May 17th, 5:20 PM

CFD Simulation Of The Air-Water Flow In The Bottom Outlet Of Ituango Hydroelectric Project

David Calderon
Integral SA, dcalderon@integral.com.co

Aldgo Germán Benavides
National University of Colombia, agbenavidesm@unal.edu.co

Francisco Mauricio Toro
National University of Colombia, fmtoro@unal.edu.co

Recommended Citation

Calderon, David. (2018). CFD Simulation Of The Air-Water Flow In The Bottom Outlet Of Ituango Hydroelectric Project. Daniel Bung, Blake Tullis, 7th IAHR International Symposium on Hydraulic Structures, Aachen, Germany, 15-18 May. doi: 10.15142/T3TW7D (978-0-692-13277-7).

7th International Symposium on Hydraulic Structures Aachen, Germany, 15-18 May 2018
ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

CFD Simulation Of The Air-Water Flow In The Bottom Outlet Of Ituango Hydroelectric Project

David Calderón Villegas[1], Aldo Germán Benavides[2] & Francisco Mauricio Toro[2]
[1]*Integral S.A, Medellín, Colombia (Senior Civil Engineer)*
[2]*National University of Colombia, Medellín, Colombia (Assistant Professor)*
[2]*National University of Colombia, Medellin, Colombia (Professor)*
E-mail: dcalderon@integral.com.co

Abstract: *This study presents a three-dimensional CFD model of the air-water flow in a free surface discharge tunnel of a typical bottom outlet with a high head gate. The numerical simulation is performed in ANSYS-FLUENT. Results of volume fraction, velocity, and pressure fields are analyzed for two gate openings, i.e. 54% and 100%. The free surface flow in the conduit is calculated by means of the VOF method. Finally, a novel methodology is presented to calculate the air demand distribution as part of the design of the aeration system in bottom outlets with high head gates.*

Keywords: *Air demand, ANSYS-FLUENT, bottom outlet, CFD, radial gate, VOF method.*

1. Introduction

The Ituango hydroelectric project is located in the Cauca River, on the so-called "Cauca Canyon". There, the river flows through deep canyons and steep margins. The Cauca River is one of the most important ones in Colombia. Its length accounts for 1,350 km, and its basin is shared by more than 180 municipalities, which in turn add up to approximately 10 million inhabitants. This river rises in the south of Colombia, and runs towards the north, down to the Magdalena River. This latter dies in the Caribbean Sea, in the northern coast of the country. The dam is located 8 km downstream the Pescadero Bridge, in the road to Ituango, and immediately upstream from the discharge of the Ituango River to the Cauca River. At the site, the river has an average flow of 1,010 m³/s.

The Project's installed capacity is 2,400 MW. The energy production is estimated to be 13,930 GWh, on an annual basis. As for the civil works, the Project comprises: *i)* the right abutment, works for temporary diversion of the Cauca River. Those are formed by two tunnels that are to be closed once the dam is constructed; *ii)* an open flow channel spillway with four radial gates; iii) a bottom outlet to control the filling of the reservoir and assure, in any event, the discharge of the minimum flow required (450 m³/s) by the environmental authority; *iv)* a 225-meters-high earth core rockfill dam; and *v)* an underground powerhouse located on the right bank of the river. The powerhouse is placed in the main cavern and accounts for electromechanical auxiliary equipment, control equipment, control building, an erection bay, office buildings, and eight Francis-type turbines, 300 MW of nominal power each, with vertical axis synchronic generators.

The dam is ECRD-type, namely, it consists of rockfill with an impervious core. Its total volume is 20,070,000 m³, and its crest is 550 m long. The maximum reservoir's volume capacity is 2,700 x 10^6 m^3/s, and its useful volume is 900 x 10^6 m3/s. The diversion of the river is made by means of two 14.0 m x 14.0 m horseshoe tunnels, which are 1,090.0 m and 1,215.0 m long, respectively. Before filling the reservoir, the tunnels will be closed by four 14.0 meter-high and 7.0 meter-width gates. The headrace system is composed by eight headrace tunnels, one for each unit, with the following main components: *i)* a submerged intake structure, *ii)* a lower penstock, *iii)* a vertical pressure shaft, and *iv)* an upper tunnel with closure gates. The operation of these latter is carried out by the vertical shafts. The water intake is 357.0 meters above sea level, and the design flow for each tunnel is of 168.8 m³/s. Figure 1 illustrates the general works with regards to the dam, the diversion, the penstock, the powerhouse, and the outlet structure.

This paper presents a computational study of the flow through the Project's bottom outlet, which is controlled by two high head radial gates placed in a chamber. Upstream the chamber, the water flows through a pressurized conduit. This latter has an 8 m wide, 8 m high, and 464 m long vault. At the chamber, it initiates a 21.4 m transition from the vaulted section to a rectangular section. 8 m downstream the conduit bifurcates into two 3 m wide and 3 m high branches up to the radial gates. Downstream of the radial gates, a free surface flow extends along 500 m. Figure 2 summarizes the above-mentioned characteristics.

