TPM Vol. 32, No. S3, 2025
ISSN: 1972-6325
https://www.tpmap.org/

Open Access

# DESIGN OF STRUCTURES WITH PROFILES IN RECTANGULAR SHAPES FOR THE REDUCTION OF VELOCITIES AND PRESSURES USING AUTODESK CFD SOFTWARE FOR HYDRAULIC WORKS

LINARES DURAND JHOSMER ANDERSSON [A], SILVA ACOSTA ANGEL JONATHAN [B], CARMONA ARTEAGA ABEL [C]
[A,B,C]UNIVERSIDAD PRIVADA DEL NORTE, AV. TINGO MARÍA 1122, LIMA 15083, PERÚ.

ABSTRACT: The study focused on the design of structures that use rectangular profiles to reduce water velocities and pressures in the context of streams and rivers due to flooding. To achieve this, the design was developed in the Autodesk inventor software, providing an easy design for its physical construction, and subsequently seven simulations were carried out using the Autodesk CFD software, each of which represents a different type of speed in an increasing manner, in the following order, 10cm/s, 20cm/s, 30cm/s, 40cm/s, 50cm/s, 100cm/s and 200cm/s. These simulations were carried out with the purpose of comparing and evaluating the impact of different velocities on the dissipative structure and how it acts on the fluid. The results showed how the use of rectangular profiles influences the speed of water flow and the distribution of pressures, where the optimality of the design and its efficiency for reducing speeds and pressures were estimated.
Keywords: Autodesk CFD, Fluid simulation, Hydraulic works, Rectangular profiles

## 1. INTRODUCTION

Throughout the world, the complex and harmful consequences of global warming have been witnessed, with populations highly impacted by climate variations and catastrophic natural events [1]. To be more specific, in Peru, the El Niño phenomenon brings about intense and prolonged precipitation, typically between December and March. In steep terrains with pronounced slopes, this can result in landslides and soil erosion. Heavy rains during the annual rainy season, coupled with the soil and rock characteristics, lead to rapid weathering, alteration, and decomposition, forming accumulations of loose material in the ravines. These materials become saturated and are swept away by the rains [2]. Unfortunately, in our country, riversides and ravine margins have been encroached upon and built upon. This increasing urbanization in river areas not only challenges sustainable urban planning but also, due to recurrent climate phenomena, exposes communities to different types of dangers and serious risks in the buildings already constructed in these areas. For these reasons, the existence of dissipative structures in rivers and ravines is necessary to reduce the water's energy as it flows through these places, and thus, in some way, mitigate natural disasters. Unfortunately, these structures or dissipative structure designs require experimental studies, and the construction of small-scale concrete models is time-consuming and costly. As a result, adequate works have not been designed. However, in recent years, thanks to improvements in hardware and software, new computer programs have been designed to solve and analyze the behavior and kinematics of fluids. The field of fluid mechanics responsible for studying these velocities and pressures is Computational Fluid Dynamics (CFD).

## 2. RESEARCH OBJECTIVE

The objective of this study is to construct a design for a dissipative structure with rectangular profiles using Autodesk Inventor, which will then be simulated in Autodesk CFD Computational Fluid Dynamics software. The purpose is to obtain an efficient prototype capable of reducing velocities and pressures in streams and rivers when they become inundated with water.

## 3. THEORETICAL FRAMEWORK

**Autodesk CFD (Computational Fluid Dynamics):**
Autodesk CFD software specializes in Computational Fluid Dynamics, generating computer simulations that are valuable for engineers and analysts in accurately predicting the behavior of liquids and gases. It is a highly useful program as it creates digital prototypes to reduce the costly need for physical models [3].

**Autodesk Inventor:**
Autodesk Inventor is a 3D CAD software that offers a range of tools for product simulation, documentation, and high-quality mechanical design. This program effectively combines parametric design capabilities, direct modeling, freeform design, and rule-based design. Additionally, it includes integrated tools for various applications such as sheet metal design, structures, pipes and tubes, cables and harnesses, presentations, rendering, simulation, machinery design, among others, making it a comprehensive solution for design professionals [4].
**Drag Forces:**
Drag force is the force exerted on a body by a fluid resisting movement in the direction of the body's displacement. At the frontal surface of the sphere lies the stagnation point and the turbulent wake behind it [5]. Figure 01 illustrates the dynamic pressure on drag.

