Utah State University

DigitalCommons@USU

International Junior Researcher and Engineer Workshop on Hydraulic Structures

6th International Junior Researcher and Engineer Workshop on Hydraulic Structures (IJREWHS 2016)

May 30th, 11:25 AM - 11:40 AM

Numerical investigation of the pressure on a round crested weir

L. Goffin
University of Liege

S. Erpicum
University of Liege

P. Archambeau
University of Liege

B. J. Dewals
University of Liege

M. Pirotton
University of Liege

DOI:10.15142/T30P4R

Numerical investigation of the pressure on a round crested weir

L. Goffin[1], S. Erpicum[1] P. Archambeau[1], B. J. Dewals[1] and M. Pirotton[1]
[1]HECE, ArGEnCo
University of Liege (ULg)
Liège, Belgium
E-mail: l.goffin@ulg.ac.be

ABSTRACT

Understanding the pressure distribution on round crested weir can help to improve their design. In this frame, we propose a finite difference method for irrotational flows. Our method, based on a previous work, determines the free surface iteratively. For best results, the computation of the velocity at the free surface is done by a bi-dimensional function fitting. Knowing the velocity, the elevation and a reference energy level, the pressure can be derived. The evolution path is determined by numerical derivatives of the pressure at the surface. The iterative method is tested on a subcritical flow. The pressure computation is compared to experimental measurements of the pressure on a spillway crest (trans-critical flow). Both results are very encouraging: the free surface moves smoothly to an equilibrium state and the pressure on the structure is very close to experiment. For this last point, the method is also able to faithfully reproduce pressure drops.

Keywords: *Finite difference, Spillway, free surface, potential flow*

1. INTRODUCTION

Spillways are important structures that should provide sufficient safety during flood events. Understanding their behavior for discharges higher than design ones is a key point for innovative design and safer structures. To do so, experimental and numerical investigations should be led in parallel for collecting specific information about these structures. This paper focuses on the implementation of a new numerical method that aims to determine quickly the free surface position, velocity and pressure fields for trans-critical flows.

In hydraulics, the Finite Volumes Method (FVM) is traditionally used to simulate free-surface flows. However it doesn't allow to compute directly the pressure on the structure due to the assumptions directly linked to this method. Numerical methods such as Finite Elements (FE) (Chatila & Tabbara 2004), Smoothed Particle Hydrodynamics (SPH) (Goffin 2013; Goffin et al. 2016; Lodomez 2014; Lodomez et al. 2014) and Particle Finite Elements (PFEM) (Larese et al. 2008) implemented in 3-D or 2-D vertical slices can handle flows over round crested weirs with information about the pressure on the structure. However they require a large amount of computing resources in order to handle accurately the flow pattern at the interface with the structure. For ordinary desktop computers, a computation for a single discharge may reach several hours. Thus, optimizing a weir profile would require too many time with such methods.

A need for simpler and quicker methods is obvious. Castro-Orgaz (2013) solves the problem using a potential flow assumption. Equations are solved in the non-physical (x,ψ), where x is the horizontal direction and ψ the stream function. A more natural way would be to express the problem in the physical spatial plane (x, z). The method presented in this paper enhances the one introduced in Goffin et al. (2014) where the potential flow is solved in a physical plane (x,z). First, improvements from Goffin et al. (2014) method will be presented. Then, test cases and practical examples are explained for a bump and a flow over a round crested weir.

2. NUMERICAL METHOD

The method developed here is based on the assumption that the flow is irrotational. This was proved by Escande (1937) for flows over spillways. The Laplacian $\Delta = \nabla^2$ of the potential ϕ or the stream function ψ can be solved in such cases. The resulting field, once derived, yields to a velocity field.

Thanks to the energy conservation principle, the pressure p at a given point can be deduced for a given energy level E :

$$E = z + \frac{p}{\rho g} + \frac{\|\vec{u}\|^2}{2g} \tag{1}$$

where z is the altitude, ρ the density, g the acceleration of gravity and $\vec{u}$ the velocity vector.

