Bottom rack intake improvement as a fluid physics application through a computational fluid dynamics model

H E Calderón¹, L M Rada¹, and J S De Plaza¹
¹ Facultad del Medio Ambiente y Recursos Naturales, Universidad Distrital Francisco José de Caldas, Bogotá, Colombia

E-mail: hecalderong@correo.udistrital.edu.co, sjsebastian@correo.udistrital.edu.co

Abstract. This research focuses on improving the hydraulic behavior of a traditionally designed bottom rack intake, from variations in roughness parameters, free height, and the inclusion of chamfers, establishing a contribution to the contrast between classical physics and the physics that takes over the partial resolution of the Navier-Stokes equations. To make possible the structure in OpenFOAM, it is necessary to use the geometric tool Salome-Meca, as well as a meshing tool (snappyHexMesh), and the InterFOAM solver in the processing stage. In the same way, through the turbulence model (K-E) local effects are evidenced in the Fluid-Structure interaction, as well as the identification of events and the development of the phenomenon of vorticity. The results show the improvement presented in some areas of the structure from the stabilization of the water flow through of the fluid-structure interaction change, the modification of the geometry and roughness, minimizing the presence of vertical vortices, cavitation, and surrounding areas. This allows us to conclude that traditional hydraulic do not consider the real physical flow behavior within the structure and neither the subsequent phenomena that develop, establishing as a starting point the need to rethink the design of the bottom rack intakes.

1. Introduction

A bottom rack intake (BRI) has the function of capturing the water in high mountain rivers. This, through a rack with bars located in the flow direction, which is responsible for diverting the water to the conduction pipeline. To prevent damage to the structure or drinking water treatment plant (DWTP), it is possible to simulate the future hydrodynamic flow behavior over structures, with the purpose of predicting phenomena presented by changes in the system variables. This, using computational fluid dynamics (CFD), from the OpenFOAM open-source software, which uses Reynolds averaged Navier-Stokes equations (RANS) [1], and provides a three-dimensional solution to practical problems. This software also allows observing the system air-water and fluid-structure interaction, using finite volumes method (VOF), and the InterFOAM solver (which has proven to be reliable for the simulation of free surface vortices in an intake) [2].

Various studies have shown hydraulic failures in dimensioned under traditional parameters (BRI), as in the case of [3], which shows the vortex development due to the momentum generated when the flow impacts on the water collection channel surface. Likewise, in that area, according to [4], there are unstable lines flow phenomena that triggers eddies by the air capture. Similarly, in [5], it is stated that, if although CFD is used to improve the knowledge of hydraulic phenomena, it is not possible to analyze effects which occur in the bars, due to their multiplicity. Furthermore, in [6], it is claimed that the surrounding velocity vectors are the effect of the square geometry of the vertex, both in channel and in water collection chamber, which leads to the drain and excesses pipeline. However, the effects generated by the flow dynamics in the main walls area, as which are appeared in the water collection channel and
chamber, have not been subjected to an exhaustive study. That is why, in this research, it is proposed to generate an alternative of hydraulic improvement in a BRI, from the Hydraulics deficiencies identification of the traditional design and structure simulation in CFD as an earlier support of traditional engineering projects that consider the physical flow behavior inside the structures.

2. Methodology and materials
To obtain results, two phases are planted that, carried out, analyze the flow behavior through the structure. The first objective focused to design the BRI under the classical hydraulics parameters, in order to establish its geometry. Then, it is proposed to carry out the 3D structure modeling in three instances; preprocessing, to create the solid geometry, processing meshing and computational modeling through OpenFOAM, and post-processing to generate the graph results.

2.1. Traditional design phase
The BRI, composed of the intake rack, channel, collection chamber, and water excess weir, is designed based on classical hydraulics parameters, in relation to sizing, flow velocity ranges, and free heights recommendations. On the other hand, the design flow is 0.026 m$^3$/s for structures dimensioning.

2.2. OpenFOAM modeling phase
The computational modeling is carried out through OpenFoam 8.0, and is executed from the following steps: solver, turbulence model, vorticity model and model initial conditions.

2.2.1. InterFOAM solver. Due to the fluids characteristics to be modeled (incompressible, isothermal, and immiscible), the interaction model between phases is carried out with InterFOAM solver, which solves the three-dimensional equations for the water-air interface [7], using the fluid volume method (VOF); this, assuming a constant water and air density [7].

Equation (1) represents the momentum conservation [2], where $u$ is the velocity, $\rho$ is the fluid density, $g$ is the gravity acceleration, and $p$ is the pressure. On the other hand, $x_i$ and $x_j$ represent a fixed Cartesian system coordinate, and $T_{ij}$ and $T_{ij}$ are the viscous and turbulent stresses. Similarly, $f_{oi}$, is the surface tension.