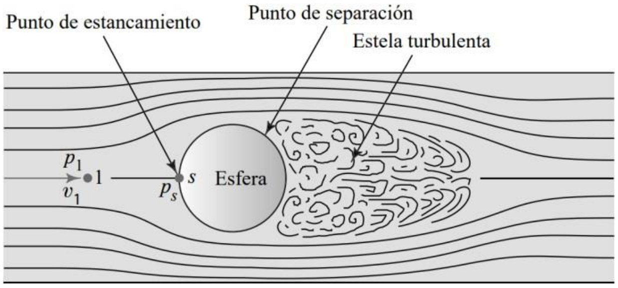

Fig. 01: Sphere in a fluid stream showing the stagnation point on the frontal surface and the turbulent wake behind it [5]

**Reynolds Number**
It is a dimensionless measure used in fluid mechanics and transport phenomena analysis to describe fluid behavior. Being a dimensionless number, it is based on a relationship or comparison. Its relevance lies in providing information about how a fluid flows, which is essential in its research and analysis. When the Reynolds number (Re) is less than or equal to 2000, it is known as laminar flow; when Re is greater, it is referred to as turbulent flow [6].

$$\mathrm{Re} = \frac{D * v * \rho}{\mu} \quad (1)$$

Where:
Re = Reynolds number
D = Diameter of the pipe
$v$ = Velocity of the liquid
$\rho$ = Density of the liquid
$\mu$ = Viscosity of the liquid

**Navier-Stokes Equations:**
The Navier-Stokes equations are a set of nonlinear partial differential equations that model the motion of viscous, incompressible, and homogeneous fluids in mathematical terms [7].
These equations are expressed as follows:

$$\rho \frac{\partial u}{\partial t} + \rho(u \cdot \nabla)u - \rho f + \nabla p - \mu \Delta u = 0, en\left[0, \infty\right) \times \mathbf{R}^m \quad (2)$$

$$\nabla \cdot u = 0 \quad (3)$$

$$u(x,0) = a_0(x) \quad (4)$$

**Describing each term:**
**The temporal derivative of velocity**

$$\frac{\partial u}{\partial t} = \left( \frac{\partial}{\partial t} u_1, \frac{\partial}{\partial t} u_2, \ldots\ldots, \frac{\partial}{\partial t} u_m \right) \quad (5)$$

**The transport term**

$$(u \cdot \nabla)u = \left( \sum_{i=1}^{m} u_i \frac{\partial}{\partial x_i} u_1, \sum_{i=1}^{m} u_i \frac{\partial}{\partial x_i} u_2, \ldots\ldots, \sum_{i=1}^{m} u_i \frac{\partial}{\partial x_i} u_m \right) (6)$$

**The pressure gradient**

$$\nabla p = \left( \frac{\partial p}{\partial x_1}, \frac{\partial p}{\partial x_2}, \ldots, \frac{\partial p}{\partial x_m} \right) \tag{7}$$

**The external force acting on the fluid**

$$f = (f_1, f_2, \ldots, f_m) \tag{8}$$

**The incompressibility of the fluid**

$$\nabla \cdot u = \sum_{i=1}^{m} \frac{\partial u_i}{\partial x_i} = 0 \tag{9}$$

## 4. METHODOLOGY

Below is the detailed process carried out in Autodesk Inventor and CFD programs. First, the area of interest was created, and rectangular profiles were placed using Autodesk Inventor software. In this case, a rectangle of 750 mm (millimeters) by 500 mm was used, as this software only operates in these units. The dimensions and spacings of the 3 designs of rectangular structures used can be seen here as well: 2 in the form of a "T" on the sides, 4 in a staggered form with unequal dimensions, and 1 symmetrically staggered in the center, as shown in Figure 02.

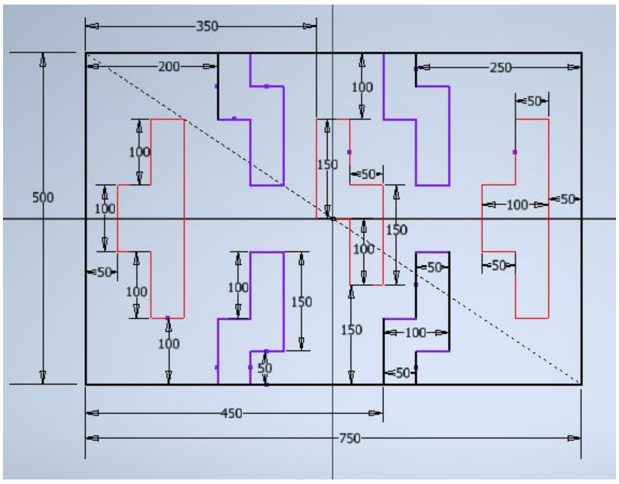


Fig. 02: 2D design showing the dimensions and spacings of the rectangular profiles.
Then, in Figure 03, the finished design is shown and ready for execution in Autodesk CFD software.

