Proceedings of the 39th IAHR World Congress
19-24 June 2022
Granada, Spain

A Coupled CFD and DEM Study to Evaluate a Trash-boom Debris Retentency

Felipe Araujo da Mata(1), João Lucas Dozzi Dantas(2), Patrick Donega Queiroz(1), Felipe Santos de Castro(1) and Gabriel Galvão Matos(1)

(1) Institute for Technological Research (IPT), São Paulo, Brazil,
felipedamata@ipt.br
(2) Department Of Naval Architecture, Ocean & Marine Engineering, University of Strathclyde, Glasgow, United Kingdom,
joao.dantas@strath.ac.uk

Abstract

Debris in rivers represents a massive problem for dams, especially hydroelectric plants. Debris can cause an obstruction of the intake grid and the penstock structures increasing the plant maintenance frequency due to the possibility of machinery damage. In the Alto Tietê system, some hydraulic power plants have to work with a great reduction in the generation capacity due to pollution in Tietê river. In this work, was developed a numerical model to calculate the hydrodynamic forces and momentum of a trash-boom system from a project developed by IPT and simulate the interactions between debris, fluid medium, and the trash boom, using a coupled CFD-DEM model, to accurately estimate debris trajectory. The calculated trajectory allows the analysis of the system efficiency by retaining and directing debris in different flow conditions. By coupling the software Ansys Fluent, used for CFD, and ESSS Rocky, used for DEM, in a one-way simulation, it was possible to model a passive collector's operation under control flow conditions and evaluate the effect of debris density and size, flow velocity, trash boom geometry and depth on debris behavior and trash boom efficiency. It was found that in most combinations of these variables, in normal flow conditions, the designed system has high efficiency. This model can be used to investigate better designs for the trash booms, identifying the relationship between the drag and retainment capacity.

Keywords: CFD Simulation; DEM Simulation; Trash-Boom Efficiency; CFD-DEM Coupling; Hydrodynamic Simulation.

1. INTRODUCTION

Tietê's river, located near to São Paulo and one of the most polluted rivers in Brazil, is used for flood control in the Great São Paulo metropolitan region. Due to this proximity to this massive urban area and lack of proper sanitization, the river is highly polluted resulting in a high-flow rate of debris, composed of trash of different sizes and densities and vegetation. Tietê's river is also part of a system that is called Alto Tietê System, shown in Figure 1, and there are three important Small Hydroelectric Plants (SHP): Porto Góes, Pirapora, and Rasgão. Hydroelectric power plants located in a flow field with a high volume of debris have great reductions in the capacity of energy generation since debris can cause obstruction of the intake grid and the penstock structures, and also the potential to damage the turbine if there is no maintenance and cleaning with enough frequency (EMAE, 2018). The trash that reaches the Dams comes from all regions of the city of São Paulo, due Tiete and Pinheiros rivers. In Edgard Souza Dam, which is the first dam in the system, highlighted in Figure 1, and in some of the plants, a trash-boom system to retain floating solid debris and vegetation is being used. However, this system only retains part of the debris for short periods, resulting in a low-efficiency solution.

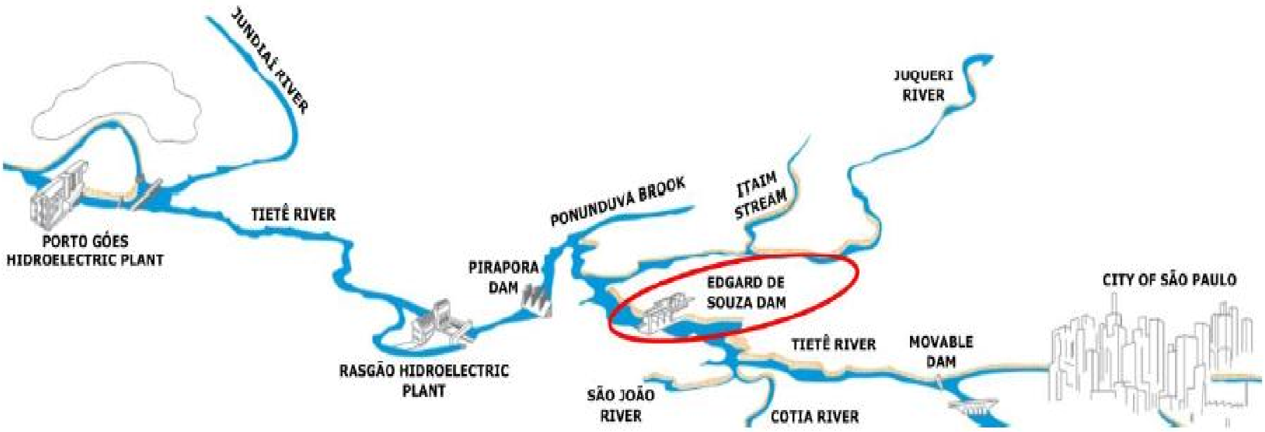

Figure 1. Alto Tietê System diagram and the location of Edgard de Souza Dam (Campos, et al., 2020).

