27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-1228
RIVERBED EFFECT INFLUENCE IN A RIVER DEBRIS REMOVAL SOLUTION FLOATING BARRIER

Patrick Donegá Queiroz
Institute for Technological Research, São Paulo, Brazil
patrickqueiroz@ipt.br

André Mitsuo Kogishi
Institute for Technological Research, São Paulo, Brazil
amkogishi@ipt.br

Felipe Santos de Castro
Institute for Technological Research, São Paulo, Brazil
fscastro@ipt.br

João Lucas Dozzi Dantas
Hydro and Aerodynamics Department, FORCE Technology, Kongens Lyngby, 2800, Copenhagen, Denmark
joda@forcetechnology.com

Abstract. *The Tietê River is located in São Paulo, Brazil, and is home to Small Hydroelectric Power Plants, despite the severe pollution. To prevent the debris to reach the equipment, the Metropolitan Water and Energy Company (EMAE) and the Institute for Technological Research (IPT) developed an active barrier to capture debris, which includes modular floating barriers called trash boom lines. The loads on these barriers are affected by different operational and environmental conditions, such as the current velocity and the line arrangement and mooring, resulting in a complex non-linear system and, since the water level changes, the distance from the water surface to the riverbed plays an essential factor in the hydrodynamic resistance. Using ANSYS Fluent to investigate the riverbed effect on the trash boom at different current velocities and incident angles and its impact on normal and lateral loads. The simulations revealed that the normal force component acting on the module is significantly affected by the riverbed height, reaching up to 2.5 times in the lesser height condition tested (3.5 m) in comparison to the deepest condition (15 m). The result shows that the riverbed height need to be considered during the dimensionalization of the trash boom lines loads.*

Keywords: *Riverbed Effect, Shallow Water Effect, Computational Fluid Dynamics, Floating Barrier, Small Hydroelectric Powerplants*

1. INTRODUCTION

The Tietê River, located in the state of São Paulo, Brazil, flows from West to East across it. The river also divides the São Paulo city, one of the largest and most populated metropolises on Earth and Brazil's most important economic center. Despite the importance of São Paulo city, its growth was not planned and many issues rose from this, such as the lack of sanitation, which led heavy pollution into the Tietê River, given that a large amount of waste is thrown into it, such as solid and non-degradable trash, plastic bags, plastic and glass bottles, tin cans, home appliances and many other sorted debris of many sizes, ranging from small bottle caps to whole pieces of furniture. Figure 1 shows the river outline and, in detail, the region of interest. And, due to the fact that the river is highly polluted, the proliferation of aquatic plants is greatly accelerated, further increasing the quantity of trash in the river.

In the work of (Campos et al., 2020) a detailed experimental investigation was conducted to understand the debris flow in this river in a particular region, near Pirapora city, assessing its composition and flow volume. An example of what was found is shown in Figure 2.

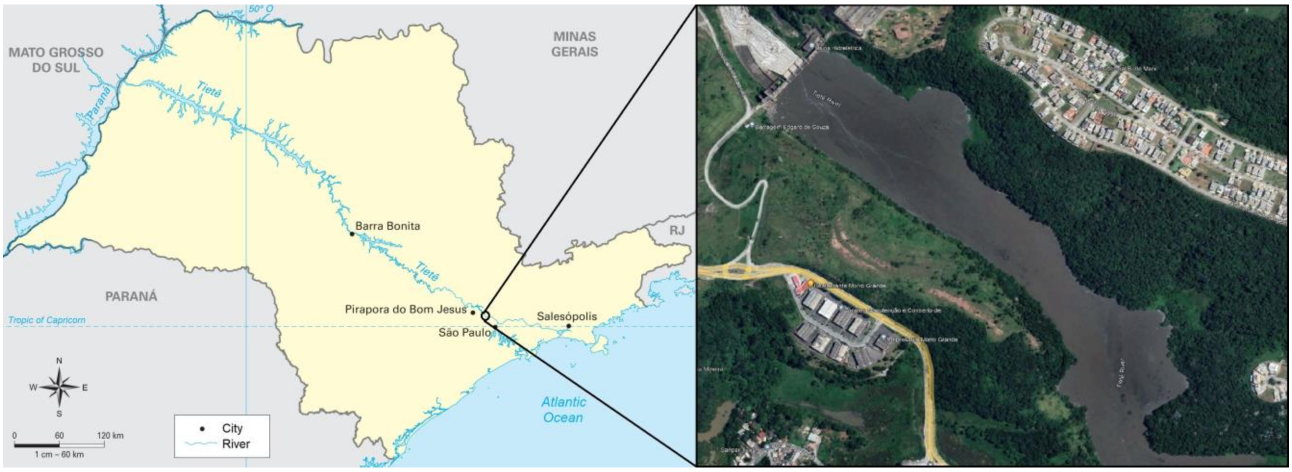

Figure 1. Tietê river outline overview and region of interest detail (Source: Queiroz, 2022)

Figure 2. Trash sampling in the Tietê River (Source: Queiroz, 2022)

Brazil is hugely abundant in hydraulic resources, which led the country's energy grid to be strongly dependent on hydroelectric generation. But, following what happened with the Tietê River, many other bodies of water got polluted, causing operational issues for the power plants installed in them. The Metropolitan Water and Energy Company (EMAE) faces these problems daily, once it is the state agency that controls the flow and energy generation in the Alto-Tietê hydrographic basin and is forced to spend more money on maintenance and clean up, while having the generation availability reduced. There are three Small Hydroelectric Power Plants (SHPs), with between 5 to 30 MW installed capacity, in Tietê River, being Pirapora, Rasgão and Porto Goés, with an average 16 m^3 of debris being collected every day, with the occasional peak that can get up to 65 m^3 (Campos et al., 2020).

