NUMERICAL SIMULATION OF MIXED FLOWS IN HYDROELECTRIC CIRCUITS WITH TEMPORARY FLOWS FLOWMIX SOFTWARE

Christian Bourdarias, Stéphane Gerbi
Laboratoire de Mathématiques, UMR CNRS 5127
Université de Savoie Mont Blanc, 73376 le Bourget du Lac, France
christian.bourdarias@univ-smb.fr, stephane.gerbi@univ-smb.fr

Victor Winckler
Electricité de France, Centre d'Ingénierie Hydraulique
73373 Savoie Technolac, le Bourget du Lac cedex, France
victor.winckler@edf.fr

KEY WORDS

Free surface, Pressurized and Mixed flows, Finite Volume, Methods, Euler equations, Saint Venant, piezometric level, discharge.

ABSTRACT

A hydroelectric powerplant can involve some adductions which are not used all the time, but temporaly. The flow in such conduits can become mixed, that is the conduit is partially concerned with a pressurized flow and a free-surface flow at the same time. The FlowMix software is a one dimensional and unsteady code, able to model this type of flow, based on the similarities between the Euler incompressible equations and the Saint-Venant equations, and an original method to associate theses two systems of equations. This paper presents the numerical methods which were developed for this code, the physical hypothesis which were choosen, and the possibilities of modelization of the software, The FlowMix software is issued from several years of researches between two partners : the Mathematical Institue of the Savoie Mont Blanc University (LAMA) and the Hydraulic Engeneering Centre of EDF (CIH), which lead to an industrial version in 2016. Finally, some industrial cases from the EDF hydropowerplants park are presented : Conduit A, Rizzanese, la Coche.

1. INTRODUCTION

In general, the main circuits of EDF hydropower schemes consist of an upstream reservoir, a headrace tunnel with pressurised flow or free surface channels, then a pressurised penstock and turbines to produce electricity, followed by a slightly pressurised tailrace channel or tunnel.

Parallel to these main circuits, there are temporary flow conduits. The flow can be mixed in these conduits, that is to say partly free surface and partly pressurised.

<u>Discharge conduits</u> allow a flow to be temporarily diverted when it is not going through the turbines following load rejection or a normal shutdown. This type of conduit can be found on the plants at Avrieux, Calypso, Les Sauces, Loudenvielle, Luz, Pralognan and Sabart.

At Les Sauces plant, upstream of Roselend reservoir in Savoie, for example, when the units shut down, the water is temporarily diverted to La Gittaz reservoir by opening gate SF3bis.

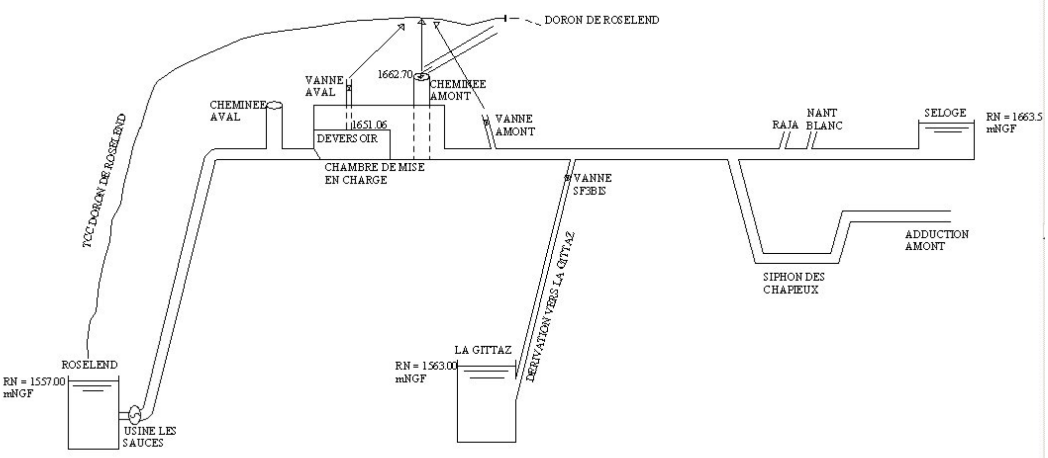


Figure 1 : Diagram of Les Sauces scheme

<u>Transfer conduits</u> allow water to be transferred from one reservoir to another, or send it under a structure: conduit A, Rizzanese.

<u>Drainage conduits</u> are used to recover water coming from diversion tunnels that cannot go through the penstock when it is undergoing maintenance work. This is the case of the La Coche drainage conduit, for example.

With the aim of designing or optimising the operation of its schemes, EDF needs to know the flow that will pass through this type of conduit. When the flow is mixed, it is not possible to make a simple calculation using a pressurised flow code (Belier) or a free surface code (Mascaret).

The FlowMix code, which models 1D mixed flows, is the result of many years of mathematical research and development at the LAboratoire de MAthématiques (LAMA), in collaboration with EDF’s Hydro Engineering Centre (CIH), both of which are located at the Université de Savoie site in Chambéry.

In January 2016, an “industrial” version of the software was handed over by LAMA to EDF : more user-friendly (clearer data files, limit conditions written in Fortran files replaced by text files) and with full documentation : an operating manual, a manual of principles, test validation documentation adapted for engineering studies and a computer manual for software developers.