Projects around the world, such as Mr. Trash Wheel (Mr. Trash Wheel) and The Ocean Cleanup (The Ocean Cleanup), are getting common. The basic idea of those projects is a trash-boom system that retains and passively directs floating debris for a structure that continually removes the trash floating in the water, resulting in a day removal of tons of trash (The Ocean Cleanup, 2021) (Mr. Trash Wheel). Although these solutions are successful in their cases, the Tietê river presents a higher flow rate and larger debris transport that make its use unfeasible, requiring a customized solution. These projects were used as a motivation to design the conceptual project of a whole system under the conditions of Edgard de Souza Dam. A preliminary sketch of the project is shown in Figure 2. The solution is in development by the Institute for Technological Research (IPT), supported by the São Paulo state company, EMAE (Metropolitan Water and Energy Company), which is responsible for the dam operations, within the scope of a scientific research and development project. This project was divided into four systems: Direction and retention, collection and removal, storage and transporting, and fixation and mooring (Campos, et al., 2020) (Castro, et al., 2022) (Galvão, et al., 2022).

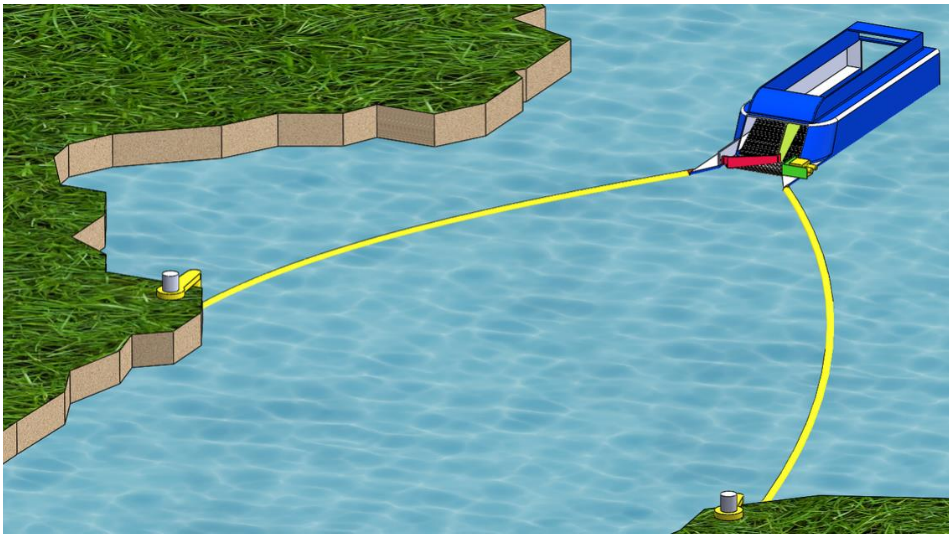

Figure 2. First sketch of the solution in development by IPT.

Computational Fluid Dynamics (CFD) models are used by the industry and academia to model and study flows with high complexity, such as turbulent multiphase free-surface flows that frequently occur in naval and hydraulic engineering. In the past 20 years, computer hardware and some CFD softwares evolved in a way that the simulations can run faster, making it possible to simulate more complex cases with high accuracy. In simulations that a free surface is present, a high-quality mesh with small elements, especially in the region of the free surface is needed, increasing the computational cost. Besides that, a well-defined free surface is necessary to avoid numerical diffusion in the mixture region and to have quality numerical results to feed the particle model that will be coupled.

The Discrete Element Method (DEM) is a model that predicts the behavior of bulk solids, known as granular media. This model can be coupled with computational methods of fluid dynamics to simulate the iteration particle-fluid.

In projects focused on debris removal from rivers, to evaluate if the system efficiently collects debris in a river, it is necessary to simulate a particle model coupled with the fluid model to check the conditions in which the system works and the conditions in which there is a drop in efficiency or total inefficiency.
This article aims in the direction and retention system, which is responsible for preventing the passage of debris through the river and carrying them passively to the collection system, describing the methodology used to design and evaluate the efficiency of the trash-boom line by a coupled CFD and DEM simulation.

2. MATHEMATICAL MODEL

CFD and DEM are numerical models responsible for solving complex problems to predict the behavior of fluids and solid particles, respectively and both can be coupled to predict how a particle will behave in a flow field.

