22nd International Conference on Renewable Energies and Power Quality (ICREPQ'24)
Bilbao (Spain), 26th to 28th June 2024
*Renewable Energy and Power Quality Journal* (RE&PQJ)
ISSN 2172-038 X, Volume No.22, September 2024

# Optimization for the inlet channel and basin of a gravitational vortex turbine through the maximization of circulation.

Eduardo D. Madrid C.[1], Juan C. Serrano R.[1] and Elkin G. Flórez S.[1]

[1] Department of Mechanical Engineering
Universidad de Pamplona
Pamplona – Norte de Santander, 36310 Colombia
Phone/Fax number: +57 304 6674187

**Abstract.** To optimize the energy generation of a gravitational vortex turbine, geometric modifications are proposed for its inlet channel and basin. These modifications include implementing a contraction in the channel, a technique commonly used in wind tunnels to accelerate the fluid and consequently improve vorticity in the basin. Three parameters were considered for optimization: the ratio between the diameter of the basin ($D$) and the length of the contraction ($L_c$), expressed as Lc/D; the size of the outlet edge ($w_2$) relative to $D$, expressed as $w_2/D$; and the height of the basin ($H$) relative to $D$, expressed as $H/D$. These parameters were systematically evaluated through numerical simulations to assess vortex circulation ($\Gamma$), established as the target variable. The behavior of the flow within the basin was simulated, which made it possible to identify the vorticity that was sought to be validated, obtaining positive results in what leads to the generation of the vortex thanks to the geometric changes made.

**Key words.** Basin, energy, optimization, turbine, vortex.

## 1. Introduction

The diversification of the energy matrix in Colombia is a crucial process aimed at reducing the country's historical dependence on oil and coal while promoting the use of cleaner and more sustainable energy sources. This strategic shift seeks to encourage the incorporation of renewable energies such as solar, wind, hydroelectric, and biomass, not only to ensure energy security but also to mitigate environmental impact and contribute to the fight against climate change [1]. Through the energy transition process, aimed at transforming the global energy matrix and reducing dependence on non-renewable sources, the realm of clean and sustainable energies is explored. This approach promotes technologies that enable energy generation without emissions, thus contributing to a more sustainable future society [2].

Gravitational vortex turbines (GVTs) propose harnessing the kinetic energy generated by the movement of fluid within a vortex formed by gravity, using a rotating rotor. This approach differs from conventional turbines, which capture the kinetic energy of wind or water flow. Although GVTs are not widely known or established in energy generation [3], they offer potential advantages that can accommodate research and development of this type of turbine, as they have a lower environmental impact on the surrounding habitat and are visually less intrusive [4]. Additionally, they can be used in locations where conventional turbines are not viable [5]. These turbines have lower manufacturing and maintenance costs, as they do not require a reservoir; they operate based on water availability and reach maximum power during rainfall, reducing generation during the dry season [6]. Given their characteristics, GVTs are suitable for distributed energy systems. GVTs installed worldwide have efficiencies ranging from 17% to 85% [7].

Usually, research on GVTs focuses on the geometric parameters of the basin [1, 3, 6], blade shapes and their positioning [1, 4, 14], as well as the optimization of the inlet channel and vortex formation [1, 4, 9]. Optimization methods stand out through geometric adjustments in the inlet channel, basin, and/or rotor design [6]. In studies aimed at optimizing channel flow, channel models with deflections in the section before entering the basin or inclinations of parts of the same channel that direct the fluid by gravity are tested [8], to different basin geometries, which can be cylindrical or conical [9]. Research extends to theoretical studies on vortex formation [3].

The proposal of this work is based on maximizing vorticity through modifications of the channel and basin geometry; under this analysis, changes are made to the inlet channel using a horizontal channel and varying the shape of the basin to maximize circulation, all through numerical simulations. Changes adapted to the inlet channel are formulated to increase the fluid velocity in the channel before entering the basin, thus a contraction commonly used in wind tunnels is proposed. The basin design generally takes on a conical shape [1] that directly induces vortex formation, but with complexity in the manufacturing of its rotor, hence a cylindrical basin is intended to be used, to utilize its space for rotor installation.