Edgard de Souza dam is located 130 km upstream of Pirapora SHP and, because of the mentioned issues, is deactivated for energy generation. Therefore, it was chosen to have a system installed uphill its damn, capable of collecting most of the floating debris before they reach the SHPs downstream. EMAE along with the Institute for Technological Research from Sao Paulo State (IPT), through an R&D project, investigated the region, looking for the best setup possible. The developed solution consists of a floating vessel to which floating debris are deflected by two floating barriers, called trash booms. A conveyor belt, installed on the vessel, removes the trash from the water and unloads it into containers, which are later retrieved and sent to their proper destination. The development took place for over three years and is composed by: collecting vessel, trash boom, mooring lines, anchors, hydraulic model, hydrodynamic model (Mata et al., 2022), laboratory tests (Castro et al., 2022) and field tests (Campos et al., 2020). Each part is investigated in an iterative manner to account for characteristics and interference. A more complete description of this system can be seen in the work of (Galvão et al., 2022). The concept is shown in Figure 3.

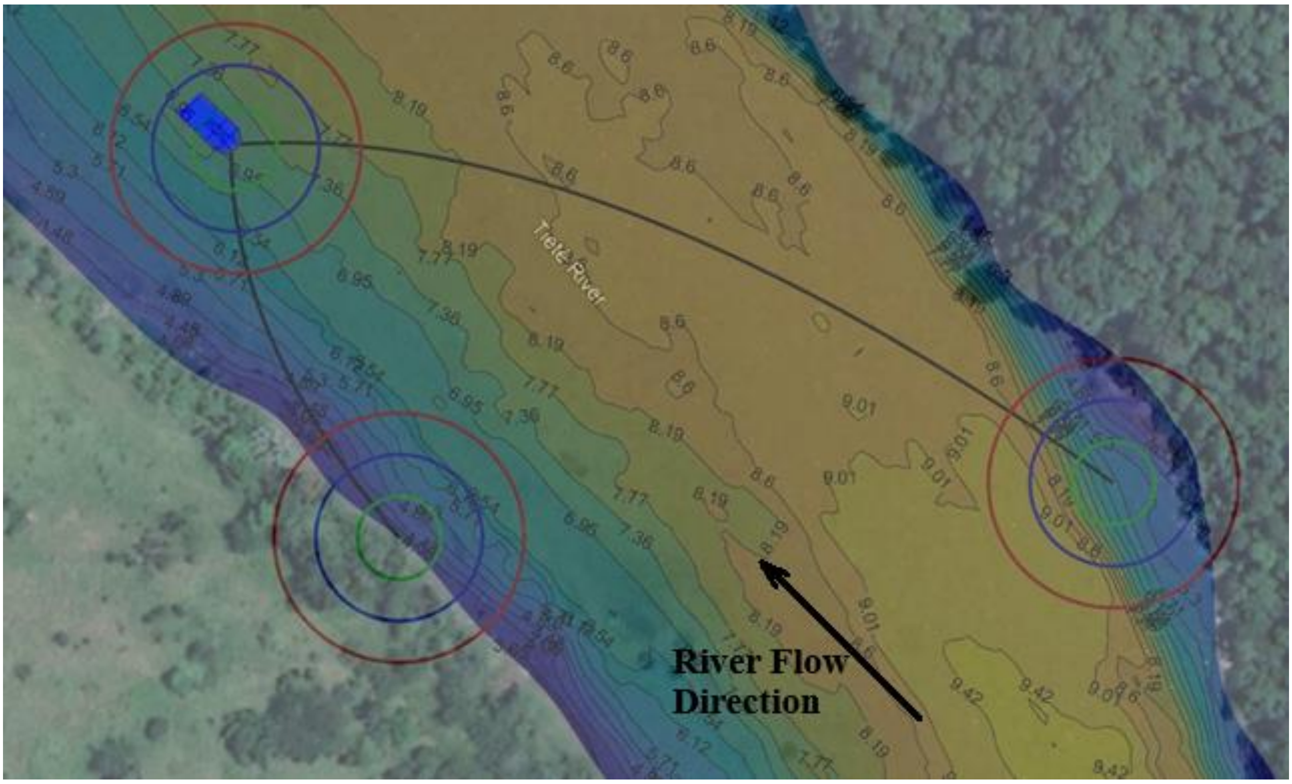

Figure 3. Concept of developed solution. The numbers on contour lines represent local depth in meters.

Floating barriers are non-linear complex systems with a combined co-dependent dynamic influenced by both, internal and external (hydrodynamic), loads and their motion. Experience with analogous structures was acquired through previous works, specifically with log booms for hydroelectric power plants (Chreim et al., 2018; Chreim and Dantas, 2019), for what was necessary to build a numerical tool to calculate the approximate loads in several different hydraulic scenarios. In such case, there were no mooring lines, and the floating barriers were moored in structural pillars with no significant change in water level.

To develop the concept for installation on the Edgard de Souza dam, a similar numerical tool was created by adapting the one previously developed. However, due to differing conditions, some adjustments were necessary. The Edgard de Souza dam is used for flow control, which means the water level varies drastically, so does the flow rate, causing the position of the floating vessel to constantly change. Each trash boom has one extremity fixed to the floating vessel, which is held in position by four mooring lines, while the other end is fixed to a pillar in the river margin, so the vessel motion causes the line to strain or loosen. During the investigations of the R&D project, it was verified that these effects have a great influence in the trash boom and mooring line loads, as well as on the dynamic positioning of the lines and vessel. It is necessary to have a good prediction during the design phase to ensure the required performance in debris management. A detailed explanation of the numerical model can be found in (Queiroz et al., 2022).

