A 2D vertical model for incompressible flows, using a level set free surface tracking

S. Detrembleur, S. Erpicum, P. Archambeau, B. J. Dewals & Prof. M. Pirotton
University of Liège - ArGEnCo Department, Belgium
corresponding author : sylvain.detrembleur@ulg.ac.be

The project in a few words

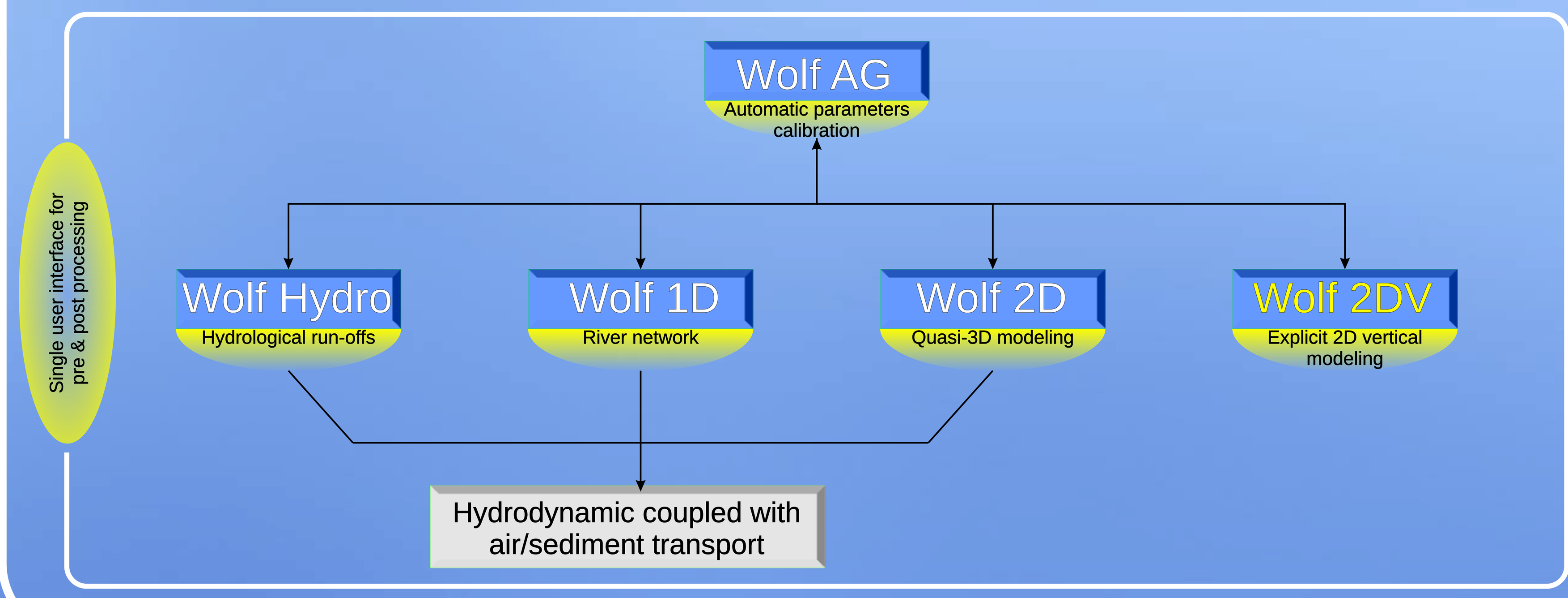
In order to solve the Navier-Stokes equations in the case of free surface incompressible flows, an original model has been developed on the basis of the finite volume technique applied to a 2D Cartesian grid in the vertical plane. The projection method has been adopted to solve the water phase.

The air phase is solved by extrapolation of the velocity field computed in the water phase and the restoration of the divergence free. A level set approach ensures the interface tracking. Numerical schemes able to solve the level set equation accurately and to respect the mass conservation has been developed carefully.

The numerical implementation of the projection method is carried out based on first or second order space accuracy. The projection step is computed by solving the Poisson's equation thanks to the iterative GMRes solver. Runge-Kutta schemes ensures time integration. The model accounts for viscous diffusive terms as well as the Smagorinsky turbulence model.
The implemented model was first validated for pressurized flows based on benchmarks from literature. Then the solver has been validated for both steady and unsteady free surface flows.

