Original Research Article: full paper

(2023), «EUREKA: Physics and Engineering»
Number 4

ANALYSIS OF A THREE-DIMENSIONAL NUMERICAL MODELING APPROACH FOR PREDICTING SCOUR PROCESSES IN LONGITUDINAL WALLS OF GRANULAR BEDDING RIVERS

Nelson Cely Calixto[2] ✉
Department of Fluid Mechanics and Thermal Sciences[1]
Department of Civil Constructions, Roads Transportation, Hydraulics and Fluids[3]
nelson.cely@correounivalle.edu.co

Melquisedec Cortés Zambrano
Department of Civil Engineering[3]

Alberto Galvis Castaño
Department of Fluid Mechanics and Thermal Sciences[1]

Gustavo Carrillo Soto[2]
Department of Civil Constructions, Roads, Transportation, Hydraulics and Fluids[3]

[1]*Universidad del Valle*
100-00, 13th str., Valle del Cauca, Cali, Colombia, 76001

[2]*Universidad Francisco de Paula Santander*
12E-96, 7th str., Norte de Santander, Cúcuta, Colombia, 540011

[3]*Universidad Santo Tomás*
11-64, 19th str., Boyacá, Tunja, Colombia, 150001

✉**Corresponding author**

Abstract

In rivers, longitudinal walls are protective structures that are directly supported on the riverbank are frequently used as hydraulic prevent the current from eroding the bank and causing scouring. However, these structures have the potential to block flow and produce erosive processes that progressively worsen scour in their area, leading to faulting and other problems. The current study used Flow-3D software to understand the scour process at the base of longitudinal walls in rivers with a well-graded granular bed. Experimental data from a physical model replicating a river with a longitudinal wall and a well-graded granular bed were used to validate the model. The investigation examined the average flow velocity and its effects on scour behavior along the longitudinal wall using the Flow-3D program. The findings showed that the Flow-3D model could improve the evaluation of debugging processes, because it provided a useful answer that closely matched the experimental data derived from the physical model. Validation with a 0.07 m mesh demonstrated that the Flow-3D model could faithfully simulate the scour process along the longitudinal wall. Overall, the findings of this study suggest that the Flow-3D software can be a useful tool for predicting the scouring process in rivers with well-graded granular beds and longitudinal walls. This is particularly important for engineers and researchers who are interested in designing and optimizing hydraulic structures to mitigate the effects of scouring, because it provided a useful answer that closely matched the experimental data derived from the physical model.

Keywords: Flow-3D software, Granular bed, longitudinal wall, scour, three-dimensional numerical modeling.

DOI: 10.21303/2461-4262.2023.002682

1. Introduction

The study of water flow and its interactions with other terrestrial factors has posed considerable uncertainty in the fields of hydraulics and civil engineering due to its profound influence on the dynamics of structures. Erosion, a widespread phenomenon, significantly impacts hydraulic structures like bridges and protective walls [1–3]. This process, also known as scour, is governed by various hydrological and hydraulic factors such as velocity, flow, width, slope, depth, viscosity, and channel type, among others [4–6]. These factors create perforations at the base of civil structures, which can lead to failure and, in the worst case, collapse [7]. The most common damages caused by erosion include land loss, channel and slope alterations, economic deficits, and, in extreme cases, human losses.

Providing formal knowledge and understanding of the erosive process that undermines longitudinal walls in rivers with well-graded granular beds is crucial to develop measures aimed at reducing its impact. In this context, numerical modeling has gained significant relevance in scientific research, as it allows for a comprehensive understanding of different physical processes through mathematical methods [8–10]. Specifically, the use of three-dimensional flow modeling tools is essential in predicting flow patterns and their interaction with bed boundaries, based on high-resolution data (spatial and temporal) obtained non-intrusively. This approach aims to find sustainable solutions or scenarios that prevent problems and ensure long-term protection of structures against the erosive effects of water.

It is important to emphasize that the study of erosion in civil structures demands a thorough analysis of the information sources used. The combination of real data acquired through field studies with numerical simulations carried out by specialized software, such as Flow-3D, forms a solid foundation for informed decision-making by engineers and hydraulic designers.

Flow-3D stands out as one of the most prominent software in this field, known for its high accuracy in numerical modeling of flow streams based on computational fluid dynamics. It facilitates the solution of hydrodynamic and sediment transport models, enabling the description and quantification of various types of sediments in the flow, including nonlinear erosion models, load transfer, and empirical equations for predicting bed sediment entrainment and erosion [11–13]. The validation of these models with in-situ data enhances the accuracy and reliability of the results obtained, helping to achieve a comprehensive and sustainable approach in addressing the problem of erosion in longitudinal walls in rivers with well-graded granular beds through protection and mitigation measures that minimize environmental impact.

This research work analyzes the erosion process in longitudinal walls in rivers with well-graded granular beds by implementing a mathematical model based on the 3D Reynolds Average Navier Stokes (RANS) equation with a *k*-epsilon model. The study considers a constant time interval and a fixed bed load coefficient within the Flow-3D software [14].

2. Materials and methods

2. 1. Experimental model

The physical modeling of a river with longitudinal walls located on the banks was carried out. The experimental physical simulation was performed on the model to verify the computational model and thus validate the results obtained by numerical simulation (**Fig. 1**).

a *b*

Fig. 1. Test channel of the Hydraulics Laboratory: *a* – the curved channel; *b* – the channel straight

In this sense, the same conditions of the experimental work were modeled in the Flow-3D software, and the results obtained for each simulation was compared with results from the physical model. The physical model was implemented in a 9.0 m long acrylic channel with a cross-section of 0.309 m × 0.450 m, and sediments corresponding to a river with a well-graded granular bed.