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\rho u_i T_{ij}) + \rho g_i + f_{oi},
\]

(1)

where $\alpha$ is the unit of water density ($\rho: 1$), and 0 represents the density of air ($\rho: 2$). At the interface between the two fluids, $\alpha$ tends to vary between 1 and 0. So, in Equation (2), if the calculation of $\alpha$ is 1, the density of the liquid in that part of the mesh is the density of water; if $\alpha$, on the contrary, is zero, it takes the density of the air in each cell, $\alpha$ being a scalar quantity, which describes the volume of a cell for a certain phase [8].

2.2.2. Turbulence model. A K-epsilon turbulence model is implemented, as it contains the governing equations developed for incompressible turbulent flows, with high Reynolds numbers.

2.2.3. Turbulent kinetic energy, $k$. The k-$\epsilon$ model contains two transport equations that consider the turbulent flow effect on the structure [9] and its dissipation rate. The first of these determines the Convection Effect, and the second, the Turbulent Energy Dissipation Rate (see Equation (3) and Equation (4)).

\[
\frac{\partial}{\partial t} (\alpha \rho e) + \nabla \cdot (\alpha \rho u \epsilon) - \nabla (\alpha \rho D_e \epsilon) = C_1 \alpha \rho G_k^2 - \left( \left( \frac{1}{3} \right) C_1 - C_3 \right) \alpha \rho \nabla \cdot u \epsilon - C_2 \alpha \rho \frac{\epsilon}{k} \epsilon,
\]

(3)
\[ \frac{\partial}{\partial t}(\alpha \rho k) + \nabla \cdot (\alpha \rho u k) - \nabla^2(\alpha \rho D_k k) = \alpha \rho G - \left( \frac{2}{3} \alpha \rho \nabla \cdot \nabla k \right) - (\alpha \rho \varepsilon k). \] (4)

where \( \alpha \) is the phase fraction, \( \rho \) is fluid density (kg/m\(^3\)), \( \varepsilon \) is the turbulent kinetic energy dissipation rate (m\(^2\)/s\(^3\)), \( u \) is the flow velocity (m/s), \( D_k \) is the effective diffusivity for \( \varepsilon \), \( G \) is the turbulent kinetic energy production rate due to the anisotropic part of the Reynolds stress tensor (m\(^2\)/s\(^3\)), \( k \) is the turbulent kinetic energy production rate due to the anisotropic part of the Reynolds stress tensor (m\(^2\)/s\(^3\)), \( C_t \) and \( C_\varepsilon \) are model coefficient (1.44), (1.92) and \( D_k \) is the fast distortion theory compression term (0.0).

In the same way, wall and roughness functions are adapted according to the type of material and the model limits. Equation (5), \( \Delta t \) approach, represents the wave velocity through the mesh (must be less than unity); Equation (6), on the other hand, represents the turbulent energy dissipation rate in m\(^2\)/s\(^2\). Lastly, Equation (7), turbulent kinetic energy is characterized by velocity fluctuations in a highly turbulent flow [5].

\[ \Delta t = C \frac{\Delta x}{u}, \] (5)

\[ \varepsilon = C_\varepsilon \frac{\mu^{0.75} k^{1.5}}{L}, \] (6)

\[ k = \frac{2}{3} \left( I(u) \right)^{\frac{3}{2}}, \] (7)

where \( C \) is the Courant number, \( \Delta x \) is the minimum mesh size (m), \( L \) is the turbulent scale length (m), and \( I \) is the turbulence intensity (0.07 %) [9].

### 3. Results and discussion

In this section, it is shown the results obtained through of the model analysis carried out under conventional hydraulics. Then, the comparison of it with the previously improvement described.

#### 3.1. Model Analysis under conventional hydraulics

The main walls sizing, based on the conventional design (at 45\(^\circ\)), originates a fluid structure interaction (FSI). This generates incident shocks, whose convergence occurs in the longitudinal focus of the intake throat [10,11]. Consequently, the turbulence intensity at this point increases due to the fluctuation imposed by the wall on the velocity vectors, producing a variation in flow depth (with a maximum upstream of 0.202 m and a minimum downstream of 0.07 m). At the same time, by gradually reducing the width of the channel, a transition is generated from a sub-critical regime upstream of the flow to a super-critical one downstream, affecting the Froude number increase (from 0.071 to 1.128); this happens, as the river flow is channeled through a smaller area, which implies downstream flow resistance. Consequently, the hydraulic jump formation effect in the throat is obtained.

