Investigation of the Behavior of the Fluid of a Micro Hydroelectric Gravitational Vortex, by Means of the Computational Dynamics of High Performance Fluids, for the Generation of Electric Power

Sánchez Ocaña Wilson, Haro Valladares Jonathan, Sanaguano Jiménez Edison
Departamento de Eléctrica y Electrónica
Universidad de las Fuerzas Armadas ESPE, ID: 60104598, Av. General Rumiñahui s/n,
Sangolquí, Ecuador, P.O.BOX: 171-5-231B
Email: wesanchez@espe.edu.ec, jonathanalexisharo@gmail.com, efsanaguano@gmail.com

Salazar Jácome Elizabeth
Departamento de Ciencias Exactas
Universidad de las Fuerzas Armadas ESPE, ID: 60104598, Av. General Rumiñahui s/n,
Sangolquí, Ecuador, P.O.BOX: 171-5-231B
Email: mesalazar2@espe.edu.ec

***Abstract*—The demand for energy is increasing, especially in developing countries. Renewable energies such as hydroelectric power, has become one of the most demanded energy sources for its generation, that is why it was studied and analyzed, through the computational dynamics of fluids "CFD" in the software ANSYS, the flow of a micro vortex gravitational hydroelectric power station for the generation of electric power. This study analyzes a structure that, by its design, has the capacity to form a gravitational vortex current from a water flow with a small difference in height. To verify results, a prototype of the system is built, which will generate energy from the formation of the gravitational vortex.**

Index Terms— ANSYS CFD, energy, simulation, gravitational vortex.

I. Introduction

In Ecuador, the National Government supports the execution of renewable energy projects, adapting to the new energy matrix. [1] To this is added that, due to its geographical position, hydraulic energy can be used in almost all of its territory. [2]

Besides encouraging the use of dynamic computational tools for fluids, since this is not intuitive, if not impossible, to predict the behavior of fluid flows in a given system. [3]

For this reason, the creation of this type of projects related to renewable energies and computational tools require the training of professionals with extensive knowledge of design and construction. [4]

One of the main problems affecting the world's population is environmental pollution, caused in part by the generation of electricity through nuclear or steam power plants[5]; which requires the incentive to investigate new forms of electricity generation that do not produce pollution and reduce environmental impact [6], such as the vortex micro-central gravitational power plant [7], in addition, computer simulation allows the person to experiment with many different policies and arguments without changing or experimenting with the existing real system and therefore guarantees economic savings since it is possible to modify variables without the need to implement a prototype or system for each one of them. [8]

Considering that it is necessary to research and implement the use of renewable energies that contribute to the ecosystem, as well as assuring alternatives to the productive matrix. This research project consolidates the study of the fluids of a micro-hydroelectric power plant.

By implementing the prototype of a vortex gravitational micro-centre, since it uses a new generation system and little knowledge at an institutional level.

Manuscript received June 3, 2018; revised December 9, 2018.

doi: 10.18178/ijmerr.8.1.79-86

A. Vortex Gravitational Hydroelectric Power Station

It is an innovative solution for the generation of electrical energy designed by the Austrian engineer, Franz Zotlötere known as a gravity hydroelectric plant with vortex; [7] its energy comes from the swirl of artificially caused water, this type of power plant is convenient from a water flow with small differences in height and places with high ecological sensitivity. [9]

Its operation is based on a round pond with a central drainpipe, the flow of water that is transported forms a stable vortex with which it moves the turbine and generates electricity [10], despite the fact that this plant has a lower yield than conventional micro-hydropower plants, its environmental impact is much lower because fish can be transported freely. [9]

B. Movement Equations

For stationary laminar flow of a viscous, Newtonian, incompressible fluid without free surface effect, the equations of motion are the continuity equation: [11]

$$\vec{\nabla} * \vec{V} = 0 \tag{1}$$

And the Navier-Stokes equation:

$$(\vec{V} * \vec{\nabla})\vec{V} = -\frac{1}{\rho}\vec{\nabla}P' + v\nabla^2\vec{V} \tag{2}$$

Motion equations can be solved by CFD for the stationary, incompressible, laminar flow case of a Newtonian fluid with constant properties and no effect of free surfaces. A Cartesian coordinate system is used. There are four equations and four unknowns: u, v, w and P'. [8]

C. Continuity:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{3}$$

The set of equations that describe the processes of movement, heat and mass transfer are known as the Navier-Stokes equations. 8] These partial differential equations originated in the 19th century and have no known analytical solution, but in general they can be discreetly and numerically solved. [12]

