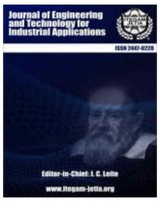
Journal of Engineering and Technology for Industrial Applications

ITEGAM-JETIA

Manaus, v.12 n.58, p. 381-388. March/April, 2026.
DOI: https://doi.org/10.5935/jetia. v12i58.3072

ISSN ONLINE: 2447-0228



RESEARCH ARTICLE OPEN ACCESS

IMPACT OF ROUGHNESS ON THE EMPTYING TIME OF A TANK: EXPERIMENTAL AND NUMERICAL ANALYSIS

Belaid Larbi[1], Bessanane Nabil[2] and Tebbi Fatima Zohra[3]

[1]Laboratory of Research Laboratory in Applied Hydraulics, Department of Hydraulics, Faculty of Technology, University of Batna 2 – Mostefa Ben Boulaid, Batna 05000, Algeria,
[2]Laboratory of Studies of Industrial Energy Systems, Department of Mechanical Engineering, Faculty of Technology, University of Batna 2 – Mostefa Ben Boulaid, Batna 05000, Algeria
[3]Natural Hazards and Territory Planning Laboratory (LRNAT Laboratory), Geography and Territory Planning Department, Earth Sciences and Universe Institute, University of Batna 2- Mostefa Ben Boulaid , Batna, Algeria,

[1]https://orcid.org/0009-0005-1710-947X, [2]https://orcid.org/0000-0002-5211-2763, [3]https://orcid.org/0000-0003-3321-5609

Email: l.belaid@univ-batna2.dz, n.bessanane@univ-batna2.dz, f.tebbi@univ-batna2.dz

ARTICLE INFO

Article History
Received: December 6, 2025
Reviewed: January 8, 2026
Accepted: January 15, 2026
Published: March 31, 2026

Keywords:
TERMS—CFD,
emptying time,
free-surface,
roughness,
tank,
VOF

ABSTRACT

This study investigates the hydrodynamic influence of surface roughness on the discharge time of free-surface tanks through a combined experimental and numerical approach. The research evaluates the mechanisms by which varying roughness magnitudes alter flow resistance, turbulence modulation, and energy dissipation during the drainage process. Experimental data were obtained from a tank with controlled roughness parameters, providing validation for Computational Fluid Dynamics (CFD) simulations performed using ANSYS FLUENT. The Volume of Fluid (VOF) method was employed to resolve the multiphase flow dynamics. Results indicate a distinct inverse correlation between surface roughness and emptying time. Specifically, the introduction of surface roughness (5–15 mm) reduced discharge duration by up to 4.67%, with the most pronounced efficiency gains observed within the lower roughness interval (0–6 mm). This phenomenon is attributed to the modification of near-wall flow structures and the reduction of laminar flow resistance. The study validates the accuracy of the numerical framework in predicting complex fluid behaviors and underscores the critical role of surface roughness optimization in enhancing the design and operational efficiency of hydraulic systems.

I. INTRODUCTION

The gravitational discharge of liquids from tanks and reservoirs is a fundamental phenomenon in fluid mechanics with extensive applications in chemical processing, civil engineering, and hydraulic system design. The accurate prediction of emptying time is critical for optimizing operational efficiency and ensuring safety in industrial processes. Historically, the theoretical basis for tank drainage has been established through classical Bernoulli equations and Torricelli's law. By[1] revisited these foundational equations to assess their practical applicability, while [2]provided comprehensive pedagogical models for the filling and emptying cycles. Further theoretical refinements have been proposed by [3], who analyzed steady-state gravity-driven inviscid flows, and [4], who focused on mathematical modeling for tank emptying scenarios.

However, real-world applications often deviate from idealized theoretical conditions due to complex fluid properties and discharge geometries, as reviewed by [5] regarding the discharge of complex fluids through orifices. With the advent of high-performance computing, Computational Fluid Dynamics (CFD) has become an indispensable tool for analyzing these complex hydraulic flows. As detailed in foundational texts by [6] and [7], numerical methods allow for the resolution of the Navier-Stokes equations in domains where analytical solutions are intractable. In turn [8] highlights the growing reliance on CFD for modern hydraulic analysis, while [9] emphasize its utility in water resources engineering. Consequently, numerical simulation has been widely adopted to study specific tank flow phenomena.