To measure the depth of scour in the outer curve of the channel, granular sediments were placed at a distance of 2.18 m from the base of the wall, with a width of 0.15 m and a depth of 0.14 m, and they were analyzed with inlet flows (Q) of 0.008, 0.017, 0.025, 0.030 m^3/s, for different bed heights determined by the slope (S) at 0.5, 1.0, 1.5, 2.0 and 2.5 %. Nine control points along the outer curve were taken to measure scour depth from the difference in bed surface level before and after the scour process.

2. 2. Three-dimensional numerical model

To determine the behavior of the fluid within the model, the calibration parameters of the numerical method were used by applying the averaged Navier Stokes equations of the Reynolds dimensionless value (1), (2) [9, 15].

In this passage, let's discuss an equation that describes the behavior of fluid. The equation is split into two parts: the left-hand side represents the acceleration of the fluid, which is determined by the rate of change of velocity and the convective acceleration. On the other hand, the right-hand side of the equation represents the forces acting on the fluid, which include pressure, viscosity, and external forces. The pressure is determined by the rate of change of pressure with respect to position, while viscosity is determined by the second derivative of velocity with respect to position. External forces can also affect the behavior of the fluid:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_i \partial x_j} - \rho u_i u_j. \tag{2}$$

Where the density of the fluid (P) is the amount of mass contained within a given volume of the substance, and it influences the behavior of the fluid as it flows in the direction of the channel. Velocity (u_j) measures how quickly the water moves in the direction of the channel and is often described in terms of its components in different directions due to the fluid's interaction with the bed boundaries. Velocity fluctuation (u_i) refers to changes in velocity that occur over time in a specific direction and can affect the fluid's movement and interaction with the bed and channel boundaries. Kinematic viscosity (μ) describes the fluid's resistance to movement, while effective pressure (p) is the pressure exerted on the fluid by its surroundings.

2. 2. 1. Numerical process

Simulation of sediment entrainment processes in the outer bend of the test channel was performed using the Renormalization Group (RNG) k-epsilon turbulence model due to its ability to predict velocity profiles and turbulent kinetic energy for flow through the test channel. Likewise, the incompressible model of the fluid is used to simulate the surface of the water in a free – non-turbulent regime, where the inertia of the air can be ignored, inhibiting the passage of liquid. On the other hand, the volume occupied by the gas is replaced by empty space to reduce the computational workload. Finally, the VOF method of the Flow-3D software was used to trace the liquid interface through arbitrary deformations and apply the correct boundary conditions at the working interface designated as the object of study [13, 15].

2. 2. 2. Sediment model

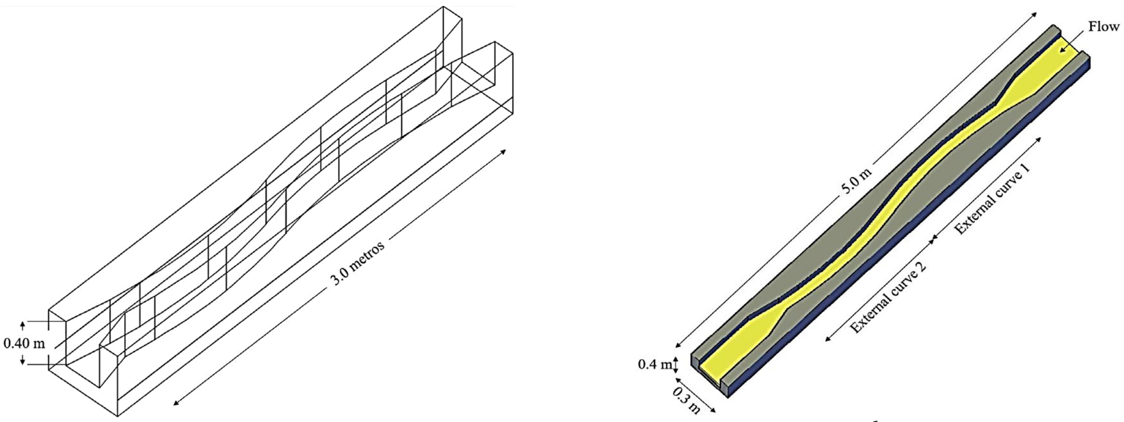
The transport of the packed sediments was defined by the Meyer-Peter & Muller equation, leaving the calculation of the Shields parameter dimensionless by the Soulsby-Whitehouse equation. Said sediments appear as a granular-type packed bed with an average diameter (Dm) of 7.08 mm and a Uniformity Coefficient (Cu) with a value of 20 evaluated as the D60/D10 ratio, guaranteeing a well-graded granular bed (**Table 1**)

Table 1
Physical model sediment characteristics section

Sediment type	Diameter (m)	Density (Kg/m^3)	Angle of repose (°)	Volume composition (%)
Fine sand	0.0009	1600	29	46
Gross sand	0.00925	1600	29	26
Gravel	0.03085	1700	31	28

2. 2. 3. Geometry

The test channel and sediment geometries were imported into the Flow-3D software in «stl» to generate the modeling according to the experimental process. A dimensional roughness of 1.5e-06 m was defined for the solid representing the channel so that the software would correctly simulate the friction exerted by the channel walls on the bounded fluid. The proportions of sediment to work with were determined in the «Packed Sediment» configuration of the subcomponent included in the moving bed layer (**Fig. 2**).



a *b*

Fig. 2. Geometry of the numerical model established in Flow-3D:
a – three dimensional Channel; *b* – general channel

