult to implement due to the hyperbolic nature of these equations and to the variety of configurations met in practice. Existing hydro-informatic software are usually specialized in a given domain. Optimized solving schemes are selected to best fit the practical performance which is required in this domain of application.

In order to meet the needs to a large variety of domains an original formulation has been developed to solve the full St Venant equations in one and two dimensions, with optional simplifications to optimize some computations, while keeping the same code architecture.

The underlying formulation is based upon an original method for solving full St Venant equations through finite volume space discretization and implicit time marching solving procedure. This method is unconditionally stable, time step is variable and is adjusted automatically within calculation in order to preserve numerical accuracy in case of discontinuities, such as shock formation.

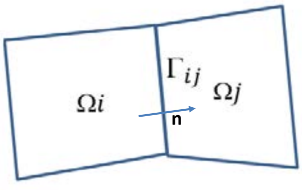
*This numerical scheme is implemented in **Hydra** software, which is developed by HYDRATEC. Various 1D and 2D tests, dam break problems and recirculation in 2D domains are presented and are compared with other referenced software in their respective domain of application, such as **Telemac** (developed by EDF).*

1 DISCRETIZATION OF TWO DIMENSIONAL ST VENANT EQUATIONS AND SOLVING PROCEDURE

St Venant equations are first expressed in classical integral form within a cell :

$$\int_{\Omega i}\frac{\partial}{\partial t}\boldsymbol{U}\,d\Omega i+\oint_{\Gamma_i}(\boldsymbol{F}\,n_x+\boldsymbol{G}n_y)d\Gamma_i=-\int_{\Omega i}\boldsymbol{S_f}\,d\Omega i+\int_{\Omega\mathrm{i}}\mathrm{ghgrad}(\mathrm{z_b})\,\mathrm{d}\Omega\mathrm{i}\quad \text{with :}\qquad (1)$$

$$\boldsymbol{U}=\begin{bmatrix}h\\ q_x\equiv hu_x\\ q_y\equiv hu_y\end{bmatrix}\qquad \mathbf{F}=\begin{bmatrix}q_x\\ \frac{{q_x}^2}{h}+\frac{g}{2}h^2\\ \frac{q_xq_y}{h}\end{bmatrix}\qquad \mathbf{G}=\begin{bmatrix}q_y\\ \frac{q_xq_y}{h}\\ \frac{{q_y}^2}{h}+\frac{g}{2}h^2\end{bmatrix}$$

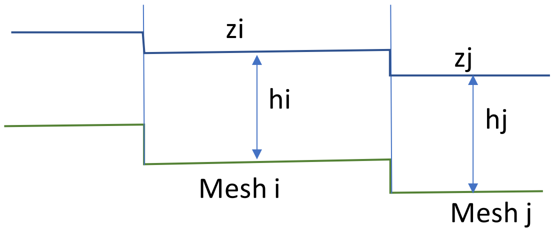


Notations are defined in appended glossary.

Pressure and gravity terms are combined and are approximated as follows :

$$\oint_{\Gamma_i}(\frac{g}{2}h^2)\boldsymbol{n}d\Gamma_i+\int_{\Omega_i}\mathrm{ghgrad}(\mathrm{z_b})\mathrm{d}\Omega\mathrm{i}\approx\sum_j\frac{g\widetilde{h_{ij}}}{2}\oint_{\Gamma_{ij}}(z_j-z_i)\boldsymbol{n}d\Gamma_{ij}\quad \text{where } \widetilde{h_{ij}}=\frac{h_i+h_j}{2}\qquad(2)$$

This expression is valid as long as differences between water depths between two adjacent cells remain small. It turns out that it remains valid in presence of shocks as will be shown in section 2 below.



Spatial discretization of equation (1) consists in decomposing the whole domain into quadrangular or triangular cells and to average components of the unknown **U** vector within each cell. Connectivity condition between cells must be ensured. Equation (1) is rewritten as follows in discretized form, using the approximation of equation (2) :

$$A_i \frac{\partial \boldsymbol{U}_i}{\partial t} + A_i \boldsymbol{S}_{fi} + \sum_j \left(\boldsymbol{F} n_x + \boldsymbol{G} n_y\right) l_{ij} = 0 \qquad (3) \qquad \text{where :}$$

$$\mathbf{F} = \begin{bmatrix} q_x \\ \frac{{q_x}^2}{h} + 0.5 g \widetilde{h_{ij}} (z_j - z_i) \\ \frac{q_x q_y}{h} \end{bmatrix} \qquad \mathbf{G} = \begin{bmatrix} q_y \\ \frac{q_x q_y}{h} \\ \frac{{q_y}^2}{h} + 0.5 g \widetilde{h_{ij}} (z_j - z_i) \end{bmatrix}$$

The terms $\sum_j \left(\boldsymbol{F} n_x + \boldsymbol{G} n_y\right) l_{ij} \equiv \text{Flux}_{ij}$ represent the flux terms across two adjacent cells.

This alternative formulation yields following advantages :

- gravity terms are source terms but they are included in the line integral terms **F** and **G** which are expressed along each cell boundary : expression (3) is best fitted to link together equations between different domains, such as 2D and 1D domain. Linkage is provided by the flux terms Flux_{ij} which have different expressions according to the physical link type. It is also easy to combine simplified and complete formulations within a model. Simplified treatment of a sub domain just requires to ignore the convective terms in vectors **F** and **G**.
- It provides numerical robustness,
- It is easy to introduce singularities within adjacent cells.