The domain is discretized by using the finite difference approach. Irregular nodes are created at boundaries in order to represent accurately their geometry. Irregular nodes result from the intersection between the boundary contour and the grid lines. Regular nodes are created at the intersection of grid lines for the inner domain. An example of discretization is depicted in Figure 1.

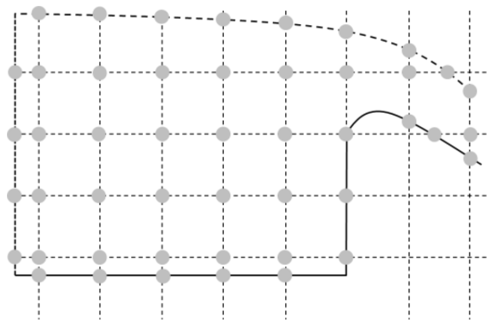

Figure 1: Schematic discretization of the computation domain

2.1. First approach and drawbacks

In Goffin et al. (2014), the Laplacian solved was based on the potential :

$$\Delta^2 \phi = 0 \tag{2}$$

This approach required to impose Neumann boundary conditions (BC), i.e. the value of a derivative is imposed at the boundary, for free surface and impervious boundaries. This was achieved by using the Green-Gauss theorem to compute derivatives at these boundaries. It led to a good approximation of the velocity at the boundary but some oscillations could be observed.

Determining the free surface requires iterations on the position of free surface nodes. This was done thanks to an approach based on curvilinear coordinates developments (Stilmant et al. 2013). The evolution of the pressure with the water depth dp/dH was linked to properties related to curvilinear cross sections and velocity profiles. The evolution of depth for each surface node ΔH was given by

$$\Delta z = \gamma \frac{p_{i+1} - p_i}{\frac{dp}{dH}} \tag{3}$$

with γ a relaxation coefficient. The derivative at the denominator is close to zero in the region of the critical section. This led to some instabilities in that area.

Main drawbacks of this method included:

- Oscillations due the velocity evaluation at boundaries
- Instabilities near the critical section
- Pressure evaluation at the free surface could present oscillations due to the velocity evaluation

2.2. New approach

The new approach described in this section aims to reduce previous method problems. The first problem that was tackled is the evaluation of velocity at boundaries. To do so, The Laplacian is now solved for the stream function ψ :

$$\nabla^2\psi = \frac{\partial^2\psi}{\partial x^2} + \frac{\partial^2\psi}{\partial z^2} \tag{4}$$

Only Dirichlet BC are now required which avoids to use a Green-Gauss derivative evaluation (Goffin et al. 2014) which led to approximations at the boundary. Velocities computed on the outer limits should be more accurate now using an imposed value for the unknown.

The second derivative is discretized according to a finite difference scheme on an irregular grid (see Figure 2):

$$\begin{aligned} \frac{\partial^2\psi}{\partial x^2} &\approx \frac{2}{a_1(a_1+a_2)}\psi_{i-1} - \frac{2}{a_1 a_2}\psi_i + \frac{2}{a_2(a_1+a_2)}\psi_{i+1} \\ \frac{\partial^2\psi}{\partial z^2} &\approx \frac{2}{b_1(b_1+b_2)}\psi_{j-1} - \frac{2}{b_1 b_2}\psi_j + \frac{2}{b_2(b_1+b_2)}\psi_{j+1} \end{aligned} \tag{5}$$

Where a_k are relative to horizontal distances and b_k are relative to vertical distances between nodes.

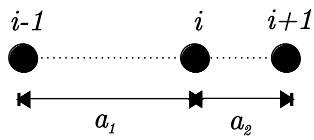

Figure 2 : Discretization scheme for a horizontal second derivative

When distances between nodes are equal, the discretization scheme has second order precision, while it is first order when nodes are not regularly spaced (Hirsch 2007).