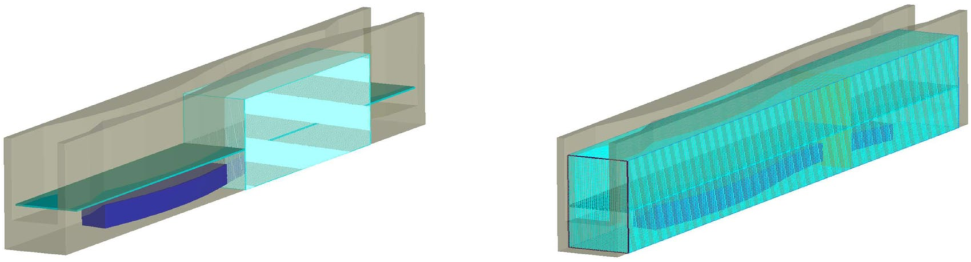
The channel model comprises two variants: a three-dimensional model with a distance of 3 m (**Fig. 2, *a***) and a general channel model with a distance of 5 m (**Fig. 2, *b***). Both models have a height of 0.4 m and a width of 0.3 m.

2. 2. 4. Meshing

A multi-block mesh was defined to represent the analysis, and geometry and reduce the total number of cells by using coarse meshes in less relevant areas and preservation of high resolution in the study sector for scour determination. In **Fig. 2**, the development of the channels' geometry simulated in Flow 3D is depicted. It is worth noting that these dimensions correspond to the experimental channels investigated in this study. Specifically, the curved channel spans 3 m in length, 0.4 m in height, and 0.3 m in width, while the straight channel measures 5 m in length, 0.4 m in height, and 0.3 m in width. The meshes employed for the simulations are also presented.

The appropriate block mesh was determined by performing a sensitivity analysis of the mesh, with which a uniform cell size was defined, representing the curvatures of the channel and the sedimentary bed. For practical reasons, the convergence of the network was limited to generating a double mesh to deepen the level of analysis in the outer curve of the second test channel.

The overall channel and curve meshing is set to a cell size of 0.014 m and 0.07 m, respectively, for a total of 1,980,013 cells in the work section (**Fig. 3**).



a *b*

Fig. 3. The meshing of the geometry configured in Flow-3D:
a – Mesh on the curve; *b* – General mesh

2. 2. 5. Edge conditions

A water inlet was established with a volumetric flow rate definition and a fluid elevation. The model's downstream (outlet) section will be treated as an outlet flow boundary condition defined as specific pressure with the same fluid elevation as in the inlet condition. The sides will follow one another like walls, that is, as a symmetrical limit; the lower edge is defined as a solid, and the well-graded sediments are located so that the scour presented by the bed can be evaluated. A VOF model was obtained in the upper limit to be specified as a free surface, treated as an imaginary frictionless plane (**Table 2**).

Table 2
Boundary conditions applied to the numerical model

Limit location		Boundary conditions applied to the numerical model	
		Symbol	Definition
X	min	*Q*	Boundary condition with defined flow
	max	*P*	Pressure outlet flow
Y	min	*S*	Symmetric limit
	max		
Z	min	*P*	The pressure boundary condition for free surface
	max		

3. Results and discussion

3. 1. Three-dimensional numerical model

The simulation process analyzes sediments dragging, specifically in the outside of the channel curve. It was possible to identify that velocity and shear stress distribution are the parameters that affect the most the transport of sediments within the flow, as shown in **Fig. 4**, which measures the maximum depth of scour reached at the limits of the structure as function of flow and longitudinal slope at the 11 control points of the straight channel.

In **Fig. 4**, the erosion behavior at different control points within the bed can be observed, highlighting the influence of the slope of the channel and the flow rate as the main factors affecting the process. It can be seen that for flow rates of 0.025 and 0.03, the erosion at control point 2 starts with values of –35 and –40 mm (**Fig. 4, *a***), respectively, and increases to reach –60 and –100 mm of erosion (**Fig. 4, *e***) for a final slope of –2.5 %. Similarly, it can be observed that for the other two flow rates, namely 0.008 and 0.017, at the same control point, erosion takes values of –10 and –20 mm (**Fig. 4, *a***), respectively, and increases as the slope becomes steeper, reaching –40 and –60 mm of erosion (**Fig. 4, *e***).

Furthermore, it can be observed that at control point 8, the experiments with higher flow rates, such as 0.025 and 0.03, decrease erosion by 30 % to 60 % compared to their lowest point at different slopes (**Fig. 4**). The final values obtained for control point 8 for flow rates of 0.025 and 0.03 are: –10 and –20 mm for a slope of –0.5 % (**Fig. 4, *a***), –40 and –45 mm for a slope of –1 % (**Fig. 4, *b***), –50 and –70 mm for a slope of –1.5 % (**Fig. 4, *c***), –15 and –25 mm for a slope of –2.5 % (**Fig. 4, *d***), and –60 and –70 mm for a slope of –2.5 %, respectively. It was also observed that for flow rates of 0.008 and 0.017, the highest erosion achieved was –0.20 and 60, respectively, with a slope of –0.25 % (**Fig. 4, *d***).