D. Computational Fluid Dynamics

Of the acronym in English CFD Computational Fluid Dynamics, [13] within the Mechanics of fluids that is based on the computational numerical analysis to solve problems that, by their complexity, can't be solved analytically and are related to the movement of fluids, heat transfer, chemical reactions etc. [14]

The computational dynamics of fluids is a technique that uses computer programs and advanced computers capable of executing a large number of calculation operations per unit of time, thus reducing time and design costs, obtaining results subject to the accuracy of the assumptions, approximations and idealizations established in the analysis. [8]

II. Methodology

A. Vortex Gravitational System

The material for the polymethyl methacrylate vortex gravitational system with its mechanical properties shown below (Table I).

TABLE I. Acrylic Mechanical Properties

Mechanical properties of polymethyl methacrylate		
Property	***Value***	***Unit***
Density	1180	Kg/m3
Young module	6000	MPa
Poisson module	0.33	-
Creep resistance	70	MPa
Density	1180	Kg/m3

A hydrostatic force was applied, which simulates the force of water resting in the system, obtaining a maximum pressure of 1000 Pa (Fig. 1).

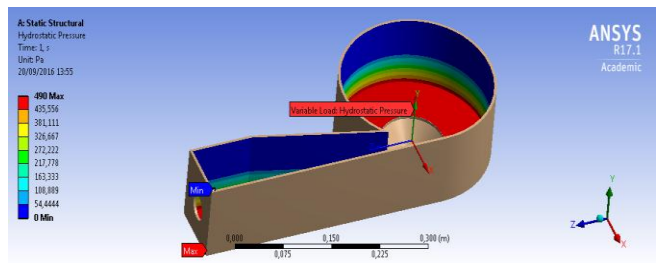

Figure 1. Hydrostatic force applies to vortex gravitational system.

B. ANSYS CFX Study

The study was carried out taking into account the domain of interest that is, isolating the geometry to be simulated.

C. Solid Model

The CAD of the system was developed in Software ANSYS ACADEMIC, in the module "Design Model" (Fig. 2).

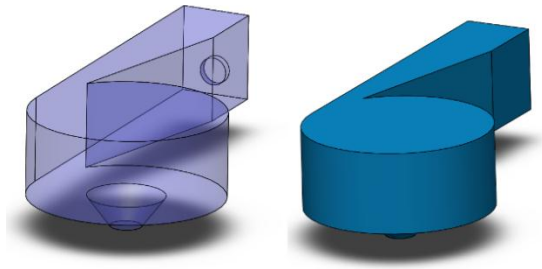

Figure 2. Solid model to use

D. Meshing

To obtain a mesh quality within the acceptable values, two meshing controls were carried out, which allow to increase the number of divisions and make it symmetric, creating only hexahedrons by means of quadrilateral / triangles (Fig. 3).

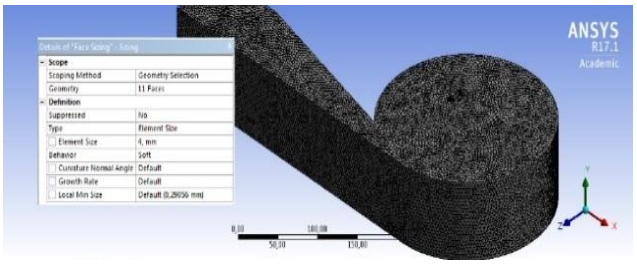

Figure 3. Mesh

E. Mesh Quality

In order to verify that the mesh is correctly made for the study, it will be determined by means of the obliqueness and orthogonality criteria, these are parameters that ANSYS CFX recommends to obtain an adequate solution of the system [14] (Table II).

TABLE II. MESH QUALITY OBTAINED

Parameter	*Software recommended*	*Obtained by simulation*	*Mesh quality*
Orthogonality	Median 0.95-1	Median 0.86	Very good
Obliquity	Median 0-0.25	Median 0.24	Prime

F. Pre-processing ANSYS CFX

ANSYS Pre-processing is used to define the simulation parameters, when starting we will have the whole system with initial parameters that are provided by the software, these parameters have to be replaced by those that most closely resemble reality, since this will help to have a faster convergence.

G. Initialization Parameters

In this parameter we need to activate the "Transient" option, once located in this section, we will enter the real time that elapses until the system is stabilized in the "Total time" tab, having made a small system, the stabilization time is 40 seconds. In the "Timesteps" tab you enter every how long you want to obtain and store a result of the internal analysis of the pre-processing, you have entered a time of 0.02 seconds; the two entered values give us the number of iterations that the software will perform, this is obtained by dividing the total time for the fractions of time, that is to say, we will have 2000 iterations.

