INTERNATIONAL JOURNAL OF
ENERGY AND ENVIRONMENT

Volume 10, Issue 1, 2019 pp.15-24
Journal homepage: www.IJEE.IEEFoundation.org

2D Visualization of flow phenomenon over individual inlet and outlet keys of piano key weir by CFD modeling

Mohammed Baqer N. Al-Baghdadi

Water Research Center, International Energy and Environment Foundation, Najaf, P.O. Box 39, Iraq.

Received 5 Nov. 2018; Received in revised form 2 Dec. 2018; Accepted 11 Dec. 2018; Available online 1 Jan. 2019

Abstract
Inlet and outlet keys are the main two components of piano key weir. Understanding the phenomena of flow over these two components is an important factor in the design of this structure. In this article, 2D investigation into the velocity and pressure distributions of the flow over individual inlet and outlet keys under different discharges has been performed using CFD technique. Analysis of the results provided information about the critical areas that should be taken into account when designing the PKW structure.

Keywords: Piano key weir; CFD modeling; Two-dimensional; Flow pattern; Velocity distribution.

1. Introduction

Piano Key Weir (PKW) is a specific type of labyrinth weirs developed between the years 1998 and 2003 as a solution to the problem of traditional labyrinth weir which is the inadequacy to construct on dams due to the large base area. PKW geometry is similar to the labyrinth one, i.e. consisting of successive repetitions of cycles, with new features introduced to form the particular shape of PKW, namely: rectangular layout, sloped floors, overhangs and reduced footprint area. These features make the PKW more economical and enable its construction on gravity dam sections. Additionally, the discharge efficiency of PKW is similar to its ancestor (the labyrinth weir) in that it reaches up to 400% to that of the linear weir [1, 2].
Formed by the combination of alternatively arranged sloped floors and a side wall, the geometrical configuration of each PKW cycle (also called unit) consists of two distinctive alveoli (or chambers) referred to as *keys*. When the sloped floor is rising toward the downstream, that chamber is called an *inlet key*; when it is falling, it is an *outlet key*. Full geometrical description and a naming convention of the PKW have been established by a group of researchers in the *International conference on labyrinth and piano key weirs PKW-2011* by Pralong et al. [3]. The reader is referred to their paper for more details. A sketch of half-unit PKW is shown in Figure 1 with its main dimensions defined in Table 1. Furthermore, Figure 2 shows the geometric details of the inlet and outlet keys of PKW.
The flow pattern over PKW is a complex 3-dimensional phenomenon due to the geometrical Figure of the PKW structure. However, researchers [4] have divided the total discharge passing over PKW into three separate parts based on the features of PKW geometry, namely, the flow over 1) inlet key, 2) outlet key, and 3) sidewalls.
To better understand the flow over PKW, it is important to study the flow of its components in separate manner. The focus of this paper it to consider the first two parts (inlet key and outlet key), hence, each part

has been modelled individually by CFD software to examine the water surface profile, velocity and pressure distributions over that key. As the flow is identical along the third dimension (key width perpendicular to the flow direction), the simulation has been performed in 2-D only.

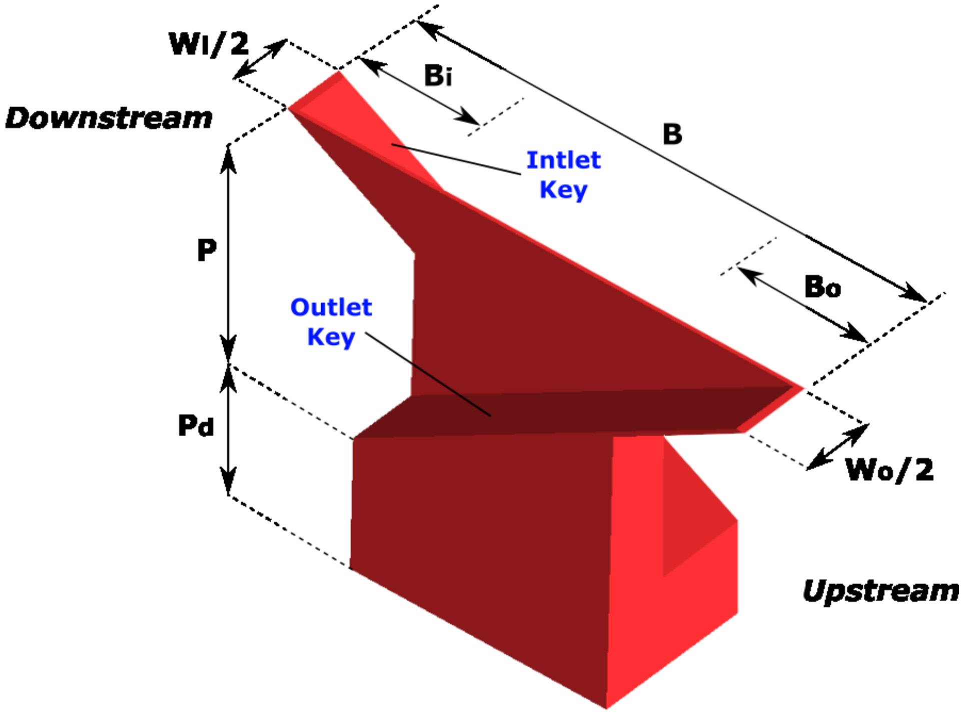


Figure 1. Geometry of a half-unit PKW.