## 2. Materials and Methods

2.1. Geometry

Considering the bibliography on aerodynamic circuits for wind tunnels, a contraction is used to increase the airflow velocity and improve its quality in terms of uniformity and turbulence. This process involves gradually reducing the cross-sectional area of the tunnel as the air approaches the test section, increasing air velocity [10]. This contraction is located in the channel that carries the flow towards the basin, allowing for an increase in fluid velocity as it enters the basin, consequently influencing vortex generation. There are different models for designing the contraction curve [12,13]; for this work, the model developed by Fang was adopted, which utilizes two third-order polynomials for two curves, as shown in equations (1) and (2), using the variables depicted in Fig. 1.

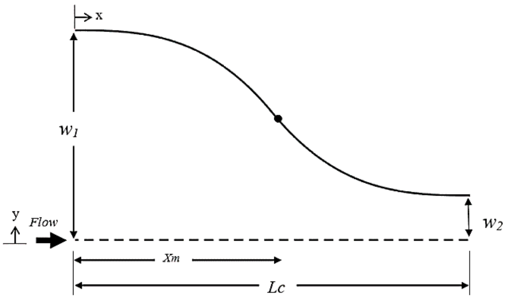



Figure 1. Contraction curve
Fuente: Fang, F. M., Chen, J. C

$$\boldsymbol{y = (h_1 - h_2)\left[1 - \frac{1}{x_m^2}\left(\frac{x}{L}\right)^3\right] + h_2, \qquad x < x_m} \quad (1)$$

$$\boldsymbol{or\ y = \frac{(h_1 - h_2)}{(1 - X_m)^2}\left(1 - \frac{x}{L}\right)^3 + h_2, \qquad x > x_m} \quad (2)$$

The proposed geometry of the GVT for analysis is shown in Fig. 2. Here, it is evident that there is an inlet channel with a uniform cross-sectional area, which then decreases according to the shape of the curve proposed by the contraction until reaching the basin, which has a cylindrical shape joined at its bottom to a truncated cone with an inclination angle proposed by Ruiz Sánchez [14].

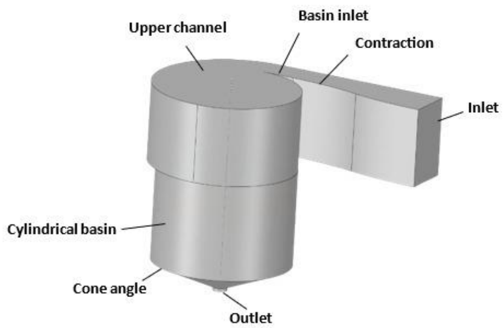



Figure. 2. Illustration of a Gravitational Turbine Vortex

Through seven variables, the geometry is defined, with three of them selected for analysis and experimentation while the remainder remains constant, including the angle of the conical section at the end of the basin. As an innovation, the basin is related to a cylindrical shape with a conical part at the end. Fig. 3 shows the parameters describing the geometry in dimensionless terms about the upper diameter of the basin (*D*). Table I describes the ranges of dimensionless variables to be analyzed in the design of experiments and consequently in simulations.

Table I-Ranges for design variables

| Independent factor | Low range | High range |
|---|---|---|
| $L_c/D$ | 0.7 | 0.9 |
| $w_2/D$ | 0.4 | 0.6 |
| $H/D$ | 0.5 | 2.0 |

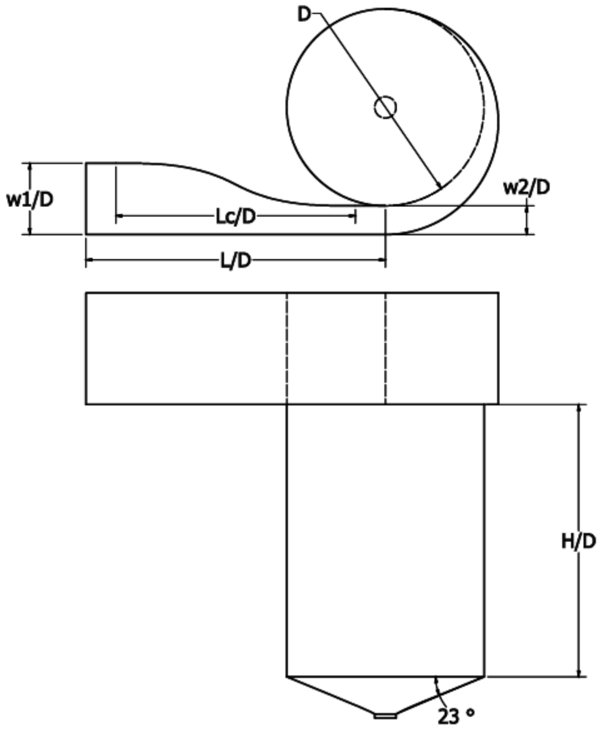



Figure. 3. The selected baseline geometry and geometric factors

In this study, the geometric parameters of the channel, contraction, and basin are modified to maximize the magnitude of the vortex. For a GVT, the circulation parameter (Γ) is determined by Eq. (3).

$$\boldsymbol{\Gamma = 2\pi r v_\theta} \quad (3)$$

Where $v_\theta$ is the tangential velocity and *r* describes the radial position located from the centre of rotation. The volumetric flow rate is calculated from the continuity equation, starting from the cross-sectional area of the flow and its average velocity.