2.1 CFD Model

The first analysis is focused on CFD. A multiphase model is necessary, then a Volume of Fluid (VoF) formulation is adopted. This model is also used in cases where wave propagation and free surface behavior are important. Another model that is required is the turbulence model. In this case, the fluid flow is governed by Reynolds averaged Navier-Stokes equations. The turbulence model $SST\ k-\omega$ is a model that has both an accurate near-wall formulation and free stream independence in the far-field from $k-\omega$ and in addition accounts for the transport of the turbulence shear stress (SST) in the definition of the turbulent viscosity (Menter, 1994). The fluid governing equations (continuity [1], momentum [2], and VoF [3]), turbulence kinetic energy [4], and specific dissipation rate [5] can be expressed as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad [1]$$

where ρ is fluid density (kgm^{-3}), t is time (s), x_i is cartesian coordinate in i direction;

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\left[(\mu + \mu_t)\frac{\partial u_i}{\partial x_i}\right] \qquad [2]$$

where u_i is velocity components(ms^{-1}), p is pressure (Nm^{-2}), μ is dynamic viscosity (Pas) and μ_t is turbulent viscosity (Pas);

$$\frac{\partial(\rho\alpha)}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i \alpha) = -\alpha\frac{D\rho}{Dt} \qquad [3]$$

where α is the volume of fluid ($\alpha = \frac{V_{water}}{V_{total}}$). If $\alpha = 0$ the cell is complete of air, while $\alpha = 1$ the cell is complete of water. The free surface is characterized by $\alpha = 0.5$;

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_j} = P_k - \beta^* \rho k\omega + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_j}\right] \qquad [4]$$

where k is turbulence kinetic energy (m^{2}s^{-2}), ω is specific energy dissipation rate (s^{-1}), P_k is production term in $SST\ k-\omega$ model, β is coefficient in k and σ_k is coefficient in k;

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \gamma P_w - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_w \mu_t)\frac{\partial \omega}{\partial x_j}\right] + 2\rho(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial dx_j}\frac{\partial \omega}{\partial x_j} \qquad [5]$$

where γ is coefficient in ω equation and F_1 is blending factor in $SST\ k-\omega$ model.
As shown in equations [4] and [5], $SST\ k-\omega$ model has some coefficients ($\beta, \sigma_k, \sigma_\omega, \ldots$) that are obtained by the formula [6]:

$$\phi = F_1\phi_1 + (1-F_1)\phi_2 \qquad [6]$$

where ϕ represents the constant in the $SST\ k-\omega$ model, ϕ_1 is the constant in $k-\omega$ model and ϕ_2 is the constant in $k-\varepsilon$ model.
The CFD simulation returns the hydrodynamic forces and momentums that are used for a numerical tool for loads and displacement (Queiroz et al., 2022). The results of force are turned into a dimensionless coefficient C_i, according to the equation [7].

$$C_i = \frac{F_i}{{}^1\!/_2\, \rho v^2 A} \quad [7]$$

where F_i is the hydrodynamic force in a given direction and A is the projected area of the structure.

2.2 DEM Model

After the CFD analysis, the results are exported for a DEM simulation. Debris are treated as solid particles with different shape formats, sizes, and materials. DEM does not solve the continuum equations of motion. The equation [8] shows the force balance:

$$\sum F_{net} = m \frac{dv_p}{dt} \quad [8]$$

where F_{net} are all the forces that act on each particle, m is particle mass and v_p is particle velocity.
Particles are subject to surface force, gravity force, and contact force. The surface force can be expressed as:

$$F_s = F_d + F_p + F_{vm} \quad [9]$$

where F_d is the drag force, F_p is pressure force and F_{vm} is virtual mass force. The formulas of those forces are shown in [10]:

$$F_d = \frac{1}{2} C_d \rho A_p v_s |v_s|$$
$$F_p = -V_p \nabla p_s \quad [10]$$
$$F_{vm} = C_{vm} \rho V_p \left(\frac{Dv}{Dt} - \frac{dv_p}{dt}\right)$$

where C_d is the drag coefficient, A_p is particle projected area, v_s is slip velocity, V_p is particle volume, p_s is fluid static pressure and C_{vm} is the virtual mass coefficient. The contact force, generated by the contact among particles and between a particle and a solid surface or a boundary, can be written as:

$$F_c = \sum_{Particles} F_{contact} + \sum_{boundaries} F_{contact} \quad [11]$$

Particles are allowed to rotate and translate, the model for particle rotation is:

$$\frac{dL_p}{dt} = \frac{d(I_p \omega_p)}{dt} = \sum_{Particles} T_{contact} + \sum_{boundaries} T_{contact} \quad [12]$$

where L_p is particle's angular momentum, I_p is particle's inertia momentum, ω_p is particle's angular velocity and T is the contact torque, which formula is written in [10].

$$T_{contact} = r_c \times F_{contact} - \mu r |r_c| |F_{contact}| \frac{\omega_p}{|\omega_p|} \quad [13]$$

where r_c is the distance between the particle's mass center to contact force and μ is the rolling friction coefficient.