Within a 2D domain the flux components are treated by :

- centered formulation for the volume fluxes,
- upwind formulation for the momentum fluxes.

$$\boldsymbol{F_{ij}}.n_x + \mathbf{G}_{\boldsymbol{ij}}.n_y = \begin{pmatrix} 0.5(q_i + q_j) \\ \boldsymbol{\alpha}_i u_{xi} q_i + \boldsymbol{\alpha}_j u_{xj} q_j \\ \boldsymbol{\alpha}_i u_{yi} q_i + \boldsymbol{\alpha}_j u_{yj} q_j \end{pmatrix} + \begin{pmatrix} 0 \\ 0.5 g \widetilde{h_{ij}} (z_j - z_i) n_x \\ 0.5 g \widetilde{h_{ij}} (z_j - z_i) n_y \end{pmatrix} \quad \text{with :}$$

$\begin{pmatrix} u_{xi} \\ u_{yi} \end{pmatrix}$: average flow velocity within one cell i

$$q_i \triangleq q_{x,i}.n_x + q_{y,i}.n_y \qquad \begin{cases} \alpha_i = 1 & \text{si } q_i > 0 \\ \alpha_i = 0 & \text{si } q_i < 0 \end{cases} \qquad \alpha_j = 1 \quad \text{si } q_j > 0$$

$$q_j \triangleq q_{x,j}.n_x + q_{yj}.n_y \qquad \alpha_j = 0 \quad \text{si } q_j < 0$$

Time integration of equation (3) is treated in a fully implicit way :

Flux terms :

$$\boldsymbol{F}_{ij}^{n+1} = F_{ij}^{n} + \left[\frac{\partial \Delta F_{ij}}{\partial \Delta U_i}\right] \Delta \boldsymbol{U}_i + \left[\frac{\partial \Delta F_{ij}}{\partial \Delta U_j}\right] \Delta \boldsymbol{U}_j$$

$$\boldsymbol{G}_{i}^{n+1} = \mathbf{G}_{i}^{n} + \left[\frac{\partial \Delta G_{ij}}{\partial \Delta U_i}\right] \Delta \boldsymbol{U}_i + \left[\frac{\partial \Delta G_{ij}}{\partial \Delta U_j}\right] \Delta \boldsymbol{U}_j$$

$$\mathbf{U}_{\boldsymbol{i}}{}^{\boldsymbol{n+1}} = \mathbf{U_i}^{\boldsymbol{n}} + \Delta \mathbf{U_i}$$

$$\mathbf{U}_{\boldsymbol{j}}{}^{\boldsymbol{n+1}} = \mathbf{U_j}^{\boldsymbol{n}} + \Delta \mathbf{U_j}$$

The Jacobian matrices : $\left[\frac{\partial \Delta F_{ij}}{\partial \Delta U_i}\right], \left[\frac{\partial \Delta F_{ij}}{\partial \Delta U_j}\right], \left[\frac{\partial \Delta G_{ij}}{\partial \Delta U_i}\right], \left[\frac{\partial \Delta G_{ij}}{\partial \Delta U_j}\right]$ are ranked 3x3 et contain differentiated terms with respect to natives variables defined in vector **U**

Inertia term:

$$A_i \frac{\partial \boldsymbol{U}_i}{\partial t} = \frac{A_i}{dt} \Delta \mathbf{U_i}$$

Friction term :

$$\boldsymbol{S}_{f\mathrm{i}}^{n+1} = \boldsymbol{S}_{fi}^{n} + \left[\frac{\partial \Delta \boldsymbol{S}_{fi}}{\partial \Delta \boldsymbol{U}_i}\right] \Delta \boldsymbol{U}_i$$

The final set of equations to be solved at each time step is put in matrix form :

$$\lfloor M_{ii} \rfloor \Delta \boldsymbol{U}_i + \sum_j \left[\boldsymbol{K}_{ij}\right] \Delta \boldsymbol{U}_i = \boldsymbol{F}_i \quad (4)$$

In practice this matrix system is formed by calculating contribution of each boundary to flux terms, then contributions of volume terms within each cell.
Solution of matrix system (4) is calculated using **Pardiso** Matrix Solver.

This formulation encompasses the 1D case, it can therefore be specialized to 1D problems or extended to mixed problems including 1D and 2D sub domains within a model without difficulty.

2. SHOCK CAPTURE

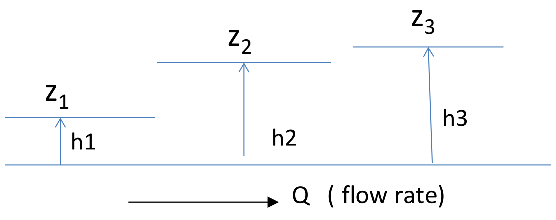
The above formulation is based on native conservation laws of hyperbolic St Venant equations. It contains all the terms which are supposed to produce discontinuities depending on physical configurations. In order to demonstrate the ability of the above algorithm to describe shocks the restricted 1D case with 3 adjacent cells is considered :

The discretized equation (3) along the x axis is written as follows in steady state flow regime for the cell M2. The following equation is obtained by neglecting friction terms :

$$-Qu_1 + Qu_2 + \frac{g}{2}(z_2 - z_1)h_{12} + \frac{g}{2}(z_3 - z_2)h_{23} = 0 \quad (5)$$

$$h_{12} = 0.5(h_1 + h_2) \qquad h_{23} = 0.5(h_2 + h_3)$$

Let us assume that a hydraulic jump occurs between cell 1 and 2 :
One has : $z_2 \approx z_3$



Equation (5) becomes :

