International Journal of
Science and Engineering Investigations vol. 7, issue 72, January 2018
ISSN: 2251-8843

# Application of a Meshless CFD Method for a Vertical Slot Fish Pass via 3D Simulation

Daniela dos Santos da Mata Gomes[1], Gabriel de Carvalho Nascimento[2]
[1,2]Universidade Federal Fluminense, Niterói, Rio de Janeiro
([1]dmata@id.uff.br, [2]gabrielcn@id.uff.br)

***Abstract***- Constructing a dam disrupts longitudinal connectivity of a river, changing its ecosystem. Thus, the dams are obstacles which fish are not able to get over and migrate to a successful life cycle. Fish passes become key elements to allow that journey upstream. The design criteria are based on the estimate of flow patterns. The aim of this work is to evaluate a meshless CFD method, Smoothed Particle Hydrodynamics (SPH), to simulate the free surface flow in a vertical slot fish pass. Its accuracy is investigated by comparing the velocity field in a scale model test, done in a previous literature, to a 3D SPH simulation conducted with the open source software DualSPHysics. As a result, velocity field of the scale model test compares well with the result of the numerical simulation which concludes that SPH is a promising method for free surface flow simulation in a fish pass.

***Keywords***- *Vertical Slot, SPH, Free Surface Flow, CFD*

## I. INTRODUCTION

One of the most severe impacts of hydroelectric dams is the interruption of fish migrations, which decreases the population size of fish species and sometimes leads to extinction. For this reason, efficient fish passes are required to restore fish passage [1]. Vertical slot fish passes (VSF) are the most efficient type since it is considered a multispecies facility and interacts well with variations in water levels. VSF consists of a sloping channel, divided into pools by baffles. Therefore, water flows between the baffles generating a jet while energy is dissipated in the next pool [2].

In order to provide an optimal design, guidelines recommend investigations of velocity field to determine an attraction flow at the fish pass entrance [1], [3], [4]. Researchers usually conduct such investigations with Computational Fluid Dynamics (CFD). In this case, to simulate the free surface flow in a VSF.

The goal of the present study is to use numerical simulation to evaluate the benefits and accuracy of a meshless CFD method, Smoothed Particle Hydrodynamics, by investigating velocity field in the system. In this method, physical particles with mass and density are used to represent water and, unlike methods that require mesh, SPH interacts well with the interface between air and water, and no mesh implies that no information is lost [5], [6]. Moreover, only a few published investigations document SPH simulations applied to fish passes [5], [7].

In the next sections, the SPH method will be presented, focusing on detailed knowledge of flow characteristics in a vertical slot fish pass versus a scale model test done in previous literature.

## II. METHOD

### A. General

The Smoothed Particle Hydrodynamics (SPH) is a (Lagrangian) meshless method developed in 1977 to solve astrophysical problems that mesh-based methods were difficult to apply [8]. SPH method decomposes the continuum into a set of particles which discretized Navier-Stokes equations are integrated for each of these particles by an interpolation function, converging the integral form into finite summation, providing estimated values at a specific point [9], [10].

One of the main advantages of this method is to simulate the motions without any mesh, due to the kernel interpolation function. The SPH method has been presented advances in the field of CFD, with continuous modifications to reliable results for its use in engineering problems [11].

### B. Case Study

The tests were conducted in a previous literature [12] on a scale model developed at FCTH-USP laboratory to represent the VSF of the Small Hydroelectric Plant, Paranatinga II, located at Culuene river, Mato Grosso, Brazil.