In the present case, the main purpose of the analyzed bottom outlet is to regulate the environmental outflow during the filling of the reservoir and sudden shutdown of the hydroelectric power plant. In order to comply with the

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

environmental regulations, the maximum flow to be discharged through the bottom outlet is 450 m³/s. Moreover, the gradual filling of the reservoir increases the pressure on the gate. To guarantee an environmental outflow controlled discharge, the opening of the radial gate varies from a fully open position up to 54% partial opening. It is worth mentioning that the analyzed bottom outlet becomes a world record in terms of pressure, as it will be subjected to a hydrostatic head of 147 mwc (meters of water column). Such a pressure has not yet been registered in other projects worldwide.

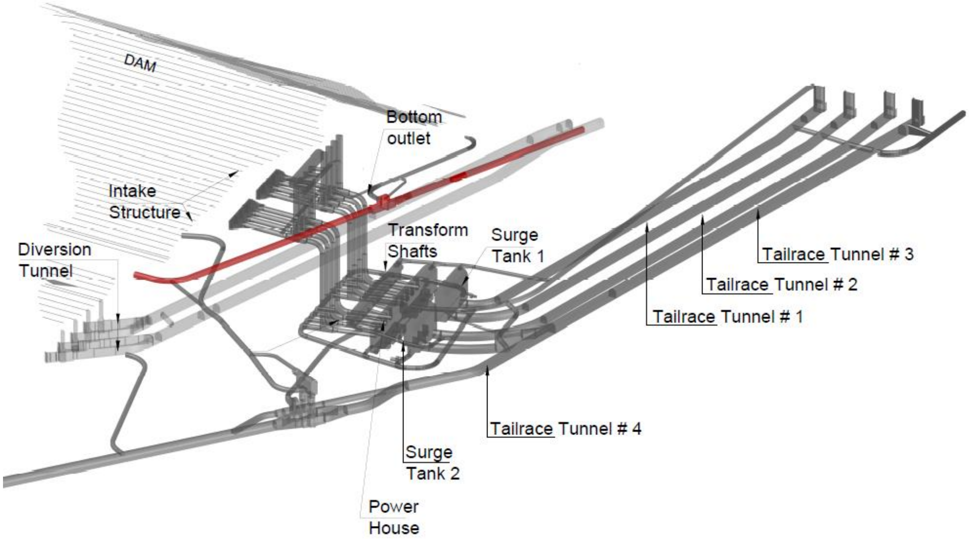

Figure 1. 3D view of the hydroelectric project.

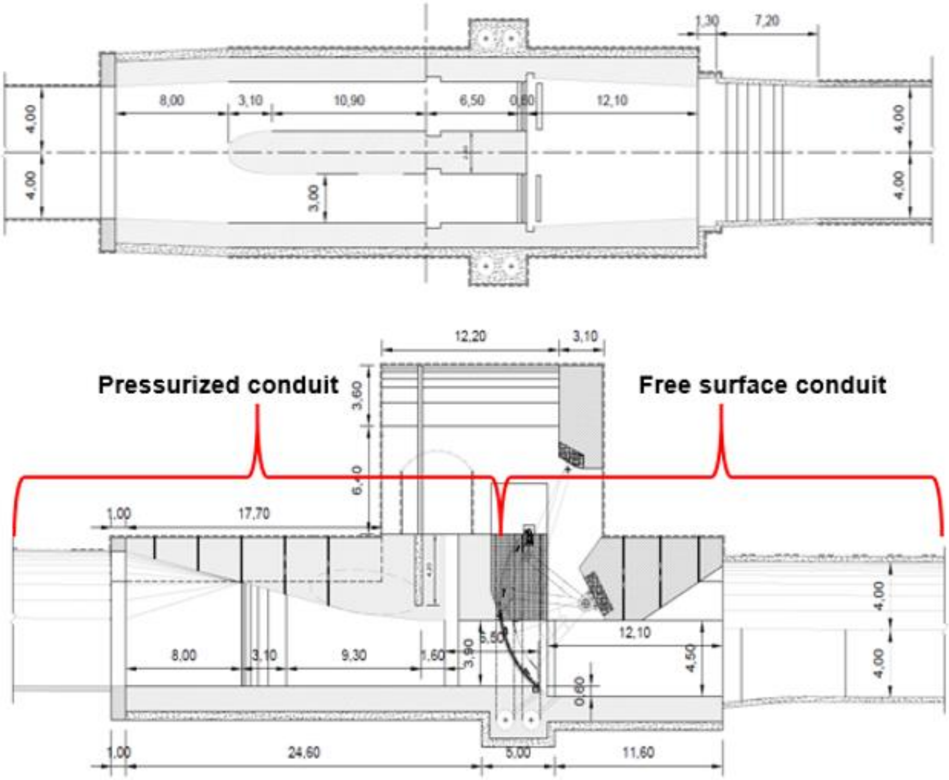

Figure 2. Plan and profile view of the gate chamber of the bottom outlet (units in meters).