The objective of the present work is to analyze how the river depth influences the loads and dynamic of a trash boom module, which was called riverbed effect, and extrapolate these findings to the loads of the entire trash boom line. For that, a series of CFD simulations were performed for a trash boom module, in various scenarios for many different values of water surface-riverbed distance. With the available results, it became possible to incorporate a correction factor for the hydrodynamic loads of the trash boom lines into the numerical method. This study was a necessary to conduct because a differentiating factor is the fact that the region from the original model had a deep-water channel, so the hydrodynamic loads would be independent from the distance to the riverbed at any given point. The region upstream of Edgard de Souza dam features a shallow water channel with substantial variations in depth. This characteristic is represented in Figure 3, where a measured bathymetry is plotted over the map. The left trash boom line spans a region with the minimum of 4 m depth and, was later verified that, in some hydraulic scenarios, the water level becomes so low that the line touches the riverbed or even completely dry up.

2. CFD MODEL

The present work uses a CFD model developed using the software ANSYS FLUENT (Mata et al., 2022). The model used was previously built and, for the present work, a riverbed model was added to consider the interference effect. The original model simulated one module of the trash boom at different current speeds and incidence angles and compared the results to experiments conducted in the IPT's Towing Tank. The next subsection describes the model adopted in this work.

2.1 GEOMETRY

The trash boom line is composed of several floating modules, which were developed and optimized in other stages of this project. The trash boom lines are around 120 m long, which makes it impracticable to simulate, due computational costs. Instead, the model only considered one module of the trash boom, using periodic conditions to emulate the effect of interference from the adjacent modules. The module designed is composed of a flat plate with 1m width and 1.3 m height, acting as the barrier and a hexahedron with 0.5 m width, 0.25 m length and 0.5 m height, acting as the floating device. Figure 4 presents the module and the fluid domain, which were divided into sub-zones to apply different refinements around the module and free surface. The riverbed effect is simulated changing the domain height.

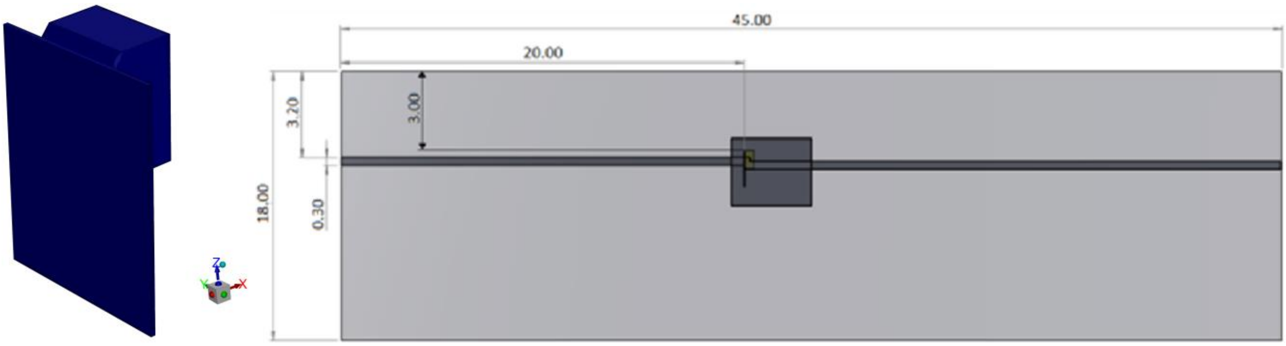

Figure 4. Trash boom module and fluid domain, with dimensions in meters (Source: Mata, 2022)

2.2 MESH

A refined mesh is important to capture the phenomena adequately, but more the mesh is refined, higher will be its computational cost, demanding a balance. Three different meshes were built, with 800.000, 1.200.000 and 2.300.000 elements. The results were roughly the same, with the only difference being the simulation duration, so the model for the conclusion of the simulations was built with 800.000 elements, shown in Figure 5, using a hybrid mesh (hexahedral and polyhedral) method. Two refinement zones can be seen around the module and free surface: the first is necessary to better model the flow small eddies, detachment and turbulence, in order to enhance the hydrodynamic forces prediction; and the latter is a mixture zone where waves and vortices are bound to happen, and a finer mesh helps captures such phenomena. There is also an outer zone that comprehends the remaining domain, that can receive a less refined mesh and the surface of the module, which receives a thin surface refinement, since it is the surface where all the important parameters will be assessed.

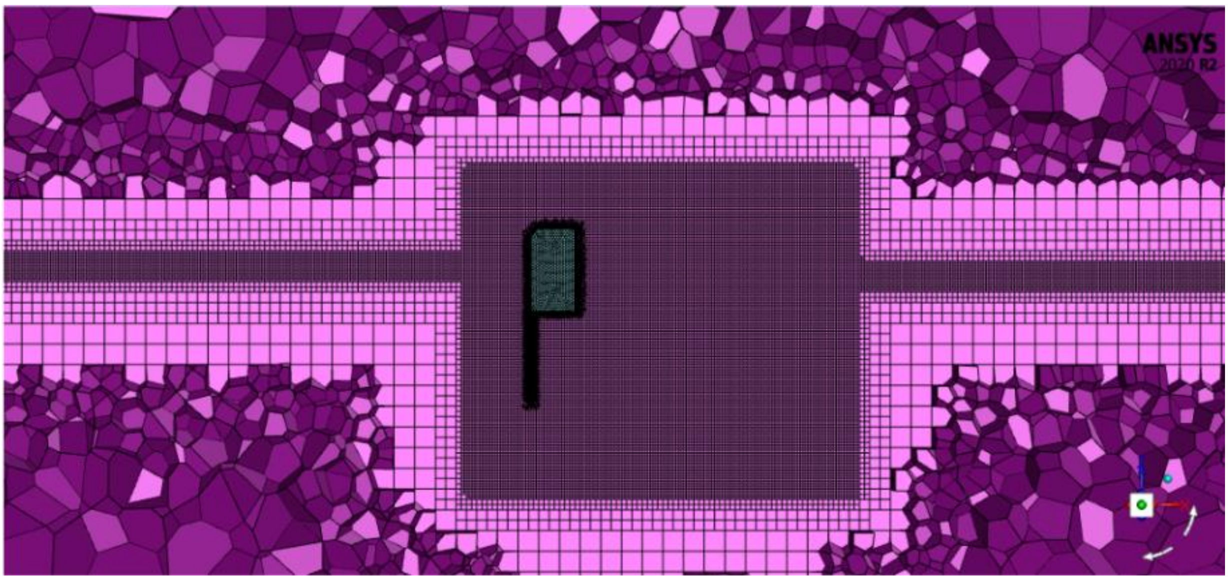

Figure 5. Hybrid mesh generated by fluent meshing and refinement zones (Source: Mata, 2022)