H. Definition of Initial Parameters

In order to carry out the simulation, it must be taken into account that there will be interaction between water and air, therefore, two types of materials must be created with the necessary characteristics.

In the model of the domain the reference pressure is placed in the model of the domain that is equal to an atmosphere, in the following sections the axis in which gravity will act with its respective density will be selected; these values are standard.

In the "Fluid Models" tab, the following parameters are placed: "Multiphase" and "Homogeneous Model", which will allow to represent the two flows independently, in the turbulence option we select the option "SSG Reynols Stress", this model is based on the equation of movement of all its components, and we keep the other parameters constant since they are independent of the modeling that we are going to perform.

The characteristics of "Fluid Specifics Models" are maintained, in "Fluid Pair Models" the "Surface Tension Coefficient" option is selected, the surface tension coefficient is entered which is 0.072 N/m in the international system. Once the configuration is complete, we apply and accept the changes (Fig. 4).

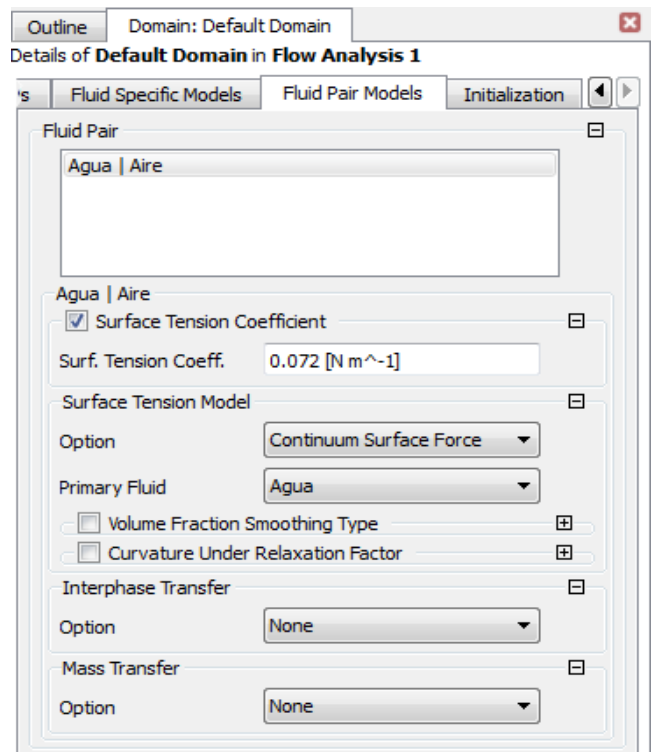

Figure 4. Definition of characteristic parameters

I. Definition of Domains

In this section, 4 different domains will be created: entrance, exit, free surface and walls. Each of these domains need different configurations that are detailed below.

- Entry.- The place where the fluid will enter is selected. "
- Departure.- In basic configurations, the place where the fluid will exit is selected and it is specified that it will be "Opening".
- Free surface.- The place that is going to have a free surface is selected and it is specified that it will be open; that is, it will be in direct contact with the air "Opening".
- Walls.- In this section the software recognizes all the faces that have not been previously used as a wall. When using polymethyl methacrylate in the construction of the system, one of the main characteristics of this material is to present a minimum friction, so it is placed as null before the fluids.

J. Expressions

To define a value based on the variation of the fluid, the following equations are created.

K. Continuous Water Flow

It is defined as (Water Flow), this expression allows us to keep the water flow constant during the established time which is 2 minutes. A flow of 0.7 kg / s that was obtained experimentally with the prototype of the gravitational vortex system will be used. [15]

$$if\,(t < 2[min], 0.7\left[\frac{kg}{s}\right], 0\left[\frac{kg}{s}\right]) \tag{4}$$

L. Water Pressure

It is defined as (Water Pressure), this expression defines the pressure exerted by the fluid in the walls of the vortex gravitational system and is given by: [15]

$$997\left[\frac{kg}{m^3}\right] * 9.80665\left[\frac{m}{s^2}\right] * (2.5[cm] - y) * aguaVF \tag{5}$$

$9.80665\left[\frac{m}{s^2}\right]$	Gravitational acceleration
$2.5[cm]$	Water Inlet height
y	Existing water level
$aguaVF$	Volume Fraction
$997\left[\frac{kg}{m^3}\right]$	Water Density