Numerical environment

for practical implementation of Wolf 2DV



WOLF is a modeling system fully developed within the HACH unit and dedicated to free surface and pressurized flows. The numerical scheme is based on the finite volume method. An original flux vector splitting assures a strong robustness and first or second order accuracy in space. Time integration is ensured up to the fourth order.

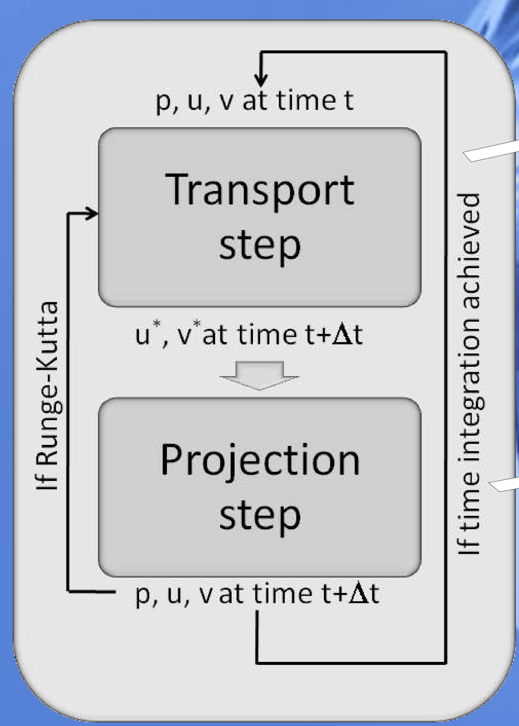
Water phase numerical implementation

Set of equations

$$\begin{cases} \dfrac{\partial u}{\partial t} + \dfrac{\partial u^2}{\partial x} + \dfrac{\partial uv}{\partial z} + \dfrac{1}{\rho}\dfrac{\partial p}{\partial x} = \nu\left(\dfrac{\partial^2 u}{\partial x^2} + \dfrac{\partial^2 u}{\partial z^2}\right) + a_x \\ \dfrac{\partial v}{\partial t} + \dfrac{\partial uv}{\partial x} + \dfrac{\partial v^2}{\partial z} + \dfrac{1}{\rho}\dfrac{\partial p}{\partial z} = \nu\left(\dfrac{\partial^2 v}{\partial x^2} + \dfrac{\partial^2 v}{\partial z^2}\right) + a_z \\ \dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial z} = 0 \end{cases}$$

Incompressible Navier-Stokes equations are composed of the 2 momentum conservation and the mass conservation. a_x and a_z are the acceleration along the 2 axis, u and v the 2 velocity components, p the pressure, ρ the density and ν the dynamic viscosity. Usually only a_z is used and is equal to the gravity acceleration

Resolution method



u* and v* are approximated velocities.

$$\frac{u^* - u}{\Delta t} + \frac{\partial u^2}{\partial x} + \frac{\partial uv}{\partial z} + \cancel{\frac{1}{\rho_0}\frac{\partial p}{\partial x}} = \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial z^2}\right) + a_x$$

$$\frac{v^* - v}{\Delta t} + \frac{\partial uv}{\partial x} + \frac{\partial v^2}{\partial z} + \cancel{\frac{1}{\rho_0}\frac{\partial p}{\partial z}} = \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + a_z$$

The transport step consists in solving the equations by neglecting the pressure term.

First or second order space accuracy is used with an upwind scheme ensuring the numerical stability in absence of diffusive terms.

$$\frac{\partial^2 p}{\partial x^2} + \frac{\partial^2 p}{\partial z^2} = \frac{\rho_0}{\Delta t}\left(\frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial z}\right)$$

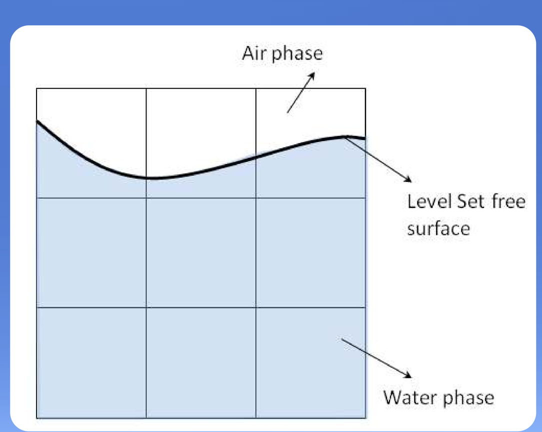
Pressure is calculated by solving a Poisson 's equation with a source term depending on the approximated velocities.