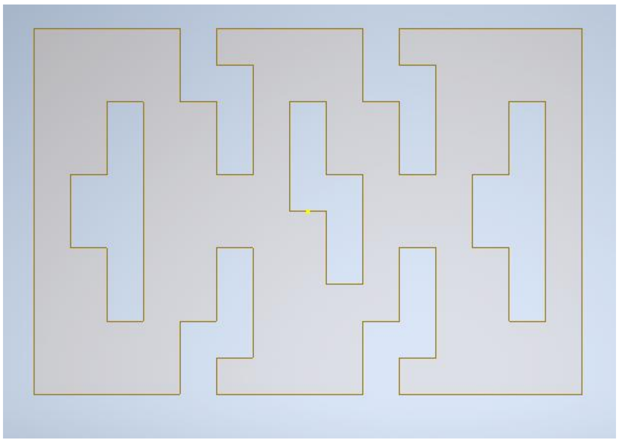

Fig. 03: 2D design of the rectangular profiles

Subsequently, the design was transferred to Autodesk CFD software to begin the simulation. In Figure 04, the "material" command was chosen to select the type of fluid. In this case, "water" was selected and "Apply" was pressed.

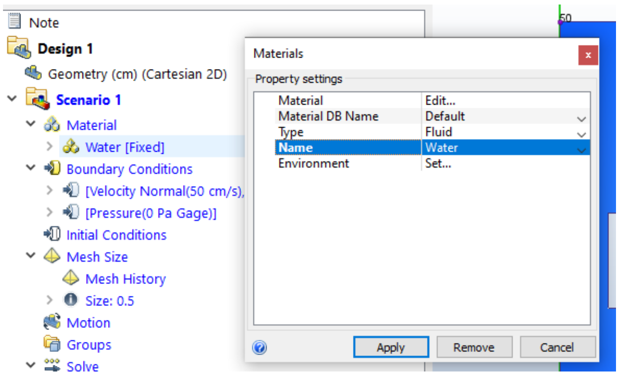

Fig. 04: Configuring the simulation fluid

In Figure 05, the "Boundary conditions" command is chosen to create the desired velocities, with units changed to cm/s. For velocity No. 01, the initial part of the design is first selected, then 10 is entered into "velocity Magnitude," followed by "Apply".

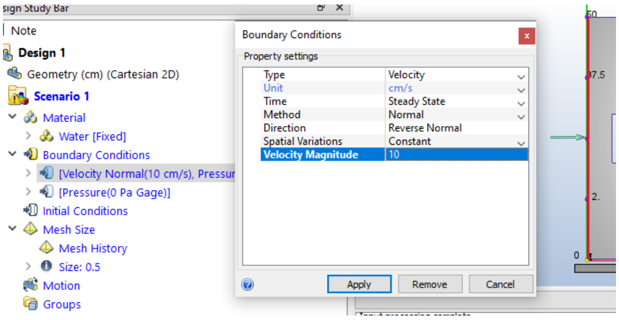

Fig. 05: Setting up the simulation velocity

In Figure 06, the "Boundary conditions" command is continued to create the desired pressures. For the inlet pressure, the initial part of the design is selected, with units changed to Pa (pascals), entering 10 into "Pressure," and then clicking "Apply."

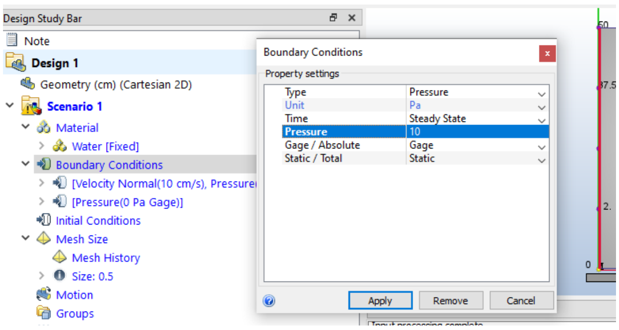

Fig. 06: Setting up the simulation inlet pressure

In Figure 07, the "Boundary conditions" command is continued to create the desired pressures. For the outlet pressure, the final part of the design is selected, with units changed to Pa, entering 0 into "Pressure," and then clicking "Apply".