M. Water Pressure

It is defined as (waterVF), this expression takes the value of 1 when the water level of the simulation does not exceed the level of water input height [15].

$$if\,(y < 2.5[cm], 1{,}0) \tag{6}$$

III. Solution Control

In the solution control algorithmic discretization is defined for the terms previously set in initialization parameters, ANSYS recommends the option, "Second Order Backward Euler", this option is applicable for constants and variables in time steps dimension, and ideal for transient regime, First Order Backward Euler solves turbulence equations. In the convergence control, the number of sub-iterations that the software will perform for each iteration proposed above is selected; there will be a total of 20,000 sub-iterations, it must be taken into account that all these mentioned configurations are pre-established by the software (Fig. 5).

A. Output Data Control

In the option "Output Control" a label is created for the variables that you want to monitor, there are several types (Fig. 6), of which the following are used:

- Conservation of water volume fraction
- Absolute pressure
- Pressure
- Water speed
- Air speed
- Speed
- Fraction of water volume
- Fraction of air volume.

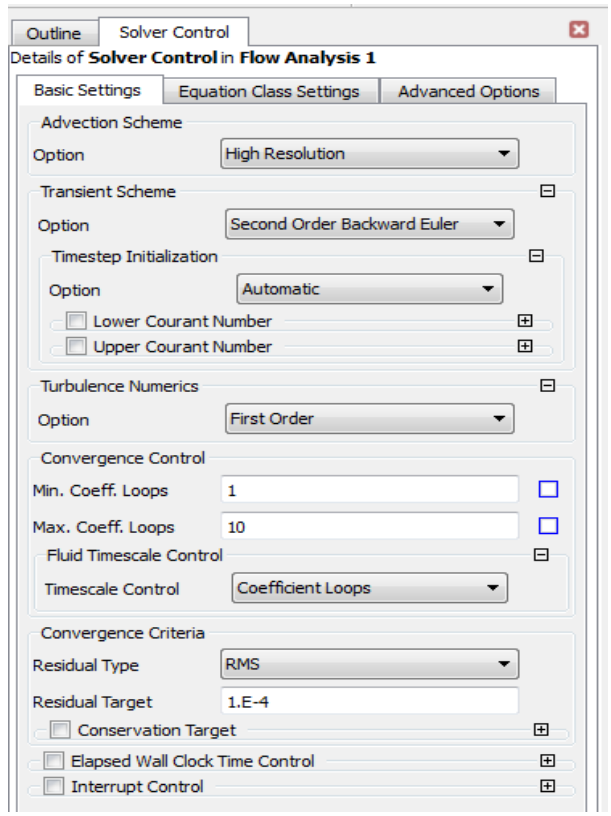

Figure 5. Solution control.