Table 1. Terminology of PKW geometrical parameters [3].

Parameter symbol	Meaning
B	Upstream-downstream length of the PKW, $B=B_b+B_i+B_o$
B_o	Upstream (outlet key) overhang length
B_i	Downstream (inlet key) overhang length
B_b	Base length
P	Height of PKW measured from the crest (including possible parapet walls)
P_d	Dam height (or any platform under the PKW)
W	Total width of the PKW
W_i	Inlet key width (sidewall to sidewall)
W_o	Outlet key width (sidewall to sidewall)
L	Total developed length along the overflowing crest axis

2. CFD modeling

2.1 Numerical model

In this study, *Flow-3D* software was used to solve the computational fluid dynamics problem. This software uses the Volume of Fluid (VOF) method to simulate the flow phenomenon which is a numerical technique for tracking and locating the free surface of fluid. The domain of solution is divided into cells to form a computational grid, then a *fractional volume function* is introduced such that it has the value of unity in cells that are full with fluid, zero in empty cells, and a value between 1 and zero when the cell contains the fluid free surface [6].

The mass continuity equation for incompressible flow (which is the case in this study) is:

$$\nabla . \mathbf{u} = 0 \tag{1}$$

Which is a vector equation that translate in Cartesian coordinates to:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{2}$$

where ∇ is the divergence, **u** is the velocity vector field, (u, v, and w) are the velocity components in the (x, y, and z) directions respectively.

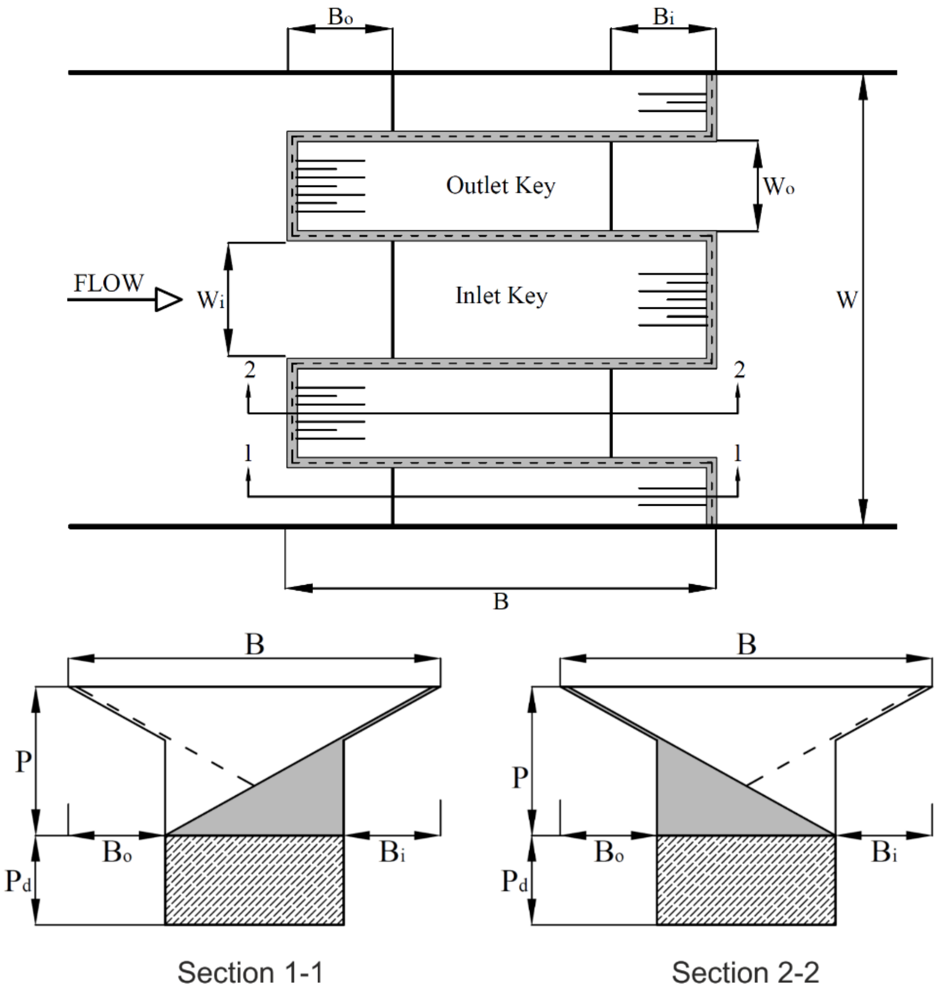


Figure 2. Plan-view of PKW (Top) with two sections for the inlet key (section 1-1) and outlet key (section 2-2) [5].