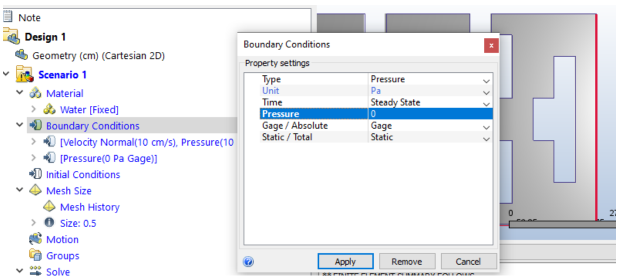

Fig. 07: Setting up the simulation outlet pressure

Then, the "Mesh size" command is selected, followed by clicking "edit." In "Type," "Automatic" is chosen, followed by clicking "Automatic Size" and then "apply," as shown in Figure 08.

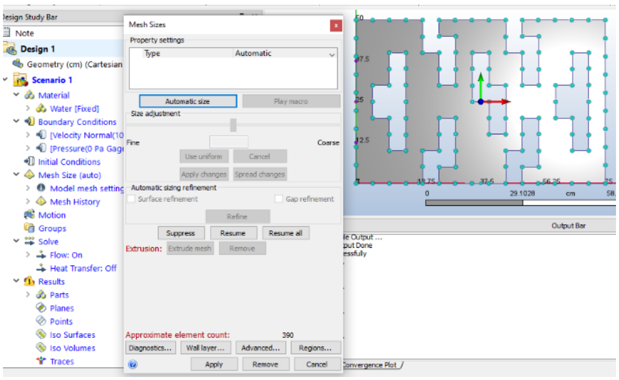

Fig. 08: Mesh setup for the simulation

For changing the mesh, in Figure 09, you deselect "model mesh" and click "edit," then select "Manual" under "Type," click "yes," and finally "apply." Then, expand the "size" command, right-click on it, press "edit," manually change 0.5 in "element size," and then click "apply".

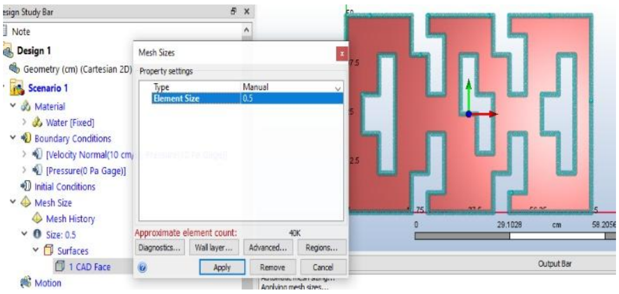

Fig. 09: Changing the simulation mesh

Finally, click on the "Solve" command, enter 1000 in "iterations to Run." Don't forget to enter 0 in "Continue From" and deactivate temperature in "Result quantities." Work in turbulent mode under "physics" and click "turbulence." To initiate the simulation, press "Solve," as shown in Figure 10.

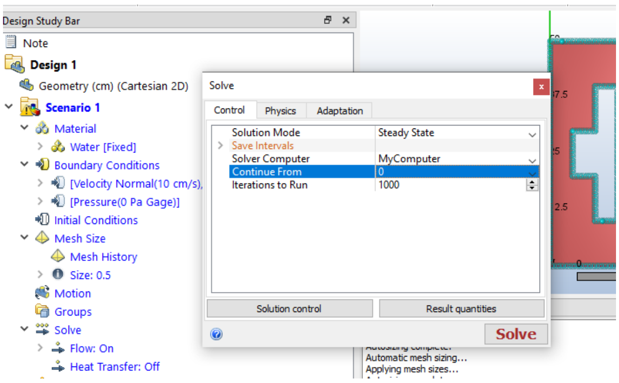


Fig. 10: Completion of setup and simulation in the CFD program

## 5. RESULTS

By comparing velocities configured in the Autodesk CFD software, the analysis of water flow behavior from left to right was obtained as follows:

As shown in Figure 11, the flow is at a velocity of 10cm/s. In certain points, the water flow has accelerated (the red color represents the highest velocity magnitude) where the structure has dissipated its energy.

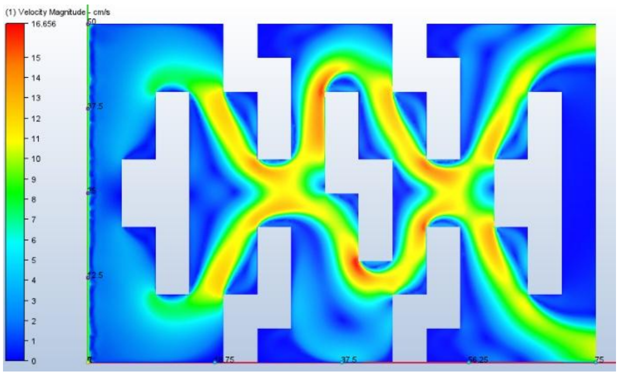

Fig. 11: Simulation with velocity No. 1 of 10cm/s

In Figure 12, the flow is at a velocity of 20 cm/s, showing a stream of light blue color indicating low velocity, with an approximate range of 8 to 10 cm/s as shown in the data table (on the left side of the image). In comparison, the blue color represents the lowest velocities.