2. OBJECTIVES AND AREA OF THE SOFTWARE

The FlowMix software is dedicated to the calculation of mixed unsteady flows. It can of course be used in the context of completely free surface flows in open channels or conduits as well as in the case of fully pressurized flows. The domain of validity is determined by a number of physical assumptions:

1. The pipes considered have a constant section or a progressive change of section. They remain in the same vertical plane and their slope is gradually varied. The walls are impermeable and non-deformable. The pipe has a preferred flow axis: the average flow velocity vector is parallel to this axis and the velocity can be considered as constant in any section perpendicular to this axis

2. In a free-surface flow, water is incompressible, the flow is irrotational and the accelerations in the direction perpendicular to the flow axis are negligible (hydrostatic hypothesis).

   In a pressurized flow, the liquid is slightly compressible and homogeneous. The law of pressure used is barotropic and is written as : $p_{abs} = p_{atm} + c^2(\rho - \rho_0)$ where p_{atm} is the atmospheric pressure, assumed to be constant for the whole circuit, ρ_0 and ρ the densities of water respectively at the pressure p_{atm} and p. The wave speed is then c. We take here $\rho_0 = 1000$ kg/m^3.

3. The effects of viscosity inside the fluid are negligible compared to friction on the walls, which are taken into account by an empirical law (Strickler's law).

The interactions between air and water are not modelized. On the other hand, there is no hypothesis of a slight slope.

3. DESCRIPTION OF THE PHYSICAL MODEL

We describe below the free surface and pressurized flow models established under the above hypotheses. Their coupling results in the mixed flow model at the base of FlowMix. For a complete description of this model, see [2]

3.1 Free surface flows

The Saint-Venant system for free-surface flows in a closed channel or pipe, based on the conservation of mass and momentum, is written in matrix form :

$$\partial_t U + \partial_x F(x,U) = G(x,U)$$

with

$$U = (A,Q)^t, F(x,U) = \left(Q, \frac{Q^2}{A} + gI_1(x,A)cos\theta(x)\right)^t$$

$$G(x,U) = \left(0, -gA\left(\partial_x Z_F + z_G \partial_x cos\theta(x) + S_f\right) + gI_2(x,A)cos\theta(x)\right)^t$$

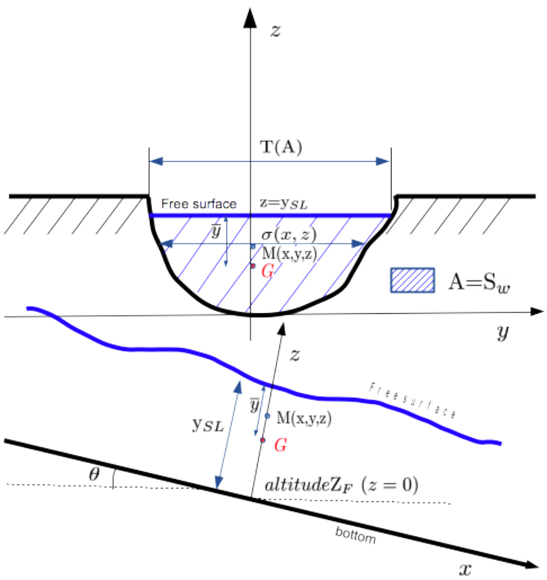


Figure 2: Free surface flow in a channel

t is the time and *x* is the curvilinear abscissa along a curve chosen as the "preferred flow axis" and defined by its elevation $Z = Z(x)$: we chose here the thread of water ($Z = Z_F$). The unknowns are the wetted area $A = A(x,t)$, also denoted S_w, and the discharge $Q = Au = Q(x,t)$. The other terms are (see figure 2 above) :

- $gI_1 \cos\theta$: pressure term in the flux.
- $gI_2 \cos\theta$: pressure source term.
- Under the hydrostatic hypothesis we get :

$$I_1 = \int_0^{y_{SL}} (y_{SL} - z)\sigma(x,z)dz = A\,\bar{y}$$

$$I_2 = \int_0^{y_{SL}} (y_{SL} - z)\partial_x\sigma(x,z)dz$$

with

- $y_{SL} = y_{SL}(x,t)$: height of water between the bottom and the free surface, measured in the direction perpendicular to the flow axis,

  - $\sigma = \sigma(x,z)$: width of the reach at position x along the pipe and at height z above the bottom in the direction perpendicular to the raft,
  - $\theta = \theta(x)$: angle between the horizontal and the base of the pipe. We have also $\partial_x Z_f = \sin\theta$,
  - y : distance between free surface and center of gravity G of the wetted area.
- g : the acceleration of gravity,
- Z_F : altitude of the abscissa point x of the raft,
- z_G : ordinate of the center of gravity G of the wetted area. We have: $h = z_G + y$.
- S_f : friction term, given by the Strickler's formula:

$$S_f = \frac{Q|Q|}{A^2 K_s(x)^2 R_h(x,A)^{\frac{4}{3}}}$$

where K_s, depends on the position x, is the Strickler's coefficient, $R_h = \frac{A}{P_m}$ is the hydraulic radius and P_m the wetted perimeter function of x and A.