For instance, [10] utilized OpenFOAM to simulate flow behavior in hydropower storage tanks, and [11] applied CFD to investigate circulation patterns and geometric optimization in aquaculture tanks. Similarly, [12] explored mixing characteristics in tanks with special geometries, and [13] reviewed Navier-Stokes approaches for oscillating water columns. A significant portion of recent literature has focused on the complexities of multiphase flows and free-surface dynamics during the emptying process. The formation of vortices at the outlet, which can significantly retard flow, has been numerically analyzed by [14]. Furthermore, the accurate resolution of the air-water interface is crucial; [15] investigated turbulent free-surface flows experimentally, while [16] examined free-surface effects on flow around obstacles.

Advanced modeling techniques have also been employed to address these challenges, including the data-enhanced model predictive control proposed by [17] and deep learning-based surrogate modeling for reservoir pressure dynamics by [18]. Additionally, [19] provided comparative studies on gravity-driven discharge involving complex pore networks. Despite these advances, the specific influence of wall roughness on the hydrodynamics of tank emptying remains an area requiring further investigation. The fundamental effects of surface roughness on boundary layers and turbulence were established in the seminal works of [20] and [21]. In modern CFD, turbulence modeling remains a central challenge, as discussed by [22], with various approaches proposed to improve accuracy, such as the modifications to Prandtl mixing length theory by [23].

While roughness is known to increase drag in external flows—as demonstrated by [24] using smoothed particle hydrodynamics—its role in internal gravity-driven tank drainage is nuanced. This study addresses this gap by investigating the impact of surface roughness on the emptying time of a free-surface tank. Unlike traditional assumptions where roughness merely acts as a resistance factor, preliminary observations suggest that specific roughness intervals may alter turbulence structures in a manner that expedites discharge. By integrating experimental validation with numerical simulations using ANSYS FLUENT and the Volume of Fluid (VOF) method, this research aims to quantify the correlation between roughness height and discharge efficiency. Specifically, this paper analyzes the flow resistance, turbulence development, and energy dissipation across varying roughness conditions to determine optimal design parameters for hydraulic engineering applications.

II. METHODOLOGY

II.1 MATERIALS

II.1.1 Tank Geometry and Roughness

The experimental facility comprised a square, prismatic cast-iron reservoir with a maximum capacity of 900 L. To facilitate precise monitoring of discharge dynamics, the tank volume was calibrated in 100 L increments relative to vertical fluid depth. As illustrated in Figure 1, the study evaluated four distinct wall surface configurations to quantify the influence of relative roughness. The baseline configuration utilized the tank's polished interior walls to establish a smooth surface condition. To simulate varying degrees of roughness, three specific configurations were generated by applying gravel particles with mean diameters of 5.15 mm, 11.25 mm, and 15.00 mm, respectively. To ensure geometric consistency and hydraulic uniformity, the aggregates were mechanically sieved and securely bonded to the interior surfaces using a high-strength adhesive. This preparation method ensured a homogeneous distribution of roughness elements across all wetted surfaces, thereby minimizing localized flow anomalies.

