Effects of confinement on the energy performance of an Archimedes screw using CFD simulation

Audrey Michel*[1], *Leandro* Duarte[2], *Guilhem* Dellinger[2], *Nicolas* Dellinger[1], *Abdelali* Terfous[3], *Abdellah* Ghenaim[3],

[1]ICUBE, Département de Mécanique, 1 Cr des Cigarières, 67000 Strasbourg
[2]ENGEES, 1 Cr des Cigarières, 67000 Strasbourg
[3]INSA, Département mécanique des fluides, 24 boulevard de la victoire, 67000 Strasbourg

Abstract. In the context of the growing development of renewable energy systems, Archimedes screw turbines represent a promising solution for harvesting the kinetic energy of shallow free-surface flows. The system investigated in this study features an innovative configuration: a horizontally oriented Archimedes screw installed at the free surface, partially submerged over half of its diameter, and operating without any hydraulic head. An experimental test rig was developed to assess the influence of key parameters on the turbine performance. The results highlight the major role of the incidence angle on energy conversion efficiency, with an optimal angle of 45° leading to efficiencies of approximately 40% under the tested conditions. Since the experimental conditions are constrained by the dimensions of the laboratory channel and the available discharge, a computational fluid dynamics (CFD) model was developed to explore a wider range of hydraulic and geometric configurations. The numerical model was first validated against experimental data, showing good agreement. Subsequently, parametric studies demonstrated that horizontal confinement is a key design parameter: maximizing the lateral confinement of the screw within the channel significantly enhances the energy efficiency of the system.

1 Introduction

The growing demand for decentralized and renewable energy is encouraging the development of solutions to complement large power plants. As energy systems become more diverse (electrification of transport, strong growth in electricity use etc), demand for electricity will continue to rise. In France, RTE estimates that renewable production capacity will need to triple by 2035 to meet energy transition targets [1]. The hydroelectric potential of river currents remains largely untapped in France and around the world. However, it represents a renewable and local source of energy and thus responds to the challenges of the energy transition. The Archimedes screw has been known since ancient times, when it was

* Corresponding author: audrey.michel@etu.unistra.fr

used for irrigation to pump and raise water from the Nile [2]. Its use in turbines to convert the potential energy of waterfalls into electricity has only emerged in recent decades. The work of G. Dellinger and X. Brunissen proposes analytical and numerical models to predict their behaviour [3; 4].

A more recent variant places the screw axis horizontally, at the free surface, to recover the kinetic energy of the watercourse. This original configuration makes it possible to exploit currents and does not require a waterfall. Because of the recent emergence of this configuration, few studies have been published on the optimization of such a device. The work of J. Betancour [5] and P.E. Slaboch [6] focused on screws whose axis is placed parallel to the flow. The first proposes a mathematical model for predicting the energy performance of a horizontal Archimedes screw turbine based on the geometric parameters of the screw (outer and inner diameters and pitch), while the second conducts laboratory experiments to analyze the influence of fairing on the energy efficiency of the system. The experimental results and mathematical model developed by A. Stergiopoulou [7] highlight another determining parameter in the search for the optimum: the variation in the angle between the screw axis and the direction of flow. Based on these observations, an experimental study of the influence of the screw's geometric parameters on its energy performance has been conducted. The experimental results highlighted the impact of the angle between the screw axis and the flow direction on efficiency [8].

As shown above, there are currently only a few studies in the literature on horizontal screw turbines used to recover hydrokinetic energy. Although, turbine confinement is a key parameter affecting performance [9], no studies have specifically addressed this topic for horizontal screw turbines. This issue is especially important because this type of turbine can be installed in highly confined flows, i.e., with low water depths or in very narrow channels. Therefore, this study investigates the impact of horizontal confinement on the efficiency of the screw turbine.

First, the CFD model developed under OpenFOAM is presented, then validated by a comparison between the numerical results and the experimental data. Next, the results of numerical studies using this model are presented. These studies focus on the influence of screw confinement in the channel to identify the configurations that maximize the device's energy efficiency.