2.3 MATHEMATICAL MODEL AND BOUNDARY CONDITIONS

Since the object of study is a floating barrier, it involves an air-water interface, necessitating the use of a multiphase model. Consequently, a Volume of Fluid (VOF or VoF) formulation is employed, which accounts waves propagation and free surface behavior. The turbulence model is the k-ω SST. The governing equations applied are the conservation of mass, E. (1), conservation of momentum, Eq. (2) and the volume of fluid, Eq. (3).

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = 0, \tag{1}$$

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla.(\rho \vec{v}\vec{v}) = \nabla p + \nabla.\left\{\mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\overrightarrow{\nabla.v}I\right]\right\} + \rho\vec{g} + \vec{F}, \tag{2}$$

$$\frac{\partial (\rho \alpha)}{\partial t} + \nabla.(\rho \vec{v}\alpha) = -\alpha\frac{D\rho}{Dt}, \tag{3}$$

On the Eq. (1), Eq. (2) and Eq. (3), ρ (kg/m^3) is the fluid density, t (s) is time, v (m/s) are velocity components, p (N/m^2) is the pressure, μ (Pa) is the dynamic viscosity, I is the unit tensor, g is the gravity acceleration and F are the external body forces. Additionally, α is the volume of fluid, defined by the water volume by the total volume, meaning if α is 0, the cell is full of air, if α is 1, the cell is full of water and if $0 < \alpha < 1$ the cell contains a mixture. The free surface, defined by the air-water interface, is characterized by $\alpha = 0.5$.

The solution methods used were: coupled for pressure-velocity coupling; Least Squares Cell-Based method for Gradient; PRESTO! for Pressure; Second-Order Upwind for Momentum; Compressive for Volume Fraction; First Order Upwind for Turbulent Kinect Energy and Specific Dissipation Rate; and Bounded Second-Order Implicit for transient formulation. Warped-Face Gradient Correction is also enabled.

Figure 6 presents the boundary conditions applied to the model. First, the use of VoF must be set to an open channel, which will create two zones automatically filled with two different fluids, flowing to the bottom the one with higher density. Additionally, the inlet is a velocity inlet, which allows the use of an open-channel wave boundary condition for the VoF model, the outlet is a pressure-outlet with the open channel. lateral surfaces are defined as periodical, the trash boom is a non-slip wall, and the top and bottom surfaces are frictionless walls.

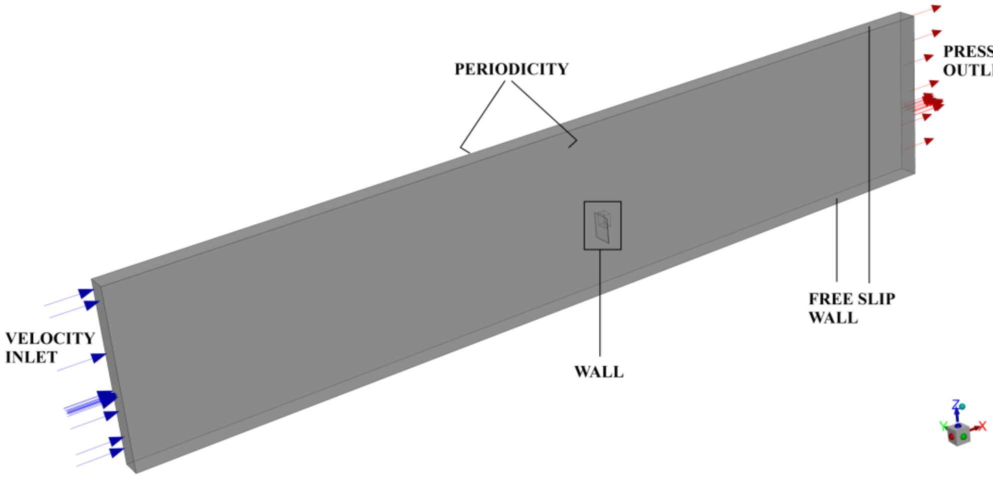

Figure 6. CFD model boundary conditions.

3. METHOD

Figure 7 shows a flow chart that shows all the sources of information for the numerical model used in the project. The environmental conditions originated in field studies, where hydraulic conditions historical series are analyzed, just like bathymetry. The information about spatial disposition and dimensions are project parameters. The velocity field is determined by the hydraulic scenario, depending on the flow rate and water level. The base hydrodynamic coefficients originated on CFD simulations. Lastly, the focus of this study centers on the highlighted yellow box. If the bathymetry indicates a shallow channel, the hydrodynamic forces will be affected by the riverbed and will apply a correction to the base hydrodynamic coefficients. However, if the channel is deep, this step is not necessary.