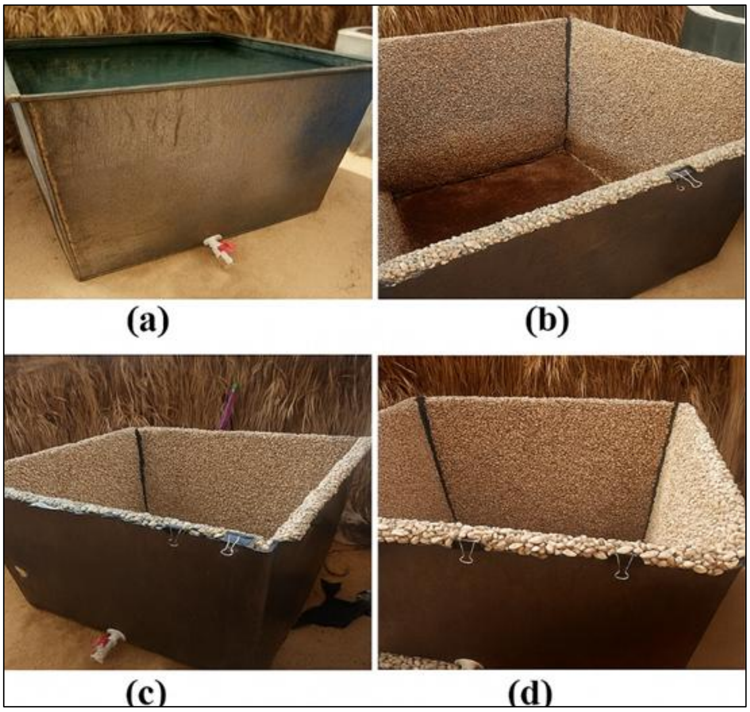


Figure 1: Tanks with different roughness(a) smooth; (b) 5.15 mm; (c) 11.25 mm; (d) 15 mm.
Source: Authors, (2026).

II.1.2 Numerical Procedure

The numerical study of three-dimensional unsteady flow was conducted using ANSYS FLUENT to analyze the influence of wall roughness on tank emptying dynamics. The Volume of Fluid (VOF) method was employed to model the multiphase flow interface (air–water), capturing the transient behavior of the free surface during emptying. The square tank geometry, shown in Figure 2, was fully reproduced based on the experimental dimensions (900 L capacity), ensuring direct comparability. A 3D computational domain was modeled, including the inner walls (with varying roughness), the free-surface zone and the tank outlet, to simulate the two-phase (air–water) interface transitions and flow hydrodynamics during emptying.

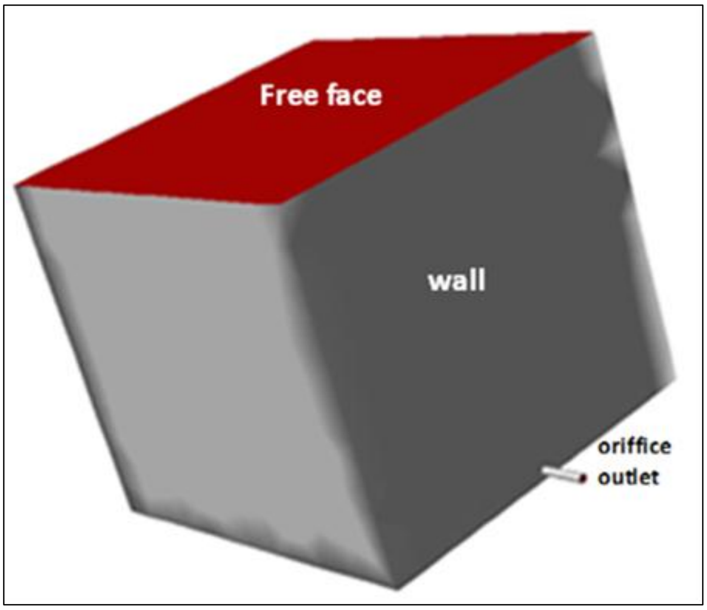


Figure 2: Geometry of tank.
Source: Authors, (2026).

An unstructured tetrahedral mesh was generated with localized refinement near the tank walls, outlet and free-surface zone to resolve boundary layer effects and interfacial dynamics (see Figure 3). Mesh independence was verified through sensitivity analysis, progressively reducing cell size until key results (e.g., emptying time, velocity profiles) varied by less than 2 %. The final mesh had a maximum cell size of 5 mm in the overall flow zone, with cells adjacent to the walls ≤ 1 mm (three inflation layers, growth rate of 1.2).