On the other hand, it is observed that the effect on the bars that make up the grid, under a super-critical flow regime, produces an undulating adhesion effect and flow drag when it passes over it. This induces the “leaf jet” effect, in which the water tends to remain bound on the bars in the rack (Figure 1), which coincides with the results obtained in [12]. Also, considering a tilt ratio, the fall of the flow is greatly contributed, producing greater turbulence and increased pressure drop over them (from 0.086 m to 0.01 m). Concerning the flow behavior over the bars, a progressive decrease in kinetic energy and the jump is evidenced, which generates a hydrostatic pressure gradient along the bars, from 1663.62 Pa, at the beginning of the rack, to 589.028 Pa at the end of this [13]. It is also observed that, at the beginning of the bars, there is a sub-critical condition (Froude 0.1) with a transition to super-critical regime at the end of the bar (Froude 7.36).

At the same time, a dissipation effect is generated at the end of the bar, as evidenced in [4], since the oscillating interaction between the velocity and the tie falls unstably on the bars. This induces a spatially varied flow (SVF) with decreasing discharge, as shown in the experimental results of [12,13] (see Figure 2), where such a flow condition is evidenced in the rack.
Since the shock current wave, it supposed the recirculation in the wall of the channel (because the angular momentum conservation). A water-air mixture is produced (Figure 3), which spreads the flow aeration over the water collection channel walls, this being relevant, since it benefits the water reoxygenation in its treatment process, up to a value of 6485 mg/L [14]. In the same way, since the discharges are variable throughout the channel length, there is an increase in the join (of 0.015 m on the opposite channel wall and 0.0931 m on the adjacent wall to the flow direction) and upward flow, whose lines are considerably curved and sloping, before delivering to the collection chamber, resulting in a flow rotational discharge. However, intense turbulence is observed, as well as a channel domain interface, which generates small waves, product of the fluctuations caused by the vertical vortex presence [15].

In addition, it is evident that the receiving shock wave wall increases the pressure, generating less sensitivity to the phenomenon already described. However, in the center and adjacent wall, tends to present both vacuum pressure (Figure 4) (on average 1292 Pa and 492 Pa, respectively), as pressures below the water vapor pressure at 15 ° (1657.27 Pa). This supposes that the water nappe hits the channel opposite wall, passing over the channel floor, and subsequently collides with the adjacent wall. Thus, it begins to adhere to the surface, as the vortex develops. This tends to detach, generating the effect of suction (Figure 4), which in turn implies the presence of pressures below atmospheric pressure, promoting the cavitation risk in the area [16].

Given that the cavitation index indicates the cavitation existence in the fluid [17], Table 1 shows the presence of this phenomenon in the channel floor and in the water collection channel adjacent wall. In addition, the accentuated layer turbulent discharge (water collection channel overhang), adopts a flow rotational pattern lines at the wall corners, generating a 0.078 m pressure drop, low kinetic energy and, on the contrary, higher potential energy.

Regarding the layer effect on the bottom of the water collection chamber, it is observed that the height of fall on the chamber increases the jet potential energy, which implies an increase in flow velocity (from 2m/s to 6 m/s) downstream of the weir (Figure 5). Likewise, it promotes the dead zones formation, due to the angular velocity of 2.86 Hz, present in the chamber corners (see Figure 6).
### Table 1. Cavitation index

|                            | Velocity (m/s) | Pressure (Pa) | Cavitation index (σ) |
|---------------------------|----------------|---------------|----------------------|
| Channel opposite wall     | 0.15           | 1678.924      | 1.86                 | No cavitation        |
| Channel floor              | 0.34           | 1292.435      | -6.73                | Cavitation           |
| Channel adjacent wall floor| 0.29           | 491.360       | -27.85               | Cavitation           |

Figure 5. Unloading the water nappe in the collection chamber.

Figure 6. Flow lines in the collection chamber.

### 3.2. Improvement to the conventional design from the computational fluid dynamics model

The proposed geometric alternative (whose characteristics are summarized in Table 2), makes possible the flow field appeasement due to the implementation of chamfers in the channel corners and chamber water collection (see Figure 7). This, in turn, favors the uniform distribution of velocities from the channel to the chamber, which implies the turbulence reduction and hydraulic improvement (Figure 8).

In the same way, from the modifications made, the dead zones reduction in the channel is evidenced, which entails the hydraulic load reduction, with respect to the one previously presented towards the collection channel corners from 0.167 m to 0.075 m, as it is possible to see in Figure 8.

Equally, about the change in roughness (from 0.03 to 2 mm), it is observed that the flow lines at the bottom are disturbed due to the rough walls at the bottom and concrete surfaces in the structure. Indeed, a behavior isotropic convergent flow in interaction with the rough wall [18]. In addition, a greater velocity gradient is obtained as a result, due to the velocities fluctuation generated by the surface rebounding, as well as the flow energy dissipation due to the roughness increase.