This system is strictly hyperbolic for $A > 0$. The Jacobian matrix of the flux is written as :

$$D_U F(U) = \begin{pmatrix} 0 & 1 \\ c^2 - u^2 & 2u \end{pmatrix}$$

with $u = \frac{Q}{A}$ (mean velocity of the liquid in a section along the flow axis) and a wave speed $c = c(x,A)$ given by:

$$c^2 = \frac{\partial g I_1 cos\theta}{\partial A} = \frac{gA}{T} cos\theta$$

where $T = T(x,A)$ is the mirror width. The eigenvalues are : $\lambda_1 = u\text{-}c$ *and* $\lambda_2 = u + c$ with the right eigenvectors :

$$r_1 = \begin{pmatrix} 1 \\ u - c \end{pmatrix} \; et \; r_2 = \begin{pmatrix} 1 \\ u + c \end{pmatrix}$$

3.2 Pressurized flows

The system describing the pressurizd flows in the case of a non-expansible pipe with cross-section $S = S(x)$ is obtained from the compressible Euler equations (with the barotropic pressure law explicated in the hypotheses) written in the coordinate system attached to the thread of water. These equations are integrated along the sections perpendicular to the water axis in a process analogous to the one leading to the Saint-Venant equations. It is written as :

$$\partial_t U + \partial_x F(U) = G(x,U) \qquad (1)$$

With

$$U = (\bar{\rho}S, \bar{\rho}Q)^t = (M,D)^t, F(U) = \left(D, \frac{D^2}{M} + c^2 M\right)^t$$

$$G(x,U) = \left(0, -gM\partial_x Z + c^2 M \partial_x \ln(S) - gMz_G \partial_x cos\theta - g\frac{1}{K_S^2 R_h^{\frac{4}{3}}}\frac{D[D]}{M}\right)^t$$

The unknowns $\rho = \rho(x,t)$ and $Q = Q(x,t) = S(x)u(x,t)$ are respectively the mean density of the water in a section and the flow rate. We have $Q(x,t) = S(x)u(x,t)$ where u is the average flow velocity in a section. Here, the quantity R_h is only dependant on the geometry. We have adopted by default the same friction law as in the free surface flow model. This system is also strictly hyperbolic for $M > 0$, which is guaranteed for a pressurized flow. The Jacobian matrix of the flux and the eigenfunctions are identical to those of the Saint-Venant system but with a constant wave velocity c.

3.3 Coupling

The two previous systems are formally very close, the differences essentially coming from the law of pressure. The search for a "natural" coupling of the two models invites to use a single set of conservative variables and requires a continuous flow through the *transition points*, that is to say where there is a transition from a free surface flow to a pressurized flow or vice versa. Let us divide the two members of each equation of the pressurized flow system by the constant ρ_0 and let us denote

$$A^* = \frac{\bar{\rho} A_{max}}{\rho_0} \; and \; Q^* = \frac{\bar{\rho} Q}{\rho_0}$$

we obtain variables that are expressed in the same units as those of the free surface model (m^2 for A^* and m^3/s for Q^*). When switching from one type of flow to another, the variables A (free surface) and A^* (pressurized) coincide, as well as Q and Q^* since we get thus: $\rho = \rho_0$. The flow speed satisfies

$$u = \frac{Q}{A} = \frac{Q^*}{A^*}$$

To perform the coupling, it is necessary to use a unique set of variables denoted $(\tilde{A}, \tilde{Q})$. We also introduce a status indicator E equal to 0 for a free surface flow and 1 for a pressurized one. Let us specify $(\tilde{A}, \tilde{Q})$:

- for $E = 0$ we have $\tilde{A} = A = S_w \leq S$,
- for $E = 1$ we have $\tilde{A} = A^* > 0$. The case $A^* < S$ corresponds to a subatmospheric pressurized flow.

With these notations, the term of pressure in the free surface flow model is $gI_1(x,\tilde{A})\cos\theta(x)$ while the pressure term in the pressurized is $c^2\tilde{A}$. When the computation of the fluxes, at the numerical level, is done on either side of a transition point, continuity is ensured by the addition of the constant $gI_1(x,S)\cos\theta(x)$ to the pressurized flux. Note that the other term of the mixed flow, namely $\frac{\tilde{Q}^2}{\tilde{A}}$ is obviously continuous at the transition points. On the other hand, the wave speed is discontinuous at the transition points: this requires an adapted method for calculating numerically flows, which is one of the key points of FlowMix.

4. NUMERICAL METHODS

In the following, we use the set of mixed variables, simply denoted *(A, Q)*. The meaning to be given to these variables therefore depends on the state of the flow but there is never any ambiguity. The scheme used is of the Finite Volumes type. After a discretization of the domain, such a scheme makes a balance of the conservative variables (mass, momentum) based on a calculation of the fluxes at the interfaces, how to define these flows (solver) characterizing the scheme. We have chosen a coupling between two specialized solvers: a kinetic solver for surface free flow zones, an upwinded VFRoe solver for pressurized flow regions. The coupling takes place at the transition points via the fluxes calculated on both sides and after application of the continuity correction specified above.

4.1 Notations

The axis of the pipe is divided into N meshes $m_i = [x_{i-1/2}, x_{i+1/2}]$, $1 \leq i \leq N$ so that if L designates the length of the pipe, we have $x_{\frac{1}{2}} = 0$ and $x_{N+\frac{1}{2}} = L$. The center of the mesh m_i is denoted x_i and its length h_i

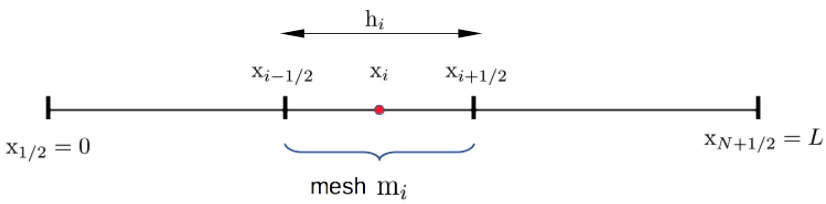


Figure 3: Mesh.

