

When CFD adds value to tailings hydraulic design: Four practical cases

Leonardo Olavarria and Pablo del Rio

1. *Project Engineering División, Paterson & Cooke Chile, Santiago Chile*
2. *Process Engineering División, Paterson & Cooke Chile, Santiago Chile*

ABSTRACT

One-dimensional (1D) hydraulic design of tailings handling systems—Manning, h/D filling criteria, Froude number, and energy balances—forms the basis of engineering practice and efficiently sizes the global components of a typical hydraulic system. At certain nodes, however, the assumptions of the 1D approach break down: multi-branch diversion fittings, boxes with high-energy jets, variable cross-section transitions, and structures with air-tailings interaction generate phenomena that cannot be captured adequately by conventional analytical tools.

Four cases of computational fluid dynamics (CFD) application in real mining projects are presented, developed with OpenFOAM v12 in transient two-phase or multiphase mode: (i) gravitational discharge spigots in series on a 36" HDPE main pipe; (ii) a "Y"-type diversion fitting in 42" HDPE; (iii) energy dissipation dropboxes on a DN 800 HDPE inlet pipe; and (iv) a thickener distribution box in an existing operation. The methodology is common to all four cases.

Four differentiated roles of CFD as a complement to 1D analysis are identified, a metric complementary to static criteria is proposed to evaluate the temporal stability of discharges, and a practical guide is derived indicating when CFD should be activated, based on criteria such as the limits of the 1D model, the absence of a reference cross-section, or the need to assess flow distribution, transient behaviour, or design verification.

INTRODUCTION

Long-distance tailings transport and handling systems integrate multiple types of hydraulic infrastructure: gravity transport pipes, energy dissipation structures, distribution boxes, special diversion fittings, and discharge points to a tailings storage facilities (TSF). The conventional hydraulic sizing of each component is well documented and follows established criteria consistently applied by the industry (Adriasola & Janssen 2018, Errázuriz 1985).

In practice, this conventional framework rests on a small set of one-dimensional tools. Gravity flow in pipes and launders is sized with uniform-flow relations of the Manning type, verifying the normal flow depth and the filling ratio h/D_i against maximum design limits; the Froude number is checked to keep the operating point away from the near-critical, unstable regime; and energy balances between control sections support the sizing of drops and energy dissipation structures. These tools are robust and efficient precisely because they share the same underlying assumptions: steady, uniform flow, a well-defined reference cross-section, and a pressure distribution that is essentially hydrostatic.

At connection points between components and in irregular geometries, 1D analysis reaches its limits. Flow is no longer uniform, cross-sections vary, jets introduce kinetic energy that does not distribute symmetrically, and the interaction between the solid-liquid phase and air generates transient phenomena not capturable in a static balance. At these nodes, CFD provides information that 1D cannot deliver.

The application of CFD to tailings handling infrastructure has gained visibility in recent literature. Hoebel et al. (2023) documented a physical and CFD model for a tailings distribution box; Olavarria & Martinson (2025) developed a CFD correlation for dissipation depth in slurry boxes; Veloso, Martinson & Olavarria (2019) presented physical scale model results for a spigot system, with demonstrated consistency between CFD, physical model, and actual plant operation. There is no work in the specialist literature, however, that documents multiple types of infrastructure within a single methodological framework and makes explicit the differentiated roles of CFD according to the nature of the problem.

This paper fills that gap. Four CFD application cases developed in real mining projects are presented. The methodology section establishes the conceptual framework and common parameters; the four subsequent sections document each case with its context, central question, and key result; the synthesis extracts the methodological pattern; and the conclusions close with recommendations for the designer engineer.

METHODOLOGY

Conceptual framework: limits of 1D analysis and the role of CFD

1D hydraulic design is necessary and indispensable: it defines the global system parameters and provides the boundary conditions without which CFD has no meaning. CFD does not replace this basic analysis; it complements the nodes where its assumptions do not apply. Table 1 presents the conceptual map of the four nodes covered in this work.