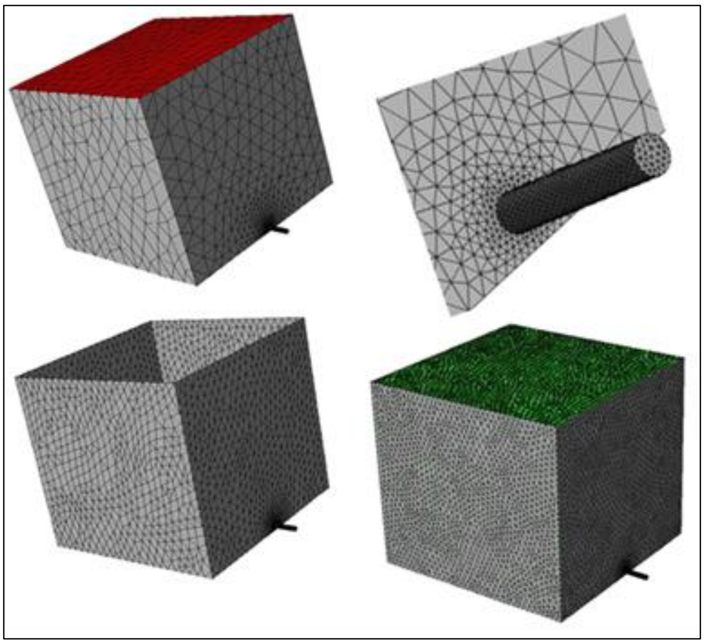


Figure 3: Meshes of the studied configuration.
Source: Authors, (2026).

The mesh contained approximately 1.2 million cells, with skewness < 0.85 and aspect ratio < 3.0, ensuring numerical stability. It accurately resolved roughness characteristics (gravel from 5.15 mm to 15.00 mm) through cell sizing near the walls proportional to the aspect ratio. The mesh effects are summarized in table 1, with errors less than 2.6 % for calculated quantities (fluid depth and outlet velocity) at 100 s in the smooth tank.

Table 1: MESH EFFECTS.

Mesh	Coarse	Medium	Error %	Fine	Error %
Depth	0,819	0,810	1,1%	0,799	2,6%
Outlet Velocity	2,986	3,005	0.63%	2,988	0.56%

Source: Authors, (2026).

The boundary conditions, summarized in table 2, were defined to replicate the experimental hydrodynamics of tank emptying while simplifying the treatment of the air phase. The outlet was set as a pressure boundary at atmospheric pressure (101 325 Pa) with backflow disabled. The tank walls used a no-slip condition with sand-grain roughness heights (0 mm to 15.00 mm) and a roughness constant (Cs = 0.5) to replicate experimental gravel surfaces. The air–water interface was tracked using the VOF method, incorporating surface tension (0.072 N/m) while neglecting air phase velocity due to its minimal impact. Turbulence was modeled using the RNG k–ε model with wall functions (y+ ≈ 30), ensuring consistency with the observed roughness-modified flow physics.

Table 2: BOUNDARY CONDITIONS SUMMARY.

Boundary	Type	Setting	Physical Basis
Outlet	Pressure outlet	P = 1 atm, no backflow	Torricelli's theorem
Walls	No-slip	Sand-grain roughness gravel diameter ks = gravel diameter	Prandtl's mixing-length theory
Free surface	VOF interface	Surface tension = 0.072 N/m	Continuum Surface Force (CSF) model

Source: Authors, (2026).

The numerical simulation of tank emptying dynamics is governed by fundamental fluid mechanics principles, including mass conservation, momentum balance, turbulence modeling and multiphase flow tracking. Below are the key equations and their physical significance in this study:

For an incompressible fluid (constant density (ρ), the continuity equation ensures the conservation of mass:

$$\partial\rho/\partial t+\nabla\cdot(\rho V)=0\Rightarrow\nabla\cdot V=0 \tag{1}$$

The Navier–Stokes equations describe momentum conservation for incompressible, viscous flow:

$$\partial V/\partial t+(V\cdot\nabla)V=-1/\rho\ \nabla P+\nu\nabla^2 V+F \tag{2}$$

In the context of tank emptying, volumetric forces play a key role in accelerating the fluid through the orifice: the pressure gradient (P), viscous effects (ν∇2V) and gravity (F = g). The RNG (Re-Normalization Group) k-ϵ model was employed to simulate turbulence effects during tank emptying, offering improved accuracy for flows with high strain rates and streamline curvature compared to the standard k-ϵ model. The RNG model modifies the turbulence dissipation equation and introduces an analytical derivation of model constants, enhancing predictions for transitional and rough-wall flows. Turbulent Kinetic Energy (k)

$$\partial(\rho k)/\partial t+\nabla\cdot(\rho kV)=P_k-\rho\varepsilon+\nabla\cdot[(\mu+\mu_t/(\sigma_k^{RNG}))\nabla k] \tag{3}$$