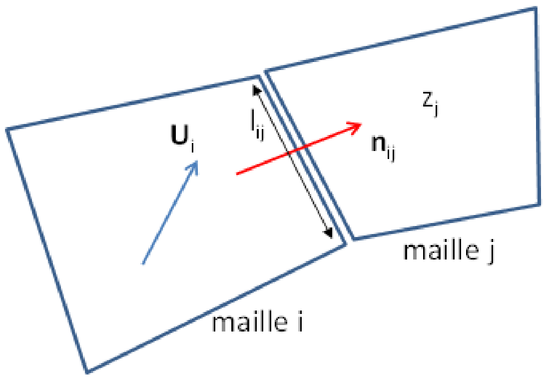
$$-Qu_1 + Qu_2 + \frac{g}{2}\,(h_2 - h_1)h_{12} = 0$$

$$Qu_2 + \frac{g}{2}\,h_1 h_{12} = Qu_1 + \frac{g}{2}\,h_2 h_{12} \quad (6)$$

Equation (6) describes a jump equation which is linearized with respect to term h_{12} . This shows that our formulation allows to capture shocks, albeit introduction of some distortion for strong shocks. The practical validity or this simplification is discussed with the tests case presented in chapter 4.
The above analysis can be extended easily to transient shock waves, such as in the dam break problem, by taking in account the inertia terms of equation (3) : it can be checked that the right expression for the shock velocity of the moving shock wave is obtained.
This shock analysis can extended to the 2D case. A further simplification is introduced by assuming that the local direction of a 2D shock wave is perpendicular to the boundary between two adjacent cells. Starting from equation (3) one obtains the equations for the oblique hydraulic jump, assuming that the boundary line between two cells is a shock line. This additional assumption yields wrong results locally but it has been found that the distortion levels out when one considers the resulting wave pattern on a scale involving a few cells together.

3. EXTENDED FORMULATION WITH HYDRAULIC SINGULARITIES

So far the formulation above applies to an homogenous 2D domain : two adjacent cells are connected by relations involving flux terms. It is fairly straightforward to generalize this concept by adapting the flux terms to hydraulic exchange laws involving singular head losses:



Exchange between the two cells is controlled by the flow rate Q et the momentum flux vector **F**. In case of a singularity linking the cells, Q and **F** are expressed as : Q(H_i, H_j) and **F**(H_i, H_j) where H stands for the averaged total head within one cell. The expressions for Q and **F** depend on the nature of the singularity.

Assuming a flow from cell i to cell j the flux leaving cell i is written as :

$$\left(\boldsymbol{FLUX}_{ij}\right) = \begin{pmatrix} Q \\ q_i u_i \\ q_i v_i \end{pmatrix}$$

And the flux entering flux j is written as :

$$\left(\boldsymbol{FLUX}_{ji}\right) = \begin{pmatrix} -Q \\ -F_x \\ -F_y \end{pmatrix}$$

As an example the gate let us consider the gate type singularity which is governed by a variety of laws according to the gate geometry and hydraulic upstream and downstream conditions.

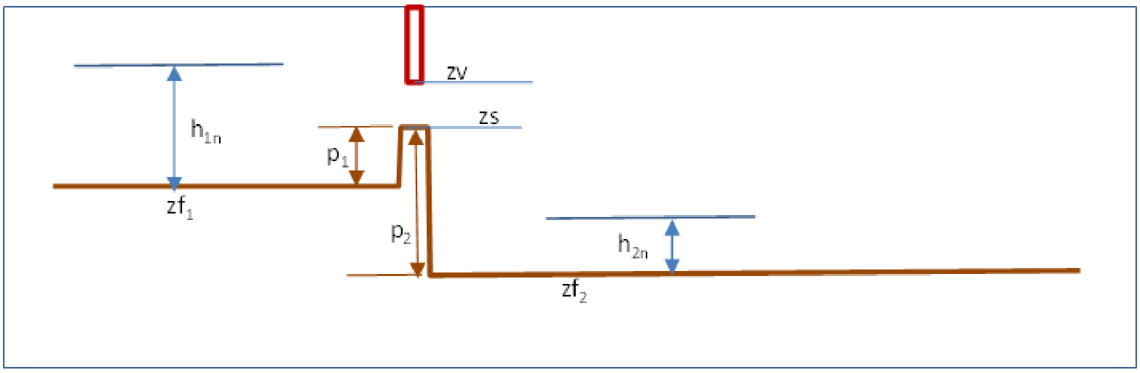
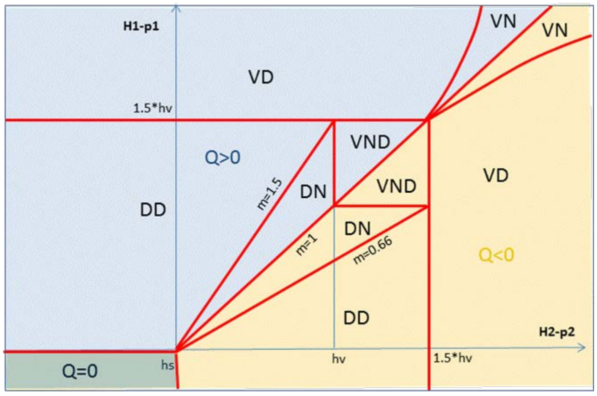


Figure 1 : definition sketch for the gate/weir singularity

Next diagrams illustrates all the laws which govern this singularity and the range of validity of each. The flow equations are defined such that flow continuity is ensured at every transition between two flow regimes. Equations are given below for 4 basic regimes:



Weir flow regime: $H_1 - p_1 < \frac{3}{2} h_v$:

$$QD_D = \frac{2}{3} C_c b \sqrt{2g} (H_1 - p_1)^{\frac{3}{2}}$$

$$QD_N = b\sqrt{2g}\left[(H_1 - p_1) - (H_2 - p_2)\right]^{\frac{1}{2}} (H_2 - p_2)$$

Gate flow regime: $H_1 - p_1 > \frac{3}{2} h_v$

$$QV_D = b C_c h_v \sqrt{2g(H_1 - p_1)}$$

$$QV_N = K b h_v \sqrt{2g} \left((H_1 - p_1) - (H_2 - p_2)\right)^{1/2}$$

Figure 2: delimitation of flow regimes for the gate/weir singularity

Special consideration must be given to the momentum flux just downstream of the gate. Iterative calculations are necessary to determine the proper flow regime and to apply the corresponding flow equations.