Spatial discretization is not necessarily uniform. The time step, variable, is noted Δt (without index, for the sake of simplicity). Then we set $t_0 = 0$, and for $n \geq 0$, $t_{n+1} = t_n + \Delta t$.
The discrete arrays are the arrays

$$U_i^n = \begin{pmatrix} A_i^n \\ Q_i^n \end{pmatrix} \; 1 \leq i \leq N, 0 \leq n \leq n_{max}$$

where n_{max} is such that $(n_{max} + 1)\Delta t > T$, where T is the final time. We may consider U_i^n as an approximation of the mean value of the exact solution $U(x,t_n)$ in the mesh m_i.

The values at the upstream and downstream bordery will be associated with two fictitious meshes numbered respectively 0 and $N + 1$ and to the arrays U_0^n, U_{N+1}^n.

4.2 Kinetic solver for the free surface flow

The kinetic solver that we have developed is adapted from the work of Perthame *et al.* ([1, 5]) in the case of a free surface flow in a pipe or in a channel that is not necessarily rectangular. It makes it possible to take into account in a natural way dry bottoms (watering on dry bottom or drying) and allows a simple upwinding of all the source terms so as to preserve the stationary states (conservation of the water at rest, in particular). It ensures the preservation of the positivity of A under the CFL condition restriction of the time step.

4.3 Well balanced VFRoe solver for pressurized flows

In this section, we recall the principle of the VFRoe explicit scheme with a well-balanced slope treatment applied to the system (1) written with the mixed variables without taking account of the boundary conditions. Consequently, it is considered here that the index of mesh, i, browses $\mathbb{Z}$, see [3]. According to an idea of Greenberg and Leroux [4], the altitude of the raft (or water line) is described by a function Z constant by mesh and the system is added to equation $\partial_t Z = 0$: this method makes it possible to integrate the term of topography in the flows, the new state variable then being $(Z,A,Q)^t$.

We extend this method by discretizing also the section S by a constant function by mesh and thus the term $Z - \frac{c^2}{g} lnS$ which appears in the equation of momentum. Finally, we can take into account frictions in the definition of the slope at each time step (notion of *dynamical slope*) by setting

$$\tilde{Z} = Z - \frac{c^2}{g} lnS + \int_0^x K(s, \mathrm{A}, E)\, u|u| ds$$

and thus upwind all source terms. We set also $Z_1 = \cos\theta$ (which is by principle constant by mesh in the context of the study) and add to the system (2) the equations $\partial_t \tilde{Z} = 0$ and $\partial_t Z_1 = 0$. The new state variable will thus be $W = (\tilde{Z}, Z_1, A, Q)^t$. We then obtain the system :

$$\partial_t W + \partial_x \Phi(x, W) + gA\partial_x \tilde{Z} + gAz_G \partial_x Z_1 = TS(W) \qquad (2)$$

with

$$\Phi(x, W) = \left(0, Q, \frac{Q^2}{A} + c^2 A\right)^t \text{and } TS(W) = \left(0, 0, -\frac{gK(A)Q|Q|}{A}\right)^t$$

This way of discretizing the slope and the geometry introduces, during the resolution of the Riemann problem at the interfaces between meshes, stationary waves linked to the jumps of Z, S and $\cos\theta$.
Let us denote W_i^n, the discrete unknown, approximation of the mean value of W on $]x_{i-1/2}, x_{i+1/2}[$.
By integrating (2) on $]x_{i-1/2}, x_{i+1/2}[\times[t_n, t_{n+1}[$ we are led to write the Finite Volume scheme in the form:

$$W_i^{n+1} = W_i^n - \frac{\Delta t}{h_i}\left(\Phi\left(W^*_{i+\frac{1}{2}}(0^-, W_i, W_{i+1})\right) - \Phi\left(W^*_{i-\frac{1}{2}}(0^+, W_{i-1}, W_i)\right)\right) + \Delta t TS(W_i^n)$$

where, following Gallouët *et al.* ([3]), $W_{i+1/2}^*(\xi = x/t, W_i, W_{i+1})$ is the exact solution of the linearized Riemann problem at the interface $x_{i+1/2}$, with W_i et W_{i+1} respectively as left and right states. The terms slope, geometry and friction are therefore not explicitly included in the scheme but will be used to calculate W^* : there lies the upwind process.

4.4 Coupling technique : the mixed solver

As a matter of principle, a transition point is always located at a mesh interface. It is then necessary to define a method for calculating the flows on each side of an interface which is a transition point. It is possible that such a point also includes one or more geometric discontinuities (slope break, change of section) or physical (change of pressurized wave velocity). The strategy consists in solving partial Riemann problems in homogeneous zones with respect to the type of flow and the geometric characteristics. There are two "generic" cases: the rise in pressurized flow and the free surface descent along the direction of propagation of the transition point and the type of flow that propagates. The estimated velocity of the transition point from an interface is denoted *w* and it is sufficient, by symmetry, to specify the cases corresponding to a positive velocity. We present here only the case of a free-surface climb, the simplest (Figure 4). The states to be taken into :

- *Ul* and *Ur* states (knowns) respectively to the left and to the right of the current interface, of the same type (FS or Press),
- *UTM* and *UTP* states to the left and right of the transition point, of distinct types,
- *UM* and *UP* states respectively to the left and to the right of the current interface: these are the states that must be calculated for the flow calculation. They are of the same type, except in exceptional cases.