After the computation of a given state, the velocities are evaluated for the inner nodes as a derivative of the stream function:

$$\vec{u} = \begin{pmatrix} u \\ v \end{pmatrix} = \begin{pmatrix} \dfrac{\partial\psi}{\partial z} \\ -\dfrac{\partial\psi}{\partial x} \end{pmatrix} \tag{6}$$

These derivatives are evaluated thanks to centered finite differences. For the nodes located on the boundaries, velocities are computed thanks to an interpolation from inner nodes. For a given boundary node, inner neighbors within a radius *r* are considered. The value

$$f_{i,j} = z_{i,j} + \frac{\left\|\vec{u}_{i,j}\right\|^2}{2g} \tag{7}$$

is computed for each of these nodes. Then, a bi-dimensional function is fitted by least squares on this set of points:

$$f(x,z) = c_1 x^2 + c_2 x + c_3 z^2 + c_4 z + c_5 x^2 z^2 + c_6 x^2 z + c_7 x z^2 + c_8 x z + c_9 \tag{8}$$

When coefficients c_i are determined, the value *f* of equation (7) can be computed using (8) for the boundary node. This method avoids velocity discontinuities at boundaries. Then, the pressure can be determined by means of (1) and a reference energy level *E*.

The approach adopted here for the free surface evolution is different from the previous paper. It is based on a Taylor development, second order accurate:

$$p_{i+1} = p_i + \frac{dp}{dH}\Delta z + \frac{1}{2}\frac{d^2 p}{dH^2}\Delta z^2 + O\left(\Delta z^3\right) \tag{9}$$

First, a classical Newton-Raphson approach (first order accurate) (Ypma 1995) is implemented by using only the first derivative:

$$\Delta z = \frac{p_{i+1} - p_i}{\frac{dp}{dH}} \tag{10}$$

Obviously, it can lead to indeterminate solution if $dp/dH = 0$. This is the case at the critical section. Nodes close to it are subject to large displacements since the pressure derivative is close to 0.

In order to avoid too large displacements of the free surface, a relaxing factor was used to limit displacements to a tenth of the discretization step. This relaxing factor was applied for the whole free surface.

Compared to a classical Newton-Raphson iteration scheme, solving the quadratic function (9) avoids to have infinite Δz since a second derivative is present. Equation (9) can be solved to find Δz that should be applied to free surface nodes:

$$\Delta z_{1,2} = \frac{-\frac{dp}{dH} \pm \sqrt{\left(\frac{dp}{dH}\right)^2 - 2\frac{d^2 p}{dH^2}\left(p_i - p_{i+1}\right)}}{\frac{d^2 p}{dH^2}} \tag{11}$$

The first root (+) was found to be the solution for supercritical flows and the second root (-) was found to be the one for subcritical flows.

In equations (10) and (11), first and second derivatives are evaluated numerically. Traditionally, this would require moving each node independently, leading to a large amount of intermediate computations. The method chosen in this approach is hybrid and consists in moving vertically the whole free surface by an increment ε. Derivatives are computed by moving the free surface upward and downward:

$$\begin{aligned} \frac{dp}{dH} &\approx \frac{p\left(H+\varepsilon\right) - p(H-\varepsilon)}{2\varepsilon} \\ \frac{d^2 p}{dH^2} &\approx \frac{p\left(H+\varepsilon\right) - 2p\left(H\right) + p(H-\varepsilon)}{\varepsilon^2} \end{aligned} \tag{12}$$

For subcritical and supercritical flows, the energy level and the pressure to reach are given as parameters. When the flow is transcritical, the energy level is set by the critical section (Castro-Orgaz & Montes 2015). This critical section tends to minimize the specific energy of the flow. The critical section can be determined by spotting the section were $dp/dH = 0$ and where the specific energy is minimized. Since the total energy of the flow is set by the user, the pressure at the critical section represents the target pressure that all nodes has to reach, which is p_{i+1} in (10) and (11).