The 1:7 scale model contains 10 pools, 4.5% slope, and the water discharge was fixed in 2.61m³/s (converted to a prototype scale). More details can be found in [12]. The geometry dimensions are detailed in Figure 1:

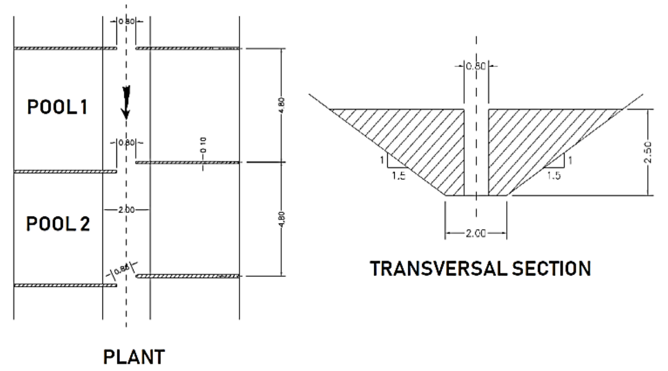



Figure 1. Plant and Transversal Section of VSF Paranatinga II. Source: [12]

*C. Numerical Model and Simulation*

The numerical model geometry reproduces the scale model, includes 9 pools, an inlet transition zone, and a reservoir. Among the pool, there are upstream pools (pools 1 and 2) and typical pools (other pools). Results analysis will focus on pool 6, considered the representative pool of the system, once it has enough pools, the flow is considered as periodic [5].

As SPH is a meshless method, boundary conditions can only be applied by predefined movement of particles located at the domain contours, which denote one of the main disadvantages of this method. The numerical simulation reproduces the hydraulic boundary conditions of the scale model, reintroducing the particles which have left the domain to the first pool. Figure 2 shows the model geometry:

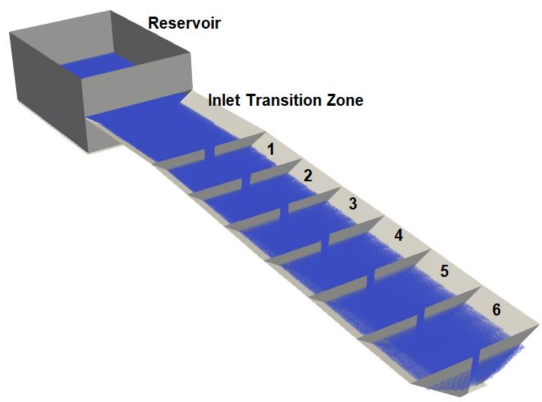



Figure 2. Model geometry.

The numerical model geometry includes 6 pools, an inlet transition zone, and a reservoir. The numerical CFD simulation is carried out using the open source software, DualSPHysics, developed from researchers at the Johns Hopkins University (US), University of Vigo (Spain) and the University Of Manchester (UK). The numerical code is based on the SPH method, developed to study free surface flow. It was used GPU (Graphics processing unit) to perform 670534 particles on DualSPHysics to simulate water. SPH formulation is detailed in DualSPHysics Users Guide [13].

## III. Results

To visualize the results was used Paraview software, which has generated post-processing images based on the numerical results imported from the DualSPHysics code after the processing time. The results show varying flow velocities, with low-velocity recirculation regions and also high-velocity main flow regions.

A differentiated behavior in the upstream pools is observed. The flow stabilization begins from pool 3, which is reached in pool 5, which has a flow pattern very similar to the pool 6. The same occurred in the case study scale model, where pool 6 was considered a representative pool of the system.

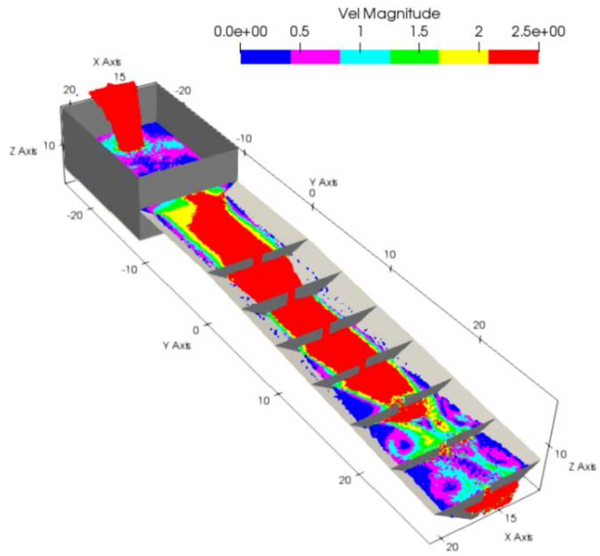



Figure 3. Flow velocity ($ms^{-1}$).