2 Computational fluid dynamics

2.1 Numerical model

The screw is modeled in SolidWorks and then exported in STL format before being integrated into the calculation domain. A series of geometric transformations, including translations of the screw's origin and rotations of its central axis, allow the screw to be positioned and oriented correctly. The screw can be seen in the channel in Figure 1.

The computational domain is defined in OpenFOAM using the blockMesh utility. The blocks are generated in a Cartesian coordinate system whose origin is carefully selected, as the entire numerical model is based on it. Initially, the mesh consists of uniform hexahedral (parallelepiped) cells. The channel (4 m long and 1.2 m wide) is discretized using 200 cells along the x-axis, 318 cells along the y-axis, and 9 cells along the vertical z-axis. The screw geometry is then incorporated into this block using the snappyHexMesh utility. A refinement region is defined around the screw. A mesh sensitivity analysis was performed using five different levels of global mesh refinement. The selected mesh contains approximately 3 million cells. The characteristic size of cells in channel is 2 cm in channel and 1 cm in refined zone near the rotor.

 https://doi.org/10.1051/e3sconf/202672004002

To reproduce the configuration of the experimental channel, the boundary conditions are defined according to the surfaces of the blocks: one inlet, one outlet, surfaces open to the atmosphere, and the remaining surfaces treated as walls. A constant velocity of 0.71 m/s is imposed at the inlet along the x-axis, while the outlet enforces the water level through a hydrostatic pressure distribution. Finally, a no-slip condition is applied on all walls.

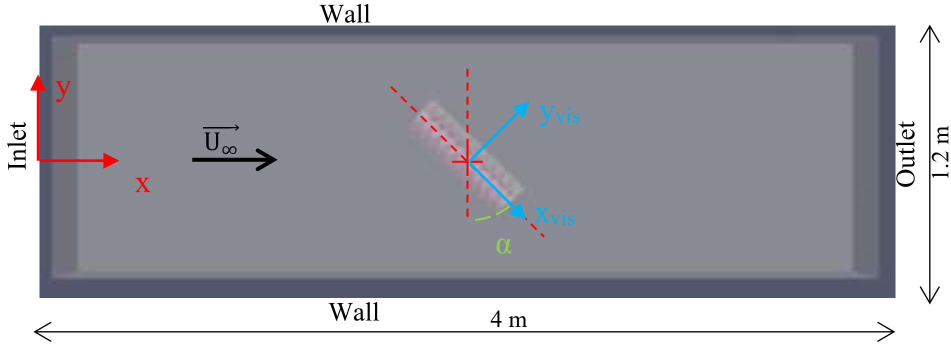

Fig. 1. Placement of the screw in the channel

The flow is modelized by resolving the Unsteady Reynolds Averaged Navier-Stokes (URANS) equation with a PIMPLE method. The system of equations is closed thanks to the k-omega SST turbulence model. The free surface flow is modeled with the Volume Of Fluid (VOF) method. It could be noticed that this turbulence model and VOF method are widely validated in the literature. An adaptive time step is used by imposing a maximal Current Number of 2.

The rotational motion of the screw is modeled using the Arbitrary Mesh Interface (AMI) method. The screw velocity is prescribed within the rotating region. The AMI zone, which corresponds to the rotating portion of the mesh, is defined as a cylindrical volume enclosing the screw, while the remainder of the mesh remains stationary.

2.2 Validation

To validate the numerical model, a comparison between experimental and simulated results is necessary.

The experimental device is installed in a channel 1.2 m wide and 30 m long. The device consists of a horizontal screw coupled to an EC Maxon motor with a gearbox (see the scheme in Figure 2). The setup allows testing of the screw's performance under various hydraulic and geometrical conditions. The experimental rig makes it possible to adjust the flow velocity (< 1 m/s) and the water depth (< 0.8 m). The angle of incidence of the screw can be selected between 0° and 90° in increments of 15°. The vertical position of the screw can also be adjusted. The rotor rotational speed is controlled using an Escon 70/50 control card. The torque is determined by measuring the motor current and using the motor torque constant. The motor current is recorded every 10 milliseconds over a period of 20 seconds, a water depth of 0.21 m, and a flow velocity of 0.71 m/s.