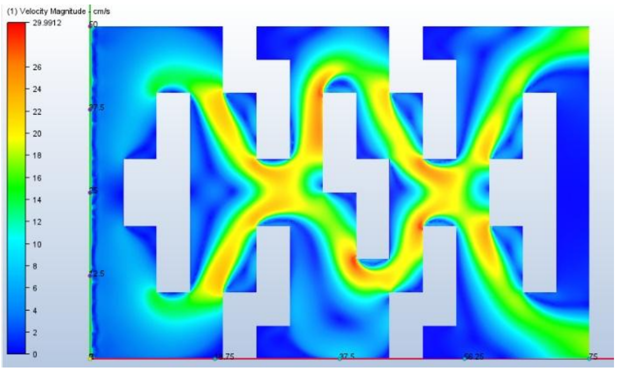

Fig. 12: Simulation with velocity No. 2 of 20cm/s

In Figure 13, the flow is at a velocity of 30 cm/s. It is observed that as the inlet velocity increases, the impacts on the structure are greater, with its value increasing in the data table. There is a relationship between these aspects.

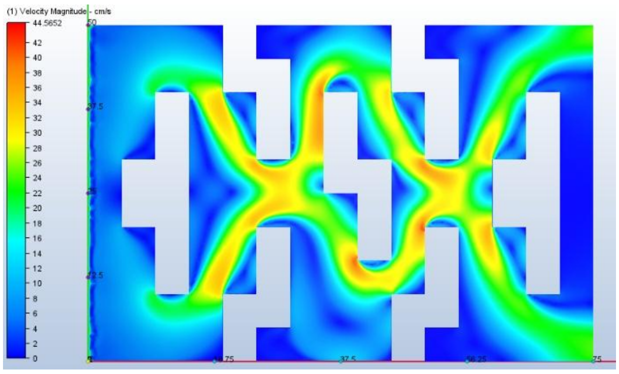

Fig. 13: Simulation with velocity No. 3 of 30cm/s

In Figure 14, the flow is at a velocity of 40 cm/s. The yellow color represents a moderate velocity, indicating that in the center of the water stream, it is approximately 35 to 40 cm/s, as indicated in the data table.

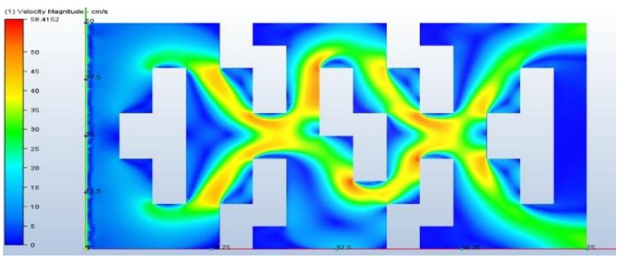

Fig. 14: Simulation with velocity No. 4 of 40cm/s

In Figure 15, the flow is at a velocity of 50 cm/s. It can be observed that at the end of the design, it reaches a green color representing an accessible velocity of 30 to 40 cm/s, which helped to decrease the initial velocity.

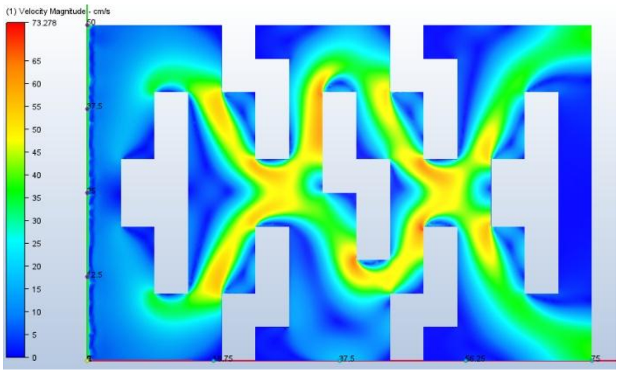

Fig. 15: Simulation with velocity No. 5 of 50cm/s

In Figure 16, the flow is at a velocity of 100 cm/s. It can be observed that the reddish part has reduced its color, suggesting a decrease in intensity. However, this is not the case, as the data in the table have increased their values.