In case of an incident flow oblique to the gate the upstream momentum flux vector is decomposed into two components, one normal to the gate, the other one parallel to the gate. The flux parallel to the gate is supposed unchanged, while the other flux component is treated by the 1D gate equation system.

3. BENCHMARKING

The above formulation has been implemented into the **Hydra** computational software and has been evaluated against analytical solutions, field data and existing referenced codes such as **Telemac** (EDF) and **Rubard** (IRSTEA). Some selected examples are presented below to demonstrate the capabilities of the code:

3.1 Hydraulic jump in rectangular channel

Channel width is discretized into 3 meshes 5m each. Channel length is discretized into 60 meshes :

- 20 cells 10x5m in the upstream part of the canal which has a 2% slope,
- 40 cells 5x5m in the downstream part which is flat.

Flow rate is 60 m^3/s. Figure below shows the computed water profile along the centerline. It is compared with the analytical solution:

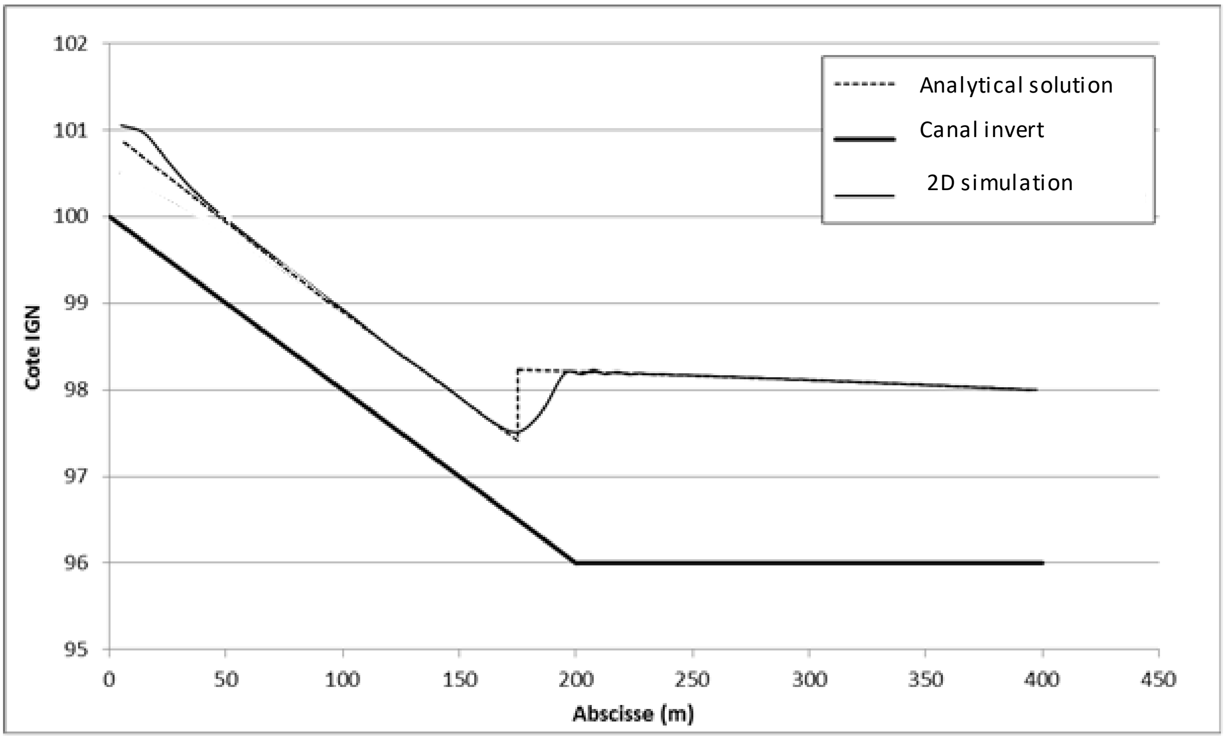


Figure 3: water profile elevation along a rectangular channel : computed and analytical solutions

The two curves are in good agreement, given the spatial discretization used for the computations. Difference on the upstream end is due to the fact that no upstream limit condition is imposed : the code selects critical flow regime. The water profile then connects gradually towards uniform flow.

3.2 flow past a gate

Channel discretization and flow conditions are the same as above but a gate is introduced at abscissa 0.4km. Gate opening depth is 50cm. the graph below shows the computed water profiles using complete formulation and simplified formulation (without convective terms) :