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

2. Problem Description

The term high head gate refers to all gates used in dams, spillways, penstocks or conduit that support and to operate under pressures greater than 25 mwc (meters of water column) (Sagar, 1995).

The bottom outlet and the spillway are essential structures to maintain the global stability and good performance of the dam. Their main functions are to maintain the operations levels in the reservoir, regulate the environmental outflow, perform reservoir cleanup, evacuate peak flows, and control the reservoir during its lifetime. The bottom outlet gates have different operating rules: they can be either completely or partially opened or completely closed. In this regard, the most critical operation is that of partial openings. In this state, the risks of harmful vibrations and high-stress levels on the gate increase as a result of fluctuating internal pressures. Likewise, the cavitation phenomenon may be triggered if the air supply at the end of the gate is depleted.

When the gate of a high head outlet conduit is partly opened, the high-velocity flow evacuated induces sub-atmospheric-pressure regions downstream. In order to avoid this effect, the conduit is connected to the atmosphere through an air intake pipe. Thus, the air needed is supplied and thereby keeps the pressure at safe levels downstream the gate (Cihat Tuna et al., 2014). The estimation of the air demand from the air vent is a paramount factor in the designing of gated tunnels. A significant number of studies have assessed the relationship between water flow characteristics and the tunnel geometry, with regards to the air flow rate (Najafi et al., 2012). Some empirical correlations available today are based on tests performed in hydraulic models, and most of them are written in terms of the air demand ratio, as follows

$$\beta = \frac{Q_a}{Q_w} \tag{1}$$

where Q_a is the air's volumetric flow rate, and Q_w is water's. Furthermore, this ratio depends on several parameters such as the conduit geometry, the gate geometry, and the water velocity. In most studies, the Froude number at the 'vena contracta' has been reported as an adequate parameter to predict the aeration ratio (Salazar et al., 2012). The most significant studies date from the second half of the 20th century onwards. All of them yield to the following form for the air demand ratio (β)

$$\beta = k(F_c - 1)^n \tag{2}$$

Where ***k*** and ***n***, are empirical coefficients, and F_c is the Froude Number at the "vena contracta" (Sharma, 1976). Table 1 lists the main correlations found in literature.

Table 1. Correlations frequently used to estimate the air demand ratio.

AUTHOR	EQUATION
Kalinske & Robertson	$\beta = 0.006(F_c - 1)^{1.4}$
Campbell & Guyton	$\beta = 0.04(F_c - 1)^{0.85}$
USACE	$\beta = 0.03(F_c - 1)^{1.06}$
Ghetti & Di Silvio	$\beta = 0.05(F_c)^{1.418}$
Sharma	$\beta = 0.09(F_c)$

Figure 3 shows a schematic representation of the air-water flow in a bottom outlet, where part of the air entrainment is supplied by an overhead aerator, and the remaining part by the secondary bottom aerator. This research aims to provide some new understanding of the air-water flow behavior in high velocity jets downstream of a high head radial gate in a bottom outlet of a large dam. Our novel methodology estimates the air distribution as part of the aeration system design in high head pressure radial gates.

Because of the high-velocity flow that occurs at the outlet of the radial gates, erosion, vibration and cavitation phenomena result, which become the main failure mechanisms of the gate. Flow-induced vibrations can jeopardize the integrity of the structure if the frequency of vortex formation is equal to the natural frequency or fundamental mode of oscillation of the gate (Naudascher, 1991). Erosion is a direct consequence of cavitation since for cavitation to be generated, it is necessary that the pressure in a certain region remain below the vapor pressure. When this

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

happens, a sudden change of phase occurs, where the water bubbles become vapor bubbles (in the gate area); as the flow continues its trajectory, these bubbles are conducted to areas where the velocity decreases, and therefore the pressure increases (downstream of the radial gates in the step area). Thus, the vapor condenses rapidly and the bubbles implode. The occurrence of this phenomenon wears out the material, either steel or concrete, and therefore generates the erosion of it. All the mechanisms mentioned above can lead to a complete failure in the structure (Novak, 2004).

Careful design of the aeration system will prevent those failure mechanisms, that not only compromise the stability of the gate, but also the general stability of the dam. We propose to characterize the aeration system of the dam through numerical simulations.