The numerical model velocities are compared to the scale model values converted to prototype scale (Figures 4 and 5).

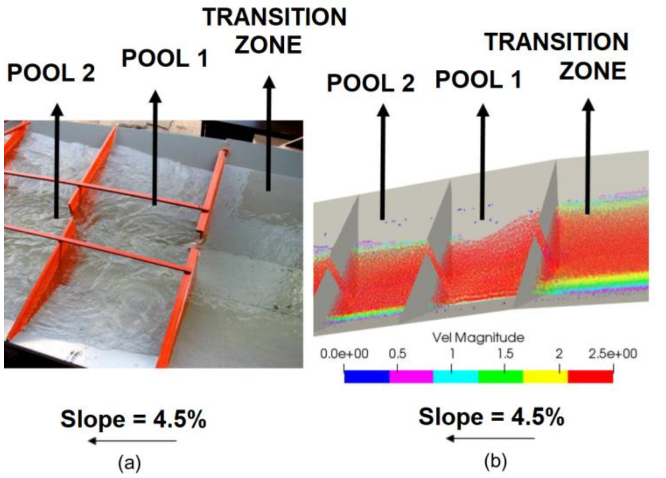



Figure 4. Upstream pools: (a) Scale model. Source: [12] and (b) Numerical model.

In Figure 4 it is possible to make a comparison for both computational and scale model flow in the upstream pools.

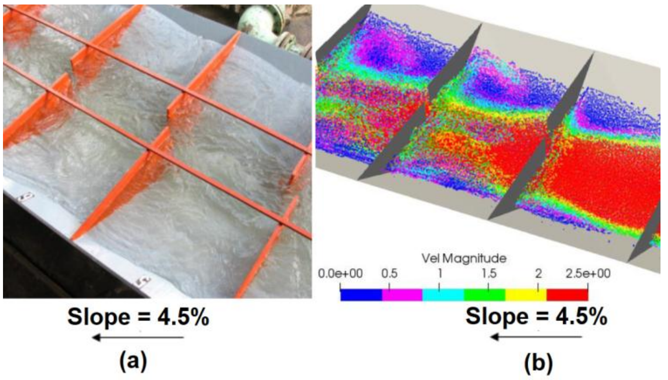
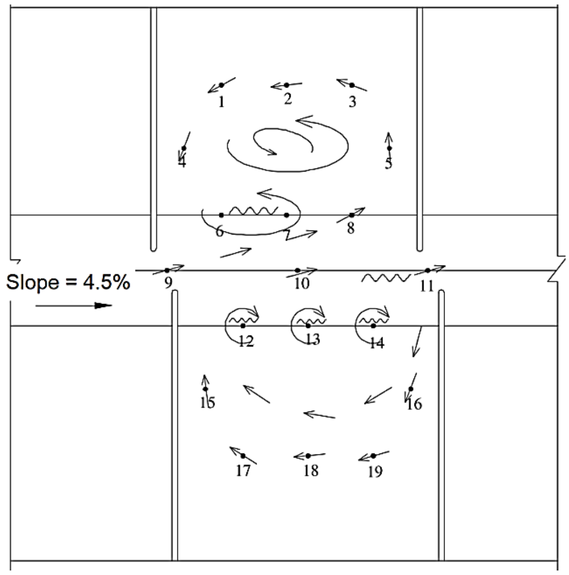


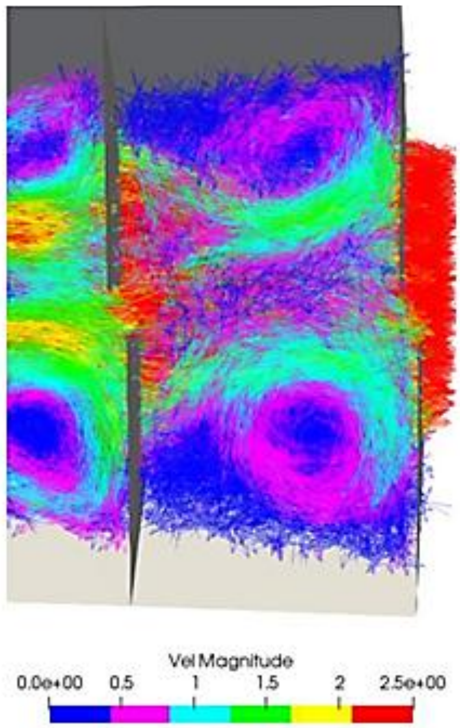

Figure 5. Typical pools: (a) Scale model. Source: [12] and (b) Numerical model.