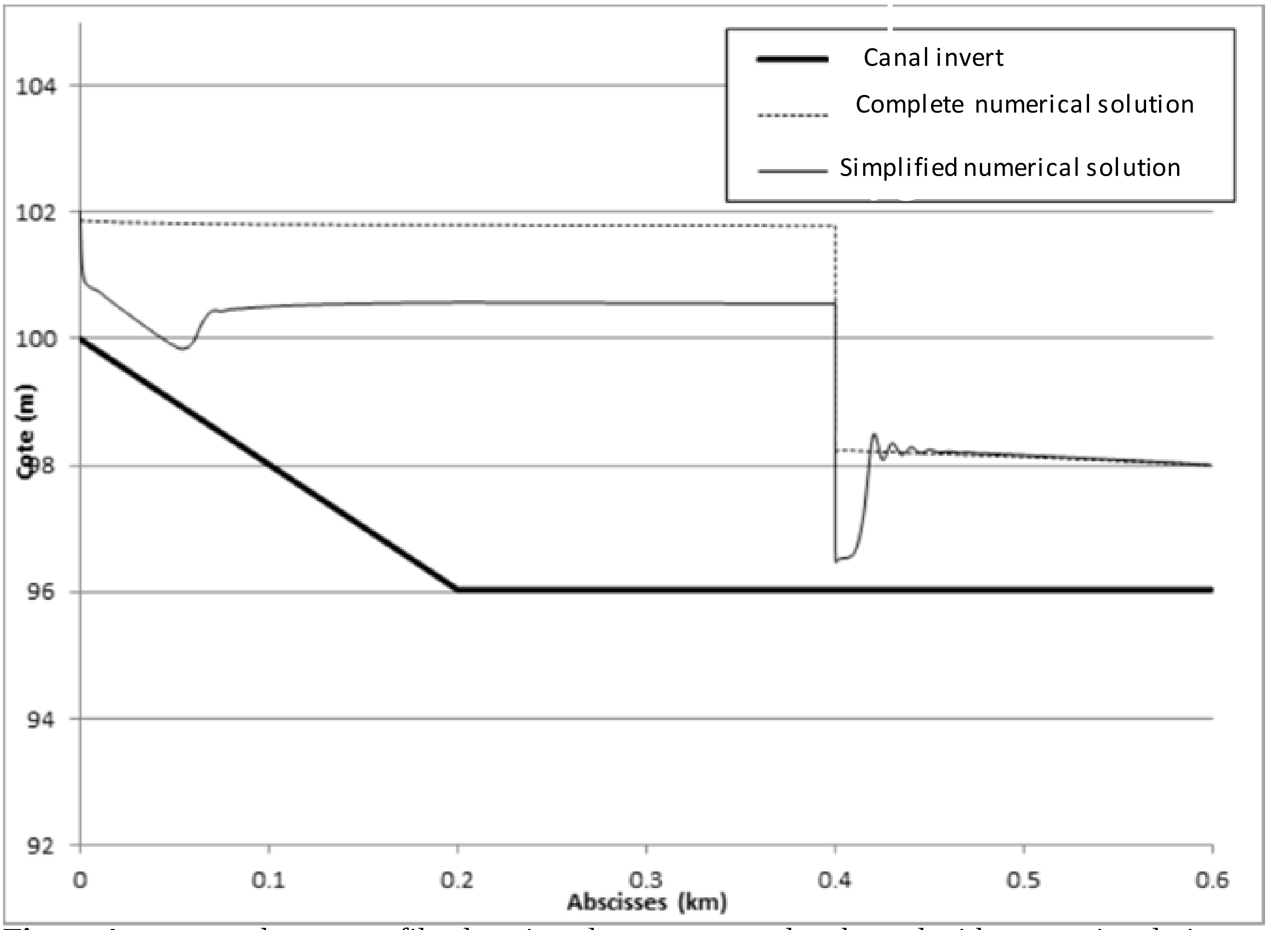


Figure 4: computed water profile elevation along a rectangular channel with a gate singularity

Water profile computed with the full equations exhibits a hydraulic jump in the upper section of the channel and another one about 25m downstream of the gate. The flow past the gate is not influenced by the downstream conditions. In the simplified calculations convective terms are ignored : super critical flow is not modelled, which results in a significant over estimates of the water profiles upstream of the gate.

3.3 2D flow in a river past a navigation weir.

A section of the Aisne river has been modelled to analyze velocity field past a navigation weir and to investigate resulting geo-morphological consequences. The model consists of about 10000 meshes arranged as follows:

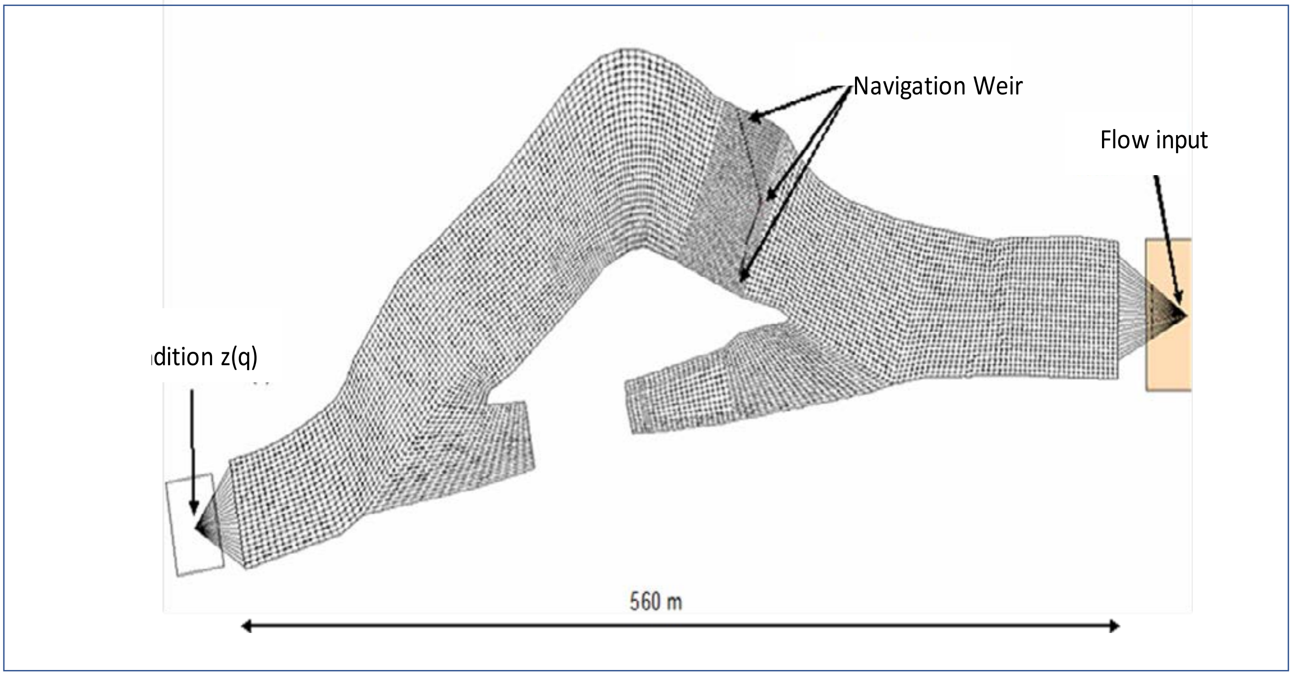
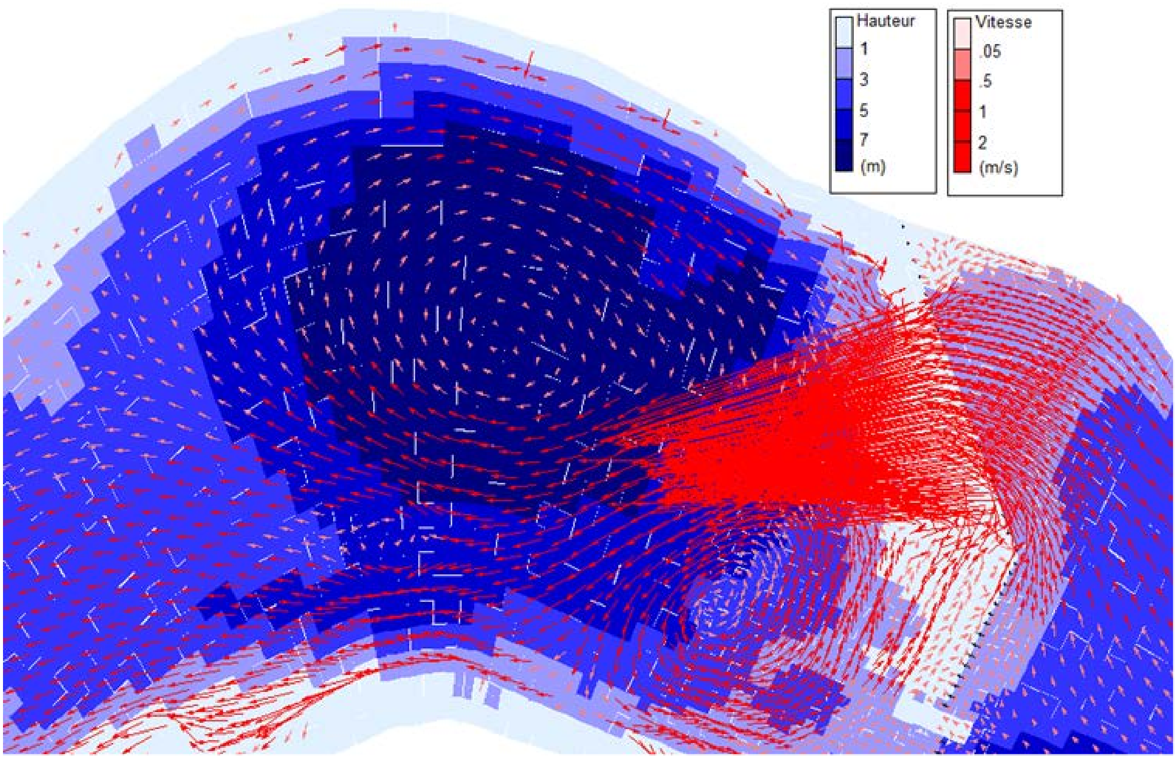


Figure 5 : 2D meshing for the river modelling pas a navigation weir

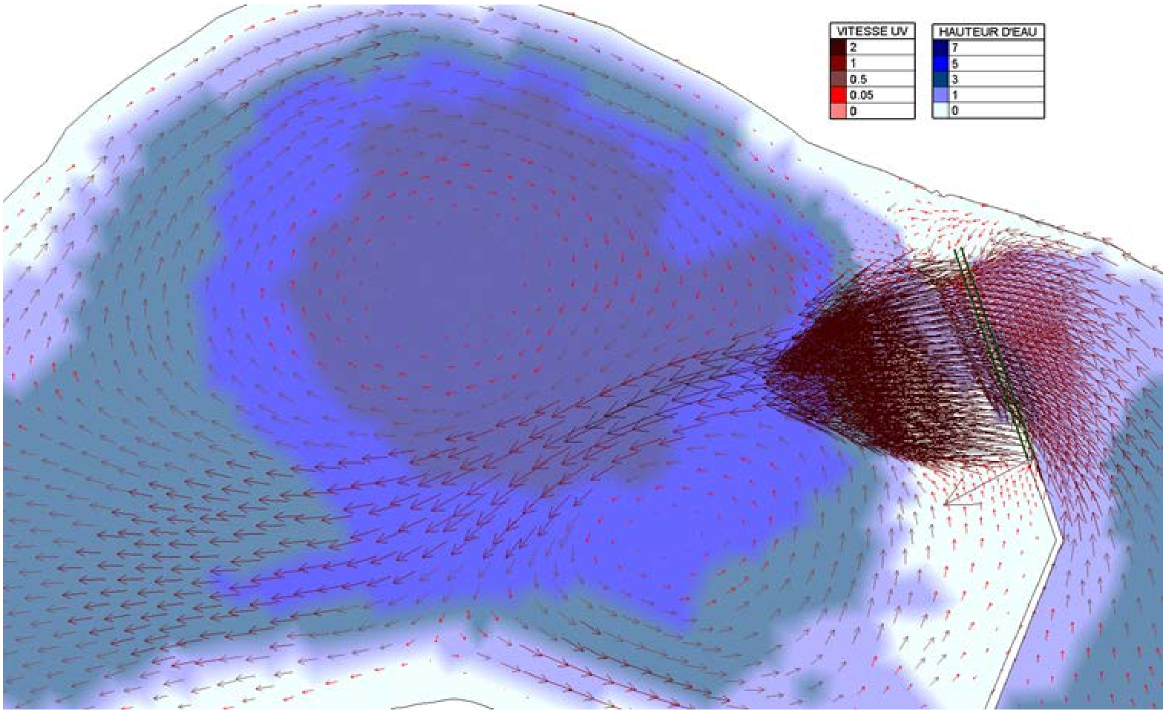
The weir is closed except along a 25m section next to the right bank. It is oblique to the upstream flow. River flow rate is 100 m^3/s. The weir elevation is adjusted so that the flow past it is critical.
A close up view of the flow velocity distribution is shown below :



hydra code

Figure 6 : flow distribution computed by **hydra** numerical code

One can see clearly the two recirculation zones induced by the jet flux past the weir. This pattern was actually observed on the site. The same calculations was performed using **Telemac** code. The computed flow distribution exhibit a very similar pattern as shown below. In the **Telemac** model the mesh size around the weir is much more refined than in the Hydra model because the weir is modelled as a geometric singularity : it is totally immerged within the 2D meshing.