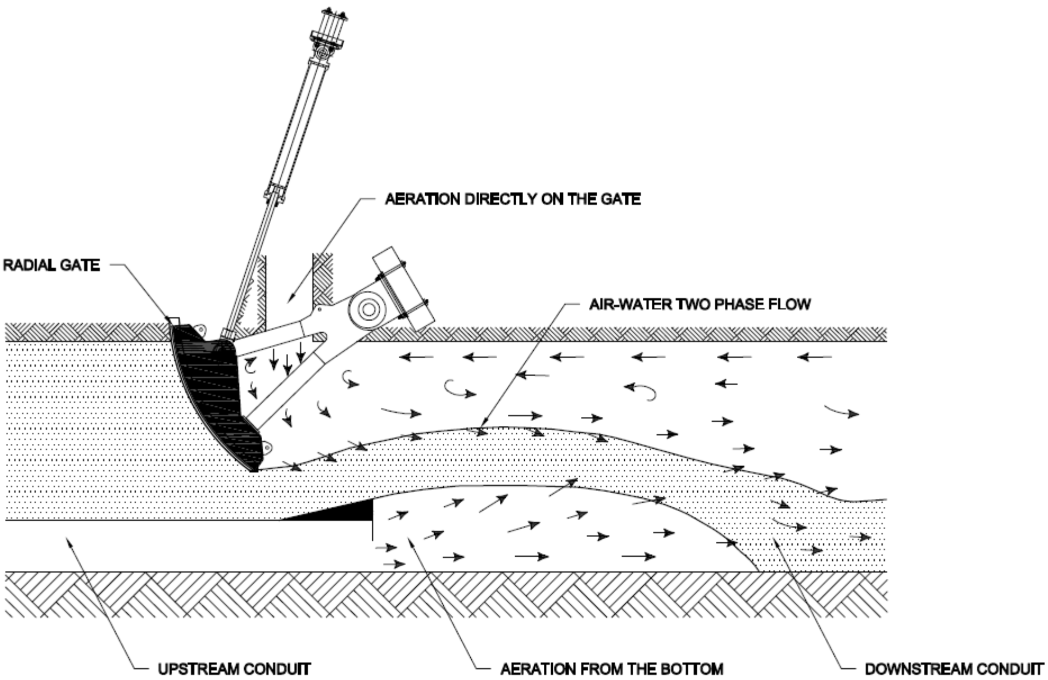

Figure 3. Typical airflow in bottom outlets.

3. CFD Model

In this research study, the CFD commercial software ANSYS-FLUENT was utilized to simulate the air-water flow downstream the radial gates. FLUENT solves the full three-dimensional Reynolds Averaged Navier- Stokes equation. A structured mesh was built with 383.064 hexahedron elements, with a max skewness of 0.457 in 257 cells. A grid size analysis was performed with grid sizes A, B, and C, summarized in Table 2. Based on the results the mesh with this amount of 383.064 cells was chosen for the subsequent calculations.

Table 2. Main characteristics of the used grids.

MESH	# OF CELLS	MAX CELL SIZE (m)	MAX SKEWNESS
1	5996	1	0.457
2	46413	0.50	0.457
3	383064	0.25	0.457

Transient simulations are performed with the PISO algorithm to couple pressure and velocity fields (Versteeg & Malalasekera, 2007). Strong streamline curvature of the flow is expected to happen at the bottom of the radial gate due to partial openings. The PRESTO interpolation scheme is used to capture the aforementioned flow pattern. Convective terms are approximated via second- order upwind schemes (SOU), due to its high precision and transportiveness. Simulation time was set to 20 seconds for the 100% opening, and 30 seconds for the 54% opening.

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

The total simulation time was set such that a fluid particle will cross the domain 98 times when the radial gate is completely opened, and 147 times for a 54% opening. Ultimately it will lead to a pseudo-steady state, and the results will be time-independent. This is in accordance with the definition of the advective time. The convergence criterion was set to $1x10^{-4}$ as minimum value, for all variables. A maximum of 60 iterations per time step was necessary to satisfy convergence.

The free surface was tracked by the VOF method, which models the motion of gas and liquid phases by solving a single set of conservation equations with shared physical properties (mixture properties). In the VOF method, the phases are treated mathematically as continuous phases, where each phase represents a continuous volumetric fraction in space and time. The sum of the volumetric fraction of the phases involved must be equal to unity. The value of the volumetric fraction is used to capture the interface. The volumetric fractions are allowed to vary between 0 and 1, being 0 or 1 the value for a cell that is completely filled with one of the specified phases. Otherwise, it will have intermediate values that represent the location of the free surface (Hirt & Nichols, 1981).

The VOF method, one of the available Euler-Euler approaches for multiphase modelling, is a surface-tracking technique applied to a fixed Eulerian mesh. It was formulated for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF method, a single set of momentum equations is shared by the fluids, and the volume fraction of each one of the fluids is calculated in every computational cell. In addition, the momentum exchange between the water and air phases needs to be accurately computed in order to estimate the air velocities and the necessary air demand. A VOF scheme with a single momentum equation will have the tendency to significantly overestimate the air velocity. However, the VOF computational cost is low compared to a full Eulerian model or a mixed model. Due to limitations on computing resources, the VOF method was used for this work.