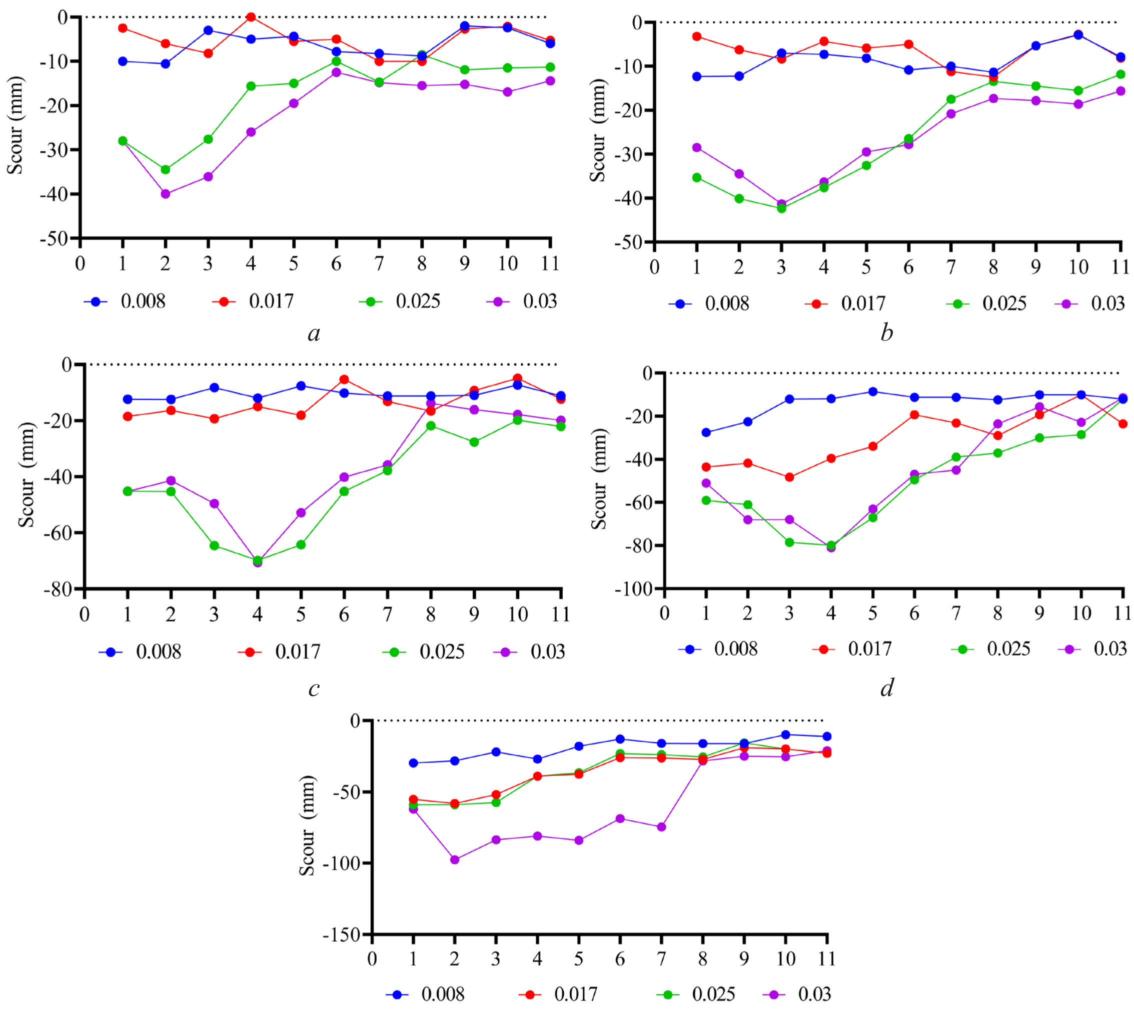


Fig. 4. Scour profiles of the curved channel for each of the slopes and flows: *a* – 0.5 %; *b* – 1.0 %; *c* – 1.5 %; *d* – 2.0 %; *e* – 2.5 %

The above is confirmed by [16], where they proposed and evaluated a simulation model of scour in walls at different flows. Likewise, the latter coincides with [17], which ensures that in the straight channel, the removal of sediments from the bottom of the channel occurs mainly at the first control points, due to the shock of the flow at the entrance that drags the sediment.

Regarding **Fig. 5** the scour profiles present a similar trend to the straight channel in terms of the influence of the longitudinal slope and the increase in the maximum scour depth. The latter agrees with [18], who, when analyzing a meandering channel with an erosion process at the limits of the channel, it was possible to determine that morphological parameter such as the depth of the foundations and the velocity of the flow directly influence the size of the undermining. Likewise, In **Fig. 5**, it can be observed that the maximum scour depth is reached as a function of the slopes tested at a flow rate of 0.025 m^3/s. The values of scour depth range from –36.35 mm to 99.83 mm for slopes of 0.5 %, 1.0 %, 1.5 %, 2.0 %, and 2.5 %, and control points 3, 4, 6, 4, and 7, respectively.