3. CFD SIMULATION MODEL

CFD simulations can be divided into three stages: Geometry, mesh, and setup and boundary conditions. The software used for CFD simulation, both setup and mesh, in this work is Ansys fluent. The simulation focused on analyzing the hydrodynamic forces and momentums

3.1. Geometry

Motivated by commercial models of trash-boom, a geometry was designed. The model is composed of a boom coupled to a barrier. The complete direction and retention system is composed of trash-boom lines of 120 meters, which makes it a problem due to the computational cost of that kind of simulation. For simplification purposes, a 1-meter long section is assumed to fit the computational costs by assuming a periodic condition at the lateral boundaries that can emulate part of an extensive trash boom line. The domain dimensions were defined after an analysis of the Tietê region, being defined 25 m upstream, 20 m downstream, 3 m above the free surface, and 10 m below. Geometry and fluid domain are shown in Figure 3.

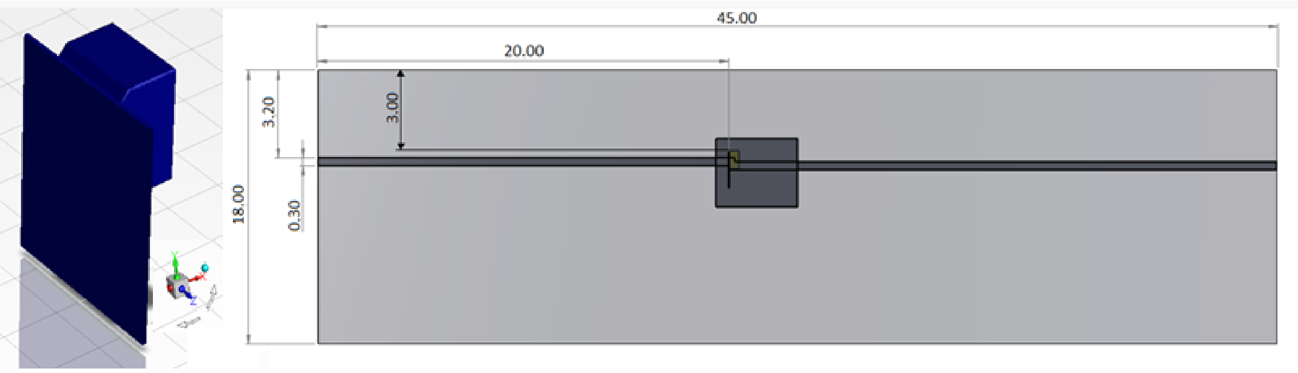

Figure 3. First designed trash-boom geometry and the detailing of the fluid domain and bodies of influence zones used for mesh refinement, with dimensions in meters.

3.2. Mesh

The size of the elements is defined by the dimension of the fluid dynamic phenomena that must be modeled. The smaller the flow characteristics, the smaller the elements must be. However, a high number of elements generates a high computational cost, requiring mesh refinements in certain regions, as shown in Figure 4, such as the free surface region, which models the interface between air and water, and in the close to the structure, which models small characteristics of the flow, such as the emission of vortices. The hybrid mesh generated for the models, from the combination of hexahedral and polyhedral elements is also shown in Figure 4. Three different meshes were generated with 800k, 1.2mil, and 2.3mil elements.

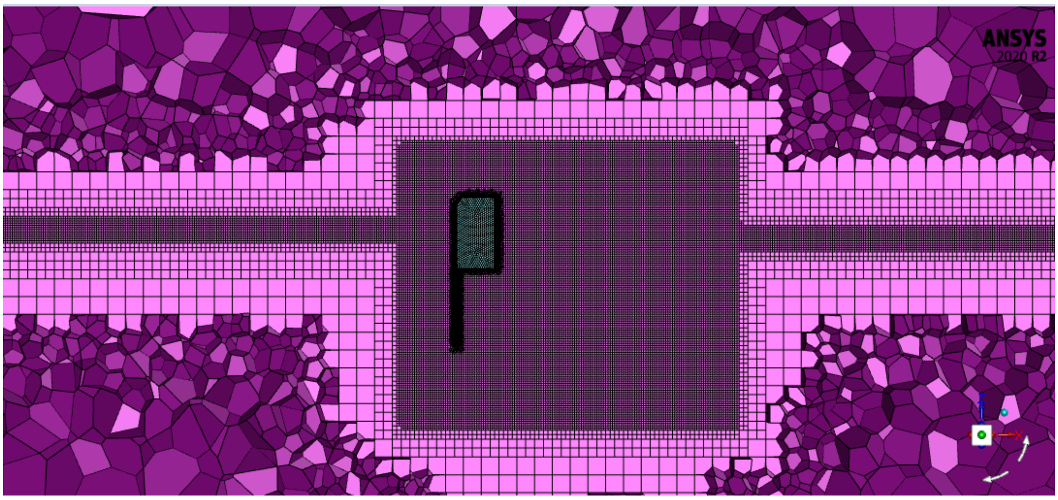

Figure 4. Hybrid mesh generated by Ansys fluent and its refinement zones.