The direction of propagation of the transition point is predicted via the sign of its estimated velocity by a jump relation associated with the discontinuity of the flux gradient:

$$w_{pred} = \frac{Qr - Ql}{Ar - Al}$$

We suppose here $w_{pred} > 0$. The red color is associated with the pressurized areas and the blue color with the free surface.

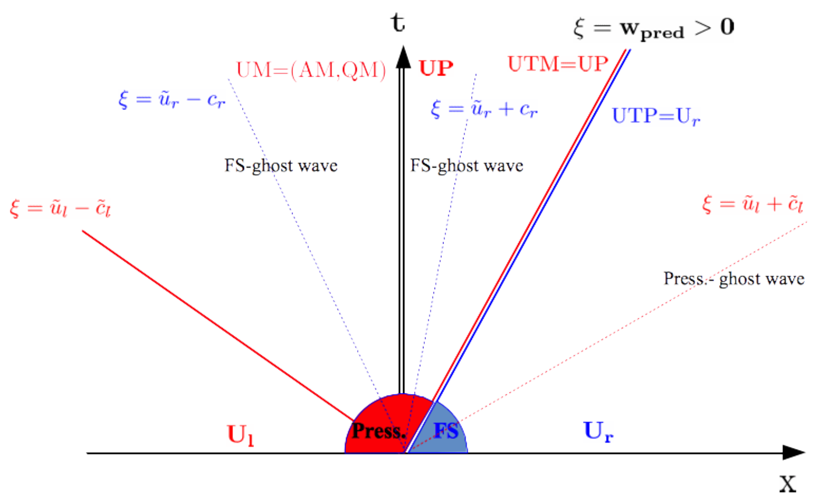


Figure 4: Rise in pressurized state with positive speed.

Three partial linearized Riemann problems (in which the states used to define the flux matrix may be part of the unknowns) are solved in the following areas:

zone1 : $-\infty < \xi = \frac{x}{t} < 0$ $(x < 0)$, **zone2 :** $0 < \xi < w$, **zone3 :** $w < \xi < +\infty$. In each zone, the altitude and the geometric and physical parameters are constant, and we do not take friction into account. As a result, only variables *A* and *Q* are to be taken into account. Zones 1 and 2 are connected by writing the conservation of the load. We search a solution satisfying the condition of Song ([6]), so that the speed of the characteristics $\xi = \tilde{u}_r + \tilde{c}_r$ (free surface) and $\xi = \tilde{u}_l + \tilde{c}_l$ (pressurized) are outside the domain of validity and are not taken into account: we speak of "ghost waves". The same applies a fortiori to the speed characteristic $\xi = \tilde{u}_r - \tilde{c}_r$ (free surface).

Thus we have *UTP* = *Ur*, *UTM* = *UP* : the only unknowns are therefore *UM* and *UP*. On the other hand, the jump relation associated with the conservation of the mass through the interface $\xi = 0$ gives $QP - QM = \xi(AP - AM) = 0$, thus $QP = QM$, denoted Q.

Finally, there are three unknowns, AM, AP and Q and a system of three non-linear equations to be solved: 1) jump through the left characteristic (pressurized), 2) charge conservation at the interface (pressurized), 3) jump through the transition line for the mass and the momentum at the same speed.

4.5 Boundary conditions

The upstream and downstream (independently) boundary conditions treated by FlowMix are the flow rate or the height of water or the load. It is also possible to take into account a downstream valve with its opening law. The imposed condition is assigned to the corresponding external fictitious mesh and is taken into account only if the flow is subcritical, which is assumed to be always the case if the flow is pressurized. Since the method requires the knowledge of a complete state (A and Q), an equation is associated with the missing data: it consists in writing that the jump (mass or momentum) through the outgoing characteristic is zero (i.e. the associated wave is phantom). Any change of upstream or downstream state is detected and taken into account by adapting the method described in section 4.4.

4.6 Mesh indicator updating process. Aeration chimney case

Consider an internal mesh of index i, whose state is known at the time n and denoted E_i^n. The updating of the variable E in this mesh, that is to say the value of E_i^{n+1} is based on a principle summarized in the figure 5 below. It allows the taking into account of meshes in load and in depression. However, the code provides for the presence of an aeration chimney by prohibiting depressions: in this case only $A_i^{n+1} \leq S \Longrightarrow E_i^{n+1} = 0$.