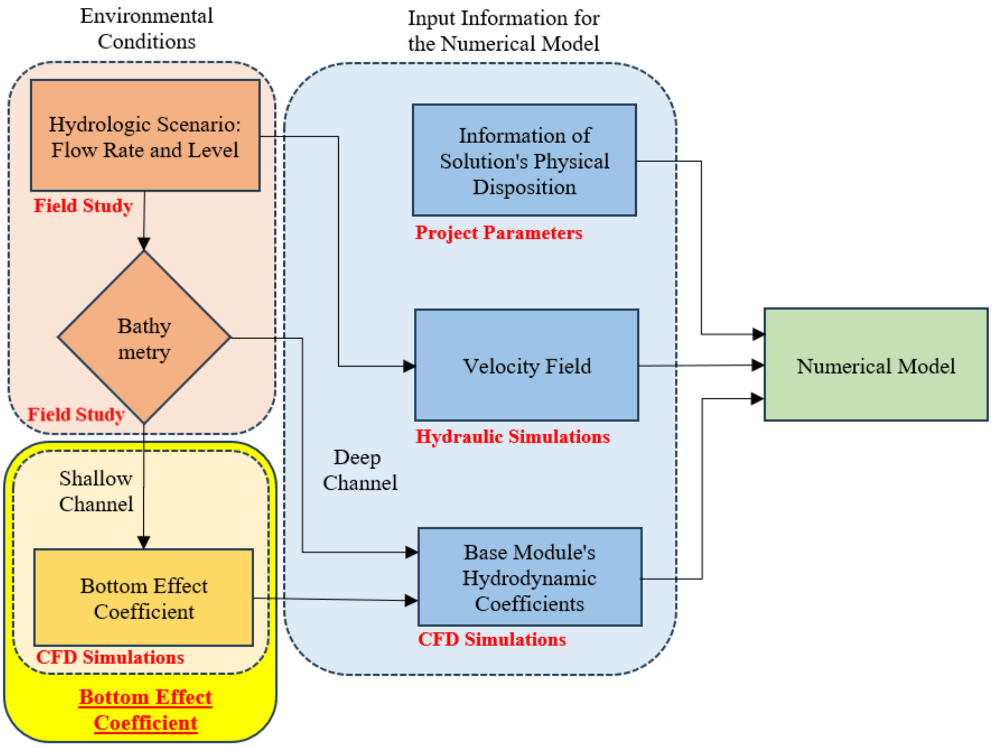

Figure 7. Flowchart for the method adopted in the present work.

Hydrodynamic forces are modelled as shown in Eq. (4), where F_i is the load, C_i is the dimensionless load factor, ρ is the water density, V is the flow velocity and A is the structure's immersed projected area. The load factors for the present work are determined through CFD simulations in various hydraulic scenarios and loaded into the code. A more detailed explanation of this numerical model is shown in (Queiroz et. al., 2022).

$$F_i = \frac{1}{2} * \rho * V^2 * A * C_i, \quad (4)$$

When a structure is exposed to hydrodynamic loads in shallow water regions, such as this, the influences the loads, intensifying their effects. In this case, it is necessary to know the degree of this influence and consider it at the calculations.

Various formulations were developed to study cases in deep water, but only a few works focus on shallow water, causing no reliable theoretical formulation to exist for a direct assessment, which is understandable once many factors influence the occurrence of such effect.

Considering these aspects, the only way to study the riverbed effect in shallow waters is to rely on experiments or numerical models. Since a validated model was available, it was used with a modification in the domain, varying the riverbed distance. This approach was used to obtain the loads in such scenarios. If the load in Eq. (4) can be thought as the reference case, i.e., a force independent from depth influence, or a load in infinite depth, then the load in any depth is given by Eq. (5), where F_{ref} is the reference load, F is the force in any given depth and C_{BE} is the riverbed effect coefficient.

$$\mathrm{F} = C_{BE} * F_{ref}, \quad (5)$$

To determine the most relevant cases to be considered, an analysis of hydraulic scenarios was made beforehand. Starting off from a field measured bathymetry, several different hydraulic scenarios were simulated with the software DELFT 3D and bathymetry and velocity field were extracted for each scenario. Considering the historic series, the level varies down to 707 m and, by analyzing the other scenarios, is possible to verify that the lines would get partially dry, touch the riverbed or get very close to it, but such scenarios are really rare and, in such cases, the flow rate would be so small that the hydrodynamic loads would be low anyway, so their magnification would not cause issues. In addition to that, the levels of flow velocity on normal operating conditions rarely surpass 1.5 m/s.

Analyzing the recurring scenarios and getting into consideration the 1-meter trash boom line draft, it was possible to notice that the closest the lines would get to the riverbed would be when the waterline-riverbed distance is 3.5 m, and the highest distance would be around 15 m, and that information was useful to determine the conditions of interest for the analysis. The CFD model has three environmental variables as inputs: flow velocity module, incidence angle from velocity on the line model (0º being the velocity aligned with the module's normal vector) and the waterline-riverbed distance. The velocity modules were 0.5, 1.0 and 1.5 m/s with incidence angles of 0º, 30º, 45º, 60º and 75º and riverbed distance from 3.5, 5, 7, 9, 11, 13 and 15 m, and the combination of these conditions amounts to 105 cases to be simulated. The reference conditions were selected to be those at a depth of 15 m, once it represents the deepest found on that channel.

Each module is under six load components, being three forces and three moments. The numerical model only uses forces as inputs, so the levels of moment will not be considered for this application, even though they are important for stability analysis made posteriorly. The three forces are the normal load, the lateral load, and the vertical load, which will influence the module's draft. For the numerical module, all three components are used as inputs, but the normal load dominates the level of line loads, and the vertical load has great influence on stability and pitch angle, aiding the stability analysis aforementioned.

The primary objective of the model is to assess the load distribution along the line and calculate the resultant force on the mooring points. The force components will be analyzed and its variation with level will be inserted in the model through the riverbed effect that was included as the riverbed effect coefficient. The base hydrodynamic loads were taken as the ones at the reference depth and the riverbed effect coefficients were calculated in relation to it, as given in Eq. (6), which is a variation of Eq. (5).

$$C_{BE} = \frac{F}{F_{ref}}, \tag{6}$$

4. RESULTS

Any floating structure causes a blockage effect on the channel, causing distortions on the velocity and pressure fields, also upstream, but mainly downstream of the module. Figure 8 and Figure 9 show the velocity and dynamic pressure fields for the cases of 3.5 m and 15 m channel depth, for 1 m/s and 0º incidence angle, but the behavior is analogous to the other hydraulic scenarios. The results are shown to the full fields and are presented with the color bar limited to the highest value of the 15 m case. Below them is shown in detail the region with higher values, which are caused by the change in depth. The alteration on pressure and velocities fields occurs especially in the region surrounding the floating structure which will lead to the magnification of the hydrodynamic loads and the alterations on dynamic behavior.