The differential momentum equation (so-called *Navier-Stoke* equation) for the fluid flow problem with constant density and viscosity may be put in the following form:

$$\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)=\rho\frac{du}{dt} \tag{3.1}$$

$$\rho g_y-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)=\rho\frac{dv}{dt} \tag{3.2}$$

$$\rho g_z-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)=\rho\frac{dw}{dt} \tag{3.3}$$

where ρ is fluid mass density, p is pressure, μ is viscosity coefficient, and t is the time [7].

Water at 20°C was considered as fluid of simulation running under the *Incompressible* flow mode with the physical features of (Gravity) and (Viscosity and Turbulence) activated. The *Flow-3D* solver offers several turbulent models to solve the flow equations (above mentioned). In this study, the two-equation *k-ε* model was used selected with the maximum turbulent mixing length dynamically computed.

ISSN 2076-2895 (Print), ISSN 2076-2909 (Online)

2.2 PKW Geometry used in simulation

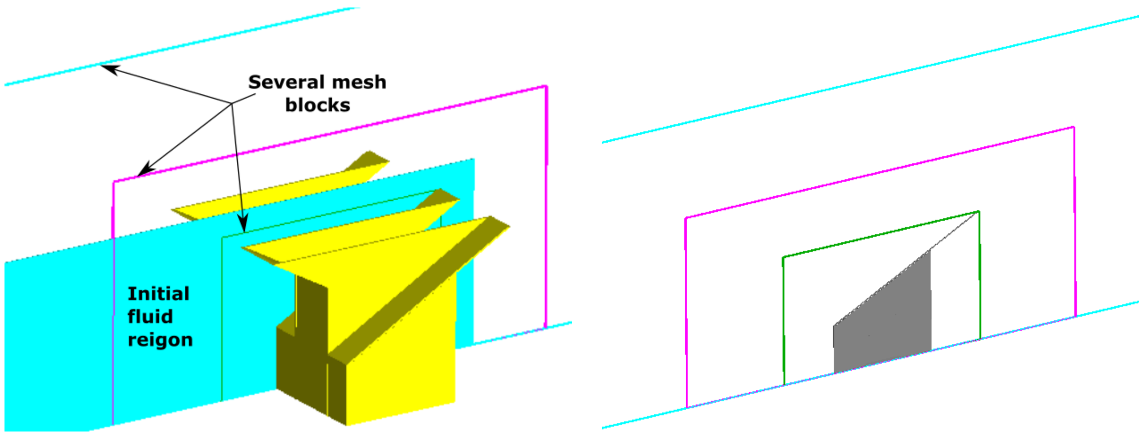
A type-A PKW structure with geometrical features corresponding to that of Lempérière [8] which are (L/W=5, W_i/W_o=1.25, B/P=2.4, B_i/B=0.25, B_o/B=0.25) was used in this model. Table 2 demonstrates the geometrical parameters of the model.

Table 2. Geometrical parameters of the PKW used in CFD simulation.

Parameter	Value (m)
B	0.303
B_i	0.076
B_o	0.076
W_i	0.0806
W_o	0.0644
P	0.126
P_d	0.076

2.3 Mesh preparation

As the simulation is done in 2 dimensions, the domain was prepared as a slice through the PKW structure having a thickness of 1 mm, see Figure 3. Several mesh blocks were employed to refine the cell sizes in important locations for accurate results. The average number of cells for all simulations was about 240,000. To get a stable water surface, sufficient distance to the upstream of the model of 1.5 m was prepared for the water to flow in. Another 1.5 m to the downstream was also prepared to receive the water until it reaches the exit boundary.



Figrue 3. View of the mesh boundaries which was prepared as 1 mm thick slice (left), and the 2-D outlet key slice that will be simulated shown by using the FAVOR tool (right).

2.4 Boundary conditions

The inlet boundary was given a specific flowrate and water elevation for each simulation. The outlet boundary was selected as "*Outflow*" which represents a mathematical continuation of the flow beyond the computational domain. This can be put in the following form (the so-called Sommerfeld radiation boundary condition) [9]:

$$\frac{\partial Q}{\partial t} + c\frac{\partial Q}{\partial x} = 0 \quad (4)$$

where Q is any flow quantity, x is directed out of the boundary, and c is the local phase speed of the assumed wavelike flow. This means that any wavelike disturbances will leave the computational mesh smoothly without backwater effects.

The top boundary was selected as pressure-specific boundary with atmospheric pressure (101.3 kPa), while the boundary on the bottom was considered as wall boundary. The lateral boundaries (perpendicular to the flow direction) were selected as "symmetry" boundaries so that no wall effect occurs. As to the solid surface of the PKW model, *no-slip* wall shear boundary was selected.

2.5 Flow conditions

Both inlet and outlet keys were tested in this study. Each key was operated under three flow conditions of discharge and specific water depth ($d_{initial}$) at the upstream boundary, Table 3 shows these three conditions. Values of discharge vs. upstream water depth were approximated based on previous experimental research [5] of PKW of the same geometry as in Table 2. This approximation has proved to be acceptable although the behavior of a single inlet or outlet key is obviously different from the behavior of the 3-D PKW structure. There were some differences between $d_{initial}$ and the actual head just upstream the PKW model, but these differences were neglected since their effect was limited in the upstream region only, and the flow passing the model was running satisfactorily.