2.2. Experimental Design

The Latin hypercube method is an effective technique in the design of experiments that aims to maximize the information obtained with a relatively small number of trials. This approach is based on dividing the experimental space into subspaces, each represented by a point in a Latin hypercube, where each variable of interest varies at a specific level. By uniformly distributing the combinations of variable values in this hypercube, complete coverage of the design space is ensured, avoiding redundancies and minimizing correlation between variables. This allows for efficient exploration of the relationship between variables and their effects on the studied system, facilitating the identification of trends and informed decision-making based on the results obtained [11].

Through the Statgraphics software, the number of simulations with random data related to the geometric variables is obtained according to the design of experiments method explained earlier. Table II shows the values used to carry out the simulations, according to the random values related to the design of experiments.

Table II-Experimental values

| *RUN* | *Lc/D* | *$w_2$/D* | *H/D* |
|---|---|---|---|
| *1* | 0.7 | 0.6 | 2 |
| *2* | 0.9 | 0.4 | 0.5 |
| *3* | 0.7 | 0.6 | 0.5 |
| *4* | 0.9 | 0.4 | 2 |
| *5* | 0.9 | 0.6 | 2 |
| *6* | 0.7 | 0.4 | 2 |
| *7* | 0.9 | 0.6 | 0.5 |
| *8* | 0.7 | 0.4 | 0.5 |

2.3. Numerical simulation

*CAD development.* The CAD of the inlet channel and basin of the GVT was developed using Autodesk Inventor software, where the curves defined in equations (1) and (2) were implemented using the "equation curve" tool. Subsequently, the files are exported to SpaceClaim software, where their repair and adjustment are carried out to develop the mesh.

*Computational domain.* The computational domain and boundary conditions are shown in Fig. 4, where the boundary conditions include the velocity at the inlet wall of the channel, the outlet pressure on the upper face exposed to the environment and at the outlet of the basin, and the wall condition for the rest of the geometry.

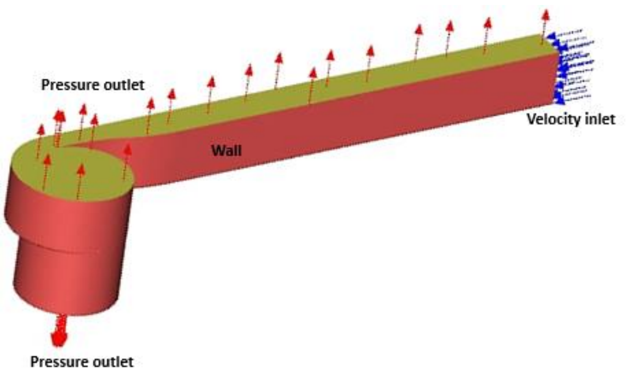



Figure 4. Computational domain and boundary conditions

*Mesh.* The Fluent Meshing tool from Ansys® is used to generate both surface and volume meshing, the latter using the Poly-Hexcore method, achieving a minimum orthogonality in the quality of its elements of 0.3. This novel approach combines polyhedral and hexahedral elements, offering greater accuracy and computational efficiency. Fig. 5 shows the volume mesh developed with the feature of having polyhedral elements in the surface flow zone and tetrahedral elements in the interior flow zone located in the basin. To improve the characterization of the flow boundary layer on the walls, the Smooth-transition model with four layers was used for all walls.