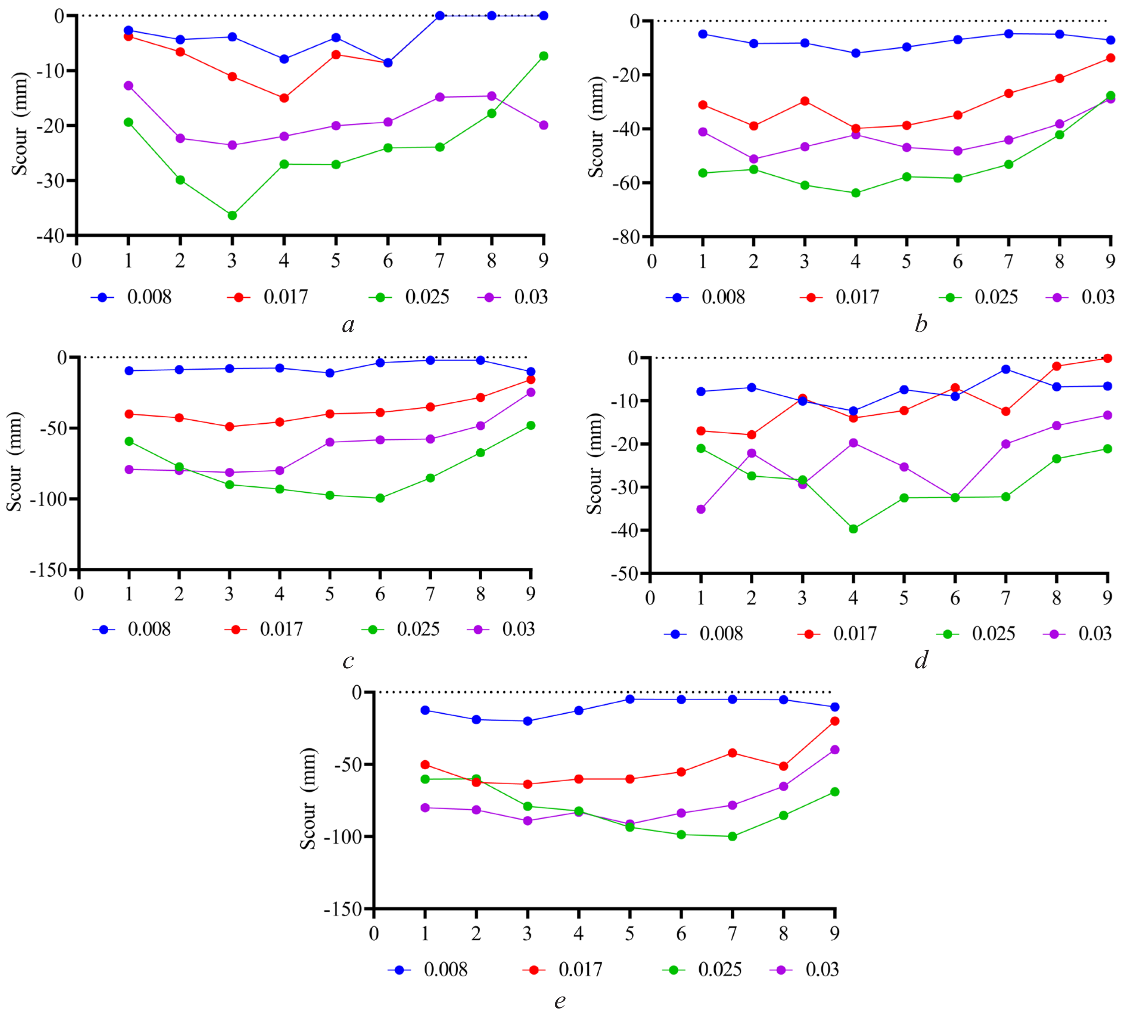


Fig. 5. Scour profiles of the curved channel for each of the slopes and flows: *a* – 0.5 %; *b* – 1.0 %; *c* – 1.5 %; *d* – 2.0 %; *e* – 2.5 %

Moreover, it is noteworthy that in **Fig. 5**, control point 1 generally has the lowest scour depth values, except for the flow corresponding to 0.008 m^3/s, in which the scour depth values after control point 7 are lower than the initial value, resulting in a decrease of up to 100, 3, 79, 67, 61, and 12 percent for slopes of 0.5 %, 1.0 %, 1.5 %, 2.0 %, and 2.5 %, respectively. The corresponding scour depth values range from –2.65 to 0.00, –4.82 to –4.72, –9.38 to –2.03, –7.38 to –2.65, and –12.5 to –4.98.

Results obtained in the «Application of computational fluid dynamics to evaluate the hydraulic performance of spillways in Australia» the meshing process determines an optimal and precise modeling solution since it allows to establish how the factors that affect the simulation process are variations as is the proposed geometry altered, increasing or decreasing the operation times, thus explaining a higher computational expense [19].

To verify the mesh characteristics effect on the simulation, a sensitivity analysis was carried out to obtain the function of the depth of the scour using the size of the cell. The Flow-3D model uses a numerical scheme to discretize the total computational domain and convert it into small sections, thus reducing the errors generated from the discretization process. **Fig. 6** shows how scour is directly proportional to the flow rate, reaching its maximum at 0.025 m^3/s. This occurs mainly because, as the flow rate increases, the amount of kinetic energy also increases, which would cause changes in the normal flow pattern of the channel, leading to the formation of new currents, vortices and increasing the dragging capacity of soil particles, eroding the bed and banks of the river or channel. In fact, **Fig. 6** illustrates that the maximum scour depth of 0.0883 mm is reached at a flow rate of 0.03 m^3/s, across all meshes used in the simulation process.

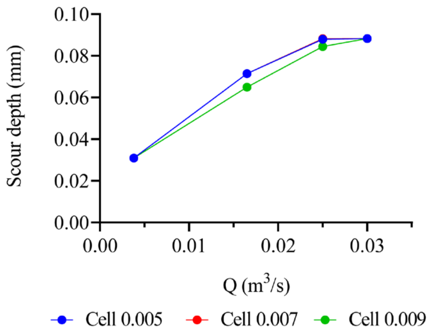


Fig. 6. The meshing of the geometry configured in the Flow-3D

Furthermore, the data shows that meshes with cell sizes of 0.05 and 0.07 m have similar trends as the flow rate increases, producing scour depths of 0.0310 and 0.0715 mm for flow rates of 0.0038 m^3/s, respectively. It was also possible to identify that, with a cell size of 0.05 m and 0.07 m, the scour depth approaches the values observed in the experimental model.

The value of 0.07 m is considered the optimal cell size based on the accuracy of the results in terms of scour depth of the longitudinal wall, as well as the representation of the lower bed. Simulation with a cell size of 0.07 m produced results in scour depth as the finest cell size (0.05 m), but with significant savings in simulation time. In contrast, the results obtained in the simulation performed with a cell size of 0.09 m were far from the observed values. It should be noted that the information provided in the graph was based on tests limited to a physical model; therefore, the validation of the cell size is considered as approximate conditions to absolute values, according to some authors [18].