Table 3. Flow conditions applied to both inlet and outlet key in the simulation.

Flow Condition	$d_{initial}$ (m) at inlet boundary	Q (m^3/s)
No. 1	0.232	0.042×10^{-3}
No. 2	0.252	0.072×10^{-3}
No. 3	0.275	0.109×10^{-3}

3. CFD Results

Simulation results are discussed in this section.

3.1 velocity distribution

Figures 4 to 9 shows the velocity distribution of the inlet and outlet key models under the flow conditions described in Table 3. For the outlet key model (Figures 4 to 6), the following points were observed:

- A hydraulic drop occurs immediately when the flow reaches the sloped floor of the outlet key and, hence, becomes supercritical. At the downstream side, however, a region of flow circulation develops just beside the downstream end of the structure.
- Separation and circulation of flow occurs at the upstream side under the sloping floor. However, this circulation declines and disappears as the discharge increases, see Figure 6.
- At the upstream edge of outlet key and as the water is running down the sloped floor, flow separation takes place. This separation region gets larger as the discharge increases.

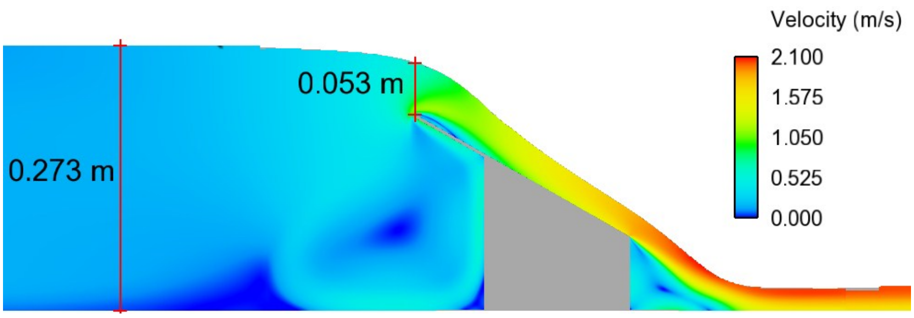


Figure 4. Velocity distribution of the flow over outlet key under flow condition No. 1.

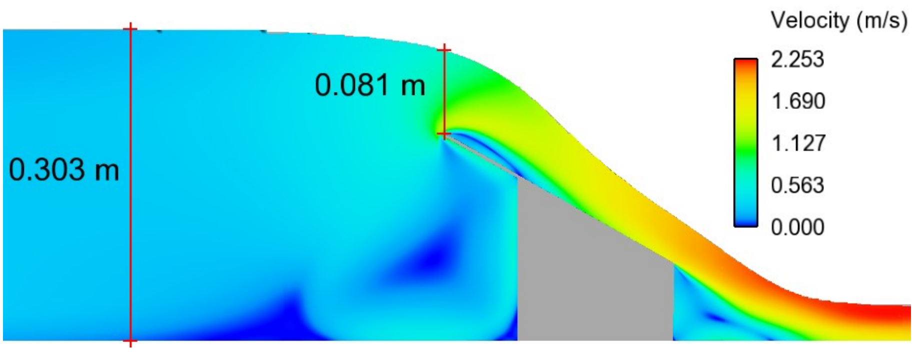


Figure 5. Velocity distribution of the flow over outlet key under flow condition No. 2.

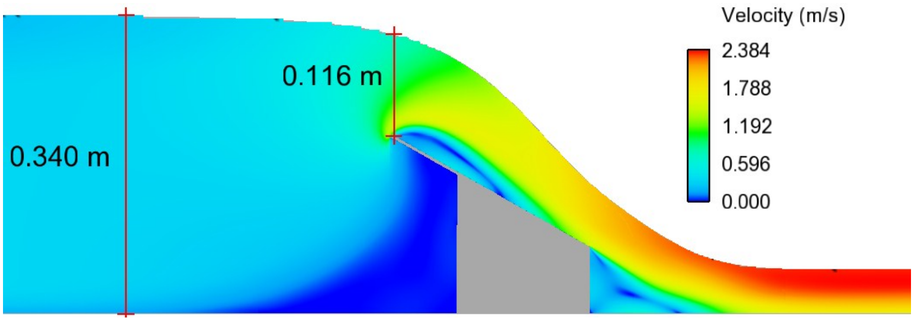


Figure 6. Velocity distribution of the flow over outlet key under flow condition No. 3.

For the inlet key model (Figures 7 to 9), the observations were as follows:

- The water flows smoothly over the sloped floor to the downstream side. The drop occurs when the water reaches the edge of the sloped floor and falls in the downstream channel.
- On the downstream side, a region of confined air under the nappe develops. The water under this region experiences circulation. Because the weir is not ventilated, this region of air gets smaller as the flow increases until it disappears and the weir becomes "drowned".
- The weir became drowned under the flow condition No. 3 as shown in Figure 9. Flow circulation is observed underneath the inlet key overhang.