3. TEST CASE

The test case in this section show first results of the currently implemented method for a subcritical flow on a bump. The bump has equation $0.3\left(1-x^2\right)$ in the range $\left[-1;1\right]$ m. In the range $\left[-5;-1\right[\cup\left]1;5\right]$ m, the bottom is flat and at the altitude 0 m. The domain is discretized with a fine 2×2 mm² grid in order to concentrate on the results of the iterative method and not on the effects linked to a rough discretization.

The initial condition is a horizontal free surface set at 0.47 m. The target pressure is 0 m and the energy level is imposed at 0.4723 m. An analytical solution was computed. It considers a uniform vertical velocity profile and the energy is conserved on all free surface points. Given the energy conservation equation (1), a water level can be derived. The uniform vertical velocity profile assumption is not in total agreement with the model implemented here. Some differences may be noticed. The velocity and pressure on the free surface were computed according to the fit of function (8) and neighbors were taken in a radius $r = 8\Delta x$. The problem was solved with equation (10) leading to results given in Figure 3 where the initial condition as well as the final solution obtained by the model are depicted.

It can be seen that the free surface resulting from the computation fits closely the analytical solution in shape and in values. However, this analytical solution is computed for a uniform velocity profile, which is not representative above the bump. This inconsistence can be noticed in the rightmost plot in Figure 3.

These encouraging results show that this method is able to move a free surface. It should also be tested on different flow regimes.

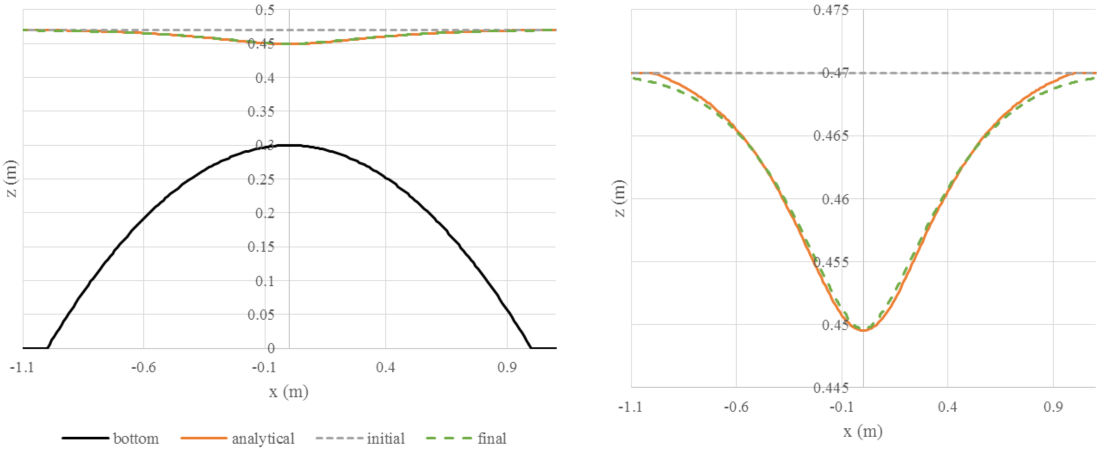

Figure 3: Free surface position for a subcritical flow over a bump

4. PRESSURE PROFILE ON A ROUND CRESTED WEIR

Once a free surface profile is determined, velocity fields and pressure fields can be computed. This section is focusing on the determination of the pressure on the structure with a free surface profile measured experimentally.

An experimental campaign was led about pressure diagrams on WES (USACE 1987) round crested weir for high head ratios (H / H_d), with a design head H_d = 15 cm. Numerical simulations were led for a domain spreading from the reservoir to the spillway itself. Results shown in Figure 4 were computed for $H / H_d = 2$ and are focusing only on the weir crest. Very good agreement between experimental measures and numerical results can be observed. This figure shows that the method implemented is able to faithfully reproduce the pressure on a weir crest. Even pressure drops can be reproduced with a high level of fidelity.