The HRIC (high-resolution interface capturing) method was used to reconstruct the free surface. This method combines two schemes: Compressive and Geo-reconstruct methods, which avoid numerical diffusion when the flow direction is considered. The explicit model was used for the time discretization of the VOF method equations. When an explicit method is used, numerical stability of the solution must be guaranteed. This latter implies setting the Courant Number (CN) to 1 and using an adaptive time stepping, namely, the time steps are adjusted based on the maximum velocity of each cell. Table 3 summarize the setup of the simulated scenario. For the 54% opening, the parameters are the same, being only modified the solution time, from 20 s to 30 s.

Inlet boundaries are defined as ‘velocity inlet’ and the exit boundary as ‘pressure outlet’. A uniform velocity profile is set at the inlet section as the liquid phase completely fills the conduit; gage pressure is specified at the exit boundary as the flow discharge to the atmosphere. The domain is initialized for both phases. At the inlets, the domain is filled only with the liquid phase. On the other hand, downstream the gates, the domain is filled with *i)* liquid up to the wall height and *ii)* gas phase for the remaining volume. The no-slip condition was specified at all solid boundaries. The K-epsilon RNG model was used to estimate turbulence quantities, such as the turbulent kinetic energy and its dissipation rate.

For the initialization, turbulence intensity and hydraulic diameter were used. Field-measured data that was not available for this research is conventionally required to estimate the turbulence intensity at the inlet outlet. However, since the distance between the real inlet and the gate zone is 464 m, which is more than 50 times the tunnel’s hydraulic diameter, a fully developed flow can be assumed. Hence, a low-medium turbulence intensity was chosen for the initialization. The turbulence length scale is a physical quantity related to the size of large eddies, which contain the energy in turbulent flows. In a fully-developed flow (as the study flow in this research), the length scale is restricted by the size of the conduit, for the turbulent eddies cannot be larger than the conduit itself. Thus, a turbulence intensity of 3% and hydraulic diameter of 3.86 were used at the inlet boundaries to initialize the simulation. Standard wall functions were specified to reduce grid refinement at the walls.

In the simulation, a simple pressure boundary in all air vents was specified. At the pressure inlets the domain is filled only with the gas phase and gauge pressure equal to zero. While the air flow is always associated with a drop pressure, for this case the drop pressure will be associated with the recirculation region and a void just downstream the gates, in the step located downstream the radial gate, generated, most likely, by the detachment of water flow from the bottom wall. With the pass of the turbulence current, the air is driven through the air vents into the flow water. The velocity obtained in each aerator is integrated and multiplied by the cross section, making it possible to obtain a conservative data of the volumetric flow rate, and therefore, a preliminary value of the air demand. It is important to clarify that implementing an atmospheric boundary at the air vent resembles a completely loss free air vent, and this implementation on itself, leads to an overestimation of the air demand by the numerical model. Consequently, it will be required to make field measurements during the operation of the bottom outlet.

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

Table 3. 100% gate opening setup.

Discretization Scheme	Explicit
Turbulence Model	K-epsilon RNG
Solution Schemes	SOU
Volume fraction reconstruction	Modified HRIC scheme
Residuals	$1x10^4$
Courant Number	1
Solution time	20 s
Time step	Adaptive
Iterations by time step	60

4. Results

Figure 4 shows streamlines of the flow in the bottom outlet for 100% gate opening. At the end of the radial gates, a sharp deflection of the streamlines is observed at half of the channel height. It generates a recirculation region and a void just downstream the gates. Figure 5 shows a pressure iso-surface, where zones of sub-atmospheric pressure become visible in the step located downstream the radial gate, which may be due to the detachment of water flow from the bottom wall. For this operation, a sub-atmospheric pressure of 10 mwc it is estimated.

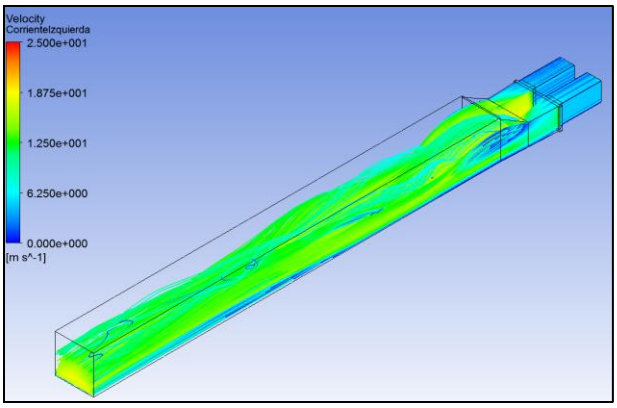

Figure 4. 3D view of the streamlines colored by the velocity magnitude of the 100% gate opening.