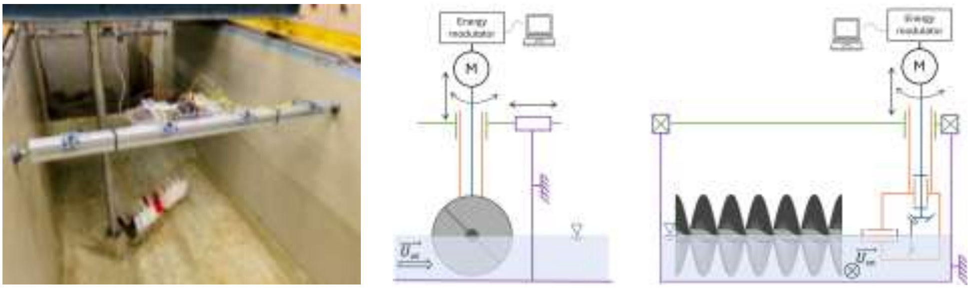

Fig. 2. Real and schematic experimental apparatus test rig.

The mechanical power $\overline{P_{meca}}$ at the screw rotation axis is calculated by multiplying the torque C_{hc}at the screw axis by the rotation speed of the screw ω.

$$\overline{P_{meca}} = C_{hc} \times \omega \tag{1}$$

The available hydrokinetic power $\overline{P_{hk}}$ is defined by:

$$\overline{P_{hk}} = \frac{1}{2} \rho \, S \, U_{\infty}{}^{3} \tag{2}$$

ρ is the density of the fluid, U_{∞} is the average flow velocity, and S is obtained by calculating the area of the screw projected onto the fluid section.

The experiment makes it possible to determine the conversion efficiency η, defined as the ratio between the mechanical power generated by the rotation of the screw $\overline{P_{meca}}$ and the available hydrokinetic power $\overline{P_{hk}}$:

$$\eta = \frac{\overline{P_{meca}}}{\overline{P_{hk}}} \tag{3}$$

This efficiency is presented as a function of the advance factor λ, calculated using the following formula:

$$\lambda = \frac{\omega \times \frac{\emptyset_{out}}{2}}{U_{\infty}} \times \sin \alpha \tag{4}$$

where ω is the rotational speed of the screw, $\emptyset_{out}$ is the outer diameter of the screw, U_{∞} is the fluid flow velocity and α is the angle between the screw axis and the flow direction.

Figure 3 shows the evolution of efficiency as a function of λ, obtained experimentally and numerically.

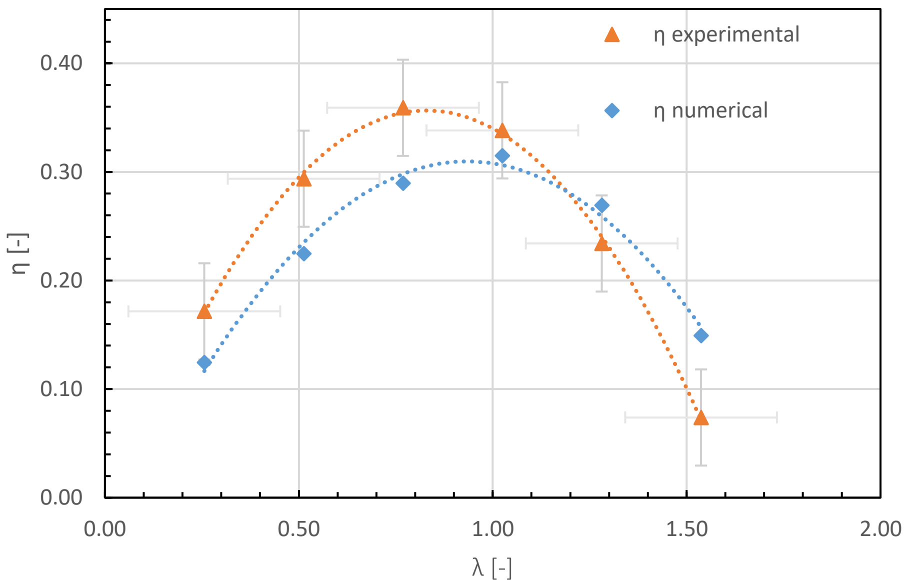

Fig. 3. Comparison of measured and simulated efficiencies as a function of λ

Qualitatively, we observe that both curves have a bell shape, characteristic of a second-order polynomial equation.