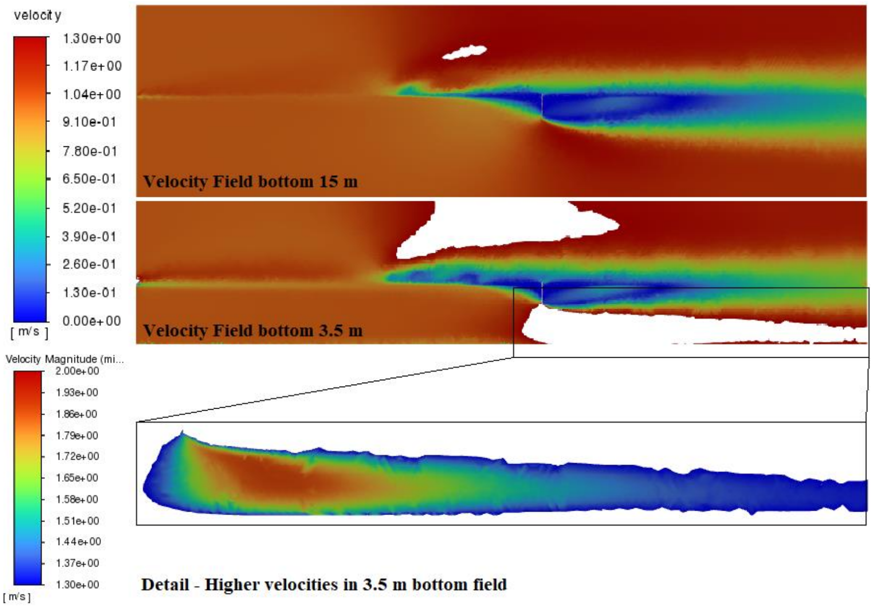

Figure 8. Velocity fields on 3.5 m and 15 m depth cases.

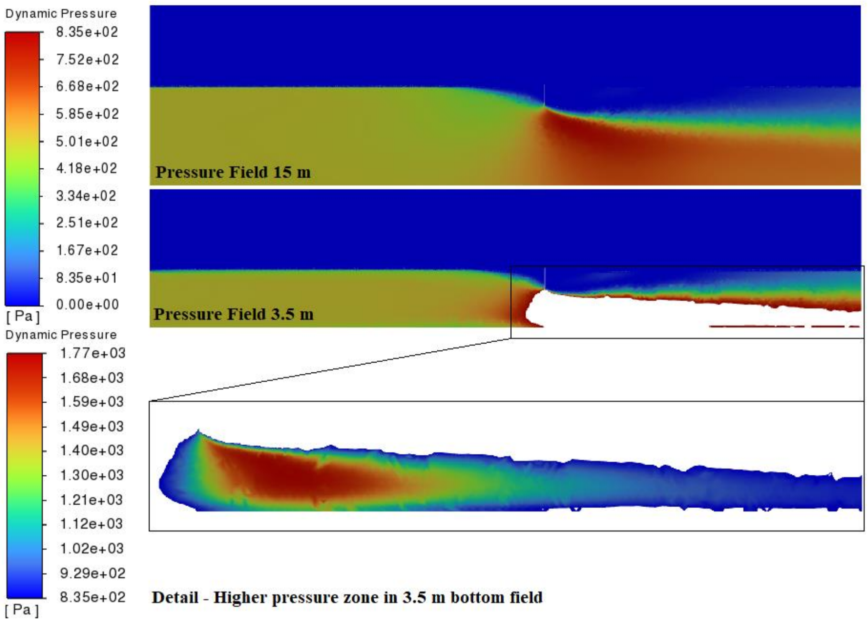

Figure 9. Pressure fields on 3.5 m and 15 m depth cases.

Figure 10 shows the water levels, calculated by the model, near the module for the conditions of 3.5 m and 15 m depth. Because of the blockage effect in the shallow channel case, the upstream water level is higher than the one for the deep channel, but the reverse occurs for the downstream water level. Since the balance between upstream and downstream levels is the one responsible for the module buoyance, if the level unbalances, then the vertical force maintaining the module afloat diminishes, and, with the higher upstream level, the module tends to suffer a pitch ahead, causing the structure to achieve equilibrium in an overturned position. That behavior is the reason the vertical load is relevant for the stability analysis. If this effect is overaccentuated, then the module might overturn to the point of completely sinking, rendering the barrier useless.

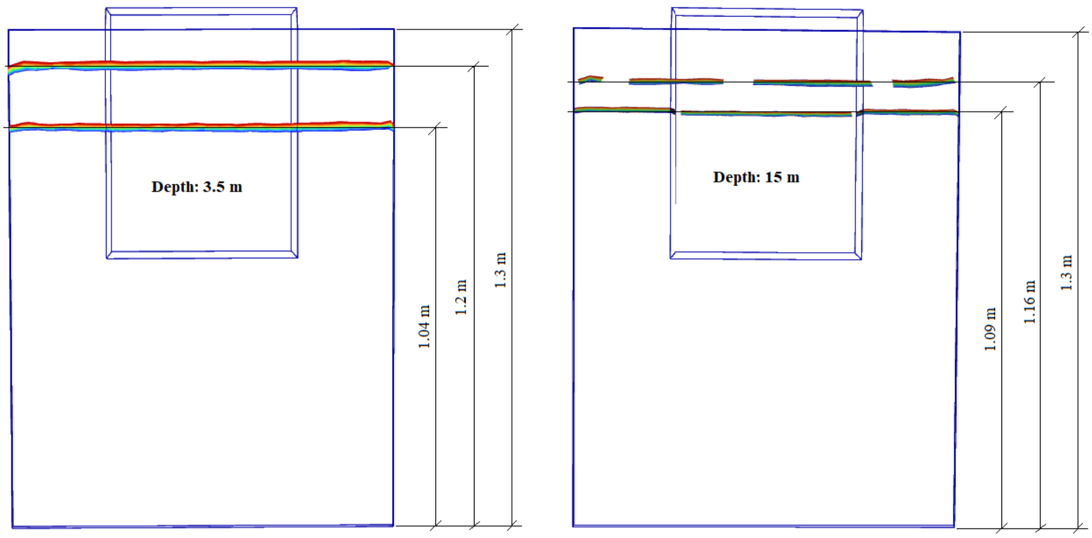

Figure 10. Water level on the module for 3.5 and 15 m depth.