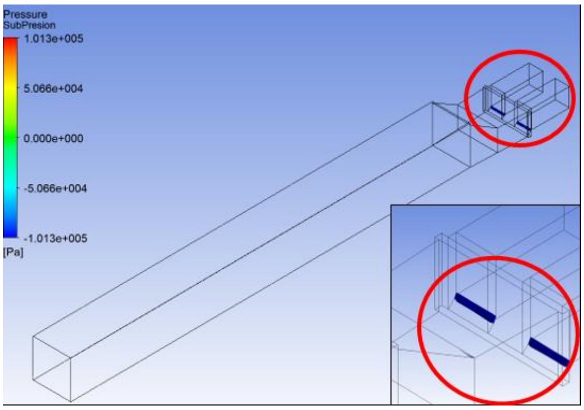

Figure 5. Maximum sub-pressure zone, downstream of the gate for a 100% opening.

Partial opening of radial gates is a critical operation scenario as flow velocities can be quite high. This operation may be induced by pressure drop to levels much lower than gates completely open. Volumetric fraction of the computed flow filed is shown in Figure 6. It can be seen that the velocity at the bottom of the free surface flow channel reaches values close to 35 m/s, and the streamlines are contracted at the end of the radial gate, which causes a decrease in the hydraulic depth and therefore a sudden increase in the velocity. For this partial opening, the free surface location is below the wall of the tunnel (4 m high), so the tunnel roof is not hit by the high-speed jet.

Flow vectors throughout each aerator for both simulated gate openings are shown in Figure 7 and Figure 8. For a 100% opening, it is observed that the air demand is low and the air velocity in the aerators is less than 5 m/s, since the flow velocity downstream the gate is small, leading to low sub-atmospheric pressures. However, for a 54% opening, the flow velocity increases considerably and so does the sub-atmospheric pressure. As a consequence, the air demand is higher and the air velocity throughout aerators reaches values up to 25 m/s. With the current results (just two openings, 100% and 54%) it is not possible to have a conclusive remark about the air distribution, so 4 additional cases were simulated to cover the operative range of radial gates of: 100%, 90%, 80%, 70%, 60% and 54%. The results are summarized in Table 4, where it is evident that for all analyzed cases, more than 90% of the air demand is supplied by the upper aerator. From the analysis of air demand distribution obtained for these gate openings and the mean air

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

velocity obtained for each aerator, the computed Mach number is less than 0.1, satisfying the incompressible flow assumption. The simulated air demand values are compared to those estimated values by the empirical formulations shown in Table 1. The comparison is shown in Table 5.

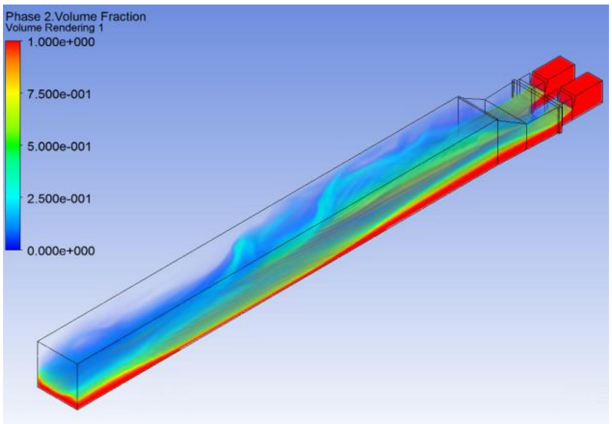

Figure 6. 3D view of the volumetric fraction of the 54% gate opening

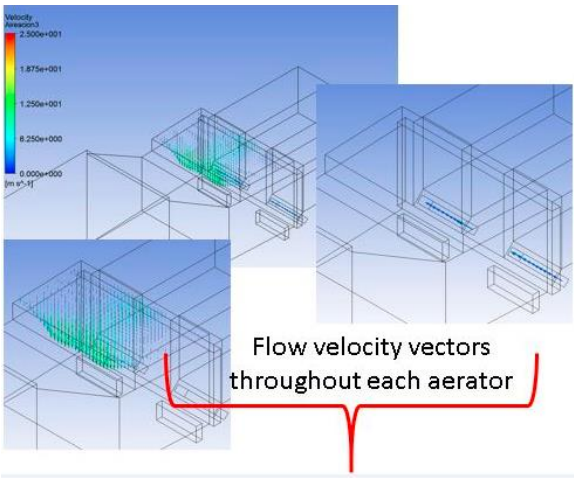

Figure 7. Flow velocity vectors throughout each aerator. 100% opening.