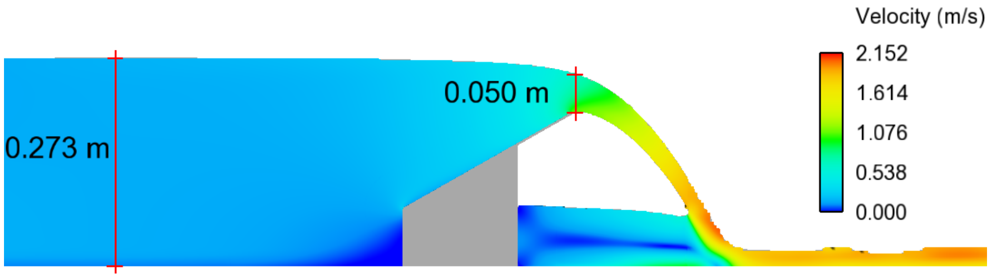


Figure 7. Velocity distribution of the flow over inlet key under flow condition No. 1.

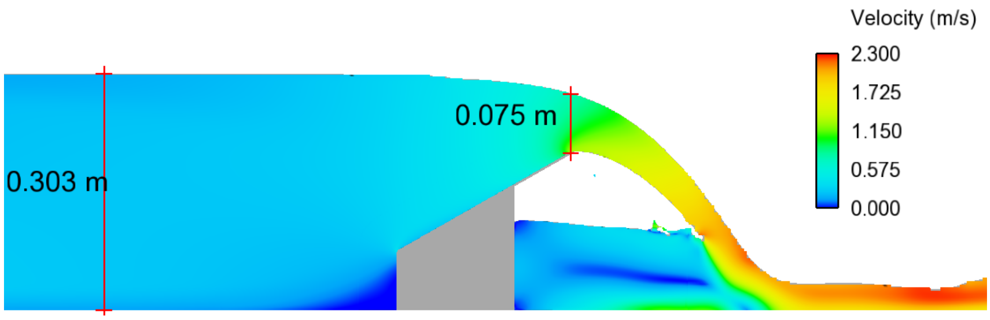


Figure 8. Velocity distribution of the flow over inlet key under flow condition No. 2.

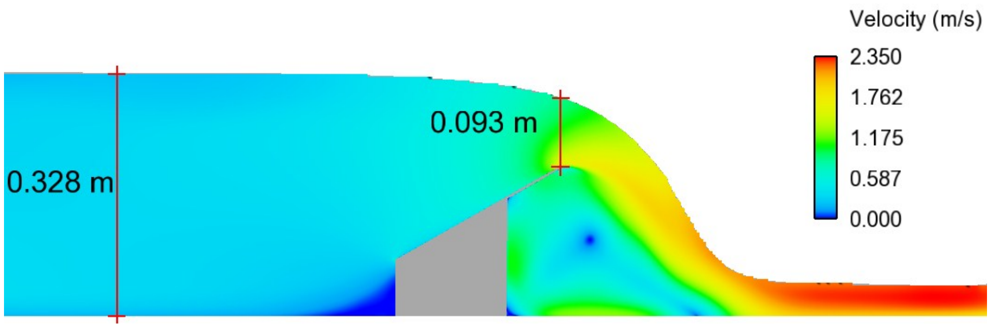


Figure 9. Velocity distribution of the flow over inlet key under flow condition No. 3.

In order to provide some insight for hydraulic calculation, measurements of water depth were taken at two locations: at the crest of the inlet (and outlet) key, and at 0.3 m to the upstream direction. The measurement at 0.3 m to the upstream direction is considered far enough from the water drop effects to represent the upstream water head of the weir [5]. Furthermore, the velocity distributions at the section 0.3 m to the upstream of the weir have been determined in Figures 10 and 11.