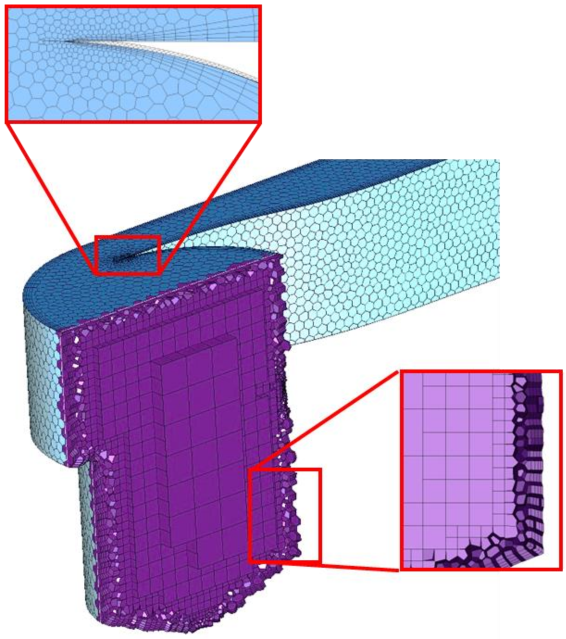


Figure 5. Poli-hexcore mesh

To validate mesh convergence, the calculation of flow vorticity inside the basin was performed. In Fig. 6, it can be observed that initially a mesh with 209,716 elements was used, which was refined in a second iteration, resulting in a variation in the vorticity value equivalent to 141%. Subsequently, the mesh was refined to reach a total of 851,626 elements, where the variation in vorticity concerning the previous coarse mesh (Mesh 3) was only 0.35%. This allowed validation that the fourth mesh met the necessary quality conditions to perform all simulations.

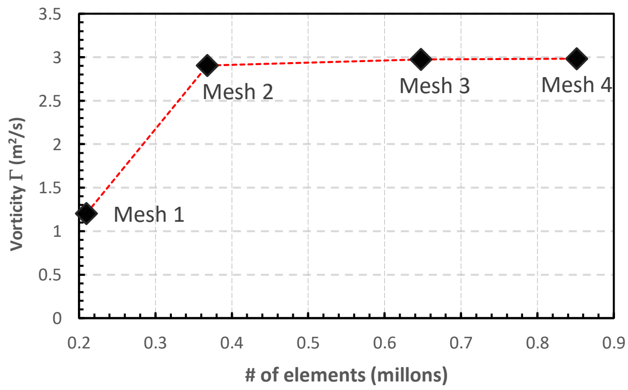



Figure 6. Mesh convergence study

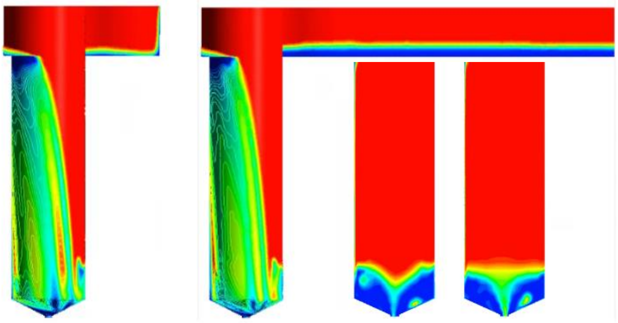
*Simulation models.* To obtain results consistent with experimental data, it was chosen to include within the simulation the Volume of Fluid (VOF) model for multiphase flows, aiming to improve the liquid-air interaction, and the k-ω SST turbulence model, to provide a better description of the turbulence phenomenon within the flow.

The VOF multiphase model is implemented under an implicit scheme, where the body force formulation is used, as well as the open channel sub-model. For phase interaction, the surface tension model was considered, with the option of wall adhesion and a constant surface tension coefficient of 0.073 N/m.

The turbulence model used for the solution was the two-equation k-ω SST, without considering the correction for low Reynolds numbers and with the active option to limit production. This model has been designed as an improvement over the k-ω BSL, as it considers the transport of turbulent shear stresses in defining the term for turbulent viscosity. This feature makes the k-ω SST model more accurate and reliable than the k-ω BSL model for a wide range of flow types. Like any model based on the Boussinesq hypothesis, it has deficiencies when the flow is highly anisotropic, but it has the advantage of accurately capturing flows in free stream and near-wall regions, even within the boundary layer. Additionally, it is considered the most complete model in the class of two-equation models based on the Boussinesq hypothesis [15].

## 3. Results

Initially, a numerical study was carried out to determine the influence of the length of the inlet channel on the behavior of the vortex in the basin.



(a) (b)

Figure 7. Water volume fraction simulation; (a) all channel, (b) Vertical plane of the basin

Fig. 7(a) shows the results obtained from the flow behavior at the entrance of the basin, where liquid water is represented in blue. For the long channel, we can note that for the long channel, the flow inlet to the basin is a totally developed profile. On the other hand, figure 7(b) shows the behavior of the vortex at the exit of the basin, where the one formed by the simulation of the long channel is the one with the best symmetry.