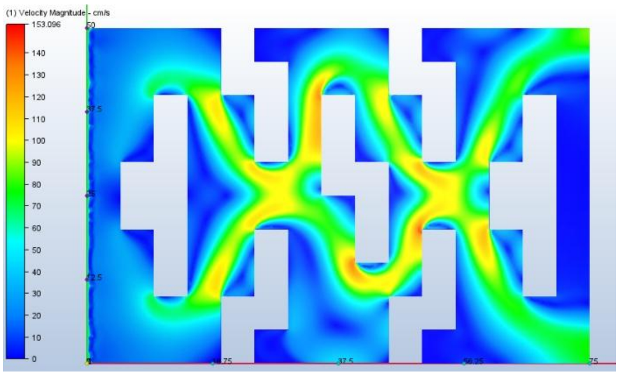

Fig. 16: Simulation with velocity No. 6 of 100cm/s

In Figure 17, the flow is at a velocity of 200 cm/s. It was identified that the areas of highest velocity impact are located at the same points for all velocities, represented in red color.

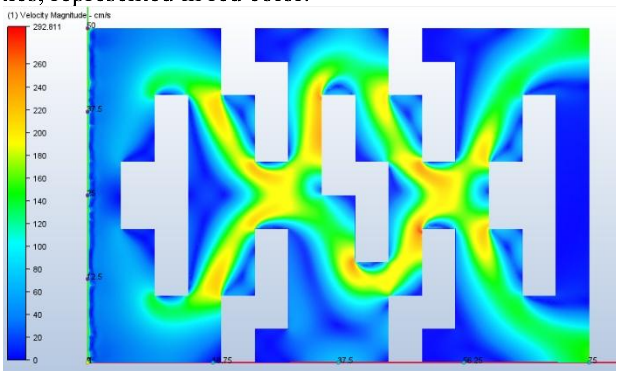

Fig. 17: Simulation with velocity No. 7 of 200cm/s

In Figure 18, the behavior of velocity vectors is observed, indicating that areas of red and yellow colors have increased their velocity, thus generating greater fluidity and increasing energy dissipation.

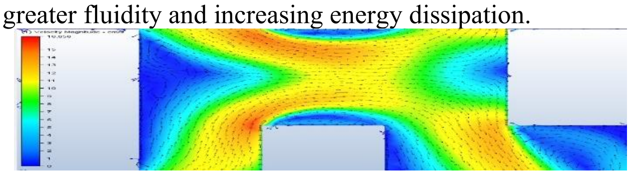

Fig. 18: Simulation with velocity No. 1 of 10cm/s

The solutions provided by the CFD are represented in the form of triangular meshes, where at each intersection, the software had to solve using the Navier-Stokes equations, as shown in Figure 19.

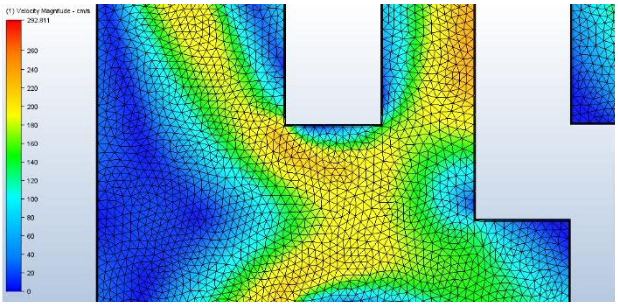

Fig. 17: Simulation with velocity No. 7 of 200cm/s

Next, the pressure graphs obtained for each velocity are shown. It is worth mentioning that a pressure of 10 Pa was used for all velocities. For the velocity of 10 cm/s, see Figure 20. The increase in entering pressure is greater (reddish color), while the exit pressures are negative (blue color), indicating a decrease in the flow mass.

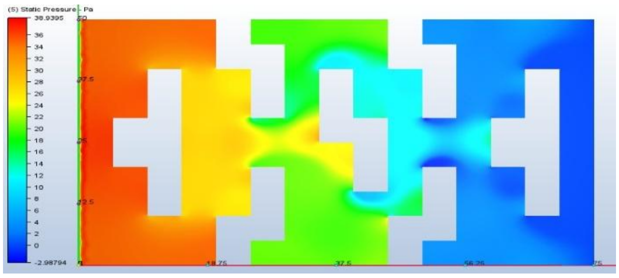

Fig. 20: Pressures with velocity No. 1 of 10 cm/s

For the velocity of 20 cm/s, the yellow color indicates a decrease in pressures from 133 to 80 Pa, as shown in the data table on the left side of Figure 21.