Fig. 7 shows the influence of the longitudinal slope (*S*) on the average speed of the flow (*U*) at different flow values. This concept increases when a higher Slope is used. In **Fig. 7**, it is evident that the increase in flow rate results in an increase in average flow velocity, reaching a maximum value of 1.036 for a slope of 2.5 % and a flow rate of 0.0233 m^3/s. When the slope decreases by 20 % compared to the maximum value of 2.5 %, the average flow velocity decreases by 3.5 %, 4.6 %, and 9 %, respectively, reaching values of 0.9978, 0.9883, 0.9425, and 0.9425 for an average flow rate of 0.028 and a slope of 2.0 %, 1.5 %, 1.0 %, and 0.5 % slope, respectively.

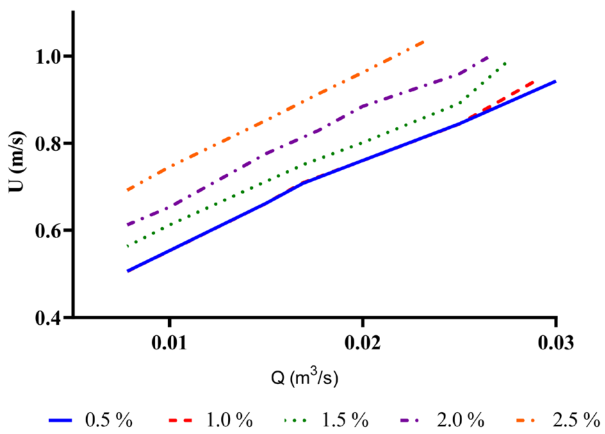


Fig. 7. Average flow velocity at different flow values and longitudinal slopes

Furthermore, it is possible to observe that the behavior of slopes 0.5 % and 1.0 % is similar to that of average velocity as the flow rate increases, with minimum values of 0.5061, 0.5534, 0.6621, 0.7077, 0.7602, 0.8444, and 0.9425 m/s for flow rates of 0.0078, 0.01, 0.015, 0.0169, 0.02, 0.025, and 0.03 m^3/s, respectively. Likewise, and in agreement with [16]. This shows that, in clear water conditions, scour in hydraulic structures is mainly due to the velocity distribution and the bed's shear stress. That is, the depth of scour of the structure is directly proportional to the flow velocity, thus, demonstrating that the geometry of the bottom of the channel influences the scours processes [4, 20].

Fig. 8 shows the flow velocity pattern along the test channel under slope conditions of $S = 2.5$ % Both the experiments and the numerical simulation develop a downward flow along the outer curve of the wall longitudinal. This process occurs due to the decrease in the width of the channel and the degree of curvature exerted along the longitudinal wall, producing high flow velocity levels near the bed, resulting in progressive deformation and undermining. Furthermore, the increased velocity in the channel bed leads to a scour hole of higher magnitudes [17, 21].

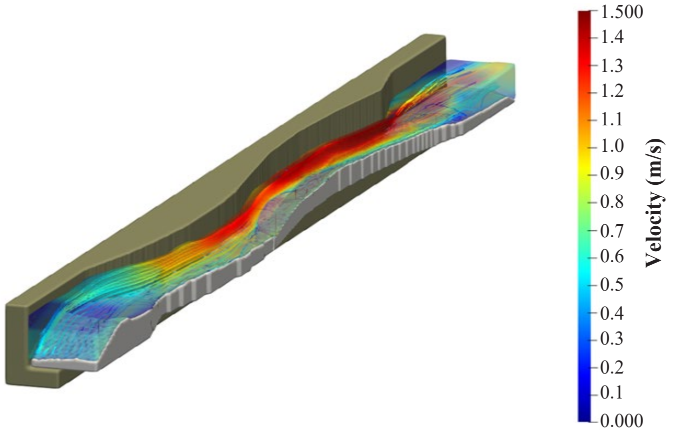


Fig. 8. Distribution of flow lines in the numerical model

In the execution of the experiments, the formation of vortices in a clockwise direction, as evidenced in some sections of the test channel, which, according to [19, 22], acted as high-speed secondary flows, producing greater scour levels. In the bed channel, specifically in the outer curve of the channel.

Finally, the Flow-3D numerical model estimated the maximum scours along the longitudinal wall established in the test channel. The analysis of the scour's depth concerning the flow's average speed proves that the numerical simulation shows a trend similar to that obtained experimentally.

«**Fig. 9** illustrates a positive correlation between volumetric flow and scour behavior, with maximum values of 0.0910 mm observed at a velocity of 1.37 m^3/s, and a minimum of 0.0150 at 0.59 m^3/s for the experimental model. Similarly, the predictive model in the Flow-3D software exhibited a comparable trend, with a maximum of 0.089 at 1.4 m^3/s and a minimum of 0.0180 at 0.66 m^3/s, and two intermediate points at a flow rate of 0.86 and 1.1 m^3/s, corresponding to erosion rates of 0.053 and 0.078, respectively». The values obtained numerically are lower than those obtained experimentally, yielding a mean square error value of 0.008. A mean square error (MSE) value of 0.008 in the scour prediction indicates that the model predictions are relatively accurate. The MSE is a measure of the root-mean-square difference between the predicted values and the actual values. However, it is important to note that the significance of the MSE value depends on the specific context of the problem. For example, in some applications, an MSE value of 0.008 may be considered high, while in others it may be considered low (**Fig. 9**).