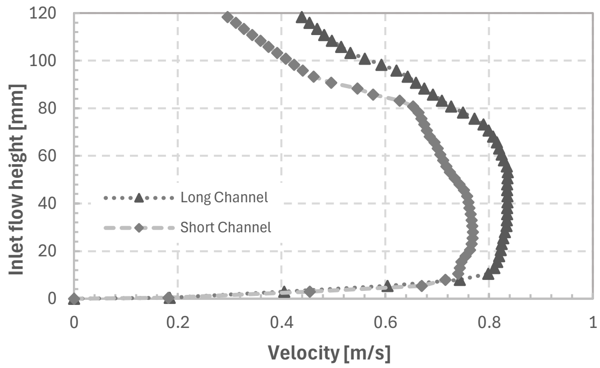
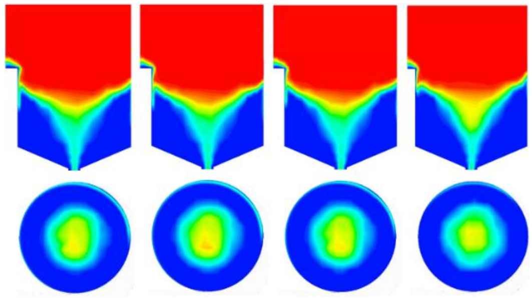



Figure 8. Velocity profile to the basin inlet

Furthermore, Fig. 8 shows the behavior of the flow velocity at the entrance of the basin, taken on a vertical line in the central coordinate of the channel. This profile allows us to validate that the long channel model can be considered a developed flow, since the curve best fits a parabola, with a maximum velocity of 0.82 m/s.



(a) (b) (c) (d)

Figure 9. Water volume fraction for $H/D$=0.5; **(a)** $L_c/D$ =0.9 y $w_2/D$=0.4, **(b)** $L_c/D$ =0.7 y $w_2/D$=0.6, **(c)** $L_c/D$ =0.9 y $w_2/D$=0.6 y **(d)** $L_c/D$ =0.7 y $w_2/D$=0.4

The Fig. 9 shows the behavior of the liquid water fraction within the basin, for all experimental values of $H/D$ = 0.5. The upper of the figure shows a vertical plane located in the centre of the basin and at the bottom for a horizontal plane located at the height of the entrance to the conical area of the basin.

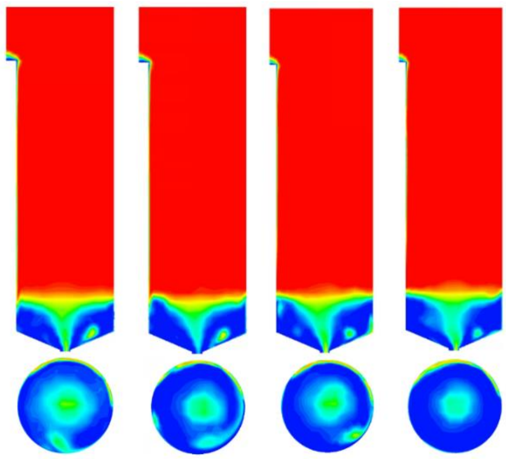


Figure 10. Water volume fraction for $H/D$=2; **(a)** $L_c/D$ =0.7 y $w_2/D$=0.6, **(b)** $L_c/D$ =0.9 y $w_2/D$=0.4, **(c)** $L_c/D$ =0.9 y $w_2/D$=0.6 y **(d)** $L_c/D$ =0.7 y $w_2/D$=0.4

Similarly, the Fig. 10 shows the behavior of the liquid water fraction within the basin, for all experimental values of *H/D* = 2.

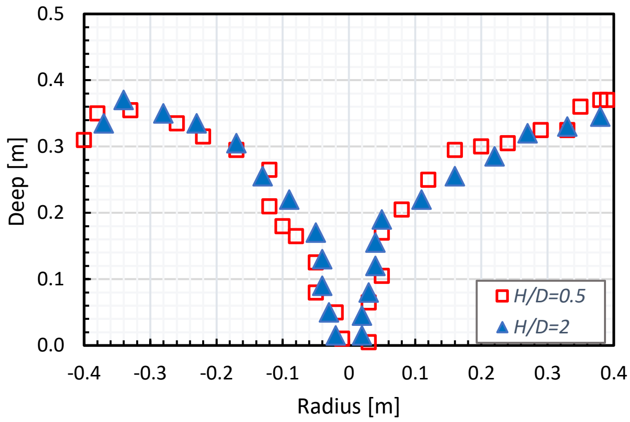



Figure 11. Vortex profile for $L_c/D$ =0.7, $w_2/D$=0.4

The Fig. 11 shows that both geometries produce a symmetrical and stable vortex in the conical zone. Being the short basin *H/D*=0.5, the one that presents the best symmetry among the simulations of this work