### Table 2. Improvement characteristics summarize

|                            | Value (m) |
|---------------------------|-----------|
| Slope (m/m)               | 0.05      |
| Chamfer radius            | Collection channel 0.25  |
|                           | Collection chamber 0.10  |
| Free boundary             | 0.17      |

Figure 7. Unloading the water nappe in the collection chamber.

Figure 8. Flow lines in the collection chamber.
4. Conclusions
The geometric proposal allowed to improve the hydraulic-structural performance in the bottom rack intake, minimizing the eddies presence from the incorporation of chamfers, and the consideration of an additional free edge to that considered in traditional hydraulics. This, if applied, would prevent damage to the bottom rack intake concrete structure. The research carried out allows us to infer that the traditional hydraulics for the bottom rack intakes design does not consider the fluid-structure interaction, ignoring the vorticity effects and recircumbent zones that this generates, as well as the spatially varied flow presented in the collection channel.

Likewise, it is suggested to apply a Lagrangian model to consider the sands interaction with the structure, and thus demonstrate the effect of viscous forces on energy dissipation. This, to adjust the phenomenon to the physical model. Also, it is recommended to run this under a direct numerical simulation or large eddy simulations model type, which would provide a closer approximation to the turbulent effect of flow on the structure.

Finally, it is recommended to add energy dissipation structures in the water collection channel to reduce flow energy to the chamber collection.

References
[1] Eichhorn M, Doujak E, Waldner L. 2016, Investigation of the fluid-structure interaction of a high head Francis turbine using OpenFOAM and Code_Aster IOP Conf. Series: Earth and Env.Sci. 49(1) 1
[2] Domfeh M, Gyamfi S, Amo-Boateng M, Andoh R, Ofosu E, Tabor G 2020 Numerical simulation of an air-core vortex and its suppression at an intake using OpenFOAM Fluids 5(4) 221
[3] Brunella S, Hager W, Minor H 2003 Hydraulics of bottom rack intake J. of Hydraulic Engineering 129 2
[4] Tokyay N, Velioglu D, Dincer 2015 A numerical and experimental study on the characteristics of hydraulic jumps on rough beds Proceedings of the 36th IAHR World Congress (Hague: International Association for Hydro-Environment Engineering and Research)
[5] Carrillo J, Garcia J, Castillo L 2018 Experimental and numerical modelling of bottom intake racks with circular bars Water 10 5
[6] Marghazar S, Montazerin N, Rahimzadeh H 2003 Flow field, turbulence and critical condition at a horizontal water intake Journal of Power and Energy 217 53
[7] Ferrer P M, Causon D M, Qian L, Mingham C,Ma Z H 2016 A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank Computers & Fluids 125 116
[8] Spitzenberger A, Neumann S, Heinrich M, Schwarze R 2020 Particle detection in VOF-simulations with OpenFOAM SoftwareX, 11 100382
[9] Tsan-Hsing S, Nan-Suey L, Kuo-Huey C 1998 A non-linear k-epsilon model for turbulent shear flows 34th Joint Propulsion Conference and Exhibit (Cleveland: American Institute of Aeronautics and Astronautics)
[10] Defina A, Pietro D 2010 Open channel flow through a linear contraction Physics of Fluids 22 036602
[11] Tosi L P, Colonius T 2019 Modeling and simulation of a fluttering cantilever in channel flow Journal of Fluids and Structures 89 174
[12] Chan S N, Wong K C C, Lee H W J 2018 Hydraulics of air–water flow in a supercritical bottom rack intake Journal of Hydro-Environment Research 21 60
[13] Castillo L G, Garcia J T, Carrillo J M 2017 Influence of rack slope and approaching conditions in bottom intake systems Water 9(1) 65:1
[14] Tsai C H, Yeh C P, Lin C H, Yang R J, Lin C H, Lin C H, Fu L M 2012 Formation of recirculation zones in a sudden expansion microchannel with a rectangular block structure over a wide Reynolds number range Microfluid Nanofluid 12 213
[15] Chen Y L, Wu C, Ye M, Ju X M 2007 Hydraulic characteristics of vertical vortex at hydraulic Journal of Hydrodynamics, Ser. B 19(2) 143
[16] Turalina D, Yenembergenova D, Alibayeva K 2015 The experimental study of the features of water flowing through a sharpcrest weir in channel EPJ Web of Conferences ed. Dančová P, Vít T (Český Krumlov: EDP Sciences)
[17] Hosseinpour Shafaghi A, Rokhsar Talabazar F, Zuvim M, Talebian Gevari M, Villanueva L G, Ghorbani M, Kozar A 2021 On cavitation inception and cavitating flow patterns in a multi-orifice microfluidic device with a functional surface Physics of fluids 33(3) 032005
[18] Orlandi P, Leonardi S 2006 DNS of turbulent channel flows with two- and three-dimensional roughness Journal of Turbulence 7 N73:1