Table 1 Nodes where 1D analysis is deficient and the role of CFD in each case

Critical node	Assumption 1D that breaks	Role of CFD	Case in this article
Open surface spigot operation	1D flow with no lateral distribution between branches	Quantify distribution + reusable regressions	Case 1 – Spigots
Variable transverse geometry ("Y" piece)	Uniform section for h/Di calculation	Verify h/Di where no reference section exists	Case 2 – Y fitting
Post-dropbox temporal stability	Static state; does not capture transient fluid dynamics	Reveal invisible instability	Case 3 – Dropboxes
Asymmetric distribution between parallel outlets	1D cannot assess the asymmetric problem	Diagnose geometric cause and compare alternatives	Case 4 – Box (operation)

CFD platform and parameters common to all cases

Table 2 summarises the common modelling parameters. Differences between cases—turbulence model, number of phases, mesh size, and simulation time—are documented in each section. The reliability of the CFD approach used for this class of problems has been validated in prior works through triangulation with physical scale models and real plant operational data (Veloso et al. 2019).

Table 2 Common parameters

Item	Description
Software CFD	OpenFOAM v12 / ParaView 5.11.2
Analysis type	Two-phase or multiphase (air + tailings); Eulerian formulation
Flow regime	Transient
Turbulence model	k-omega, k-omega SST or LES

CASE 1 – GRAVITATIONAL DISCHARGE SPIGOTS

System and problem

A 36" DR21 HDPE main pipe delivers combined tailings from two plants over the crest of an iron ore storage facility embankment, at 0.8% gradient. Groups of four discharge spigots (14" DR26 HDPE) are located along the pipe, connected via 36"→20" eccentric steel tees with 14" pinch valves. Combined flowrates—with froth factor 1.2—range from 1,523 m³/h (minimum, $C_w = 9.4\%$, $\rho = 1.11$ t/m³) to 4,546 m³/h (maximum, $C_w = 6.3\%$, $\rho = 1.03$ t/m³). The 1D analysis verifies h/D_i and velocities in the main pipe under uniform flow but cannot predict how much flow each spigot within a group discharges: flow tends to remain in the main pipe by inertia and extraction at each branch depends on the tee angle and local hydraulic conditions. Uneven distribution leads to unfulfilled storage filling plans and irregular beaches at the TSF.

CFD application and results

An individual spigot was simulated for four inlet flowrates (481 to 5,070 m³/h), with slope and rotation angle incorporated as components of the gravity vector. Mesh of ~480,600 elements; results at $t = 15$ s. Three angle combinations were evaluated (45°-50°-50°-50°, 28°-42°-42°-42°, and uniform 50°), and regressions of discharge and pass-through flowrate per spigot were built, reusable for any flowrate within the range without new simulations. Table 3 shows the resulting distribution for the optimal configuration.

Table 3 Resulting distribution for the optimal configuration

Scenarios	S1 (%)	S2 (%)	S3 (%)	S4 (%)	Variation S1–S4 (pp)
Minimum (1.523 m ³ /h)	32,3	27,9	23,8	16,0	16,3
Nominal (2.062 m ³ /h)	27,1	23,7	20,4	28,8	8,4
Maximum (4.546 m ³ /h)	18,5	16,8	15,0	49,7	34,7

The nominal scenario shows a variation of 8.4 percentage points between the lowest- and highest-discharging spigot, considered acceptable for the filling plan. In the maximum scenario, the fourth spigot accumulates nearly half of the total flow, attributable to the capacity limitation of the eccentric tee at high flowrates.