Dissipation rate (ε)

$$((\rho\varepsilon))/\partial t+\nabla\cdot(\rho\varepsilon V)=C_1^{RNG}\ \varepsilon/k\ P_k-C_2^{RNG}\ \rho\ \varepsilon^2/k+\nabla\cdot[(\mu+\mu_t/(\sigma_\varepsilon^{RNG}))\nabla\varepsilon]-R_\varepsilon, \tag{4}$$

Were:

$$R_\varepsilon=(C_\mu^{RNG}\rho\eta^3(1-\eta/\eta_0)^{(\varepsilon^2)})/(1+\beta\eta^3),\eta=k/\varepsilon\sqrt{(2S_{ij}S_{ij})},\mu_t=\rho C_\mu^{RNG}\ k^2/\varepsilon_e \tag{5}$$

The VOF model is used to track the evolution of the interface between water and air in the tank during emptying. It relies on an indicator function () defining the volume fraction of a fluid in each cell of the numerical mesh:

$$\partial\alpha/\partial t+V\cdot\nabla\alpha=0 \tag{6}$$

Were:

- $\alpha=1$: full presence of the fluid (e.g., water).
- $\alpha=0$: absence of the fluid (e.g., air).
- $0<\alpha<1$: Interface between the two fluids.

The Navier–Stokes equations are adapted for multiphase flow:

$$\frac{\partial(\rho u)}{\partial t}+\nabla\cdot(\rho uu)=-\nabla p+\nabla\cdot(\mu\nabla u)+\rho g+F_s, \tag{7}$$

$$\frac{\partial(\rho u)}{\partial t}+\nabla\cdot(\rho uu)=-\nabla p+\nabla\cdot(\mu\nabla u)+\rho g+\mathrm{F}_s \tag{8}$$

Where the density is interpolated according to α :

$$\rho=\alpha\rho_{cau}+(1-\alpha)\rho_{air} \tag{9}$$

The viscosity follows a similar interpolation:

$$\mu=\alpha\mu_{cau}+(1-\alpha)\,\mu_{air} \tag{10}$$

The surface tension force Fs is modeled with the CSF (Continuum Surface Force):

$$F_s=\sigma k\nabla\alpha \tag{11}$$

With k, the curvature of the interface, defined by:

$$k=\nabla\cdot(\nabla\alpha/|\nabla\alpha|) \tag{12}$$

III. RESULTS AND DISCUSSIONS

The results show an inverse relationship between surface roughness and emptying time, with increased roughness leading to faster emptying. Specifically, smoother surfaces exhibit longer emptying times, whereas rougher surfaces significantly reduce them. The most pronounced reduction occurs at lower roughness levels (0 – 6.00 mm), with diminishing effects beyond 10.00 mm. Rougher surfaces, such as 15.00 mm gravel, promote a more fragmented interface and faster free-surface descent compared to smooth walls. This behavior is attributed to enhanced turbulent mixing induced by surface roughness, which disrupts laminar boundary layers and accelerates emptying. Increased turbulence promotes boundary layer separation, reducing wall adherence. These findings align with predictions from the RNG k–ε model, confirming higher energy dissipation near rough walls. The VOF method effectively captures the air–water interface dynamics during tank emptying, as shown in (Fig. 4).

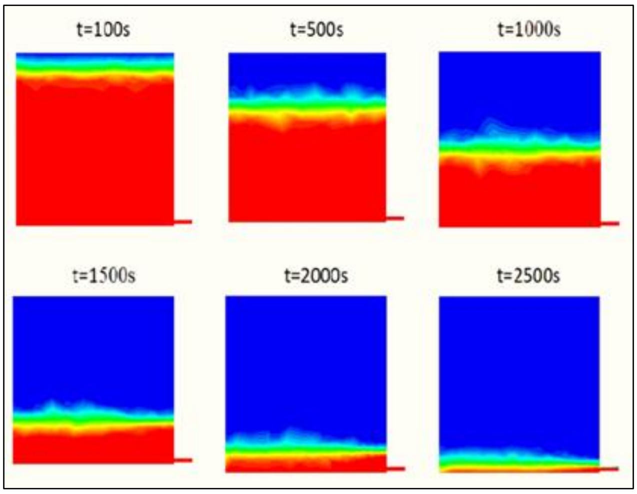


Figure 4: Contours of volume fractions of fluids at different times (s).
Source: Authors, (2026).