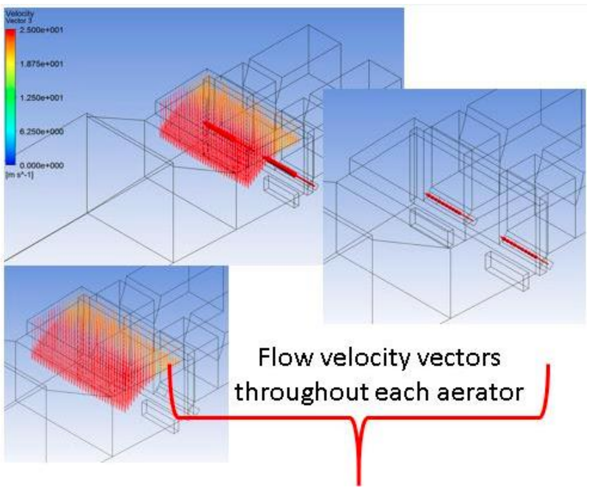

Figure 8. Flow velocity vectors throughout each aerator. 54% opening.

Table 4. Flow rate throughout each aerator for various radial gate openings.

Opening (%)	Pressure (mwc)	Flow rate A1 (m^3/s)	Flow rate A2 (m^3/s)	Flow rate A3 (m^3/s)
100.0	32.2	2.3	2.3	193.6
90.0	57.2	18.0	18.0	300.0
80.0	73.2	17.0	17.0	500.0
70.0	94.2	22.0	22.0	760.0
60.0	124.2	27.6	27.6	1056.0
54.0	147.2	30.0	30.0	1100.0

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

Table 5. Comparison between empirical estimated air demand and CFD estimated air demand.

Opening (%)	β Levin	β Ghetti & Di Silvio	β Sharma	β CFD
100.0	0.15	0.36	0.36	0.44
90.0	0.25	0.62	0.53	0.75
80.0	0.33	0.88	0.68	1.19
70.0	0.42	1.21	0.85	1.79
60.0	0.55	1.71	1.09	2.47
54.0	0.66	2.14	1.27	2.58

5. Conclusions

It can be concluded that the empirical model which most closely approximates the CFD results is the Ghetti & Di Silvio model. This empirical model has been validated in other projects in Colombia where there was a measure of the air demand in the prototype. In the San Carlos hydroelectric project (Villegas & Mejía, 1988), air measurements were recorded during the field measurements for two different gate openings (14% opening and 100% opening) with the same hydrostatic pressure of 66.5 mwc. It is valid to clarify that the gate of the bottom discharge of the San Carlos hydroelectric project is a sluice gate and came into operation in 1981, while the gate analyzed in this investigation is a radial gate and will start operating in July 2018.

From the summarized information shown in Table 4, it is possible to obtain a relationship between the Froude number at the "vena contracta" and the ratio of air flow to water flow. The correlation is shown in Figure 9. When analyzing the air demand curve calculated by CFD, we can conclude that all empirical correlations predict a steady increment in the air demand as the Froude numbers increases, whereas the correlation obtained by CFD exhibits a reduced slope for Froude numbers greater than 12.

With the calculated aeration based on the CFD simulation results, curve fitting leads to the following air demand equation:

$$\beta = 0.0572(F_c)^{1.484} \quad (3)$$

At this point, it is important to consider that the methodology followed to estimate the air demand integrates the velocity profile obtained in each aerator at its cross section, making it possible to approximate the volumetric flow rate, and therefore, a preliminary estimation of the air demand. Additionally, as a result of specifying a zero-gauge pressure boundary for all air vents in the CFD model, the air demand is overestimated in comparison with the results predicted by empirical models. Therefore, the air demand calculated by equation (3) provides a conservative value, so it exclusively applies for the analyzed case which will have to be validated with measurements "in-situ."

It is concluded that the curve shown in Figure 9 may be used to estimate the air demand only in the analyzed high head radial gate of Ituango hydropower project, as long as the Froude number in the "vena contracta" is less than 12.

A pure VOF scheme does not allow for air entrainment into the flow. It is possible to change the situation implementing a mixture model because this one differs from the volume of fluid in three aspects: *i)* the mixture model allows the phases to be interpenetrating; *ii)* the mixture model allows the phases to move at different velocities, using the concept of slip velocities; *iii)* there is interaction of the inter-phase mass, momentum and energy transfer (Eghbalzadeh & Javan, 2012). However, the computational cost is very high and for this research, the authors do not have the available required resources for that kind of simulation.

Part of the intention with this manuscript is to show the operators and the owners of the hydropower projects in Colombia, the relevance of the data and information that they possess. In Colombia, at the moment, only two works in bottom outlets have been made before. The first one is the mentioned in the manuscript written by (Villegas & Mejía, 1986), where they compare the analytical estimation of the air demand with the filed measurements and data taken at laboratory scale hydraulic model, but only for two gate openings. Finally, the second one is described in the present research, where the analytical and computational results, are expected to be compared against future field measurements.