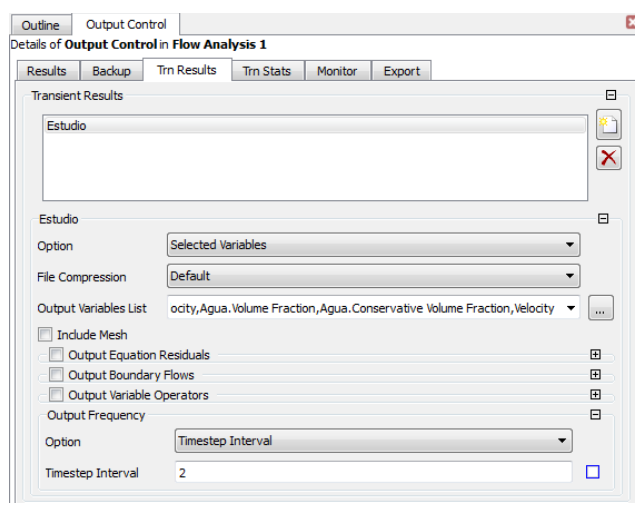

Figure 6. Output data

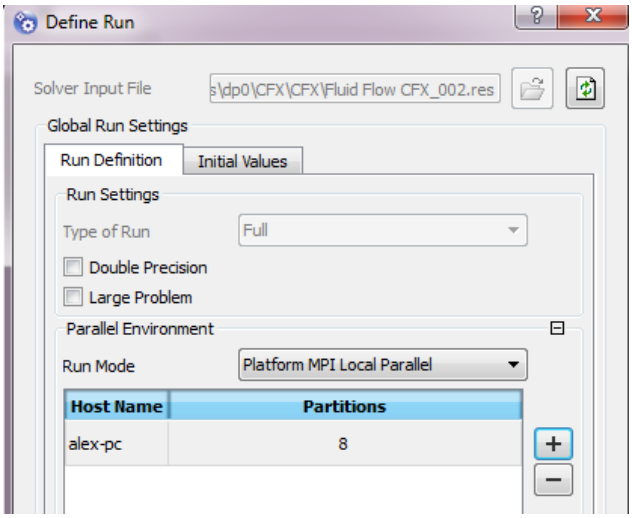

Figure 7. Defining partitions

B. *CFX Solver Manager*

To be able to simulate correctly, choose to start the simulation using the initial values; for faster processing speed, it is recommended to use all processor cores using the "Platform MPI local parallel" (Fig. 7).

IV. Analysis of Results

The "CFX-Result" module is generated from the "CFX-Solver" module, this section contains the data generated above, as well as the entire description of the fluid including the meshing of the volume and the solution of the system.

A. *Mesh Statistics*

Mesh statistics summarize the specific and global domains they are:

Diagnosis of mesh quality.- Mesh quality diagnostics include the values of mesh orthogonality, expansion and radius aspect, for each of these values there is a range defined by "good (OK), acceptable (ok), questionable (!)" (Figure 8).

```
+-------------------------------------------------------------------+
|                        Mesh Statistics                            |
+-------------------------------------------------------------------+
| Domain Name          | Orthog. Angle | Exp. Factor  | Aspect Ratio |
+----------------------+---------------+--------------+--------------+
|                      | Minimum [deg] |   Maximum    |    Maximum   |
+----------------------+---------------+--------------+--------------+
| Default Domain       |       22.2 ok |        326  ! |        14 OK |
+----------------------+---------------+--------------+--------------+
|                      |  %!  %ok  %OK | %!  %ok  %OK | %!  %ok  %OK |
+----------------------+---------------+--------------+--------------+
| Default Domain       |   0    5   95 |  4   13   83 |  0    0  100 |
+----------------------+---------------+--------------+--------------+
```

Figure 8. Mesh statistics

The orthogonality angle has a domain of 22.2 ° and is defined as acceptable due to the acceptance ranges shown in table 3.

TABLE III. Minimum Orthogonality Angle

Minimum orthogonality angle	***Ok***	***>50 °***
	Ok	50 °> 20 °
	!	< 20 °

The mesh expansion factor has a domain of 326 and is defined as questionable due to the acceptance ranges shown in Table IV.

TABLE IV. Mesh Expansion Factor

Mesh expansion factor	***Ok***	***< 5.0***
	Ok	5.0 < 20.0
	!	> 20.0

The maximum radio aspect has a domain name of 14 and is defined as good due to the acceptance ratings show in the Table V.

TABLE V. Maximum radius Aspect

Maximum radius aspect	***Ok***	***< 100.0***
	Ok	100.0 < 1000.0
	!	> 1000.0

B. *Total Number of Nodes, Elements and Boundaries in the Grid*

This section details the number of nodes that depending on the type and quality of mesh were created, this quantity has to be contrasted with the number of nodes allowed by the software, since it is an academic version and allows a maximum of 512 thousand nodes (Fig. 9).

```
+-------------------------------------------------------------------+
|                        Mesh Statistics                            |
+-------------------------------------------------------------------+

Domain Name : Default Domain

    Total Number of Nodes                              =      114739

    Total Number of Elements                           =      122315
        Total Number of Tetrahedrons                   =        7890
        Total Number of Prisms                         =        1221
        Total Number of Hexahedrons                    =       99483
        Total Number of Pyramids                       =       13721

    Total Number of Faces                              =       17876
```

Figure 9. Total number of nodes

The total number of nodes is 114739 and the total number of elements is 122315, the largest number of elements is found in the hexahedron section; this is because when making the mesh is selected this option to have a better quality of mesh, the other elements are used mainly in corners and rounding of the system, the total number of faces created is 17876.

C. Vortex Simulation Based on Time

The results obtained will be visualized in the course of time to examine and extract useful data from the simulation, these results will be analyzed for the subsequent generation of energy.

D. View 0.5 Seconds

At 0.5 seconds the impulse of the water hits the left side wall of the system, for the initial formation of the artificial vortex with a speed of 0.75 m / s; the pressure exerted by the water at the beginning, causes that at the entrance to the cylinder there is a small overflow, which is almost immediately eliminated (Fig. 10).