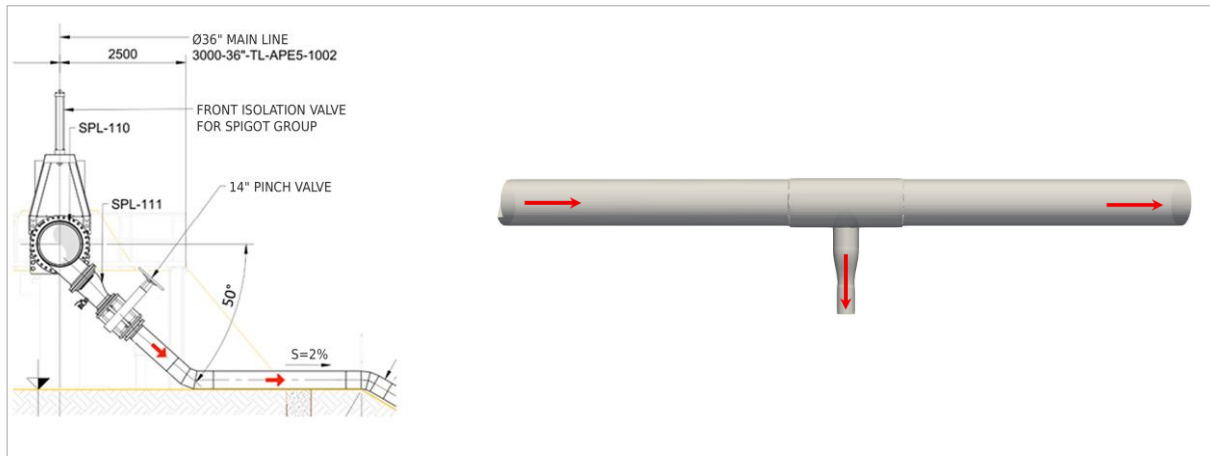


Figure 1 Spigot Geometry

CASE 2 – "Y"-TYPE DIVERSION FITTING

System and problem

A special 42" nominal diameter "Y" diversion fitting with eccentric reductions at inlet and outlets and 20 mm internal rubber lining allows flow from the 36" DR21 HDPE main line to be directed towards the embankment crest or the rockfill sector of the storage facility depending on the construction stage. Flowrates with froth factor 1.2 range from 1,523 m³/h (minimum) to 4,546 m³/h (maximum), with slurry densities between 1.03 and 1.11 t/m³. In the "Y" fitting the cross-section varies continuously: the 36" pipe transforms into a 42" body that bifurcates into two 36" branches. At the intermediate sections—the diversion expansion (section B) and the concentric reduction at the outlet (section C)—no normal flow depth is defined, and no 1D formulation is applicable, so the risk of local pressurization cannot be assessed analytically.

CFD application and results

Transient two-phase model with mesh of ~347,000 elements; three scenarios (minimum, nominal, maximum); four control sections (A: upstream; B: at the expansion; C: at the reduction; D: 3 m downstream); results at $t = 40$ s. Table 4 summarizes the h/D_i ratios.

Table 4 h/D_i ratios in control sections — "Y" branch piece (* sections B and C without reference normal height; h/D_i calculated directly from CFD)

Scenarios	Sec. A	Sec. B	Sec. C	Sec. D	h/D_i max.	Design Limit
Minimum (1.523 m ³ /h)	0,34	0,41	0,38	0,35	0,41	< 0,60

Nominal (2.062 m ³ /h)	0,41	0,47	0,45	0,41	0,47	< 0,60
Maximum (4.546 m ³ /h)	0,55	0,51	0,60*	0,59	0,60*	< 0,60

The maximum value is obtained at section C in the maximum scenario: $h/D_i = 0.60$, exactly at the normal operating criterion limit. The system maintains the free-flow condition. This result cannot be obtained with 1D analysis.