Telemac code

Figure 7: flow distribution computed by **telemac** numerical code

3.4 Transient dam break problem in a rectangular channel.

This problem is classic and has an analytical solution. The case investigated is defined by the following definition sketch :

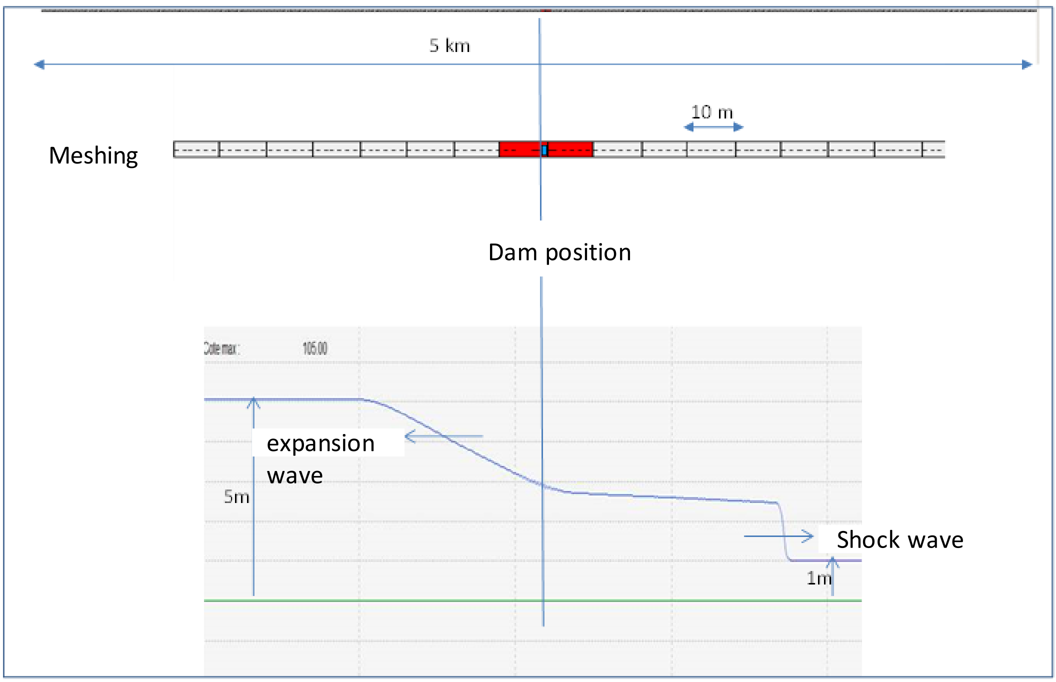
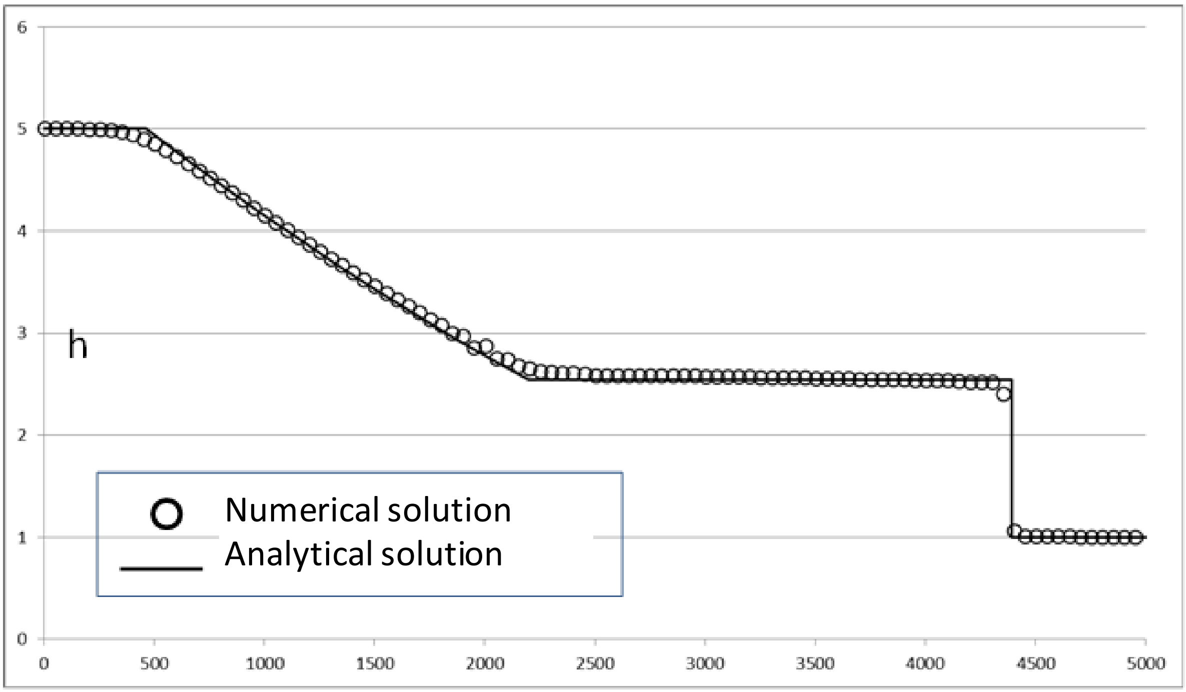


Figure 8 : definition sketch for the dam break problem

Initially water depth is 5m upstream of the dam and 1m downstream. The dam breaks at time=0. Figures below show the longitudinal water and velocity profiles along the canal at time t = 288seconds.