The GMRes is exploited to solve iteratively the linear system.

$$\begin{cases} u^{t+\Delta t} = u^* - \dfrac{\partial p}{\partial x}\dfrac{\Delta t}{\rho_0} \\ v^{t+\Delta t} = v^* - \dfrac{\partial p}{\partial z}\dfrac{\Delta t}{\rho_0} \end{cases}$$

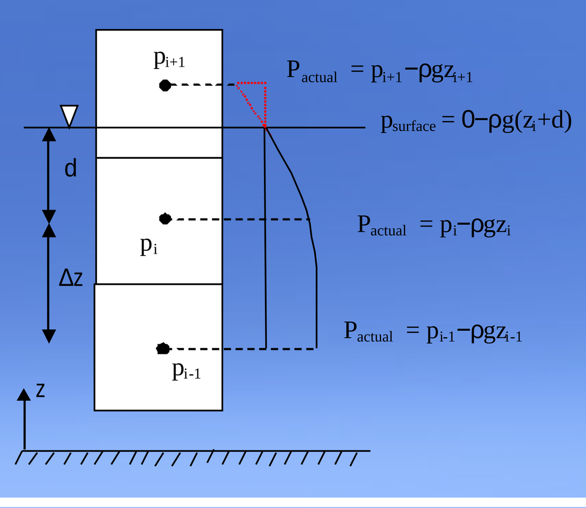
Final velocities are obtained by correcting approximated ones.

Air phase numerical implementation



$$\frac{\partial \varphi}{\partial t} + \frac{\partial u\varphi}{\partial x} + \frac{\partial v\varphi}{\partial z} = 0$$

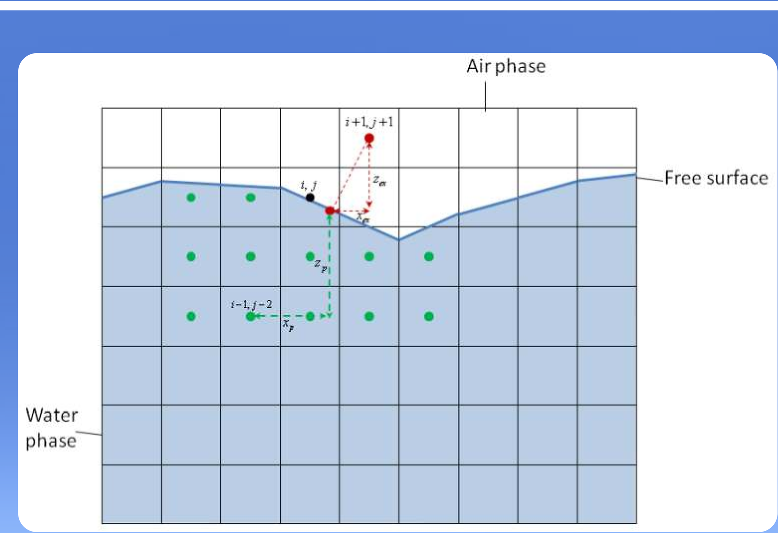
The Level Set approach is used to track the free surface with an original conservative formulation ensured through the incompressibility constraint. φ is the Level Set value



Boundary conditions at free surface are imposed following a continuous second order approximation according to Poisson's equation.

The marching square algorithm enables to compute the position of the free surface based on Level Set node values.

The distance d is then geometrically computed.



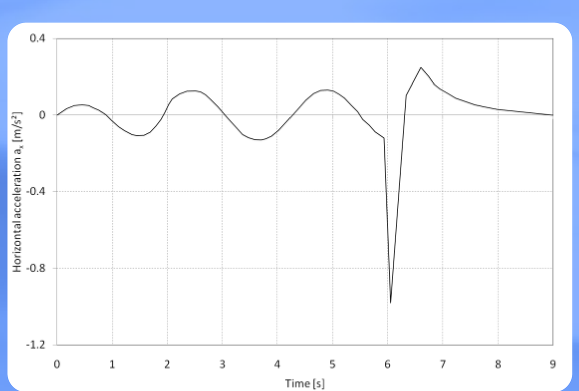
Velocity field in the air phase is extrapolated from directionnal velocity values computed previously in the water phase.