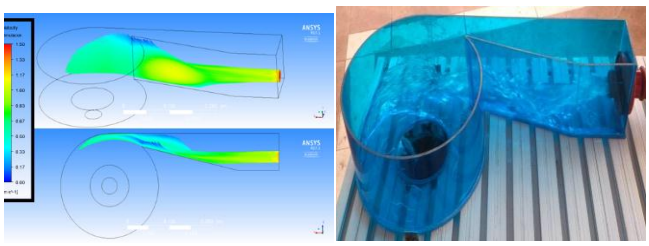

Figure 10. Simulated and real view at 0.5 seconds

E. View 1.5 Seconds

At 1.5 seconds the water travels almost the entire profile of the cylinder without overflowing, in addition to already forming the vortex in a small fraction, with an average speed of 0.52 m / s; in the section of the start of artificial vortex, the water that does not enter the cylinder returns to the entrance (Fig. 11).

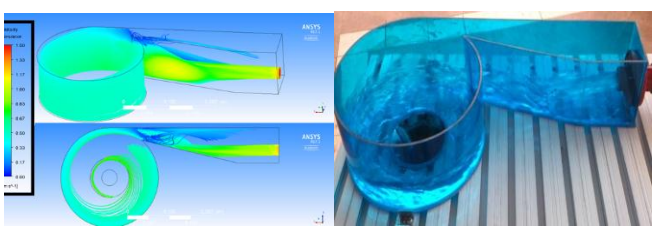

Figure 11. Simulated and real view at 1.5 seconds

F. View 6 Seconds

At 6 seconds the whole system is stabilized, maintaining a constant flow in the gravitational vortex with a velocity of 0.9 m / s, this being the speed with which electric power will be generated by a DC generator; the water that does not enter the cylinder remains in constant motion until it is driven again (Fig. 12).

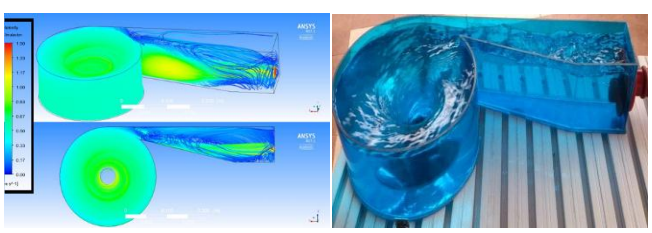

Figure 12. Simulated and real view at 6 seconds

G. Volume Fraction of Water and Air in the Gravitational System

In the Figs. 13, 14 and 15, show the sectional views of the system shown; the blue color shows the volume fraction of water and the gray volume fraction of air.

Figure 13. Top view of volume fraction of water

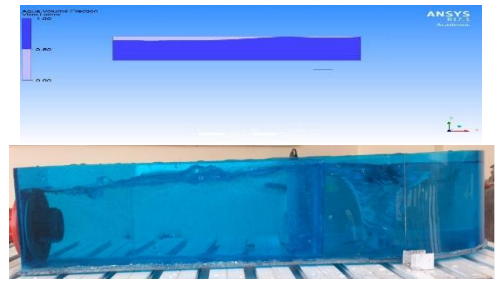

Figure. 14 Top view of volume fraction of water

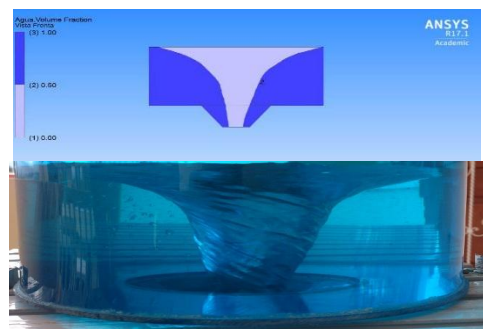

Figure 15. Front view volume fraction of water

H. Surface Pressure

The water exerts a pressure on the walls of the gravitational system, being 1050 pascales the maximum pressure that is mainly exerted on the edges of the lower part of the cylinder, there is a slight variation of pressure in the section where the water collides when entering the system between 583 and 700 pascales (Fig. 16).