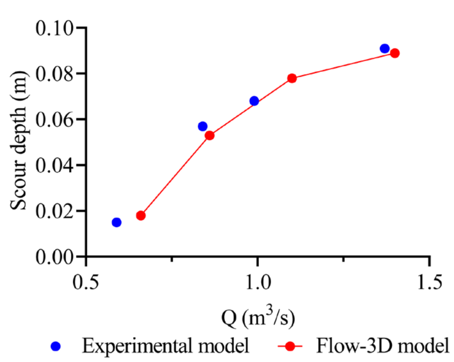


Fig. 9. Response of maximum scour depth to mean flow velocity

In scour prediction, a low MSE value indicates that the model can accurately predict the amount of scour that is likely to occur around a bridge or other hydraulic structure. This can be useful in the design and construction of structures that are less likely to experience scour-related failure. It is important to note that the MSE value should not be the only criteria used to assess the accuracy of a model. Other measures should also be considered, such as the coefficient of determination (R-squared). Additionally, the validity of the model must be assessed through independent testing and validation before it is used in real-world applications.

Despite the significant advances achieved through the use of numerical modeling and the application of Flow-3D software in the stu dy of erosion in civil structures, this work presents certain limitations that should be considered when applying the results in practice or when designing future theoretical studies. First, it is important to recognize that although the numerical model provides a valuable approximation of the hydraulic and sediment transport processes, there are still challenges in obtaining real and representative data to fully validate the model. The combination of field data and numerical simulations remains essential to improve the accuracy of the results and ensure better application in practical situations.

Furthermore, the accuracy of the numerical model and its ability to predict the erosion process on longitudinal structures in rivers with well-graded granular beds are highly dependent on the input parameters and model configuration. Proper selection of the mesh and boundary conditions are crucial to obtain accurate and representative results. Appropriate sensitivity analyses and adjustments are necessary to ensure that the numerical results correspond adequately to the physical conditions observed in the field.

Another important limitation is related to the simplification of some flow characteristics and geometry in the numerical model. Despite efforts to accurately represent hydraulic phenomena, computational simulation still involves certain simplifications and approximations that may affect the accuracy of the results. The representation of the interaction between the stream and the channel walls, as well as the presence of obstacles or vegetation, may not be completely faithful to the real conditions in the field.

Also, the research focused on a specific type of well-graded granular bed and did not take into account the variability of sediment characteristics in different geographic locations or river environments. Consideration of different sediment types and their properties could provide a broader view applicable to different situations in reality.

Based on the investigation of erosion in hydraulic structures in rivers with well-graded granular beds and the use of numerical modeling with Flow-3D software, possible future developments are proposed. These include investigating the effects of different sediment types on erosion, studying the interaction between vegetation and erosion, investigating the impact of climate change on the erosion process, and exploring the combination of different erosion protection methods. Also, it's recommended to investigate the connection between erosion and the lifespan of hydraulic structures, this will lead to a greater understanding of how to effectively maintain and rehabilitate these structures. These future investigations would expand on the subject and provide more informative and practical solutions to address the effects of erosion in civil engineering. Numerical modeling will still be of great importance in the evaluation of different scenarios and conditions, and will continue to be instrumental in the optimization of the design and protection of hydraulic structures against erosion.

4. Conclusions

The development of a new mathematical expression based on experimental data has made it possible to estimate local scour in relation to the increase in water flow and kinetic energy in a river or channel. These lead to an increase in scour, especially if there are other factors contributing to soil erosion, such as changes in water level, terrain topography, vegetation, and soil composition. Three-dimensional numerical modeling is a powerful tool for predicting the scour process around hydraulic structures. This approach allows engineers to simulate the complex interactions between water flow, sediment transport and hydraulic structures, identifying potential scour problems prior to construction to optimize design features that reduce scour or design countermeasures such as riprap protection, scour holes or velocity baffles.

This study has been able to determine that the maximum scour occurs when the velocity, flow and slope factors increase, and shows that the optimum cell size is when it reaches a value of 0.07 m, since at this point it is like the values obtained in the test model. On the other hand, the influence of the morphological factors of the different causes was also evidenced, producing high velocity secondary flows, and increasing the scour of the granular bed.

Conflict of interest

The authors declare that they have no conflict of interest in relation to the present research, whether financial, personal, authorship or otherwise, that may affect the research or the results presented in this paper.

Financing

The study was performed without financial support.

Data availability

Data will be made available on reasonable request.

Acknowledgments

The authors express their gratitude to the Santo Tomas University Tunja headquarters for the opportunity to carry out the physical modeling in the hydraulics laboratory and for accompanying the experimental process.

References

[1] Pereira Mendonça, I. S., Daniel Leitão Canilho, H., Maria Senafael, C. (2019). Flow-3D modelling of the debris effect on maximum scour hole depth at bridge piers. 38th IAHR World Congress – «Water: Connecting the World». doi: https://doi.org/10.3850/38wc092019-1850

[2] Toapaxi, J., Galiano, L., Castro, M., Hidalgo, X., Valencia, N. (2015). Analisis de la Socavacion en Cauces Naturales. Revista Politecnica, 35 (3). Available at: https://revistapolitecnica.epn.edu.ec/ojs2/index.php/revista_politecnica2/article/view/368/pdf

[3] Bento, A. M., Gomes, A., Viseu, T., Couto, L., Pêgo, J. P. (2020). Risk-based methodology for scour analysis at bridge foundations. Engineering Structures, 223, 111115. doi: https://doi.org/10.1016/j.engstruct.2020.111115

[4] Xiang, Q., Wei, K., Li, Y., Zhang, M., Qin, S. (2020). Experimental and Numerical Investigation of Local Scour for Suspended Square Caisson under Steady Flow. KSCE Journal of Civil Engineering, 24 (9), 2682–2693. doi: https://doi.org/10.1007/s12205-020-2343-9