As depicted in (Fig 5), the steepest reduction in emptying time (up to 4.67 %) occurs at lower roughness levels (0 –6.00 mm), with a “roughness threshold effect” suggesting diminishing returns beyond 10.00 mm, likely due to fully developed turbulence. This matches the RNG k–ε model’s strain-rate term for high-roughness flows.

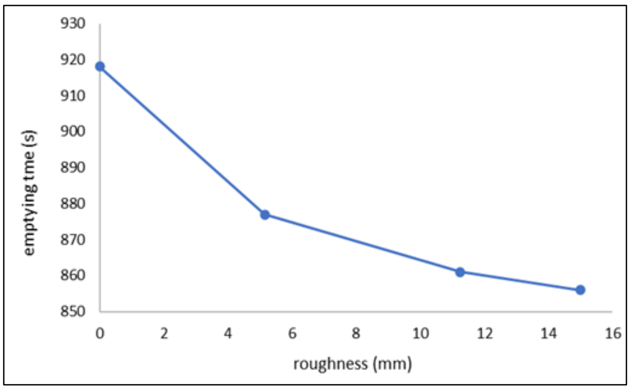


Figure 5: Experimental variation of tank emptying time as a function of wall roughness.
Source: Authors, (2026).

The numerical framework shows strong agreement with experimental data (Fig 7), particularly for roughness levels of 5.15–11.25 mm, with deviations < 3 %. A slight underestimation of emptying times at low fluid heights ((h < 0.2,), Figure 10) suggests limitations in modeling the air–water interface. The RNG k–ε model accurately reproduces the roughness threshold effect, as shown in (Fig 6).

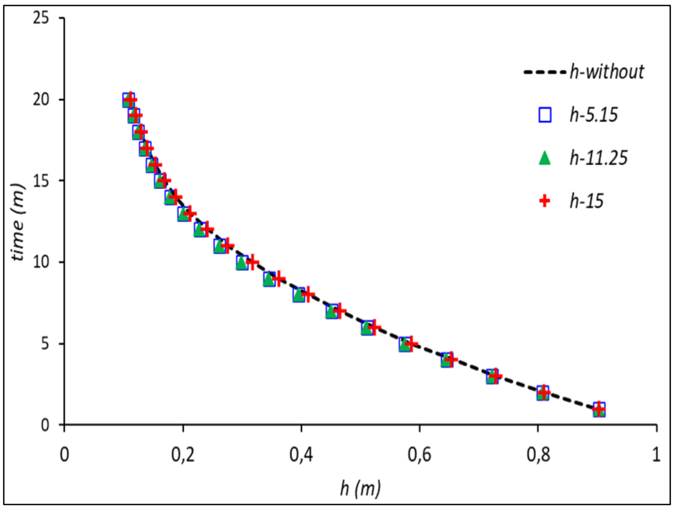
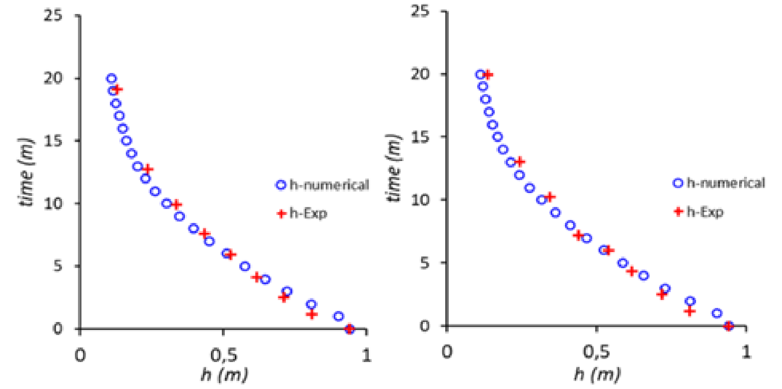