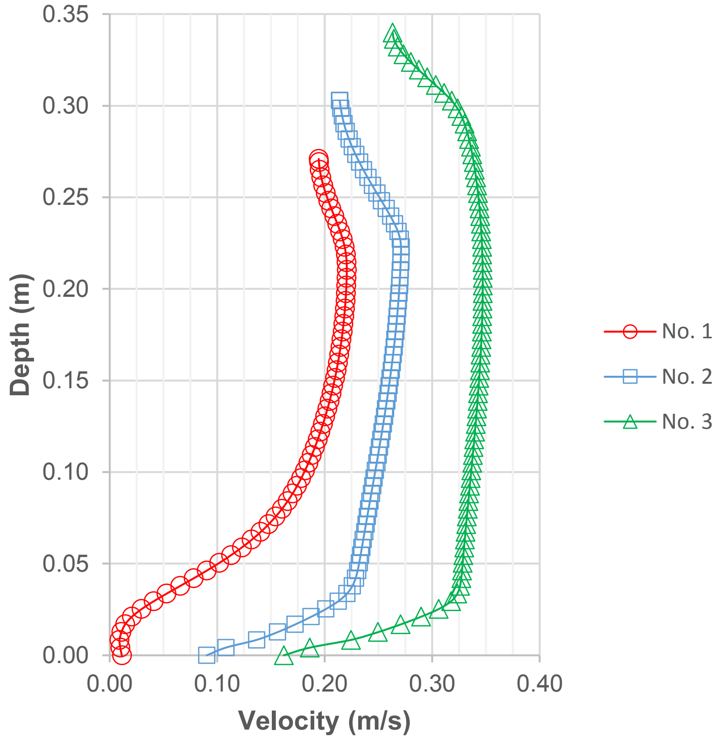


Figure 10. Velocity profile of the flow reach the taken at 0.3 m upstream of the outlet key model for the three flow conditions defined in Table 3.

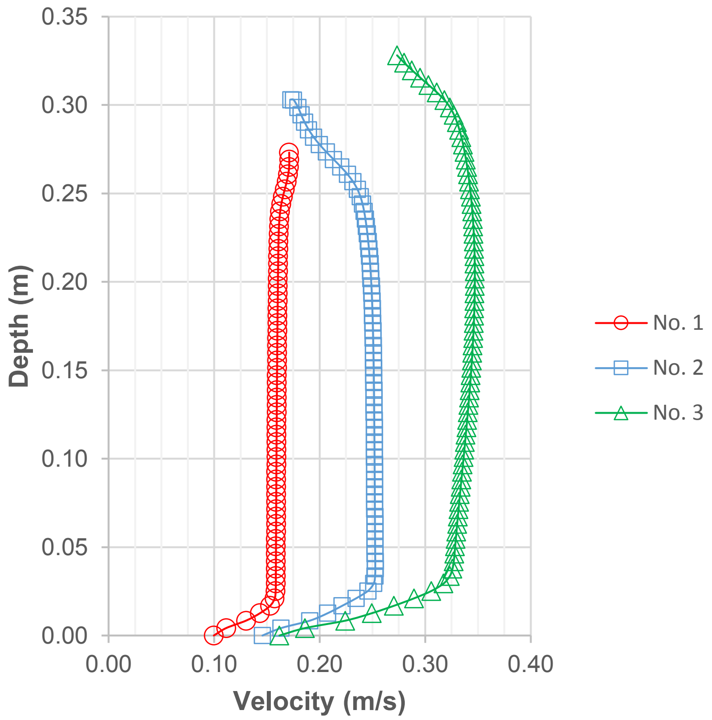


Figure 11. Velocity profile of the flow reach the taken at 0.3 m upstream of the inlet key model for the three flow conditions defined in Table 3.

3.2 Pressure distribution

Figures 12 to 17 shows the pressure distribution for the inlet and outlet models. It is important to consider the weir overhang in both models because of its small thickness and being under pressure variation between the its upper and lower surfaces. When the separation region occurs on the overhang of the outlet key, negative gauge pressure. This conditions also happens beneath the inlet key overhang when the flow is drowned (Figure 17). On the upstream side, the pressure seems almost hydrostatic because the flow is uniform.

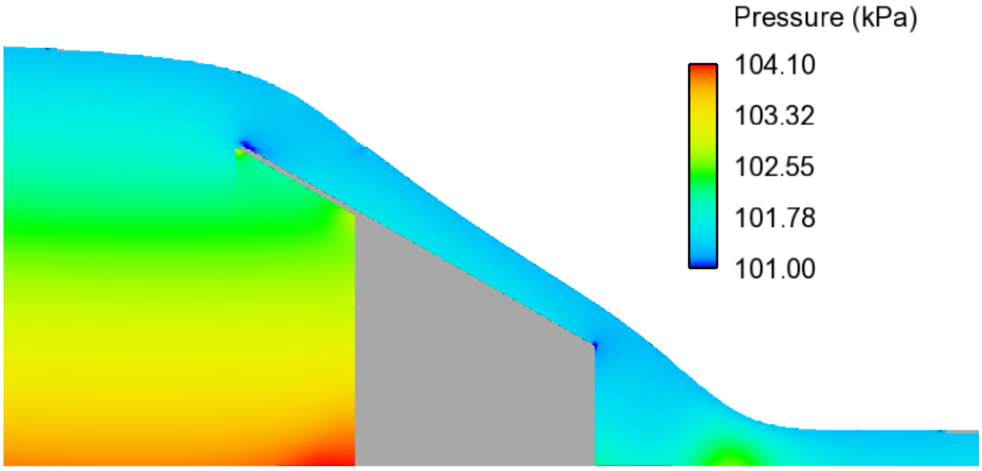


Figure 12. Pressure distribution of the flow over outlet key under flow condition No. 1.