The extrapolation is based on a distance-weighted from free surface and solved at the least square.

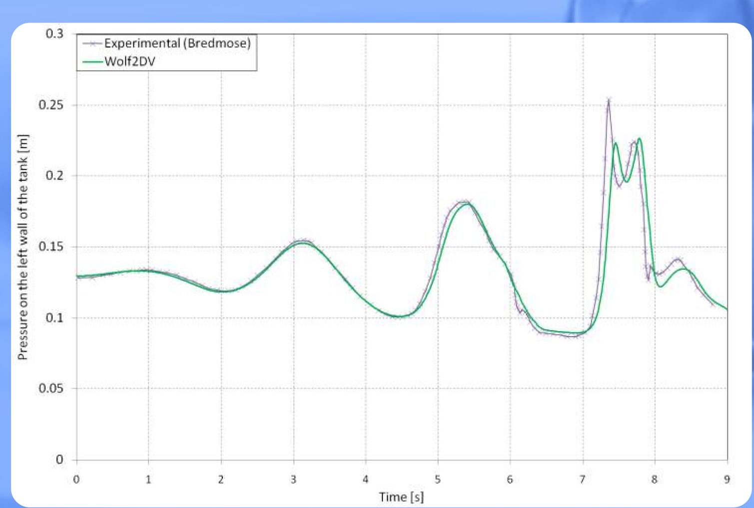
The extrapolated values are then put in a projection step to ensure free divergence.

Flow in a sloshing tank

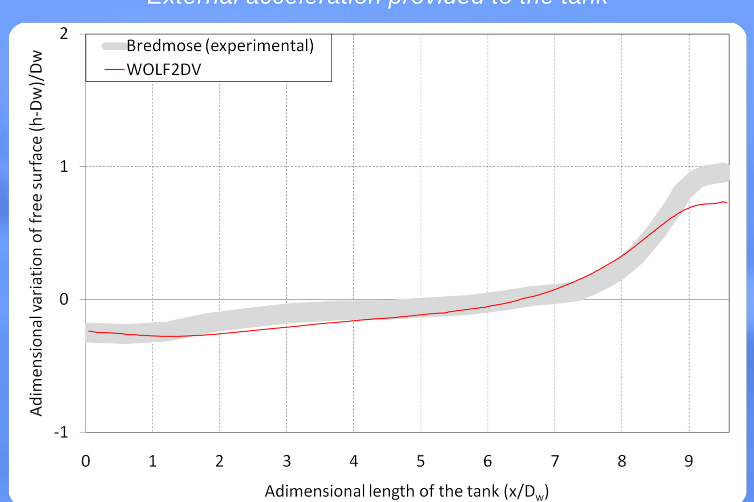
Liquid dampers are usually used on high constructions subjected to wind.
Their role is to damp the effects due to wind cycles . To achieve this goal the damper must be designed such as the flow induced by the movement of the construction provides forces opposing the wind action.
We propose here to validate the developed code on the experimental results of Bredmose. The benchmark consists in shaking a tank with an horizontal force for which the acceleration is defined



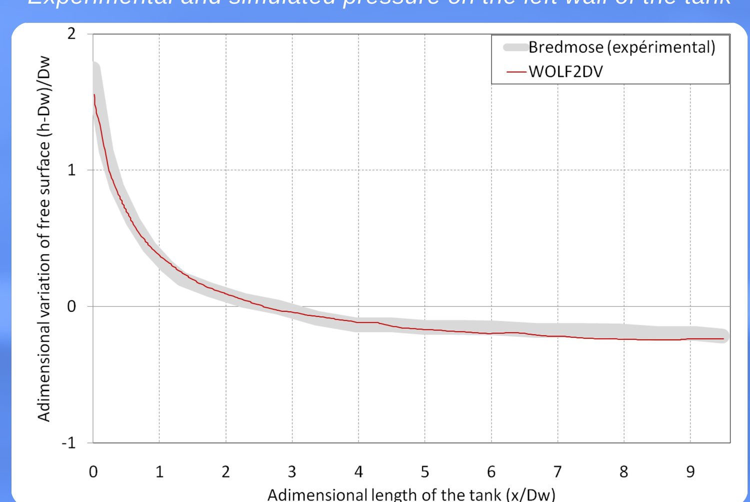
External acceleration provided to the tank