[5] Alemi, M., Maia, R. (2016). Numerical Simulation of the Flow and Local Scour Process around Single and Complex Bridge Piers. International Journal of Civil Engineering, 16 (5), 475–487. doi: https://doi.org/10.1007/s40999-016-0137-8

[6] Gupta, L. K., Pandey, M., Raj, P. A., Pu, J. H. (2023). Scour Reduction around Bridge Pier Using the Airfoil-Shaped Collar. Hydrology, 10 (4), 77. doi: https://doi.org/10.3390/hydrology10040077

[7] Coronación, S. (2017). Evaluación de impactos por la extracción de agregados para la construcción en el cauce del río Achamayo, Concepción – Junín. Huancayo. Available at: https://repositorio.upla.edu.pe/handle/20.500.12848/279

[8] Illidge-Araujo, J., Chacon Velasco, J. L., Chacon Velasco, A. J., Romero Piehadraita, C. A. (2020). Diseño y simulación de un sistema pico-hydro para la generación de energía eléctrica en zonas rurales, mediante un software de mecánica de fluidos computacional. Revista UIS Ingenierías, 19 (1), 155–170. doi: https://doi.org/10.18273/revuin.v19n1-2020015

[9] Liu, M., Wang, H., Tang, G., Shao, F., Jin, X. (2022). Investigation of local scour around two vertical piles by using numerical method. Ocean Engineering, 244, 110405. doi: https://doi.org/10.1016/j.oceaneng.2021.110405

[10] Delgado-Mejia, A. L., Olmos-Villalba, L. C., Rivero-Mejia, A. E. (2014) Estudio comparativo del ciclo indicado de un motor diesel mediante simulación CFD y datos experimentales. *Revista UIS Ingenierías*, 13 (1), 23–31. Available at: https://revistas.uis.edu.co/index.php/revistauisingenierias/article/view/23-31

[11] Basante-Bolaños, E. A., Villacrés-Martinez, M. A., Cruz-Velasco, L. G. (2021). Simulación numérica 2D subrasantes de baja resistencia, estabilizadas con material granular de sobretamaño (rajón). Revista UIS Ingenierías, 20 (3). doi: https://doi.org/10.18273/revuin.v20n3-2021004

[12] Taha, N., El-Feky, M. M., El-Saiad, A. A., Fathy, I. (2020). Numerical investigation of scour characteristics downstream of blocked culverts. Alexandria Engineering Journal, 59 (5), 3503–3513. doi: https://doi.org/10.1016/j.aej.2020.05.032

[13] Monroy-González, H. (2021). Evaluación de eficiencia hidráulica en rejillas de sumideros mediante investigación numérica tridimensional y verificación de resultados de experimentación física de prototipos. Universidad Santo Tomás. Available at: https://repositorioslatinoamericanos.uchile.cl/handle/2250/3680967

[14] Kaurav, R., Mohapatra, P. (2019). Effect of rheology on earthen embankment breaching. 38th IAHR World Congress – «Water: Connecting the World». doi: https://doi.org/10.3850/38wc092019-1505

[15] Cevallos Cevallos, P. E., Ochoa Rodriguez, D. M. (2020). Modelación numérica (CFD) del lavado de sedimentos en el embalse de la presa Toachi. Programa FLOW-3D. Quito: EPN. Available at: https://bibdigital.epn.edu.ec/handle/15000/21320

[16] Ghasemi, M., Soltani-Gerdefaramarzi, S. (2017). The scour bridge simulation around a cylindrical pier using Flow-3D. Journal of Hydrosciences and Environment, 1 (2), 46–54. doi: https://doi.org/10.22111/JHE.2017.3357

[17] Di Pietro, P., Mahajan, R. R. (2021). Erosion Control Solutions with Case Studies. Lecture Notes in Civil Engineering, 71–94. doi: https://doi.org/10.1007/978-981-16-4783-3_6

[18] Yamaguchi, S., Kyuka, T. (2019). A hydraulic model experiment on the relationship between sediment transport characteristics and changes inwatercourses arounda low-water revetment or spur dikes. 38th IAHR World Congress – «Water: Connecting the World». doi: https://doi.org/10.3850/38wc092019-0245

[19] Ho, D. K. H., Riddette, K. M. (2010). Application of computational fluid dynamics to evaluate hydraulic performance of spillways in australia. Australian Journal of Civil Engineering, 6 (1), 81–104. doi: https://doi.org/10.1080/14488353.2010.11463946

[20] Jalal, H. K., Hassan, W. H. (2020). Three-dimensional numerical simulation of local scour around circular bridge pier using Flow-3D software. IOP Conference Series: Materials Science and Engineering, 745, 012150. doi: https://doi.org/10.1088/1757-899x/745/1/012150

[21] Shamohamadi, B., Mehboudi, A. (2016). Analyzing Parameters Influencing Scour Bed in Confluence Channels Using Flow-3D Numerical Model. Civil Engineering Journal, 2 (10), 529–537. doi: https://doi.org/10.28991/cej-2016-00000055

[22] Al-Zubaidy, R. A., Hilo, A. N. (2022). Numerical investigation of flow behavior at the lateral intake using Computational Fluid Dynamics (CFD). Materials Today: Proceedings, 56, 1914–1926. doi: https://doi.org/10.1016/j.matpr.2021.11.172

Received date 22.03.2023
Accepted date 19.07.2023
Published date 27.07.2023

How to cite: *Calixto, N. C., Zambrano, M. C., Castaño, A. G., Soto, G. C. (2023). Analysis of a three-dimensional numerical modeling approach for predicting scour processes in longitudinal walls of granular bedding rivers. EUREKA: Physics and Engineering, 4, 168–179. doi: https://doi.org/10.21303/2461-4262.2023.002682*