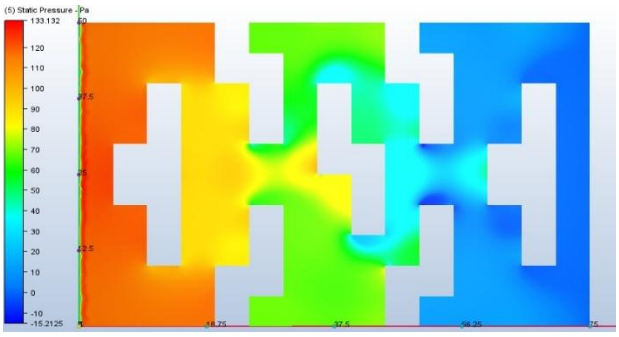

Fig. 21: Pressures with velocity No. 2 of 20 cm/s

In Figure 22, with a velocity of 30 cm/s, it was observed that the higher the velocity, the greater the increase in inlet pressures, as well as the decrease in the flow mass.

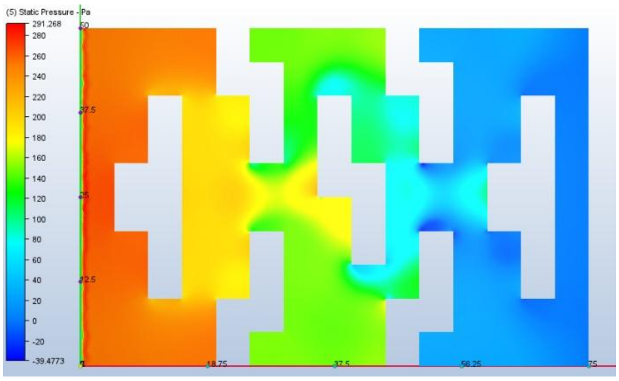

Fig. 22: Pressures with velocity No. 3 of 30 cm/s

In Figure 23, with a velocity of 40 cm/s, it was observed that the light blue color decreases in tone at the upper central position of the design, indicating an increase in pressure in that area.

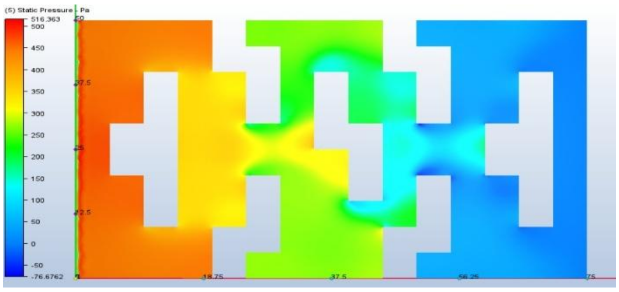

Fig. 23: Pressures with velocity No. 4 of 40 cm/s

In Figure 24, with a velocity of 50 cm/s, it shows a decrease in color intensity compared to Figure 20, indicating that higher velocity generates higher pressure in the flow mass. Additionally, variations in colors in the central part indicate abrupt changes in pressures.

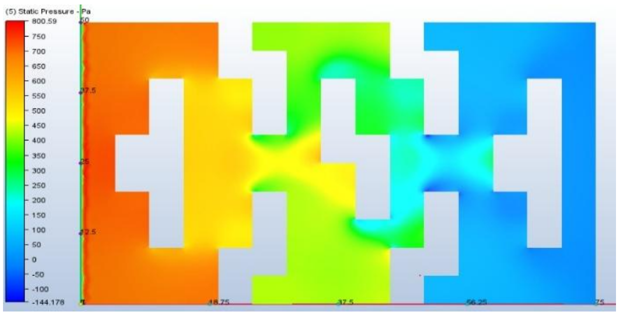

Fig. 24: Pressures with velocity No. 5 of 50 cm/s

In Figure 25, with a velocity of 100 cm/s, increases in outlet pressures were observed compared to Figure 24 (as seen in the data table). As the velocity increases, the outlet pressures also increase.

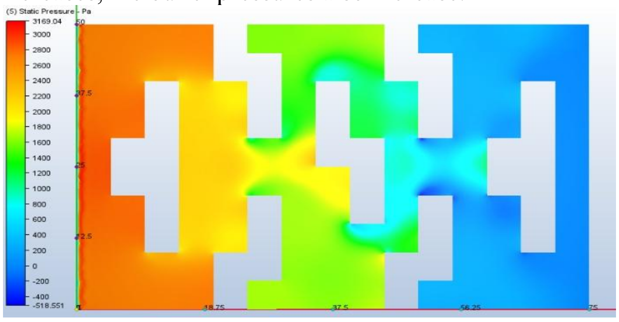

Fig. 25: Pressures with velocity No. 6 of 100 cm/s

In Figure 26, with a velocity of 200 cm/s, blue points are shown indicating negative pressures, where it was determined that regardless of the increase in velocities, these points will always be dead.