Experimental and simulated pressure on the left wall of the tank



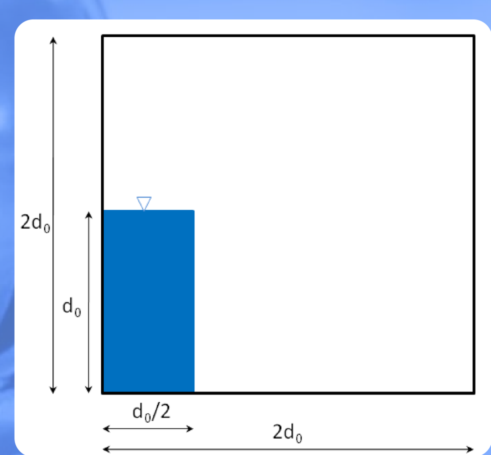
Free surface at t = 6,52 s. D_w is the initial water depth.



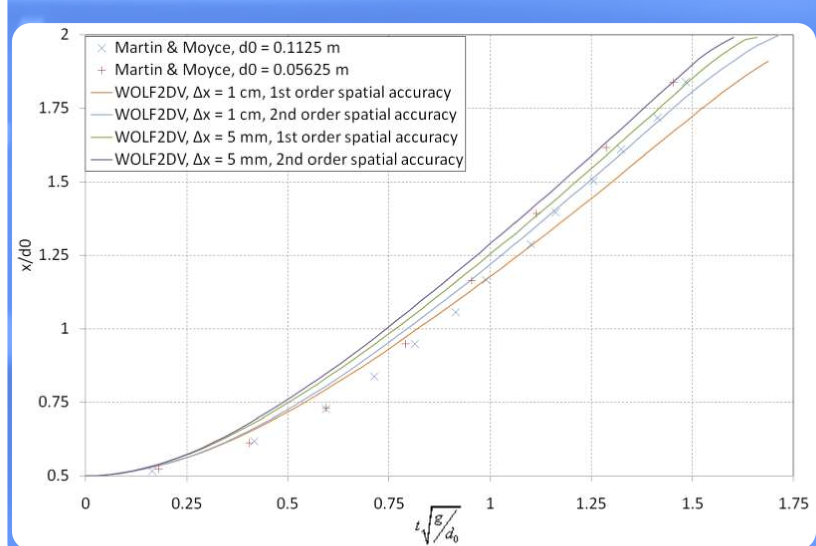
Free surface at t = 7,64 s. D_w is the initial water depth.

Dam break flow

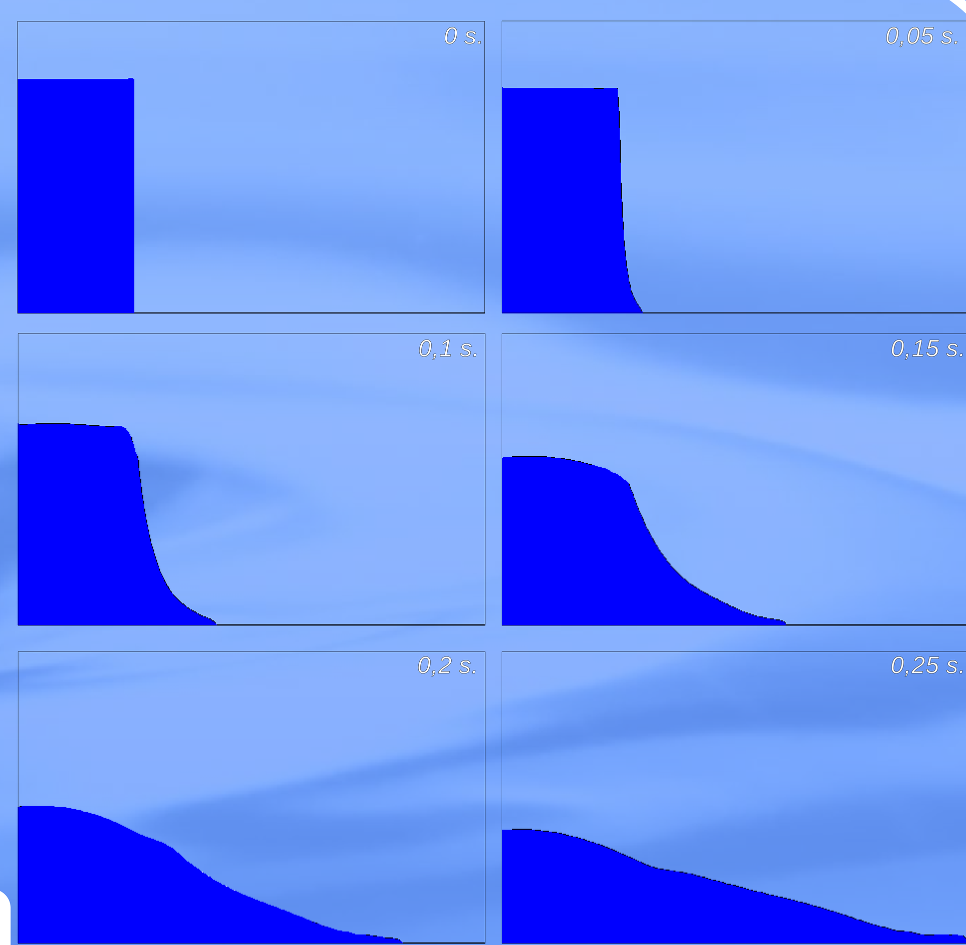
Modeling the flow that occurs after a dam break is a challenge due to the high gradient that appears in the flow.
The developed solver has been tested on the experimental solution of Martin & Moyce considering a volume of water released in a tank with d_0 = 30 centimeters.