With the qualitative aspects known, the data can be treated quantitatively and can be analyzed if is necessary to be loaded into the numerical model or not. For that, Figure 11 presents the three components of forces and each one respective riverbed effect coefficient, which were calculated using Eq. (6) and the results at 15 m depth as reference values. Figure 11 shows the results for the flow velocity of 1 m/s, but as the velocity and pressure fields, the behavior of forces and coefficients for the other scenarios are analogous and there was no need to show the results for every flow speed, but the analysis was made using the complete dataset, as shown in Figure 12.

The first analysis is about the values of Fx (normal load) and Fy (lateral load). Once both components are analyzed separately, it is possible to notice the difference in magnitude, with the normal component dominating. That dominance, combined to the fact that the riverbed effect coefficients for the lateral component are all around Cy = 1.0, especially in the cases this component is more relevant, makes it necessary to only correct the normal component.

Following, the vertical component grows smaller in shallower regions, meaning that if a correction were not applied to it, the calculations would only consider a slightly higher vertical component. The vertical component is the one that determines how much the module will sink, and a positive component mean that the waterline, defined in the volume of fluid method, was determined above from what it should. A round of correction for waterline levels was performed afterwards on the loads that were inserted into the model, but such step is not relevant for the riverbed effect correction factor. The numerical model is tridimensional, and since the line and mooring points follow the waterline level, unless the modules from the extremities of the line are forcibly sunken, the vertical component does not amount much on the line load distribution and reaction on the mooring points.

In conclusion, the only load component that needs riverbed effect correction into the numerical model is *Fx*, which applies very high levels of tension on the line and can be raised more than 2.5 times in shallow regions when compared to the reference case. Figure 12 presents all riverbed effect coefficients for all cases simulated, it is possible to realize that, independent from depth, the *Fx* load will have, approximately, the same behavior.

This coefficient could be dependent of three variables, being depth, flow velocity and incidence angle, but that would complicate the analysis and implementation in the model. So, it was decided, for simplicity, to make the riverbed effect coefficient independent from flow velocity and incidence angle by using mean values for each depth, instead of performing a tridimensional interpolation of values, what would increase significantly the computational cost of the model. The values used are presented in Table 1 and represent the mean values extracted from the results shown in Figure 12.

Table 1. Values of riverbed effect factor implemented into the code.

Depth [m]	**3.5**	**5**	**7**	**9**	**11**	**13**	**15**
Coefficient	2.38	1,62	1,29	1,13	1,08	1,04	1