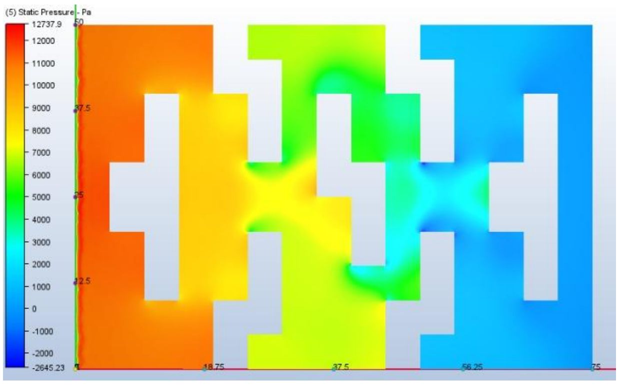

Fig. 26: Pressures with velocity No. 7 of 200 cm/s

Below, Table No. 1 shows the Reynolds number for each velocity, in order to identify whether they are laminar or turbulent flows.

TABLE 1 REYNOLDS NUMBER

| Velocidad (cm/s) | Temperatura (°C) | Densidad (kg/m3) | Viscosidad dinámica (Pa.s) | Diametro interior (m) | Reynolds |
|---|---|---|---|---|---|
| 10 | 20 | 998.57778 | 0.00102 | 0.019607843 | 1919.60 |
| 20 | 20 | 998.57778 | 0.00102 | 0.019607843 | 3839.21 |
| 30 | 20 | 998.57778 | 0.00102 | 0.019607843 | 5758.81 |
| 40 | 20 | 998.57778 | 0.00102 | 0.019607843 | 7678.41 |
| 50 | 20 | 998.57778 | 0.00102 | 0.019607843 | 9598.02 |
| 100 | 20 | 998.57778 | 0.00102 | 0.019607843 | 19196.04 |
| 200 | 20 | 998.57778 | 0.00102 | 0.019607843 | 38392.07 |

It can be stated that by varying the velocities, we have the ability to categorize the flow as laminar or turbulent. In this case, there would only be laminar flow for the velocity of 10 cm/s, while the rest of the cases (20 cm/s, 30 cm/s, 40 cm/s, 50 cm/s, 100 cm/s, and 200 cm/s) are turbulent flows.

## 6. CONCLUSIONS

Based on the results of the CFD simulation, we can conclude that the design of the dissipative structure with rectangular profiles demonstrates its ability to effectively reduce the velocity and pressure of water flow under turbulent conditions. This suggests a promising approach for mitigating risks associated with extreme weather events and urbanization in riverside areas in Peru. However, further studies and testing in real-life situations are required to fully validate the effectiveness of this solution.

The current research is still ongoing; while it is currently based only on the proposed design using simulations in Autodesk CFD, there are plans to build the prototype in the future and determine its functionality in comparison to the simulations already conducted.

## 7. REFERENCES

[1] ARMAS, Daniel Ramón Chirinos, et al. Prototipo de sistema de alerta temprana para la prevención de huaicos, Chosica, Perú. Dilemas contemporáneos: Educación, Política y Valores, (2022). Recuperado de https://dilemascontemporaneoseducacionpoliticayvalores.com/index.php/dilemas/article/view/3163/3150

[2] DELGADO ASCARZA, Carol Janet; TAMAYO LOPEZ, Guiliana Andrea. Plan de gestión integral para reducir daños ocasionados por huaicos, quebrada El Pedregal, Chosica. Recuperado de http://doi.org/10.19083/tesis/654022

[3] Autodesk, 2023. Autodesk CFD. Recuperado de https://www.autodesk.com/products/cfd/overview

[4] Autodesk, 2023. Autodesk Inventor. Recuperado de https://www.autodesk.com/products/inventor/overview

[5] R. Mott and J. Untener. Mecánica de fluidos (7na ed.), editorial Addison Wesley (2015), pp. 434

[6] JARAMILLO DIAZ, Julian David; CARDENAS BAÑOL, Hector Alonso. Numero de Reynolds. 2015. Recuperado de https://repository.uniminuto.edu/handle/10656/4849

[7] MACHACA, Magdalena Huacasi. El problema de valor inicial para las ecuaciones de Navier-Stokes en Lm (Rm). Pesquimat, 2019, vol. 22, no 1, p. 9-29. Recuperado de https://dialnet.unirioja.es/servlet/articulo?codigo=9404913&orden=0&info=link