3.3. Setup and Boundary Conditions

The solution setup and definition of the boundary conditions of the control volume are created. For the multiphase model, the VoF model with open channel is used, which allows the creation of a boundary condition that already considers the region filled by each fluid. There is also the $SST\ k - \omega$ turbulence model.
Finally, solution methods are selected, such as Coupled Scheme with Coupled with Volume Fractions for pressure-velocity coupling, in spatial discretization, some methods need to be chosen, is used 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.
For this case, the boundary conditions, shown in Figure 5, are Velocity-Inlet, which allows the use of the Open-channel Wave BC model for the multiphase VoF model; The outlet surface is defined as pressure outlet with the open channel; The side walls were defined as periodicity; The upper and lower walls were defined as sliding (frictionless) walls, while the boom wall was defined as a non-slip wall.

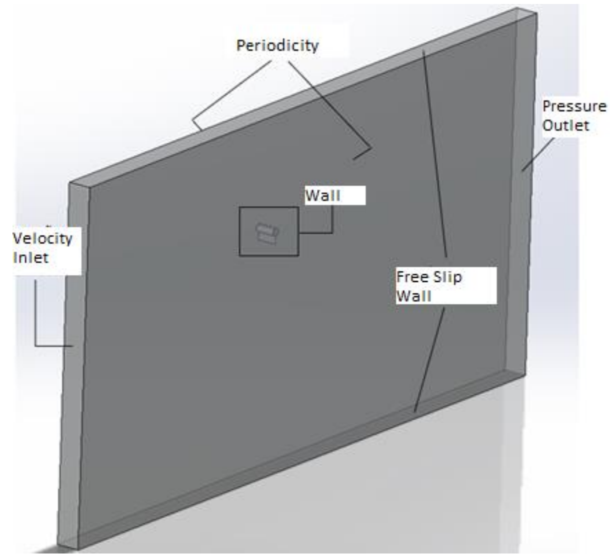

Figure 5. Boundary conditions for CFD simulation.

4. DEM SIMULATION MODEL

To evaluate the efficiency of the designed trash boom, the CFD simulation is connected with Rocky DEM software to simulate how particles are behaving in the flow field. The ESSS Rocky import the flow field calculated in the last iteration from fluent simulation, such as pressure gradient, velocity flow field, and VoF information for each cell, Figure 6 shows phases loaded from fluent.
The CFD coupling mode used is the one-Way Fluent Steady State, which means that the fluid influences the behavior of the particles, however, particles don’t interfere in the flow field, and the flow field is considered a steady-state. This mode was chosen due to the lower computation cost when compared to the two-way and that is not expected that the debris effect will change significantly the results in this study scope.
The simulations were carried out by several different geometries and densities of particles, representing the principal type of debris that can be found in the Edgard de Souza Dam (Campos et al., 2020). The particle shapes are simplified for spheres and cylinders of different sizes that resemble the distribution of the real debris, due to the impossibility to model all shapes and sizes of the trash. This study used the following DEM models for particle interaction: Hysteretic Linear Spring for normal force, Linear Spring Coulomb Limit for tangential force, adhesive force is constant, and Constant Moment is used for rolling resistance model.

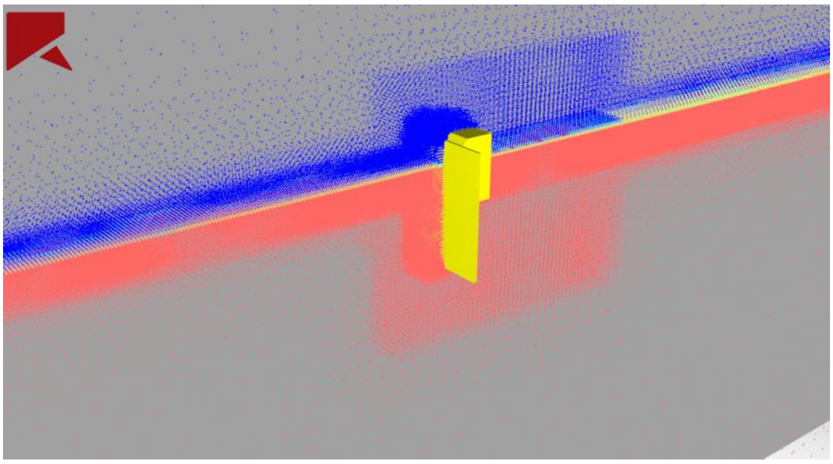

Figure 6. Fluid phases are shown in Rocky after importing data from fluent.

All scenarios were simulated to evaluate the efficiency relating the particle density to the flow velocity and angle of incidence, and also to evaluate de retention capacity for large accumulations of debris in front of the trash boom. The post-processing of Rocky allows you to observe whether debris has passed through and, in addition, it allows the generation of regions that the software can compute some information, for example, the amount of debris, total mass, residence time, etc.