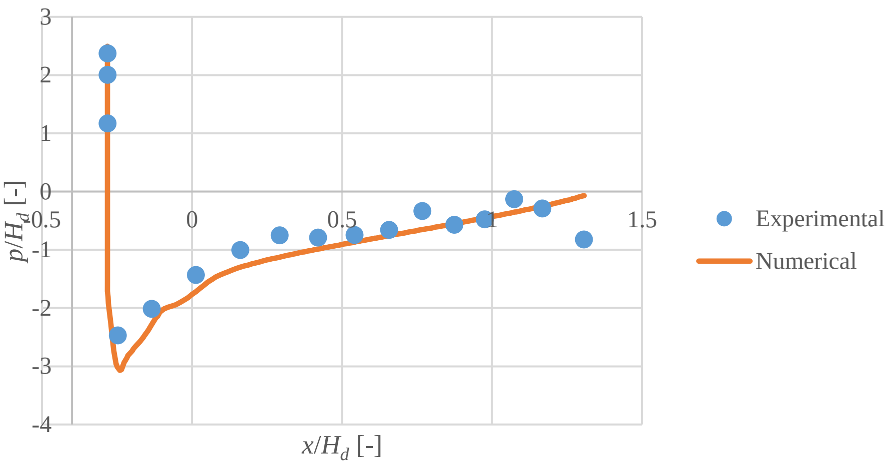

Figure 4: Comparison between experimental measurements and numerical computation of the pressure on a WES weir crest

5. CONCLUSIONS

In this paper, we presented an original method which goal is to determine the free surface profile of irrotational flows. This work was based on first developments presented in Goffin et al. (2014). Improvements were presented in order to reduce drawbacks noticed in the previous approach.

The Newton-Raphson scheme was successfully tested on a subcritical flow over a bump. A more advanced method was also proposed in order to deal with transcritical flows.

With a given free surface, the method showed that it was able to faithfully reproduce the pressure diagram on a round crested weir.

Further developments should focus on transcritical flows and the critical section issue. In this frame the second order method should bring benefits.

6. REFRENCES

Castro-Orgaz, O., 2013. Potential flow solution for open-channel flows and weir-crest overflow. *Journal of Irrigation and Drainage Engineering*, 139(7), pp.551–559.

Castro-Orgaz, O. & Montes, J.S., 2015. Minimum specific energy in open-channel flows: the Salas–Dominguez contribution. *Journal of Hydraulic Research*, 53(2), pp.151–160.

Chatila, J. & Tabbara, M., 2004. Computational modeling of flow over an ogee spillway. *Computers & structures*, 82(22), pp.1805–1812.

Escande, L., 1937. *Barrages* H. & Cie, ed., Hermann & Cie.

Goffin, L., 2013. *Development of a didactic SPH model*. University of Liege.

Goffin, L. et al., 2014. How to simulate quickly and efficiently a flow over a spillway? In *5th IAHR International Junior Researcher and Engineer Workshop on Hydraulic Structures*.

Goffin, L. et al., 2016. Validation and Test Cases for a Free Surface SPH Model. In *Advances in Hydroinformatics*. Springer, pp. 175–187.

Hirsch, C., 2007. *Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics* E. Ltd, ed., Butterworth-Heinemann.

Larese, A. et al., 2008. Validation of the particle finite element method (PFEM) for simulation of free surface flows. *Engineering Computations (Swansea, Wales)*, 25(4), pp.385–425.

Lodomez, M. et al., 2014. Comparison between Experimental and SPH Models over a Sharp-crested Weir. In *5th IAHR International Junior Researcher and Engineer Workshop on Hydraulic Structures*.

Lodomez, M., 2014. *Determining the characteristics of a free jet in 2-D by the SPH method*. University of Liège.

Stilmant, F. et al., 2013. Depth-averaged flow modeling in curvilinear coordinates. In *2013 IAHR World Congress*.

USACE, 1987. *Hydraulic design criteria*,

Ypma, T.J., 1995. Historical Development of the Newton-Raphson Method. *SIAM Review*, 37(4).