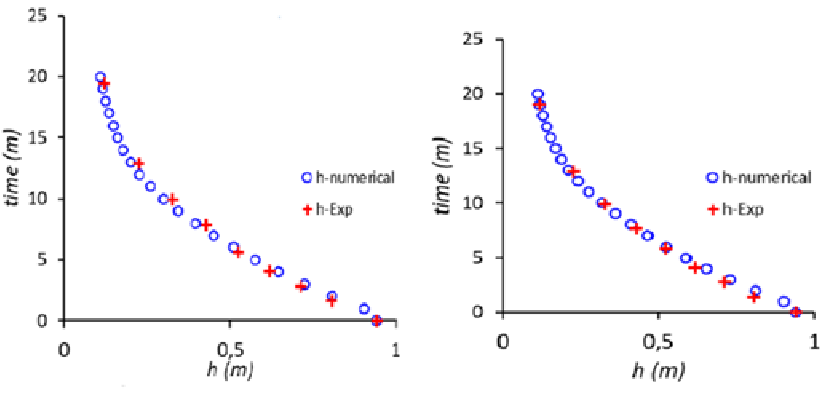
Figure 6: Variation of emptying time versus the free-surface level for different roughness values.
Source: Authors, (2026).

(Fig 7) validates the numerical results against experimental data for all roughness conditions, showing deviations < 5 %. Minor discrepancies at late-stage emptying (low fluid heights) may result from unmodeled air entrainment or VOF limitations in resolving micro-scale bubbles. The RNG k–ε model's accuracy is confirmed for rough-wall flows. This figure also plots the relationship between fluid height (h) (in meters) and time (t) (in minutes). Experimental data ((h)–Exp), represented by red (+) signs, closely match numerical results ((h)–Numerical), shown as blue circles. Both datasets follow a non-linear decreasing curve, indicating that emptying time decreases rapidly as fluid height increases. The close agreement reflects the accuracy of the numerical simulations, with minor differences likely due to experimental errors or simplifications in the mathematical model.



(a) Smooth Tank

(b) Roughness of 5.15 mm



(c) Roughness of 11.25 mm

(d) Roughness of 15 mm

Figure 7: Comparison of experimental ((h)–Exp) and numerical ((h)–Numerical) fluid height versus time for different roughness conditions: (a) smooth tank, (b) roughness of 5.15 mm, (c) roughness of 11.25 mm, (d) roughness of 15.00 mm.
Source: Authors, (2026).

The graphs demonstrate excellent agreement between experimental and numerical results across all four roughness conditions, validating the accuracy of the numerical model employed. Total drainage time decreases with increasing roughness, dropping from approximately 20 seconds for the smooth tank to about 18 seconds at 15 mm roughness. All curves follow a typical nonlinear behavior characteristic of gravity-driven tank drainage. The deviation between experimental and numerical values diminishes at higher roughness levels, indicating improved simulation accuracy under increased turbulence conditions.

IV. CONCLUSIONS

This study quantified the influence of wall roughness on the discharge kinetics of free-surface tanks through a synergistic approach combining experimental validation and Computational Fluid Dynamics (CFD). The findings demonstrate a distinct non-linear relationship where increased surface roughness significantly accelerates the emptying process. This effect is most pronounced within the lower roughness regime (0–6.00 mm). Conversely, a critical threshold was identified beyond 10.00 mm, where the influence of roughness on flow enhancement plateaus due to the onset of fully developed turbulence.

The numerical framework, utilizing the Volume of Fluid (VOF) method and the RNG k–ε turbulence model, exhibited high fidelity in reproducing experimental trajectories, achieving a deviation margin of less than 5%. However, discrepancies were noted at low fluid depths ($h < 0.2$), attributed to the limitations of the VOF method in resolving complex air entrainment and micro-scale bubble dynamics during late-stage drainage. Collectively, these results underscore the potential of surface roughness as a passive control parameter for optimizing hydraulic system design. Future research should prioritize advanced multiphase modeling to resolve interface dynamics at shallow depths and expand the roughness topology spectrum to further characterize the identified turbulence threshold.

V. AUTHOR'S CONTRIBUTION

Conceptualization: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Methodology: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Investigation: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Discussion of results: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Writing – Original Draft: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Writing – Review and Editing: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Resources: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Supervision: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.
Approval of the final text: Belaid Larbi, Bessanane Nabil and Tebbi Fatima Zohra.

VI. REFERENCES

[1]Joye, D. D., & Barrett, B. C. (2003). The Tank Drainage Problem Revisited: Do These Equations Actually Work? The Canadian Journal of Chemical Engineering, 81(5), 1052–1057.

[2]Otto, J., & Mungan, C. E. (2022). Filling and emptying a tank of liquid. European Journal of Physics, 43(5), 055003.