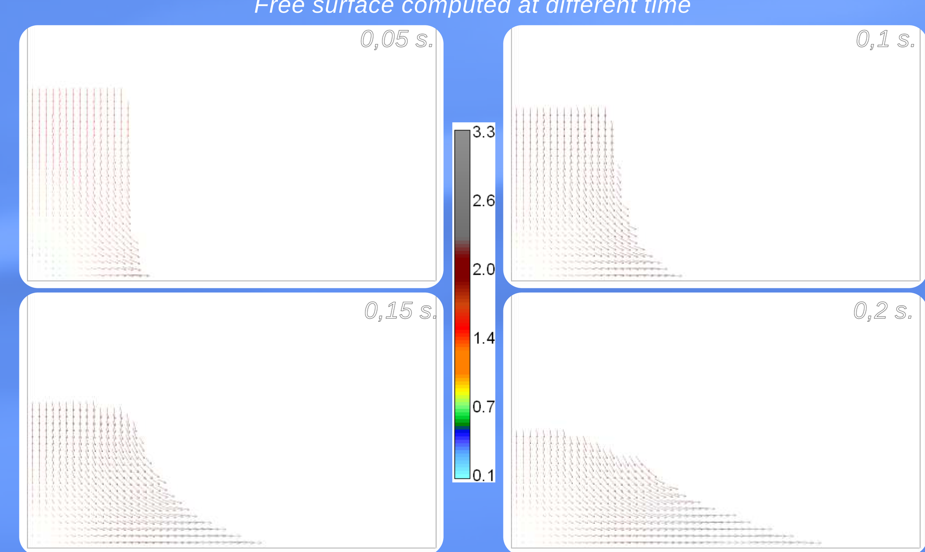
Initial condition of the experience of Martin & Moyce



Non-dimensional leading edge position



Free surface computed at different time



Velocity field [m/s] computed at different time

Conclusions

The 2D incompressible Navier-Stokes equations are solved by means of the projection method. The free surface tracking is ensured by the Level Set method. The algorithm enables to deal with up to 400 000 computed cells and requires moderate calculation time.Two benchmarks are presented among the large number of applications modeled by this new software, such as flows on crested weirs, through botton outlets...The robustness and the accuracy of the model are presented here in these applications.The first one is the flow induced in a tank by an external force applied. It has a direct practical application in the tuning of liquid dampers. The second one is the flow that occurs directly after a dam break. The developed code shows in both cases a very good aggreement with experimental results.

University of Liège (ULg - www.ulg.ac.be)
Department of Architecture, Geology, Environment and Constructions (ArGEnCo - www.argenco.ulg.ac.be)
Sector of Solid, Fluid and Structure Mechanics (MS²F)
Hydrology, Applied Hydrodynamics and Hydraulic Constructions (HACH - www.hach.ulg.ac.be)