## 4. Conclusion

The inlet channel design and basin are important parameters for effective vortex generation in the gravitational Water Vortex Turbine (GWVT); therefore, CFD analysis were conducted on the inlet channel (Contraction curve function) and the basin (short and long cylindrical) with conical outlet, in order to discern the effect of the combination of geometrical configuration of these parameters.

The Latin hypercube method was the technique used in the design of experiments to identify the factor values and the number of combinations to be carried out in the simulation. Specifically, three dimensionless parameters were taken into account, for a total of eight computationally simulated configurations.

The results allowed us to validate that for the values of *H/D*=0.5, *Lc/D* =0.7 and $w_2/D$=0.4, a more symmetrical vortex is present, which can be better used for energy generation in vortex turbines. gravitational.

To simulate the behavior of the flow within the basin, it must be taken into account that the inlet channel must have a minimum length, according to the literature for open channels, since this allows a better approximation to the real conditions of this kind of flows.


## Acknowledgement

The authors acknowledge the financial support provided by the announcement No. 890 de 2020 “Convocatoria para el fortalecimiento de CTeI en Instituciones de Educación Superior (IES) Públicas 2020” (Contract No. 2022-0453)


## References


[1] Velásquez, L., Posada, A., & Chica, E. (2023). Surrogate modeling method for multi-objective optimization of the inlet channel and the basin of a gravitational water vortex hydraulic turbine. Applied Energy, 330, 120357.

[2] Prías, Omar, (2010). "Programa de uso racional y eficiente de energía y fuentes no convencionales–PROURE." Informe Final. Plan de Acción 2015.

[3] Kiviniemi, Olli, (2009).and Gregory Makusa. "A scale model investigation of free surface vortex with particle tracking velocimetry".

[4] Velásquez, L., Posada, A., & Chica, E. (2022). Optimization of the basin and inlet channel of a gravitational water vortex hydraulic turbine using the response surface methodology. Renewable Energy, 187, 508-52.

[5] Boyle, G. (1993). Renewable Energy, 2004, ISBN: 9780199261789;(b) TB Johansson, H. Kelly, AKN Reddy and RH Williams. Renewable energy: sources for fuels and electricity.

[6] S. Mulligan, (2015). Experimental and Numerical Analysis of Three-Dimensional FreeSurface Turbulent Vortex Flows with Strong Circulation, Institute of Technology Sligo, Ireland.

[7] Timilsina AB, Mulligan S, Bajracharya TR. (2018). Water vortex hydropower technology: a state-of-the-art review of developmental trends. Clean Technol Environ Policy. 20(8):1737–60.

[8] Velásquez, L., Posada, A., & Chica, E. (2021). Advances in the Development of Gravitational Water Vortex Hydraulic Turbines

[9] Velásquez, L., Posada, A., & Chica, E. (2020). Numerical analysis of the inlet channel and basin geometries for vortex generation in a gravitational water vortex power plant.

[10] Fang, F. M., Chen, J. C., & Hong, Y. T. (2001). Experimental and analytical evaluation of flow in a square-to-square wind tunnel contraction. Journal of wind engineering and industrial aerodynamics, 89(3-4), 247-262.

[11] S. McLeod. (2019). What a p-value tells you about statistical significance, simply psychology. Indian Journal of Psychological Medicine, 41(3):210-215.

[12] Whitehead, L. G., Wu, L. Y., & Waters, M. H. L. (1951). Contracting ducts of finite length. Aeronautical Quarterly, 2(4), 254-271.

[13]. Morel, T. (1975). "Comprehensive Design of Axisymmetric Wind Tunnel Contractions." ASME.*J. Fluids Eng*. June 1975; 97(2): 225–233

[14] Ruiz, A., Sierra, J., Correa, E., Sanín, D. (2023). numerical comparison of Savonius turbine as a rotor for gravitational vortex turbine with standard rotor.

[15] Schiestel, R., Wiley. (2008). Modeling and simulation of Turbulent Flows.