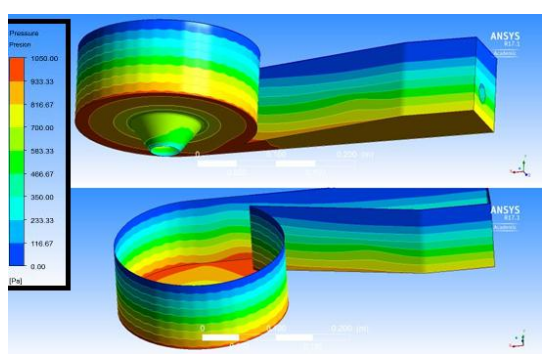

Figure 16. Pressure on the walls

I. Theoretical Power

For the calculation of the electric energy, the potential energy of the water is required, the maximum theoretical hydraulic power value is given by the formula:

$$P(W) = Q * g * \delta * h \tag{7}$$

$$P(W) = 0.0007\frac{m^3}{s} * 9.80665\frac{m}{s^2} * 997\frac{kg}{m^3} * 0.05m$$

$$P(W) = 0.34W$$

J. Actual Power

To obtain the actual power of the system, the voltage and current is measured with a digital multimeter, using as load 4 coreless mini motors with a resistance of 9 ohms each, thus obtaining a voltage of 0.28V and 0.1A as shown in the Figs. 17 and 18.

Figure. 17. Digital multimeter

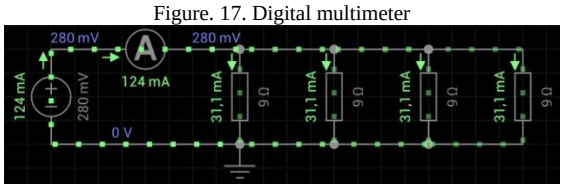

Figure 18. Circuit diagram

A total power of:

$$P_{real} = V \times I \tag{8}$$

$$P_{real} = 0{,}28\,V \times 0{,}124A$$
$$P_{real} = 0{,}03472\,Vatios$$

With an efficiency of:

$$Efficiency = \frac{P_{theoretical}}{P_{real}} \times 100\%$$
$$Efficiency = \frac{0{,}03472}{0{,}34} \times 100\%$$
$$Efficiency = 10{,}21\%$$

V. Conclusions

The study was carried out, and analyzed by means of computational fluid dynamics in ANSYS CFX, the flow behavior in a prototype of a gravitational vortex power plant, demonstrating that in simulation as in real life, water flow acts in the same way.

With the software license, it limits the construction of a larger prototype, since the meshing restricts to a specific number of nodes.

Mesh quality will not always be better if you refine it further, it is important that the mesh expansion measures, orthogonality and symmetry are in the right ranges, because if not met you will have erroneous results or in turn the simulation will not finish and show errors.

In the simulation it presents a small splash of water, which, in order to correct it, a small lid was placed on the upper part of the vortex gravitational system entrance.

The gravitational vortex system was developed in polymethylmethacrylate, since it allows the visibility of the water flow behavior and thus, be able to compare with the simulation results.

For the generation of voltage, turbine number 1 proved to be the most suitable, because it has six blades, unlike the others that have a larger number.

The convergence time of the analysis performed depends to a large extent on the computational tools available and the initial conditions established.

The behavior of the fluid is turbulent in the first 3 seconds, then stabilizes forming a stable vortex within the cylinder, and remains turbulent in the inlet section.

The different time-dependent results obtained match exactly with the behavior in the gravitational prototype of the vortex.

The highest water velocity was located in the lower part of the discharge cone of the vortex gravitational system, being this 0.9 m/s.

The greater amount of pressure that is exerted on the surface of the polymethyl methacrylate is given in the inferior vertices of the cylinder of the system, since the weight is in constant movement on them, another section is in the face that is in front of the water entrance, because it enters and collides there, first of all, before continuing the course during the whole system.

It was justified that the design of the developed vortex gravitational system was adequate for the established needs.

VI. Limitarions of Research and Future Work

With the software license it has limits the construction of a prototype, since the meshing restricted to a specific number of nodes.

Developing this type of projects in the future opens the possibility to design more sophisticated water plants presenting an economic saving and using sources of renewable energies that help to preserve the environment.

References

[1] M. D. E. Y. E. Renewable, Accountability Report, 2012.

[2] University of the Espe Armed Forces, "Research lines," Research Management Unit, 09 February 2015. [Online]. Available: http://ugi.espe.edu.ec/ugi/lineas-de-investigacion/. [Last accessed on 23 November 2016].

[3] ANSYS, "Fluids," ANSYS ACADEMIC, September 10, 2016. [On line]. Available: http://www.ansys.com/es-ES/products/fluids. [Last accessed on 15 October 2016].

[4] UNIVERSITY OF THE ARMED FORCES ESPE, " Career of Electromechanical Engineering," Public information extension Latacunga, 20 June 2009. [Online]. Available: http://webltga.espe.edu.ec/site/index.php?option=com_content&view=article&id=52&Itemid=63. [Last accessed on 28 July 2016].