5. RESULTS AND DISCUSSIONS

Several scenarios have to be simulated in order to create a database in which its behavior can be analyzed in different situations. The scenarios are classified according to two parameters: the average velocity of the flow that is acting on the module and the angle of incidence of the velocity vector with the normal vector of the trash-boom barrier.

5.1. CFD Simulation Cases

The speed limits were analyzed through the scenarios developed in a numerical-hydraulic model developed (Queiroz, et al., 2022). It was noticed that flow velocity never exceeds 2 m/s, rarely reaching values very close to this limit. Thus, four flow velocity values were defined: 0.5 m/s; 1.0 m/s; 1.5 m/s; and 2.0 m/s. These values are enough to make the interpolation of the database when you want to make scenario estimates with intermediate values.
The next parameter taken into account is the angle of incidence of the velocity in the module. It is defined as ranging from 0º, when the velocity is perpendicular to the module, which is the critical case of operation, up to 90º, a condition in which the velocity is parallel to the module. To generate a load variation curve at a given speed, as the incidence varies, angles of 0º, 15º, 30º, 45º, 60º, 75º, and 80º were assumed.

Table 1. Simulated cases.

Flow Velocity (ms^{-1})	Angle of incidence - β (0)						
0.50	0	15	30	45	60	75	80
1.00	0	15	30	45	60	75	80
1.50	0	15	30	45	60	75	80
2.00	0	15	30	45	60	75	80

5.2. Flow Topology

The post-processing of CFD shows the impact of flow speed and angle of incidence on the free-surface level. The impact of the flow velocity is that for higher flow velocity, the free surface level upstream increases. While the impact of the angle, for a higher angle of incidence, less the free surface level upstream increase. For the 2.00 ms^{-1} and low angle (between 0^0 and 45^0), the free surface reaches the limit of the trash-boom barrier. For higher angles there is no imminence of failure due to the passage of debris over the barrier, these effects are shown in Figure 7.

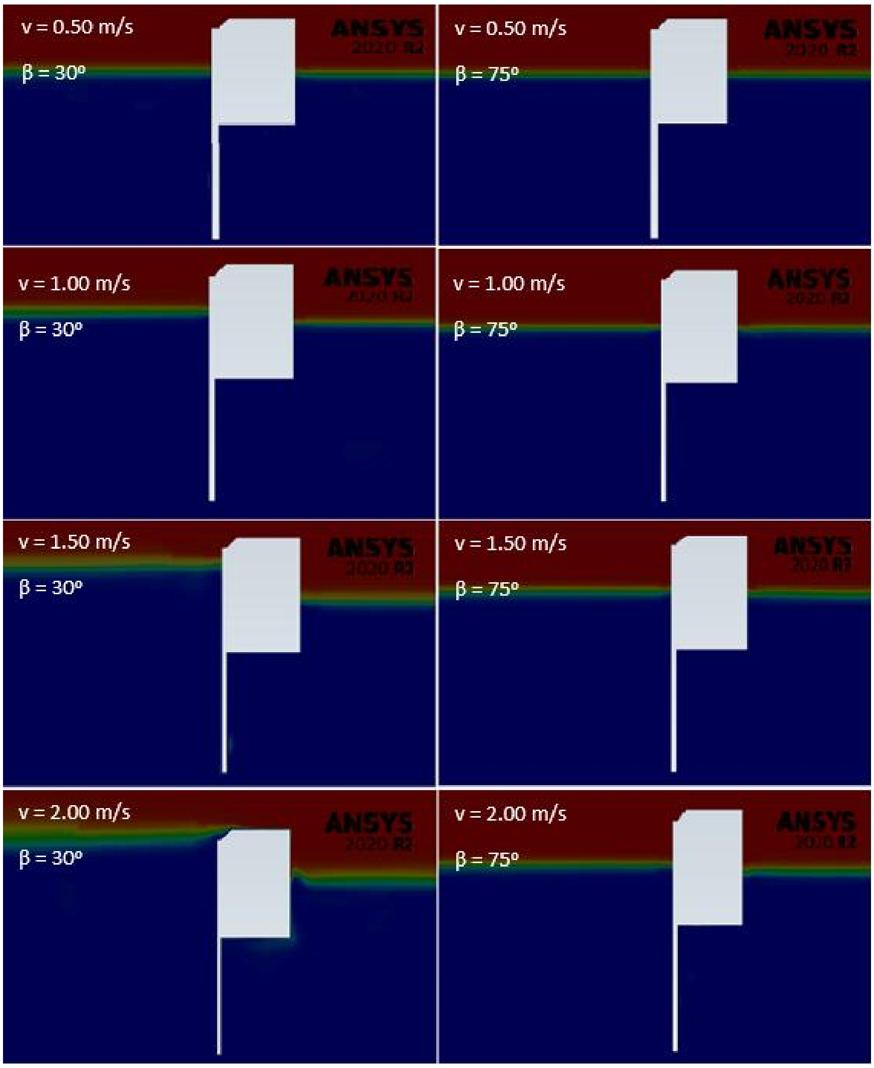

Figure 7. Phases contour of different cases that show the impact of velocity and incidence angle.