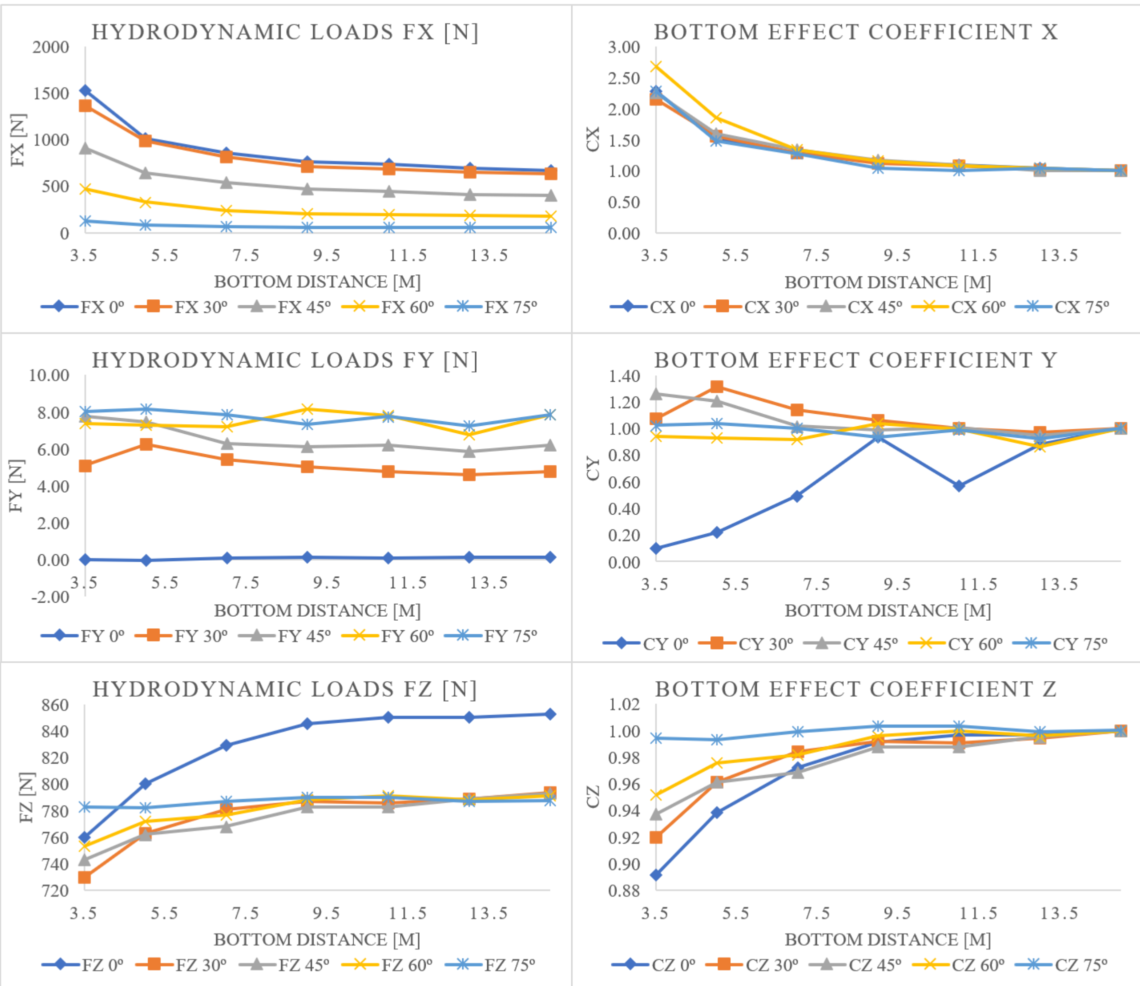

Figure 11. Force components and riverbed effect coefficients for flow velocities of 1m/s.

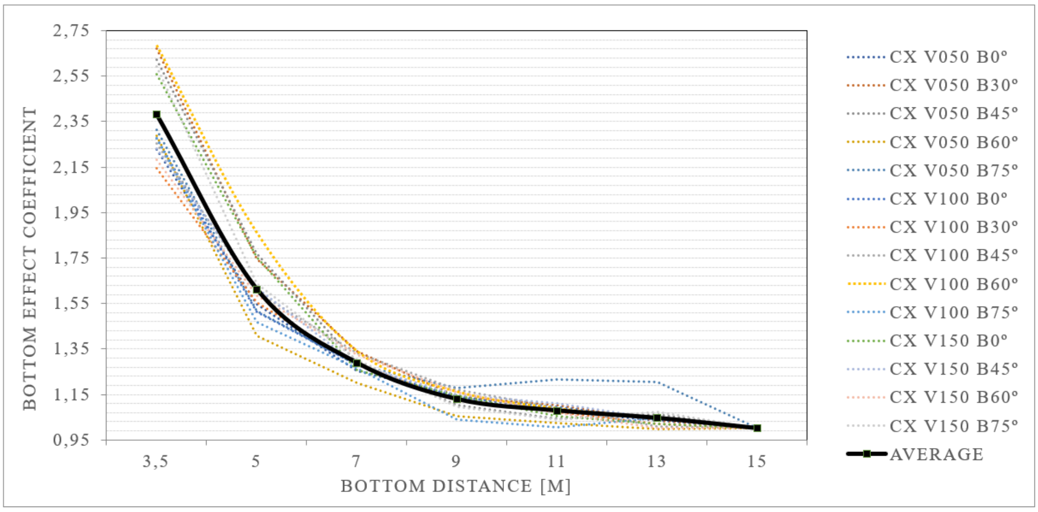

Figure 12. Riverbed effect coefficients for every scenario simulated.

5. CONCLUSION

The hydrodynamic forces are influenced by the distance between the waterline and the riverbed in shallow water regions. The blockage from the floating structure influences the velocity and pressure fields on the flow around the module, hence the alterations on the loads. The normal load component is significantly increased, up to almost 2.5 times and, since the module of the lateral component is negligible in comparison, the only correction needed into the numerical model is on the normal load component by the riverbed effect coefficient. The vertical load is influenced by the waterline-riverbed distance with reverse effect, being smaller in shallower regions. But the vertical load does not influence the line load distribution mooring reaction force, making unnecessary the correction in the numerical model. But the vertical load is relevant for stability analysis, made separately from the present work. The loads become nearly independent from depth distance at around 11 meters distance from water level to the riverbed, or, since the draft for the trash boom line modules is 1-meter, for a riverbed that is distant 11 times the module draft.

6. ACKNOWLEDGEMENTS

The authors would like to express their gratitude to the IPT laboratory technicians and researchers, EMAE for funding the project (PD-00393-0014/2019), through the Research and Development fund of Brazilian Electricity Regulatory Agency (in Portuguese, Agência Nacional de Energia Elétrica, ANEEL), and Institute for Technological Research Foundation (in Portuguese, Fundação de Apoio ao IPT, FIPT).

7. REFERENCES

Campos, G.C., Dozzi, J.L.D., Castro, F.S. and Kogishi, A.M., 2020. Debris Removal solution in a long river in São Paulo Brazil. In: Fourth International Dam World Conference, Lisbo, Portugal. v. 1. p. 109- 119

Castro, F.S., Dantas, J.L.D., Matos, G.G., Mata, F.A., Queiroz, P.D., 2022. Experimental Analysis of Floating Debris Barrier Employed on Polluted Rivers. 39th IAHR (International Association of Hydraulic Engineering and Research) World Congress - From Snow to Sea, Granada, Spain.

Mata, F.A., Castro, F.S., Dantas, J.L.D., Matos, G.G., Queiroz, P.D., 2022. A Coupled CFD and DEM Study to Evaluate a Trash-boom Debris Retentency. 39th IAHR (International Association of Hydraulic Engineering and Research) World Congress - From Snow to Sea: Granada, Spain.

Matos, G.G., Queiroz, P.D., Dantas, J.L.D., Castro, F.S., Mata, F.A., Campos, G.C., Junior, R.R. and Kogishi, A.M., 2022. Debris Removal System in the Tietê River – Brazil – A Case Study. 39th IAHR (International Association of Hydraulic Engineering and Research) World Congress -From Snow to Sea: Granada, Spain.

Queiroz, P.D., Dantas, J.L.D., Castro, F.S., Mata, F.A., Matos, G.G. and Kogishi, A.M., 2022. Development of Numerical Tool for Loads and Displacement Prediction in a River Debris Removal Solution. 39th IAHR (International Association of Hydraulic Engineering and Research) World Congress -From Snow to Sea: Granada, Spain.

8. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.