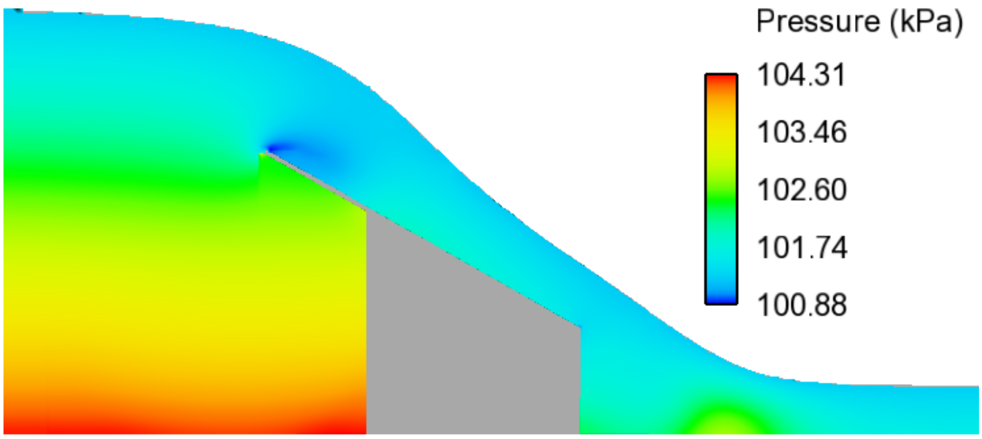


Figure 13. Pressure distribution of the flow over outlet key under flow condition No. 2.

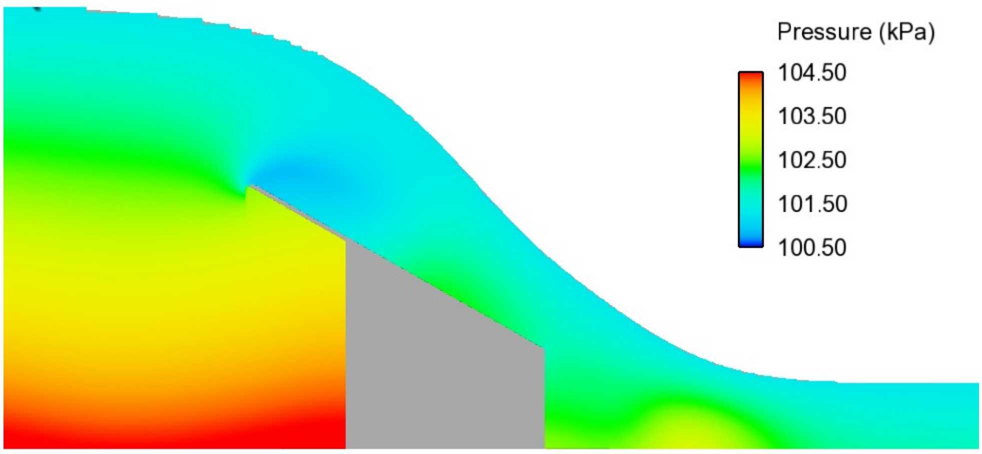


Figure 14. Pressure distribution of the flow over outlet key under flow condition No. 3.

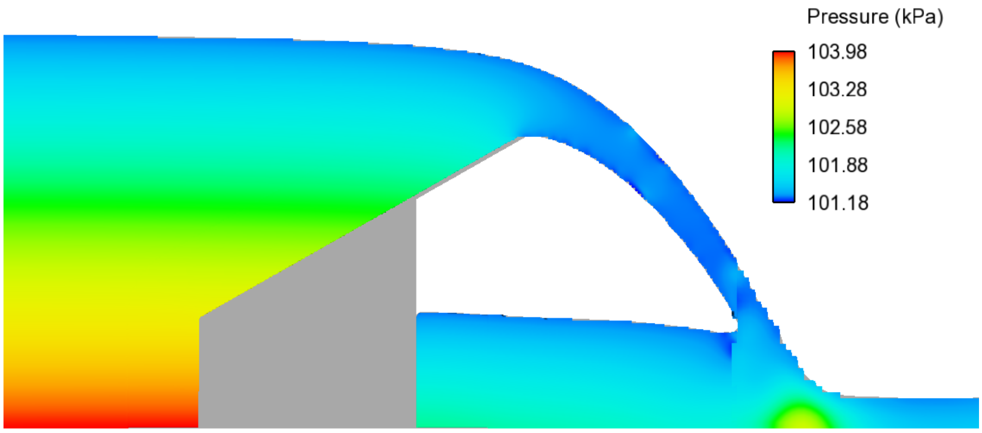


Figure 15. Pressure distribution of the flow over inlet key under flow condition No. 1.