For low values of λ, the simulation underestimates the efficiency compared to the experiment, while for $\lambda > 1.2$, the trend is reversed.

The range of use appears to be wider in numerical simulation than in experimentation.

It is essential to quantify the error between the measured results and those obtained from numerical simulation. Table 1 shows that the relative error between the optimal values of the simulated advanced factor obtained numerically and that measured experimentally is 12.5%. Similarly, the relative error between the simulated and measured maximum efficiencies is 18.4%. Both differences are less than 20%, which is a satisfactory accuracy for a CFD model applied to free surface flow.

Table 1. Optimal values of λ et de η.

	Lambda	Efficiency
Experimental	0.83	0.38
Numerical	0.934	0.31
Relative error	12.5%	18.4%

This numerical model can be considered sufficiently representative to predict the optimal performance of a horizontal Archimedes screw in a free-surface channel. The remaining discrepancies between numerical and experimental results may arise from measurement uncertainties, from the imposed constant rotational speed in the simulation (whereas in the experiment the screw speed, although controlled, varies with the fluid loading), and from differences in flow characteristics, particularly the assumption of a uniform velocity field across the channel width and isotropic turbulence in the model, while significant transverse velocity gradients and anisotropic turbulence were observed experimentally.

Furthermore, when considering the measurement uncertainties associated with the experimental data, these discrepancies are further reduced. The margin of error of the numerical model should therefore not be interpreted as a strict error but rather as a combined effect of modeling assumptions and experimental variability

3 Numerical results.

3.1 Impact of confinement

Few studies have directly examined the influence of the confinement of an Archimedes screw turbine in the channel section. In other words, the effect of the channel width (horizontal confinement) has rarely been studied as independent variables.

In this chapter, the horizontal confinement is analyzed separately through parametric study. All other geometric and operating parameters are kept constant at their experimental values to isolate and highlight the specific influence of this confinement on the overall performance of the screw.

The values of the parameters kept constant are summarized in Table 2:

Table 2. Constant parameters of the simulation.

PARAMETER	SYMBOLE	VALUES
OUTER DIAMETER	Φ_{OUT}	0.18 m
INNER DIAMETER	Φ_{IN}	0.035 m
LENGTH	L_V	0.6 m
PITCH	P	0.18 m
NUMBER OF THREADS	N_F	3
INCIDENT ANGLE	α	45°
INCIDENT SPEED	U_∞	0.71 $m.s^{-1}$
SCREW ROTATION SPEED	ω	9.56 $rad.s^{-1}$

3.2 Horizontal confinement

The horizontal confinement ratio τ_{ch} is defined as the ratio between the channel width l_c and the length of the screw projected onto the water section l_{vp}. A confinement of 100% corresponds to a channel width equal to the projected length of the screw, which means that the screw touches the side walls. With numerical simulation, this means moving the side walls of the channel closer to or further away from the screw, as shown in Figure 4.

$$\tau_{ch} = \frac{l_c}{l_{vp}} = \frac{l_c}{l_v sin\,\alpha\, +\, \emptyset_{out} cos\alpha} \tag{5}$$

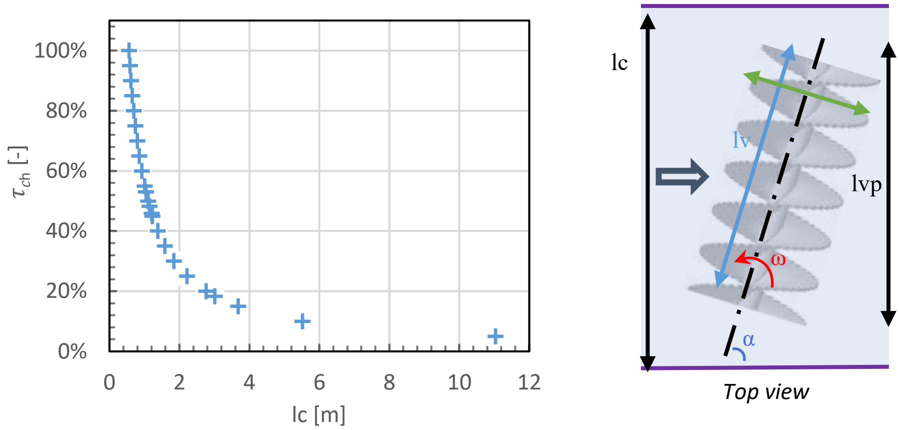

Fig. 4. Horizontal confinement rate

It should be noted that the confinement of the experimental configuration is $\tau_{ch} = 46\%$ corresponding to a channel width $l_c = 1.2\ m$.