[3]Adeeyo, O. A., Adefila, S. S., & Ayeni, A. O. (2023). Dynamics of steady-state gravity-driven inviscid flow in an open system. International Journal of Innovative Research and Scientific Studies, 6(1), 80–88.

[4]Gálvez, L. F. P. (2017). Modelamiento Matemático Para Vaciado De Tanques. Scientia Et Technica, 22(1), 89–94.

[5]Teoman, B., Potanin, A., & Armenante, P. M. (2022). The discharge of complex fluids through an orifice: A review. Chemical Engineering Research and Design, 179, 346–364.

[6]Anderson, J. D. (2017). Computational Fluid Dynamics: The Basics with Applications. McGraw-Hill Education.

[7]Versteeg, H. K., & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd ed.). Pearson.

[8]Ortloff, C. R. (2024). Engineering and Modeling of Water Flow via Computational Fluid Dynamics (CFD) and Modern Hydraulic Analysis Methods. Water, 16(21), 3086.

[9]Costanzo, C., Padulano, R., & Caloiero, T. (2022). Advances in Flow Modeling for Water Resources and Hydrological Engineering. Hydrology, 9(12), 228.

[10]Achig, R. S., Hidalgo, V., Simbaña, S., & Gavilanes, J. (2024). Simulation of the Flow Behavior in the Storage Tank of a Hydropower Using OpenFOAM. Systems, Smart Technologies and Innovation for Society, Cham.

[11]Mohamadi, M., Islami, H. R., Ezam, M., & Mousavi, S. (2022). Study of geometric shape and water circulation pattern in fish tanks using computational fluid dynamics. Journal of Aquaculture Development, 16(3), 91–102.

[12]Tarcsay, B. L., Egedy, A., Bobek, J., & Rippel-Pethő, D. (2019). Investigation of Mixing in Tanks of a Special Geometry. Hungarian Journal of Industry and Chemistry, 47(2), 63–70.

[13]Opoku, F., Uddin, M. N., & Atkinson, M. (2023). A review of computational methods for studying oscillating water columns – the Navier-Stokes based equation approach. Renewable and Sustainable Energy Reviews, 174, 113124.

[14]Mohseni, M., & Domfeh, M. K. (2023). Numerical Analysis of Transient Vortex Formation at the Outlet of a Tank Containing Gas-Liquid Phases. Journal of Applied Fluid Mechanics, 16(11), 2235–2248.

[15]Rousseau, G., & Ancey, C. (2022). An experimental investigation of turbulent free-surface flows over a steep permeable bed. Journal of Fluid Mechanics, 941, A51.

[16]Zhao, F. et al. (2024). Free-surface effects on the flow around two circular cylinders in tandem. Journal of Fluid Mechanics, 1001, A7.

[17]Scheurenberg, D., Schmerling, K., & Abel, D. (2022). Data Enhanced Model Predictive Control of a Coupled Tank System. Proc. 2022 IEEE/ASME Int. Conf. Advanced Intelligent Mechatronics (AIM), 1646–1651.

[18]Saberali, B. et al. (2023). Simulating dynamics of pressure and fluid saturation at grid-scale by a deep learning-based surrogate reservoir modeling based on a fast-supply hybrid database and developing preliminary insights for future gas hydrate exploitations in China. Geoenergy Science and Engineering, 222, 211415.

[19]Balankin, A. S. et al. (2018). Comparative study of gravity-driven discharge from reservoirs with translationally invariant and fractal pore networks. Journal of Hydrology, 565, 467–473.

[20] Nikuradse, J. (1950). Laws of Flow in Rough Pipes. VDI-Forschungsheft 361. Translated in NACA Technical Memorandum 1292.

[21]Schlichting, H., & Gersten, K. (2017). Boundary-Layer Theory (9th ed.). Springer.

[22] Wilcox, D. C. (2006). Turbulence Modeling for CFD (3rd ed.). DCW Industries.

[23]Luo, X., Liu, P.-l., & Luo, H.-a. (2008). Improvement of Prandtl mixing length theory and application in modeling of turbulent flow in circular tubes. Journal of Central South University of Technology, 15(6), 774–778.

[24]Korzani, M. G. et al. (2017). Parametric study on smoothed particle hydrodynamics for accurate determination of drag coefficient for a circular cylinder. Water Science and Engineering, 10(2), 143–153.