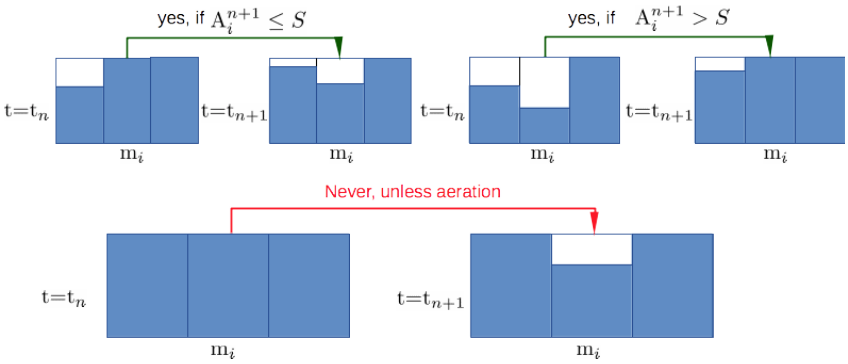


Figure 5: Update process of the state indicator

5. EXAMPLES OF INDUSTRIAL APPLICATIONS

5.1 Transfer conduit A

The aim of this study was to calculate the flow passing through a conduit relative to the level of water at the conduit inlet, which can vary from 0 (empty conduit) to 19 m during a rainy period. The results of FlowMix are then compared with an other method of calculation which is more simple and better adapted to the objectives of the project.

Dimensions (m x m)	Length (m)	Cross section (m²)	Slope (%)	Discharge (m^3/s)	Speed (m/s)
2 x 1.20	270	0.9 to 1.90	0.2 to 4	2 to 19	2 to 10

Figure 6 : Conduit characteristics

This conduit slopes more steeply in the middle with possible air intake.

For calculation stability purposes, for flows of over 7 m^3/s, an additional section has been added downstream of the conduit, representing the sudden widening at the conduit outlet. The geometry of this section has no impact on the upstream height as long as flow is torrential with a height of water lower than the height of the conduit in this section, which was checked after calculating.

Upstream limit condition: flow hydrograph (flow < 7 m^3/s) varying by 1m^3/s to the target flow, or imposed height of water (flow > 7 m^3/s), varying by 1m to the target height.

Downstream limit condition: water height of 1 m imposed for all flows. It should be noted that the flows were always torrential immediately upstream of the outlet (steep slope and free surface). This downstream limit condition does not influence the results.

For flows of less than 4.5 m3/s, FlowMix determines a free surface flow with a torrential passage in the middle of the conduit. For flows of 4.5 to 12 m^3/s, FlowMix highlights a pressurised flow to the middle then a free surface flow in torrential regime. Finally, for flows of over 12 m^3/s, FlowMix calculates a pressurised flow all through the conduit.

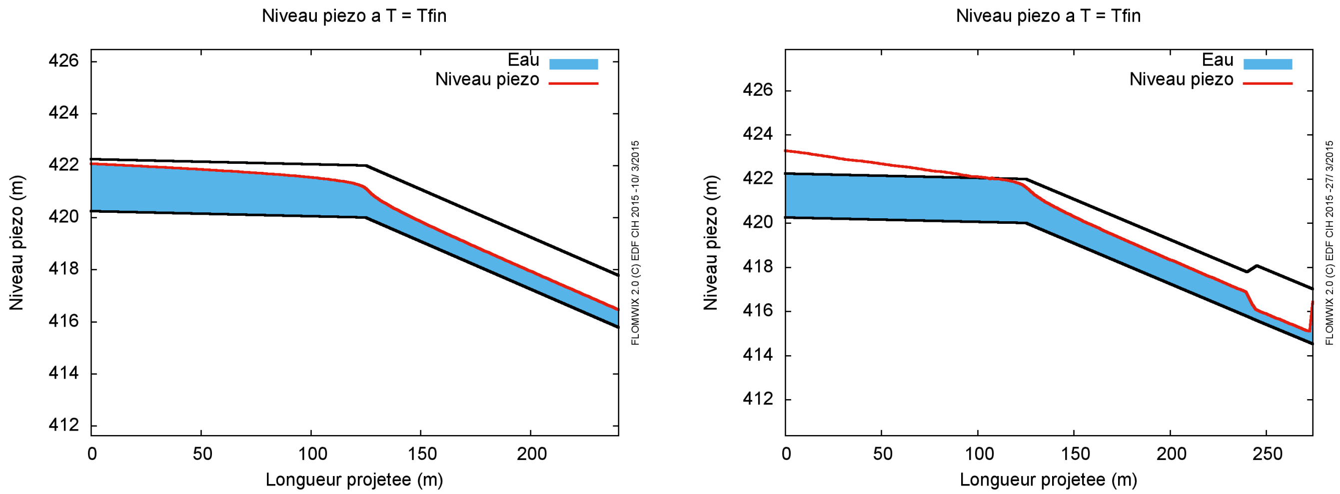
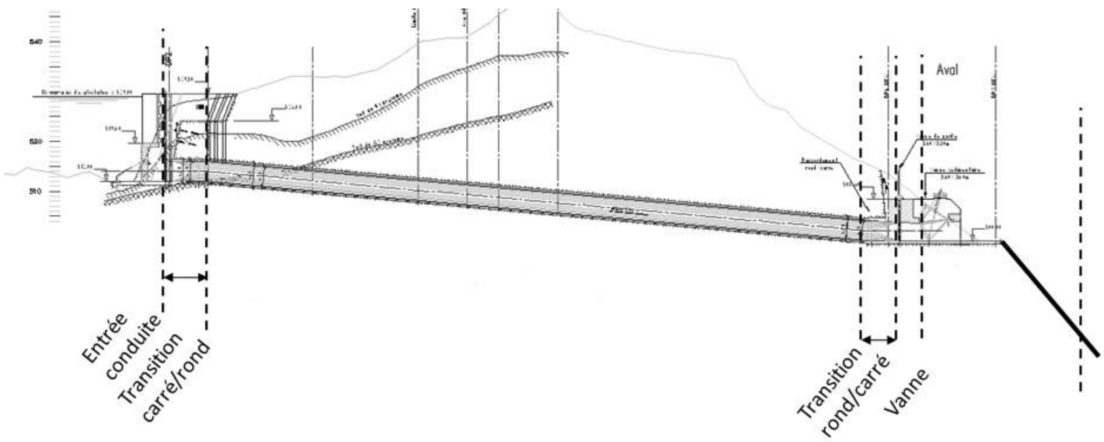