5.3. Hydrodynamics Force and Momentums

CFD simulations are also important to estimate the hydrodynamics forces and momentums. Axial (x-direction) and lateral (y-directions) coefficients are important to design structural parts, and momentum on the y-axis to assess the module stability and rotation. Graphics showing the nondimensional coefficients (C_x and C_y) and momentum (My) are shown in Figure 8, and Figure 9. Momentum in the y-direction as a function of the angle of incidence., respectively.

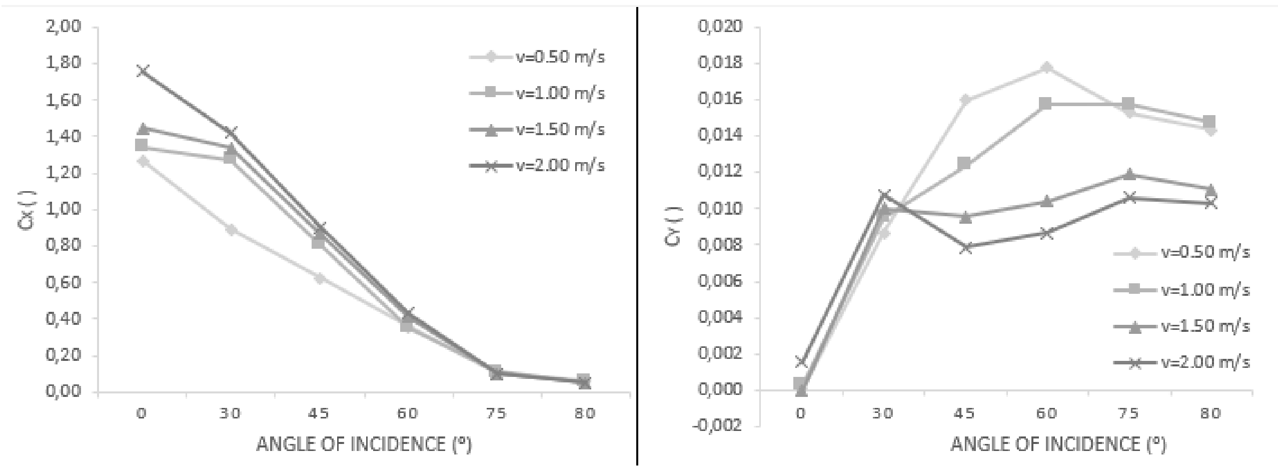

Figure 8. Axial force coefficient (left), x-direction, and. lateral force coefficient (right), y-direction, in function of the angle of incidence.

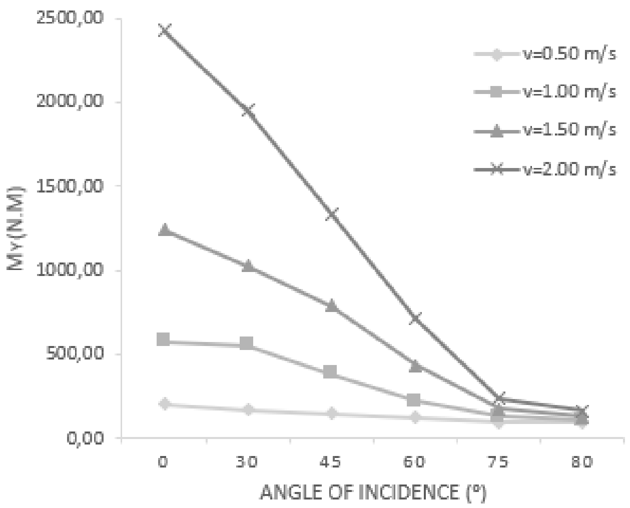

Figure 9. Momentum in the y-direction as a function of the angle of incidence.

5.4. DEM Simulation

The post-processing of CFD shows how free surface level is affected by velocity and incidence angle. The post-processing of DEM simulation shows a correlation among flow velocity, angle of incidence, and particle density. For low angles, particles with a density higher than 950 kgm^{-3} pass under the barrier, and for all flow scenarios and particles with a density lower than 950 kgm^{-3}, the barrier efficiency is 100%. Table 2 shows the barrier efficiency. Efficiency is defined as a relation of retained particles over the total particles released in the flow field.

Table 2. Trash-boom efficiency (%) for each density in different flow conditions.