A permanent regime is obtained in the typical pools according to Figure 5.

The flow velocities and flow directions were evaluated at pool 6. In Figure 6, it is possible to compare, in the representative pool, the results in both scale and computational model.




(a)




(b)

Figure 6. Velocity field (ms-1) in the representative pool: (a) Scale model. Source: [12] and (b) Numerical model.

Recirculation areas in the SPH model are very similar to those of the scale model. Moreover, 19 points of flow velocity were evaluated in the scale model [12] in Figure 6 (a), the same is done to the numerical model in Table 1.

TABLE I. AVERAGE AND MAXIMUM FLOW VELOCITIES IN PROTOTYPE SCALE

| | *Average Velocity (m/s)* | | *Maximum Velocity (m/s)* | |
|---|---|---|---|---|
| Points | *Scale Model* [12] | *Numerical Model* | *Scale Model* [12] | *Numerical Model* |
| 1 | 0.52 | 0.25 | 0.58 | 0.45 |
| 2 | 0.98 | 1.00 | 1.06 | 1.25 |
| 3 | 0.53 | 0.65 | 0.64 | 0.85 |
| 4 | 0.70 | 0.65 | 0.75 | 0.85 |
| 5 | 0.74 | 1.00 | 0.91 | 1.25 |
| 6 | 0.31 | 0.25 | 0.35 | 0.45 |
| 7 | 0.54 | 0.65 | 0.73 | 0.85 |
| 8 | 0.72 | 1.45 | 0.78 | 1.65 |
| 9 | 1.70 | 2.30 | 2.02 | 2.50 |
| 10 | 2.14 | 2.30 | 2.24 | 2.50 |
| 11 | 1.75 | 2.30 | 1.90 | 2.50 |
| 12 | 0.47 | 0.65 | 0.51 | 0.85 |
| 13 | 0.35 | 0.65 | 0.44 | 0.85 |
| 14 | 0.46 | 0.65 | 0.49 | 0.85 |
| 15 | 0.82 | 1.00 | 0.83 | 1.25 |
| 16 | 0.83 | 1.00 | 0.91 | 1.25 |
| 17 | 0.97 | 0.65 | 1.02 | 0.85 |
| 18 | 1.12 | 0.65 | 1.17 | 0.85 |
| 19 | 0.70 | 1.00 | 0.74 | 1.25 |

Notably, points 9, 10 and 11 correspond to the main flow, where the downstream flow is clearly seen. Also, the upper recirculation zone was more compatible with the scale model than the lower zone.

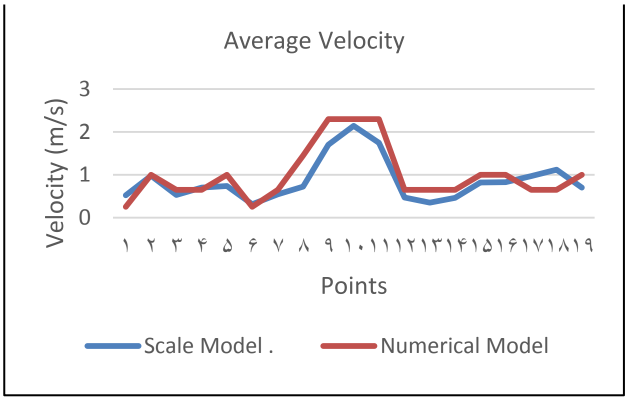
In Figure 7, two comparative velocity charts, maximum and average flow velocity, are presented to each of the 19 points.