Figure 7 : Conduite A : piezometric level versus horizontal length with a discharge of 4 m^3/s (left) and a discharge of 7 m^3/s (right)

5.2 Rizzanese sediment transit tunnel

This tunnel is part of Rizzanese scheme in Corsica, commissioned by EDF in 2013. It transports sediment from the reservoir to downstream of the dam.

A 1/30th scale model of the tunnel was given to Liège University's Engineering Hydraulics Laboratory (ULg), with the aim of checking its hydraulic function, in particular for low elevations, where vortex, air entrainment and mixed flow phenomena appear [7].



Diameter (m)	Length (m)	Slope (%)	Discharge (m^3/s)
4.20	150	7	20 to 200

Figure 8 : Scheme and characteristics of the tunnel

The maximum flow speeds observed on the model are around 20 m/s for pressurised flows and around 15 m/s for free surface flows

Square to round transitions upstream and downstream of the tunnel ensure the connection with the square and rectangular sections at the upstream bulkhead gate and the downstream radial gates. Vent pipes are planned upstream of the tunnel.

The model is supplied upstream by a flow injected into a stilling supply basin. The level in the reservoir establishes itself naturally relative to the tunnel's discharge capacity.

The model has been equipped with two electromagnetic flowmeters placed on the upstream supply conduit and 14 graduated scales positioned on the tunnel perimeter to measure the water levels for the free surface flows.

A comparison between these measurements with FlowMix calculations was made using the characteristics of the scale model. This gave very satisfactory results.

For low reservoir levels, the flows are free surface all along the conduit.

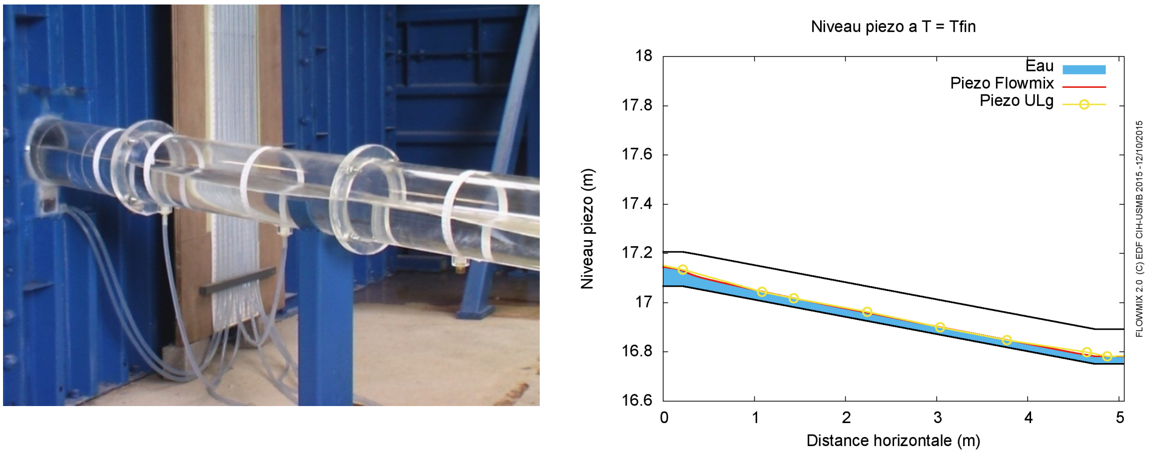


Figure 9 : Free surface flow on the model (65 m^3/s) (left) and piezometric level versus horizontal length with comparison between FlowMix results and measurements on the model (right)

When the reservoir level increases, the flows become pressurised upstream of the tunnel and a mixed flow is created with the air from the vent pipes.

When the gate is partially closed, there is pressurised flow downstream of the tunnel and a hydraulic jump appears in the conduit.