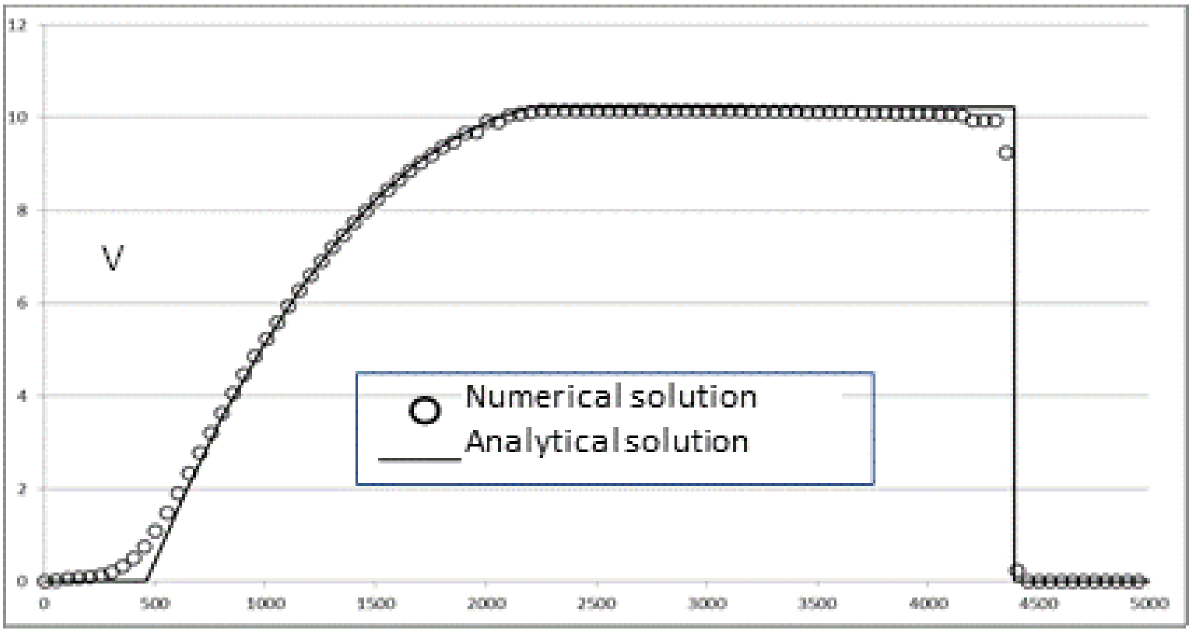


Figure 9 : water elevation and velocity profiles at time 288seconds after sudden dam break.

Examination of these graphs show near perfect agreement between computation and analytical solution.
This good agreement suggests that the approximation introduced in equation (3) is valid to solve practical problems involving shocks.

4 DISCUSSION AND CONCLUSION

The implicit formulation presented in this paper to solve full equations of free surface flow is believed to mark a progress which departs from existing methods. It combines several advantages: unconditional numerical stability, computational robustness, accuracy, flexibility for solving problems in a wide variety of fields: estuary and river hydraulics, flood propagation and networks hydraulics. Finite volume formulation is based on the native conservative form of the St Venant equations without any assumptions on the solution structure across a hydraulic shock. It includes any kind of hydraulics singularities in transient as well as steady state flow regimes and to combine simplified and full resolution of equations in the same model. Formulation is also quite suited to model 1D-2D interactions within a single model.

Calculations are fast: inclusion of singularities and constraints lines into a model enables to choose fairly loose meshing as compared with full 2D codes for the same accuracy. This results in considerable time savings in computations.

This upgraded formulation is implemented into **Hydra** software and has shown particular efficiency in all modelling problems involving floods and inundations in urban areas. Hydra is now coupled to a user's interface immerged in the Open Source **Qgis** GIS (ref[4]). Coupling between this interface and the computational code, should make Hydra attractive among the hydro informatic software available on the market.

BIBLIOGRAPHY

1. J.M. Hervouet - Hydrodynamics of free surface flow. Modelling with Finite Element method - Wiley ed. 2008
2. A Paquier - Rupture de barrage : validation des modèles numériques du CEMAGREF dans le cadre de CADAM – Ingenierie IRSTEA edition 2001
3. E. Toro – Shock capturing methods for free surface flows – Wiley ed. 2003
4. Development of a hydraulic modelling platform within an Open Source environment – C Duran, Th. Lepelletier, V. Mora - 2017 Symhydro symposium.

GLOSSARY

(u_x u_y) : velocity components in the x and y direction.
g : acceleration of gravity
h : water depth
h_i : mean water depth within cell i
l_{ij} : segment length between two adjacent cells
A_i : cell area
z_b : bottom elevation
U : vector composed of unknown variables (h q_x q_y)
(q_x q_y) : = (hu_x hu_y)
H : pressure head = $h + \frac{u_x^2 + u_y^2}{2g}$
$\boldsymbol{S_f}$: bottom friction vector with components :
$S_{f,x} = gh\,(n^2{}_x u_x \sqrt{u_x^2 + u_y^2}\, h^{-4/3})$ et $S_{f,y} = gh(n^2{}_y \sqrt{u_x^2 + u_y^2}\, h^{-4/3})$
(n_x n_y) : Manning coefficients in Ox et Oy direction.