In what concerns the upper recirculation zones and the dominant flow, the same flow patterns are noted in scale model and computational model. As shown in Figure 7, flow patterns vary slightly in the lower recirculation zone.

## IV. CONCLUSIONS

After The SPH method reproduced the hydrodynamic conditions of the fish pass in a relatively small computational cost (1522 seconds), saving time that would be taken to generate the mesh in other non-meshless methods.

The results of the scale model tests are quite compatible with the results of the numerical simulation performed in the present study. The flow pattern was the same in the upper recirculation zone and in the main flow, with a small variation in the flow pattern in the lower recirculation zone.



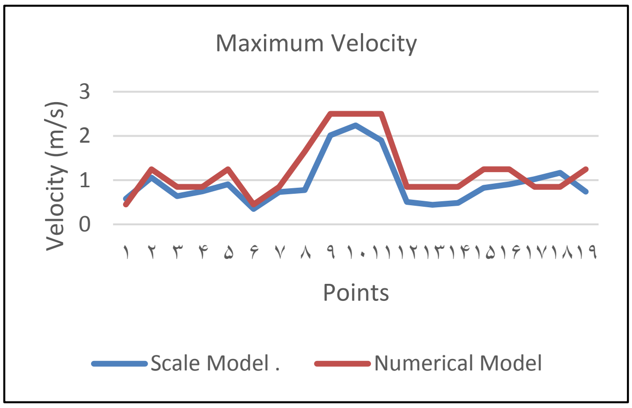




Figure 7. Average and Maximum Flow Velocities

Numerical models are highly desirable as a complementary tool to the scale models due to the importance of knowing the behavior of the fish in each zone of flow, and also for a design optimization. The behavior of different species can be associated with the hydraulic variables that can be estimated using the SPH method, saving time and financial resources.

In light of this, SPH is a promising method for engineering problems involving the complex interaction between air and water-free surface.

## REFERENCES


[1] FAO/DVWK, *Fish Passes: Design, Dimensions and Monitoring*. Rome: Food and Agriculture Organization of the United Nations, 2002.

[2] C. Katopodis, "Introduction to fishway design," *Oceans*, p. 67, 1992.

[3] C. H. Clay, *Design of Fishways and other Fish Facilities*, 2nd ed. CRC Press, 1995.

[4] U.S. Fish and Wildlife Service, "Fish Passage Engineering Design Criteria," Northeast Region R5, Hadley, Massachusetts, 2017.

[5] D. Violeau, *Fluid Mechanics and the SPH Method*, 1st ed. Oxford: Oxford University Press, 2012.

[6] A. Tafuni, J. M. Domínguez, R. Vacondio, and A. J. C. Crespo, "Accurate and efficient SPH open boundary conditions for real 3-D engineering problems," no. March, pp. 3–4, 2017.

[7] R. Marivela, "Applications of the SPH Model to the Design of Fishways," *33rd IAHR Congr.*, 2009.

[8] L. B. Lucy, "A numerical approach to the testing of the fission hypothesis," *Astron. J.*, vol. 82, no. 12, pp. 1013–1024, 1977.

[9] J. M. Domínguez, "DualSPHysics: Towards High Performance Computing using SPH technique," Universidad de Vigo, 2014.

[10] X. Dong, G. R. Liu, Z. Li, and W. Zeng, "Smoothed particle hydrodynamics (SPH) modeling of shot peening process," *J. Comput. Methods Sci. Eng.*, no. October, pp. 1–27, 2017.

[11] DualSPHysics Team, "Users Guide for DualSPHysics code DualSPHysics v4.0 April 2016," no. April, 2016.

[12] R. Junho, "Migrações ascedentes de peixes neotropicais e hidrelétricas: Proteção a jusante de turbinas e vertedouros e sistemas de transposição," Escola Politécnica da Universidade de São Paulo, 2008.

[13] G. Fourtakas and B. D. Rogers, "Modelling multi-phase liquid-sediment scour and resuspension induced by rapid flows using Smoothed Particle Hydrodynamics (SPH) accelerated with a Graphics Processing Unit (GPU)," *Adv. Water Resour.*, vol. 92, pp. 186–199, 2016.