Six confinement rates were simulated: 10 %, 25 %, 40 %, 55 %, 70 % et 85 %. It was not possible to simulate rates higher than 85 % due to numerical limitations related to the meshing of the rotating element, which comes into conflict with the channel boundaries in the AMI (Arbitrary Mesh Interface).

The screw rotation speed is set at $\omega = 9.56\,rad.s^{-1}$, which corresponds to approximately 1.5 rotations per second. To obtain at least 10 complete revolutions in a steady state, the total simulation time is set at 25 seconds. Only the last 15 seconds are used to calculate the average moments on the screw, since the first 10 seconds being excluded as they correspond to the transitional phase.

The size of each mesh cell is fixed: thus, when the channel width is reduced to increase confinement, the total number of meshes decreases. As a result, simulations with low confinement (wider channel) take longer to calculate.

The results presented in Figure 5 show an increase in efficiency with the horizontal confinement rate. The more the screw is confined laterally, the more the flow is constrained, which leads to a reduction in the effective water passage cross-section, an increase in the local fluid velocity, and, consequently, an improvement in overall efficiency.

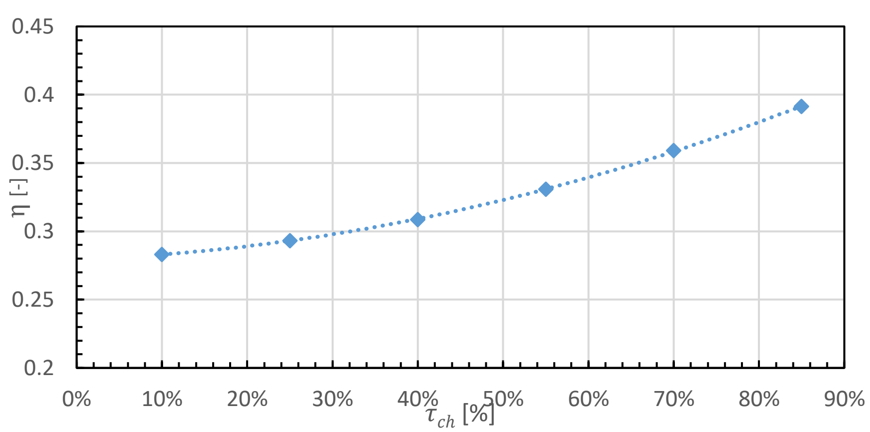

Fig. 5. Efficiency evolution as a function of horizontal confinement rate

The observed trend is consistent with physical intuition: confinement accentuates the channeling effect of the fluid around the screw.

he second-degree polynomial regression from the simulations clearly reflects this correlation between confinement and efficiency.

This study therefore provides decisive results concerning the dimensions of the screw in relation to the available width in the screw channel: to optimize efficiency, it is recommended to apply maximum horizontal confinement.

Figure 6 shows a top view of the channel representing the distribution of fluid flow velocity. The color scale, ranging from dark blue (low velocities) to dark red (high velocities), allows the areas of acceleration and deceleration around the screw to be visualized.