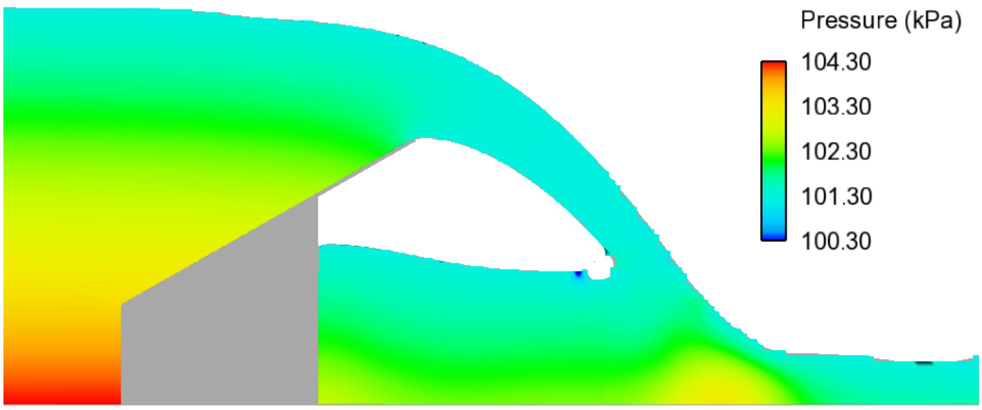


Figure 16. Pressure distribution of the flow over inlet key under flow condition No. 2.

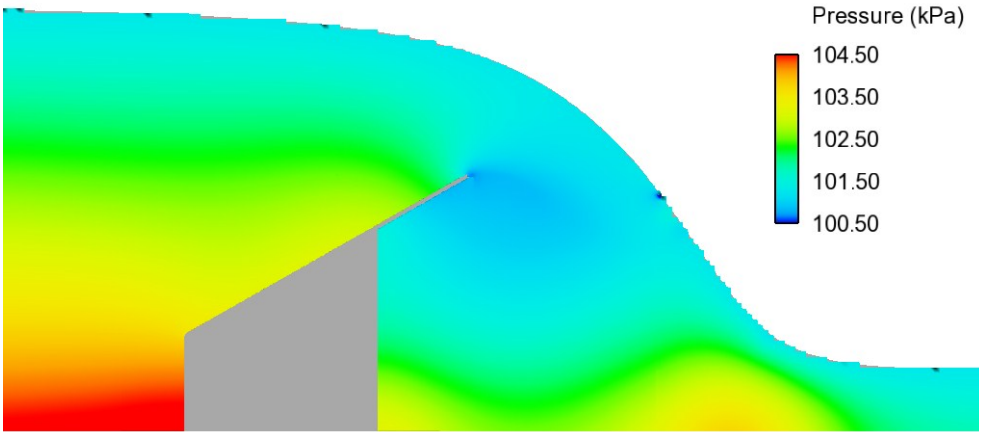


Figure 17. Pressure distribution of the flow over inlet key under flow condition No. 3.

4. Conclusion

- 2-D simulation has been performed using the CFD package *Flow-3D*. The velocity and pressure distribution of the flow over individual inlet and outlet key structures were established providing more understanding of flow behavior.
- Regions of extreme pressures and velocities were identified enabling the engineer to take them into account during the design process.
- Piano key weir is an important structure in dam rehabilitation and reduction of the damaging effect of floods. Study of flow phenomena over this structure leads to more comprehension of the design requirements not only of the PKW but of similar labyrinth structures. Because of the increasing

ISSN 2076-2895 (Print), ISSN 2076-2909 (Online)

frequency of heavy rain and flood events around the world due to climate change, more studies on this subject are required.

Acknowledgements
This work was supported by International Energy and Environment Foundation.

References

[1] Lempérière F., Vigny J.-P., Ouamane A. General comments on Labyrinths and Piano Key Weirs: The past and present. Proceedings of the International Conference Labyrinth and Piano Key Weirs – PKW 2011, Liège, Belgium. CRC Press, London, pp. 17-24, 2011.
[2] Crookston B. M., Tullis B. P. Hydraulic characteristics of labyrinth weirs. Proceedings of the International Conference Labyrinth and Piano Key Weirs – PKW 2011, Liège, Belgium. CRC Press, London, pp. 25-32, 2011.
[3] Pralong J., Vermeulen J., Blancher B., Laugier F., Erpicum S., Machiel O., Pirotton M., Boillat J.-L., Leite Ribeiro M., Schleiss A. J. A naming convention for the Piano Key Weirs geometrical parameters. Proceedings of the International Conference Labyrinth and Piano Key Weirs – PKW 2011, Liège, Belgium. CRC Press, London, pp. 271-278, 2011.
[4] Machiels O. Experimental study of the hydraulic behaviour of Piano Key Weirs (Ph.D. thesis, Université de Liège, Belgium), 2012.
[5] M. B. N. Al-Baghdadi, Physical modeling of PIANO KEY weir: detailed experimental study. International Energy and Environment Foundation (IEEF), (ISBN: 978-1-5397-1134-6), 2016.
[6] Hirt CW, Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics. 1981; 39(1):201-25.
[7] White FM. Fluid Mechanics, 4th edition. McGraw-Hill Series in Mechanical Engineering, 1998.
[8] Lempérière F. New Labyrinth weirs triple the spillways discharge – Data for an easy design of P.K. Weir, 2009. Hydrocoop company website. (http://www.hydrocoop.org), accessed on 1 August 2018.
[9] Flow Science Inc. Flow-3D v.11.2.0 User Manual, (Theory chapter, Boundary Conditions). Copyright 2016.