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

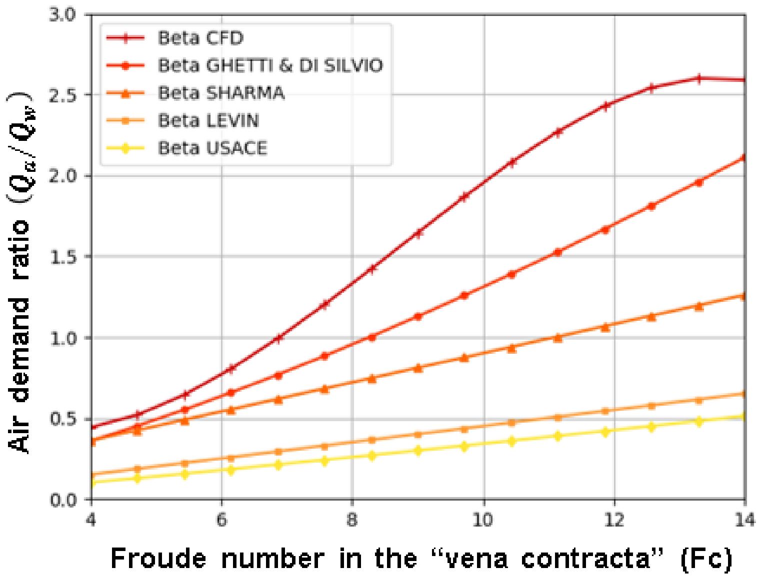

Figure 9. Correlation between air flow rate and Froude number at "vena contracta".

6. Acknowledgements

The authors wish to acknowledge the important contribution made by EPM (Empresas Públicas de Medellín), who facilitated and authorized the use of information about the Ituango hydroelectric project. Integral S.A and the authors were allowed to share the experiences, analyses, findings and all sort of data obtained in the design stage of the bottom outlet. Additionally, this work was conducted in collaboration with the professor Aldo Benavides and Francisco Toro.

7. References

Campbell, F.B. and Guyton, B. (1953) Air demand in gated outlet works. Proceedings of the 5th Congress of the International Association of Hydraulic Research, Minneapolis, Minnesota, USA, pp. 529-533.

Cihat Tuna, M., Ozkan, F., & Baylar, A. (2014). "Experimental investigations of aeration efficiency in high-head gated circular conduits". Water Science & Technology, 69(6), 1275

Eghbalzadeh, A., Javan, M (2012). "Comparison of Mixture and VOF models for Numerical Simulation of Air-entrainment in Skimming Flow over Stepped Spilways". CMHE International conference on Modern Hydraulic Engineering, Nanjing, China, 28,657-660

Ghetti, A., and G. Di Silvio. (1967). "Investigation on the running of deep gated outlet works from reservoirs". Proceedings of the 9th International Congress On Large Dams, Istanbul, Turkey, Vol. 2, Q33 (R48), pp. 837-852

Hirt, C.W. and Nichols, B.D. (1981) Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics, 39, 201-225Kalinske, A. A., and J. W. Robertson. (1943). "Closed conduit flow". Transactions of Symposium on Entrainment of Air in Flowing Water. ASCE, USA, pp. 1435-1447

Najafi, M. R., Kavianpour, Z., Najafi, B., Kavianpour, M. R., & Moradkhani, H. (2012). "Air demand in gated tunnels a Bayesian approach to merge various predictions". Journal of Hydroinformatics, 14(1), 152

Naudascher, E. (1991). Hydrodynamic forces. Hydraulic Structures design manual: Hydraulic Design considerations. The International Association for Hydraulic Research (IAHR). A.A. Balkema/Rotterdam/Brookfield

Sagar, B. (ASCE). (1995). ASCE, "Hydrogates task committee design for high-head gates". Journal of Hydraulic Engineering, 121, NO. 1(Paper No. 5480), 845–852

Salazar, F., Morán, R., Oñate, E., Toledo, M. A., & Riquelme, F. J. (2012). "Numerical modelling of air-water interaction in bottom outlets. Application to Susqueda Dam" *IAHR Iternational symposium on Hydraulic structures*

Sharma, H. R. (ASCE). (1976). AIR-ENTRAINMENT IN HIGH HEAD GATED COBDUITED. Journal of Hydraulic Engineering, 102(HY11), 1630–1645.USACE (United States Army Corps Of Engineers) (1964). Air demand-regulated outlet works. Hydraulic Design Criteria, Sheet 050-1/2/3, 211-1/2,255-1

ISBN: 978-0-692-13277-7 DOI: 10.15142/T3TW7D

Versteeg H.K, Malalasekera W, "An introduction to computational fluid dynamics. The finite volume method", Longman publications, 1995

Villegas, F. (ASCE), & Mejía, Ó. (ASCE). (1986). “Air Demand of High Head Sluices Gates: Model Prototype Comparison”. Anais, XII Congreso Latinoamericano de Hidráulica, IAHR, San Pablo, Brasil, 114, 396–407

DOI: UUUUUUUUUU