[5] EROSKI FOUNDATION, "The environmental problems that should concern us," EROSKI CONSUMER, 10 April 2014. [Online].

Available: http://www.consumer.es/web/es/medio_ambiente/naturaleza/2014/04/10/10/219707.php. [Last accessed on 24 September 2016].

[6] D. SUSTAINABLE, "New sources of energy," Juan David Montoya, 16 May 2013. [Online]. Available: http://www.desarrollosustentable.co/2013/05/nuevas-fuentes-de-energia.html. [Last accessed on 6 June 2016].

[7] Hidroenergia.net, "Hydroelectric plants," HIDROENERGIA, 5 October 2010. [Online]. Available: http://www.hidroenergia.net/index.php?option=com_content&view=article&id=103:plant-hydroelectric-de-vortice&catid=35:innovations&Itemid=63. [Last accessed on 2 June 2016].

[8] Y. A. Cengel and J. M. Cimbala, Fluid Mechanics, Mexico DF: McGRAW-HILL/INTERAMERICANAS.S.A. DE C.V, 2012.

[9] Wasserwirbe, "Naturally achieved electricity," genossenschaft, 31 October 2015. [Online]. Available: http://gwwk.ch/about/main-message/. [Last accessed on 10 June 2016].

[10] Wikipedia, "Wasserwirbelkraftwerk," 17 November 2017. [Online]. Available: https://de.wikipedia.org/wiki/Wasserwirbelkraftwerk. [Last accessed on 23 November 2016].

[11] Mott, Fluid Mechanics, Mexico: Pearson education in Mexico, 2014.

[12] S. Sebastian, " Mechanical Fluid Notes," University of the Basque Country, 25 January 2012. [Online]. Available: http://www.ehu.eus/inwmooqb/asignaturas/Mecanica%20de%20de%20fluidos/APUNTES%20DE%20MF%202011-12.pdf. [Last accessed on 20 October 2016].

[13] M. Kessler, "Computational fluid dynamics: What is it," ESSS, 24 June 2016. [On line]. Available: http://www.esss.com.br/blog/es/2016/06/dinamica-de-fluidos-computacional-que-es/. [Last accessed on 10 June 2016].

[14] ANSYS ACADEMIC 17.1, "ANSYS HELP," 01 June 2016. [On line]. Available: http://www.ansys.com/es-ES/About-ANSYS. [Last accessed on 24 August 2016].

[15] E. P. Armijo, Study of the hydraulic behavior of a flow through a gravitational vortex system using computational fluid dynamics techniques CFD, Loja: Private Technical University of Loja, 2014.

Wilson E. Sánchez, received the Electromechanical Engineer Degree in 2005, at the Polytechnic School of the Army, the Master in Production Management in 2013 at the Technical University of Cotopaxi – Latacunga, Ecuador and the Master in Design and Industrial Automation in 2017 at the National Polytechnic School, Quito, Ecuador. He worked for private companies in the oil sector and since 1997 for the University of the Armed Forces ESPE, as Principal Professor in the Department of Electrical - Electronics. His research interests include: simulation, modeling of biomechanical, mechanical and hydraulic systems, automation of industrial processes, renewable energy.

Jonathan Haro Valladares, He received the degree of electromechanical engineer in 2017 at the Polytechnic School of the Army, graduated in electrical risks in 2016, participated in the production of scientific articles in the area of simulation and modeling of fluid dynamic systems and energy efficiency. Research areas: fluid dynamics simulation, systems for saving energy consumption through the automation of industrial processes

Edison Sanaguano Jiménez, He received the degree of electromechanical engineer in 2017 at the Polytechnic School of the Army, graduated in electrical risks in 2016, participated in the production of scientific articles in the area of simulation and modeling of fluid dynamic systems and energy efficiency. Research areas: fluid dynamics simulation, systems for saving energy consumption through the automation of industrial processes

Elizabeth Salazar Jácome, received the B.S. degree in Computer Science Engineering from University of the Armed Forces ESPE in 2002 and her M.S. degree in Software and Informatics from University of the Armed Forces ESPE in 2013. She worked as Computer Expert in Consejo de la Judicatura in 2014. She has worked as Professor in University of the Armed Forces ESPE, Chief of Informatics Department in Colegio Militar No.13 Patria, Technical of Cámara de Gesell in Fiscalía Provincial de Cotopaxi. She has participated in conferences in many educational institutions.