Velocity (ms^{-1})	β (°)	Density (kgm^{-3})			
		950	970	990	1000
0.50	30	100	100	92	0
0.50	45	100	100	100	0
0.50	75	100	100	100	0
0.50	80	100	100	100	0
1.00	30	100	81	59	0
1.00	45	100	100	82	0
1.00	75	100	100	100	0
1.00	80	100	100	100	0
1.50	30	90	39	0	0
1.50	45	100	100	83	0
1.50	75	100	100	100	0
1.50	80	100	100	100	0
2.00	30	77*	48*	0*	0
2.00	45	74	54	0	0
2.00	75	100	100	64	0
2.00	80	100	96	87	0

* Debris also passing over the barrier

6. CONCLUSIONS

A model to predict hydrodynamic forces and momentums of a designed trash boom have been developed through a CFD simulation. A total of 28 CFD simulations varying flow velocity and angle of incidence, were run and the results generated a database that has been used in a simulator (Queiroz, et al., 2022) that estimates the whole trash-boom line stress and strain, also drawing the line curvature according to the flow condition. The coupled simulation results can be drawn as follows:

i. The coupled CFD-DEM allowed the verification of the efficiency of the direction and retention system in both functions, retaining and directing. The coupled model also returned the relation among all variable conditions (flow speed, angle of incidence, and particle density) with the solution efficiency. A total of 64 coupled model simulations were conducted, for each velocity, angle, and density. The results allow us to conclude that: As the angle of incidence increases, the retention efficiency of the system also increases. This behavior is maintained for high velocity, this means that most of the line has 100% efficiency.
ii. For high speed and low angle, it was noticed that the waterline reaches the barrier upper border that resulted in debris passing over the system, warning that, for these conditions, a grid over the barrier is needed to retain debris without generating a considerable increase of drag.

Given the flow velocity is normally less than 1.50 ms^{-1} and the minimum angle is 45^0 (Queiroz, et al., 2022), the minimum efficiency is 82%, also for debris with a density of 990 kgm^{-3}, which also has low quantities.
To validate the results of this work, they are compared with data from scale-model experiments conducted in the towing tank at the Institute for Technological Research (IPT) (Castro, et al., 2022).
This work will be used for a future paper that focuses on other barrier designs. The model was developed with a flat barrier, the idea of a future work adds a device that changes the format to a ‘L’ format for example looking to improve the efficiency for extremum cases.

7. ACKNOWLEDGEMENTS

The authors acknowledge the whole IPT Naval and Oceanic Engineering Laboratory, technicians, researchers, and all people who make the laboratory viable, EMAE for funding the project (PD-00393-0014/2019) through the Research and Development fund of Brazilian Electricity Regulatory Agency (ANEEL), and Institute for Technological Research Foundation (FIPT). The authors also want to thank the funding from FINEP (agreement 01.18.0082.00 - CTInfra 2014), for the simulation hardware, and Fapesp (PDIP Process 2017/50343-2), for the simulation software.

8. REFERENCES

Campos, G.C., Dantas, J.D., Castro, F.S., Kogishi, A.M. (2020). Debris removal solution in a long river in São Paulo – Brazil. Fourth International Dam World Conference, (p. 11). Lisbon.

Castro, F.S., Dantas, J.L., Mata, F.A., Queiroz, P.D., Matos, G.G., Kogishi, A. M. (2022). Experimental Analysis of Floating Debris Barrier Employed on Polluted Rivers. IAHR 2022. Granada.

EMAE. (2018). Relatório Anual da Administração - 2018. Available from: http://www.emae.com.br/arquivos/internet/Investidores/Informacoes%20Financeiras/Informacoes%20Anuais_Trimestrais/DF%20EMAE%202018.pdf. Accessed in Dez/2021;

EMAE. (n.d.). Empresa Metropolitana de Águas e Energia. Sistema Hidráulico. Avaiable from: http://www.emae.com.br/conteudo.asp?id=Sistema-Hidraulico. Acessed in Dez/2021;

Matos, G.G., Castro, F.S., Dantas, J.L., Mata, F.A.,Queiroz, P. D., Junior, R.R., Kogishi, A. M. (2022). Debris Removal System: A Case Study. IAHR 2022. Granada.

Menter, F. (1994, August). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32, pp. 1598-1605.

Mr. Trash Wheel. (n.d.). A Proven Solution to Ocean Plastics. Available from: https://www.mrtrashwheel.com. Accessed in Dez/2021;

Mr. Trash Wheel. (n.d.). Description of The Mr.Trash Wheel. Available from https://www.mrtrashwheel.com/technology/. Accessed in Dez/2021;

Queiroz, P. D., Dantas, J. D., Mata, F. A., Castro, F. S., & Matos, G. G. (2022). Development of Numerical Tool for Loads and Displacement Prediction in a River Debris Removal Solution. IAHR 2022. Granada.

The Ocean Cleanup. (2021, May 20). The Numbers Behind our Catch. Available from https://theoceancleanup.com/updates/the-numbers-behind-our-catch/. Accessed in Dez/2021;

The Ocean Cleanup. (n.d.). How it Works: The Interceptor. Available from https://theoceancleanup.com/rivers/. Accessed in Dez/2021.