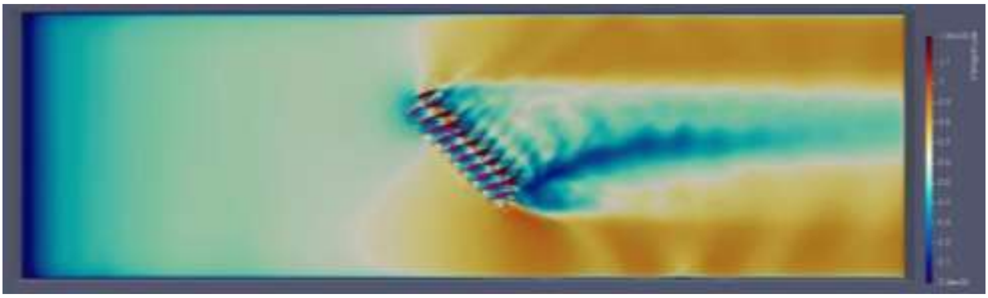

Fig. 6. Top view of flow velocity

The average fluid inlet velocity is set at $U_\infty = 0.71\ m.s^{-1}$, which corresponds to the experimental conditions. However, figure 6 clearly shows that the flow is not uniform in the channel:

❖ upstream of the screw, the fluid slows down slightly (light blue area) due to the presence of the obstacle.

❖ at the ends of the screw, there are areas where the fluid is fast (orange-red area), as the fluid flows around the obstacle.

❖ downstream, a low-velocity area (light to dark blue) forms, corresponding to the wake area where the disturbances generated by the rotation of the screw dissipate kinetic energy.

4 Conclusion

The main objective of this paper was to investigate the influence of horizontal turbine confinement on hydraulic performance. To achieve this, a methodology combining experiments and CFD simulations was employed. First, a 3D CFD model was developed using OpenFOAM, and the numerical results were validated against experimental data obtained from a newly designed experimental device.

Once the CFD model was validated, different levels of horizontal screw confinement were simulated while keeping the screw geometry, water depth, and flow velocity constant. The results show that turbine efficiency strongly depends on horizontal screw confinement. Specifically, the hydraulic efficiency reaches approximately 40% for a confinement of 85% and decreases to around 28% for a confinement of 10%. This behavior can be primarily explained by the fact that the flow is more constrained as the confinement increases. From an operational perspective, these results highlight the importance of accounting for confinement when estimating the energy potential of this type of turbine.

It should be noted that these results were obtained for a single screw geometry and a single flow velocity. It is possible that the results would vary with changes in these parameters. Moreover, the data may be strongly influenced by vertical confinement, which will be investigated in future work.

Nevertheless, the numerical model developed here is a robust tool for conducting optimization studies that are difficult to perform experimentally, particularly on screw geometric parameters such as pitch, diameter, or length. This approach thus paves the way for further optimization of energy recovery devices based on horizontal-axis Archimedes screws.

References

1. RTE (2023). French renewable generation needs to rise threefold by 2035, according to RTE. En-former, RWE. Disponible sur : https://www.en-former.com/en/french-renewable-generation-needs-to-rise-threefold-by-2035-rte/
2. C.Rorres, "The turn oh the screw : optimal design of an Achimedes screw", in Journal of Hydraulic Engineering, 2000, p 73-80.
3. G.Dellinger, “Etude expérimentale et optimisation des performances hydrauliques des vis d'Archimède utilisées dans les micro centrales hydroélectriques”, Ph.D. dissertation, INSA, Strasbourg, 2015
4. X.Brunissen, “Analytical model for Archimedes screw generators with rotating trough”, in Journal of hydraulic Research, 2025, 63:3, 357-373.
5. J.Betancour, L.Velásquez, L.Y.Jaramillo, E.Chica, A.Rubio-Clemente, "Application of a central composite face-centered design in the optimization of an Archimedean hydrokinetic turbine", Renewable Energy and Power Quality Journal, 19 : 418-422, 2021
6. P.E.Slaboch, J.Coday, "Design and analysis of small scale horizontal archimedean screw for electric power generation", International Mechanical Engineering Congress and Expopsition, 2017
7. A.Stergiopoulou, V.Stergiopoulo, E.Kalkani, "Experimental and theoretical research of zero head innovative horizontal axis archimedean screw turbines", International Journal of Energy and Environment, 6 : 471-476, 2015
8. A.MICHEL, Journée Société Hydrotechnique de France conference: Cavitation, Machines, and Hydraulic Transients highlighted the impact of the angle between the screw axis and the flow direction on efficiency, 2026
9. T.KINSEY, G.DUMAS, "Impact of channel blockage on the performance of axial and cross-flow hydrokinetic turbines", Renewable Energy, 2017, p 239-254