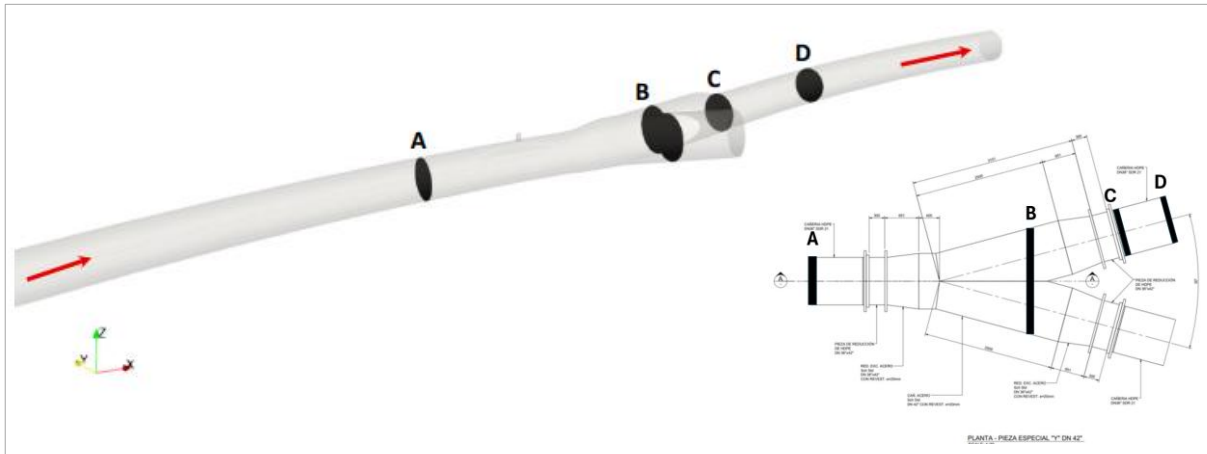


Figure 2 "Y" branch piece Geometry

CASE 3 — ENERGY DISSIPATION DROPBOXES

System and problem

Three dropboxes on a DN 800 SDR 17 HDPE pipe ($D_i = 705.2$ mm, slope 1.5%) dissipate energy at slope breaks in the alignment. The slurry has $\tau_y = 3.88$ Pa, $K_b = 0.0132$ Pa·s, density $1,450$ kg/m³ and $C_w = 50\%$ w/w. The design flowrate is 0.704 m³/s (2,536 m³/h), with $h_n/D_i = 0.475$ in the discharge pipe. The 1D dropbox sizing meets all conventional criteria: $H_c/H_t \approx 0.68 < 0.75$ (transition piece not submerged) and $h_n/D_i \approx 0.475 < 0.60$ (no pressurization). The 1D analysis, however, is static: it certifies that the system does not drown but does not characterise the temporal stability of the tailings volumetric fraction downstream of the transition piece, where the interaction between the falling jet and the air pocket generates transient fluctuations.

CFD application, proposed metric and results

The geometric modification in the solution case (1 m extension of the vertical discharge and the addition of the vent) optimizes the hydrodynamics of the system by confining the falling jet and releasing trapped air. This eliminates the "piston" effect and cyclic pulsations of the base case.

Both cases satisfy the 1D hydraulic design criteria equally well. However, the alternative involving the extended pipeline delivers superior performance in terms of flow stability and homogeneity.

Consequently, transient fluctuations in the tailings volumetric concentration are significantly reduced, converting an unstable flow regime into a quasi-steady and more homogeneous one. This improvement helps protect the HDPE pipeline from cyclic abrasive wear and minimizes the risk of unintended pressurization. These comparisons are shown in the following Figure 3:

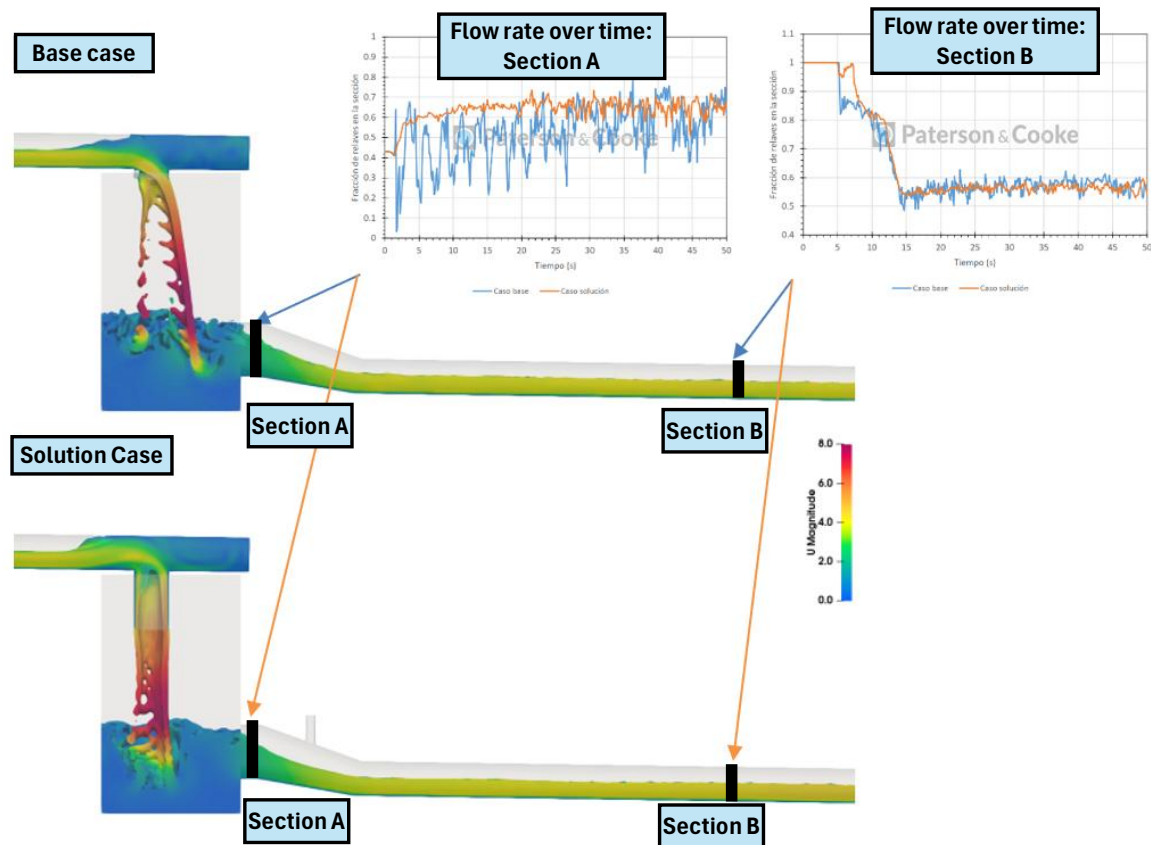


Figure 3 Energy dissipation dropboxes results

CASE 4 — THICKENER DISTRIBUTION BOX (EXISTING OPERATION)

System and problem

The distribution box receives dilute tailings from the flotation plant (42" feed pipe, flowrate 9,700 m³/h, density 1.213 t/m³, Cw ≈ 26.7% w/w) and distributes it to two conventional 122 m diameter thickeners. The discharge pipes to the thickeners are located asymmetrically with respect to the feed pipe. Operations had historically exhibited systematic differences in tonnage and concentration between both thickeners. The 1D design of the box can verify operating levels and velocities at the outlets, but cannot predict the flow distribution between the two pipes or determine what fraction of each particle size class (fine, coarse, integral) reports to each thickener. The cause of the asymmetry is an inherently three-dimensional problem.

CFD application and results

Multiphase model (4 phases: air, fine fraction, coarse fraction, integral tailings), LES turbulence, mesh of ~135,500 elements. The base case and three alternatives were analyzed: (1) 2.25 m box extension, (2) alignment of discharge pipes relative to the feed pipe, and (3) incorporation of a deflector. All alternatives include raising the intermediate weir threshold. Results averaged over $t = 30\text{--}50$ s.

The base case confirms the geometric asymmetry with ~9 percentage points difference between thickeners. Alternatives 1 and 3 invert the asymmetry without eliminating it. Alternative 2—alignment of the discharge pipes relative to the intermediate baffle—achieves a difference of 0.8 percentage points with homogeneous distribution of all three fractions, without requiring operational regulation. In Figure 4 are shown the models and results.

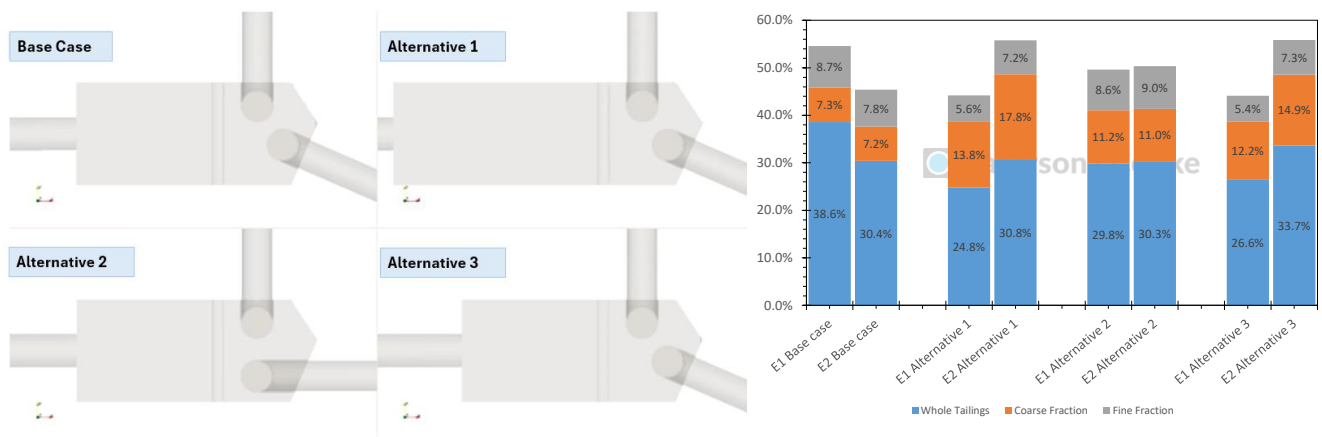


Figure 4 Percentage flow distribution by thickener (T1, T2) and phase: base case and three alternatives.