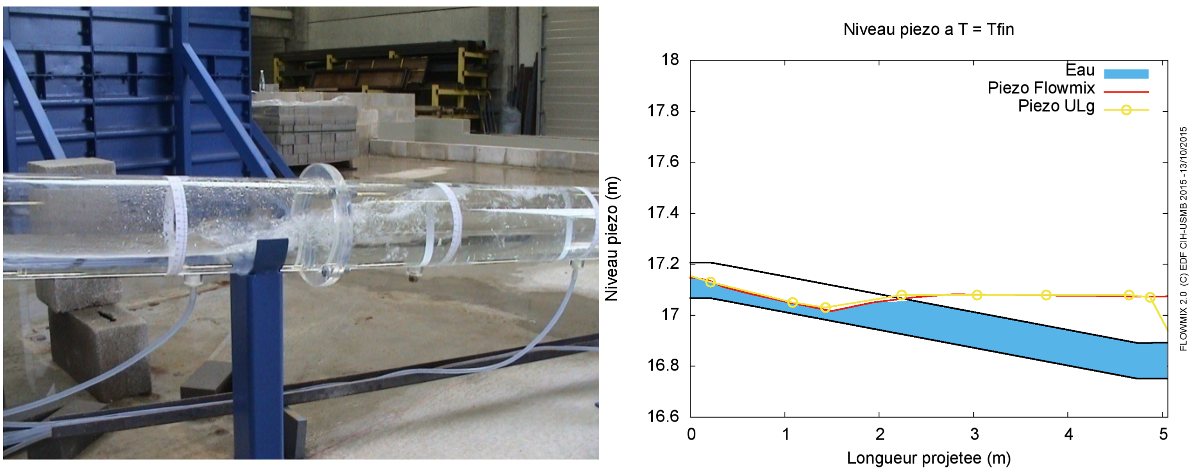


Figure 10 : Free surface flow and pressurized flow on the model (25 m^3/s, gate closed 20%) (left) and piezometric level versus horizontal length with comparison between FlowMix results and measurements on the model (right)

5.3 La Coche drainage conduit

The function of the La Coche drainage conduit is to drain the water coming from the scheme's tunnels when the plant is shut down and the upper basin is completely emptied. With a capacity of 800 l/s, the drainage conduit can discharge the water coming from leaks in the tunnels and rainfall on the basin and the catchment area:

Diameter (mm)	Length (m)	Difference in level (m)	Maximum slopes (%)	Discharge (l/s)	Speed (m/s)
400	1400	530	70 to 120	800	10 to 19

Figure 11 : Characteristics of the conduit

Figure 12 : New buried conduit

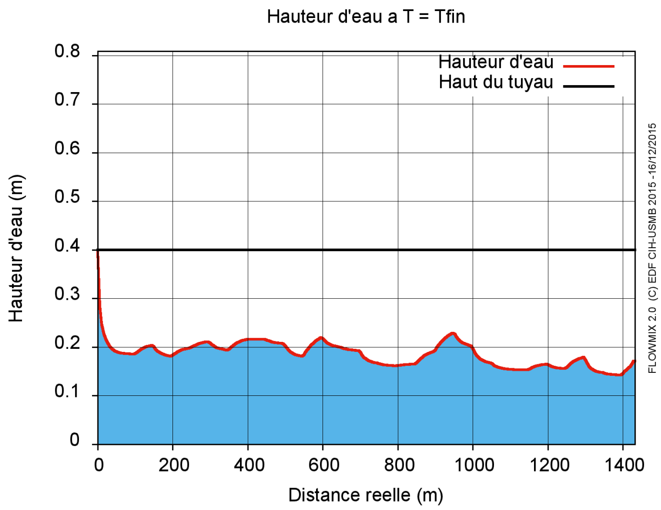


Figure 13 : Water heigth versus real distance - FlowMix results

A new conduit was installed in 2014 - 2015 and the FlowMix software was used to design it, to ensure that the flow stayed on the free surface all through the conduit, in accordance with the hypothesis chosen for its function. With FlowMix the desired calculations could be made because the shallow water equations (St Venant equations) programmed in the code do not use the approximation of a slight slope.

The criterion chosen was to have a "simulated" water height of less than 60% of the conduit's diameter, which was checked through the calculations.

The tests carried out in August 2015 were done gradually increasing the flow from 200 to 750 l/s. For this flow, all the air valves were sucking air in except air valve V4, situated on a nearly flat section of the conduit (x=960 m), where intermittent water discharge was observed but without any pressure increase. These observations showed that the conduit's function was quite satisfactory.

6. CONCLUSION

It remains to EDF to qualify the code, i.e. to use it in design studies and to make a comparison with measurements taken on real conduits, to be able to determine its effective field of application and the uncertainties to associate with the calculation results.

Using FlowMix to study hydraulic transients for mixed flows is possible but still needs to be tested.

The simulation of special physical phenomena is currently being developed: air entrainment on a steeply sloping conduit, energy consumed by a hydraulic jump.

REFERENCES

[1] R. Botchorishvili, B. Perthame, and A. Vasseur (2003). *Equilibrium schemes for scalar conservation laws with stiff sources*. Math. Comput., 72 (241) : 131–157.

[2] C. Bourdarias, M. Ersoy, and S. Gerbi (2014). *Unsteady mixed flows in non uniform closed water pipes : a full kinetic approach*. Numerische Mathematik, 128 (2) : 217–263.

[3] T. Gallouet, J. M. Hérard, and N. Seguin (2003). *Some approximate Godunov schemes to compute shallow-water equations with topography*. Comput. Fluids, 32 : 479–513.

[4] J.M. Greenberg and A.Y. LeRoux (1996). *A well balanced scheme for the numerical processing of source terms in hyperbolic equation.* SIAM J. Numer. Anal., pages 1–16.

[5] B. Perthame and C. Simeoni (2001). *A kinetic scheme for the Saint-Venant system with a source term*. Calcolo, 38(4) : 201–231.

[6] C.S.S. Song, J.A. Cardle, and K.S. Leung (1983). *Transient mixed-flow models for storm sewers*. Journal of Hydraulic Engineering, ASCE, pages 1487–1503.

[7] S. Erpicum (2008). *Rizzanèse - Projet de vanne de transit sédimentaire - Réalisation d'un modèle réduit* **.** Etude réalisée pour Electricité de France – Centre d'Ingénierie Hydraulique (EDF-CIH)