LIMITATIONS OF THE CFD APPROACH AND WAYS FORWARD

The four cases show where CFD adds value, but it is worth being clear about its scope. Each simulation depends on the rheological parameters, mesh, and turbulence model chosen for that case. Simulation times (15–50 s) were set to capture the phenomena of interest, not long-term operational cycles. Direct validation against physical scale models and plant data, as reported by Veloso et al. (2019), is available for Case 1. The other cases build on this same validated framework and on the team's accumulated modelling experience.

Two steps would strengthen this practice going forward. First, extending physical or field validation to other node types—for example, comparing dropbox behaviour against published experimental work such as Padulano, Del Giudice & Carravetta (2013) on vertical drop shafts. Second, developing reusable correlations for recurring geometries, following the approach already used for spigots in Case 1, to reduce turnaround time on future projects.

CONCLUSION

Four cases of CFD application as a complement to 1D hydraulic design in tailings handling systems were documented. The main conclusions are:

CFD does not replace 1D hydraulic modelling. Rather, 1D analysis remains an essential and indispensable component of the whole design process, while CFD provides additional value when applied selectively to locations where the underlying assumptions of the 1D approach are no longer valid.

Four distinct roles of CFD can be identified: (i) quantification of flow distribution through the development of reusable regression-based correlations; (ii) extension of design verification to variable cross-sectional geometries for which analytical formulations are unavailable; (iii) identification of dynamic flow behaviour that remains undetected in conventional steady-state analyses; and (iv) independent verification of designs, enabling iterative refinement and risk reduction.

A practical framework for implementing CFD is proposed. CFD should be considered when the operating point approaches the limits of applicability of the 1D model, when the system incorporates variable-section geometries lacking analytical solutions, when the engineering question concerns flow distribution or transient behaviour, or when an independently developed design requires verification prior to capital commitment.

Complex hydraulic structures require a specialized team of engineers that has been involved in several designs providing a straight forward view for the best approach to conduct at each design phase.

ACKNOWLEDGEMENTS

The authors thank the engineering and operations teams of the described projects for their collaboration and for facilitating access to technical information. The methodological development documented herein is the result of accumulated work by the process engineering team at Paterson & Cooke Chile.

NOMENCLATURE

C _w	Mass solids concentration (–)
d ₅₀	Particle diameter at 50% passing cumulative (μm)
D _i	Internal diameter pipe (m)
H _c	Critical energy on the Dropbox transition piece (m)
h _n	Normal flow depth in the pipe (m)

Ht	Total height of the transition piece (m)
Q	Volumetric flow rate (m ³ /s o m ³ /h)
τ_y	Bingham yield strength (Pa)
K _b	Bingham plastic viscosity (Pa·s)
ρ	Pulp density (t/m ³)

REFERENCES

Errázuriz, PP (1985) Estudio experimental de flujo sólido-líquido a alta concentración en canales, Tesis de Magíster, Pontificia Universidad Católica de Chile, Santiago.

Adriasola, JM & Janssen, RHA (2018) 'Best practices for design of slurry flow distributions', 7th IAHR International Symposium on Hydraulic Structures, Aachen, Germany, 15–18 May.

Hoebel, D; Concha, G; Martinson, R; Jacobs, J & Flores, JL (2023) 'Tailings distribution box scale modeling', Tailings 2023: Proceedings of the 9th International Conference on Tailings Management, Gecamin, Santiago.

Olavarria, L & Martinson, R (2025) 'Mathematical correlation for determining required depth to dissipate inflow in a mineral slurry box by using CFD', Tailings 2025: Proceedings of the 11th International Conference on Tailings Management, Gecamin, Santiago.

Padulano, R; Del Giudice, G & Carravetta, A (2013) 'Experimental analysis of a vertical drop shaft', Water, vol. 5, no. 3, pp. 1380–1392, doi:10.3390/w5031380.

Veloso, C; Martinson, R & Olavarria, L (2019) 'Hydraulic scale modeling of gravitational tailings spigot discharge', Tailings 2019: Proceedings of the 7th International Seminar on Tailings Management, Gecamin, Santiago.