Indian Journal of Science and Technology, *Vol 11(22), DOI: 10.17485/ijst/2018/v11i22/123055,* June 2018 *ISSN (Print) : 0974-6846*
ISSN (Online) : 0974-5645

Cavitation Analysis with CFD Techniques of the Impeller of a Centrifugal Pump

Wilson Sánchez Ocaña[1*], Christian Carvajal[1], José Poalacín[1], Monserrate Intriago Pazmiño[2], Elizabeth Salazar Jácome[1] and Luis Basantes[3]

[1]Departamento de Eléctrica y Electrónica, Universidad de las Fuerzas Armadas ESPE, ID: 60104598, Sangolquí, Ecuador; wesanchez@espe.edu.ec, cc.andres9316@gmail.com, pocito182@hotmail.com, netzonews@hotmail.com
[2]Departamento de Informática y Ciencias de la Computación, Escuela Politécnica Nacional, Quito, Ecuador; monserrate.intriago@epn.edu.ec
[3]Departamento de Ciencias Exactas, Universidad de las Fuerzas Armadas ESPE, ID: 60104598, Sangolquí, Ecuador; lfbasantes@espe.edu.ec

Abstract

Background/Objectives: The present work evaluates the phenomenon of cavitation in the impeller of a centrifugal pump and observes the behavior of said phenomenon computationally in a simulation as in a physical way in a real pumping system. **Methods:** The method used is the CFD simulation techniques of ANSYS that allowed designing and analyzing the impeller of the centrifugal pump with convergence criteria of 1000 iterations, an rms value off low lines of 1e-4, a physical scale time of 0.00276 and a conservation value of 1%. In order to analyze the phenomenon of cavitation in the eye of the pumping system impeller, the ball valve V1 was closed an angle of 45°, located in the suction pipe to create an empty pression exceeding the vapor pressure of the pumped fluid. Experimentally recording the pressure reduction at discharge from 42.75 KPa to 19.31 KPa. **Findings:** The results obtained with the help of the CFD module allowed us to observe the fluid pressure distribution on the pump impeller blades with a value of 2064 Pascals (Saturation Pressure at 18 ° C), indicating the behavior of the fluid, when it passing from a liquid state to a steam state, causing the formation of vapor bubbles whose implosion causes the surface of the impeller to wear out and it is evidenced computationally the areas affected by the presence of the cavitation phenomenon. These surfaces wear out in an area close to the impeller's eye, and then we observe aware of the material produced by lack of maintenance known as average erosion and finally total impeller damage caused by vapor bubbles. **Applications/Improvements:** It is suggested for this work to perform an analysis of the other elements that suffer damage due to the erosion of the material of the pump impeller, among them the casing and other elements.

Keywords: Cavitation Analysis, Cavitation Phenomenon, Centrifugal Pump, CFD Techniques, Impeller

1. Introduction

The centrifugal pump is a machine that converts mechanical energy into pressure energy by means of the centrifugal force acting on the fluid[1]. The main characteristic of centrifugal pumps is that they allow liquids to be lifted to a lower or higher level, depending on the need for application. Compared to other types of pumps, these pumps have a high efficiency and performance and are therefore used in industrial, agricultural and domestic applications.

For this reason, the design of the impeller must consider a detailed understanding of operation in both nominal and partial conditions[2]. In order to obtain a machine with high profitability that is subject to high demands, it is necessary to predict the flow rate performance of the entire pump, which requires a thorough analysis of the flow through the pump, which is three-

**Author for correspondence*

dimensional and turbulent by nature. For this reason, the selection of a Computational Fluid Dynamics (CFD) study allows us to visualize the state of the flow inside the pump, in addition to experiencing phenomena that affect the centrifugal pump such as cavitation. Many researchers have developed studies through CFD with the case of[3], who analyzed the distribution of pressure in the passage of the blades and the respective pressure diagrams. The study by[4] showed that the decrease in the efficiency of the centrifugal pump was due to the analysis of non-uniformities in different parts of the pump[5] have carried out a performance study based on the variation in the angle of the outlet vanes, where the discharge height increased by more than 7% with an increase in the angle of the outlet vanes from 20° to 45° [6] have carried out the cavitation modeling of a centrifugal pump, since the vapor pressure is directly related to the temperature of the fluid, which allows us to observe the areas affected by cavitation and the improvements that can be made in the design of the system to prevent or reduce the phenomenon.

2. Materials andMethods

The present work seeks to develop a cavitation model applying a CFD model covering numerical techniques with several aspects: Selection of flow equations, generation of meshing, establishment of appropriate boundary conditions, and choice of turbulence model, resolution of equations in selected meshing and subsequent processing of results[7]. The fluid flow solution is performed by a numerical method based on a homogeneous liquid-vapor model[8], where the mixture is specified in each control volume for the subsequent solution. The cavitation phenomenon is analyzed by CFD, which is a dynamic process of bubble formation within the liquid, its growth and subsequent collapse as the liquid flows through the pump[9].

The formation of vapor bubbles within the mass of a liquid begins when the absolute pressure of the liquid is equal to or falls below the vapor pressure of the liquid[10]. Once the vapor bubbles have formed in the flowing liquid, the regions with the highest pressure present real structural damage[11] which can be observed. Experimental and theoretical studies have shown that the collapse of a bubble produced by cavitation can produce a high velocity microjet, resulting in eventual erosion of neighboring surfaces, specifically causing holes in the impeller[12].

2.1 General Characteristics of a Pump Impeller Design

The design of the driver is a very important aspect of CFD analysis, a good design and subsequent meshing allows for convergence of solutions. To develop the impeller design, we used the centrifugal pump data sheet as shown Figure 1, this data is entered into the CPD VISTA module, which uses a one-dimensional (1D) approach to the preliminary design of the centrifugal pumps, the module allows us to generate an optimized impeller design before it moves to a 3D geometry and after a CFD analysis.

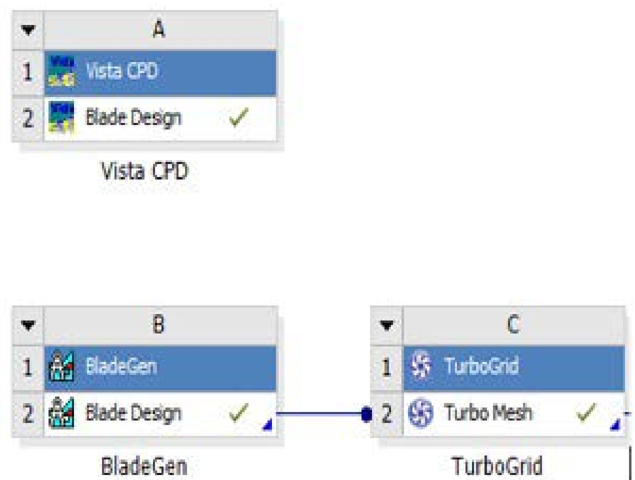

Figure 1. Impeller design modules.

Cavitation detection is not an easy task, as it depends on the design of the monitored equipment; the operating conditions, the type of cavitation and the location of the phenomenon are different[13]. For a 3D view of the impeller we support our 1D model through the BladeGen module. Once entered in BladeGen we can see the impeller in 1D and 3D geometry.

In Figure 2, the mesh quality for the optimal development of solutions is done through TURBOGRID, which is a powerful tool that uses high quality hexahedral mesh to allow a high convergence in the search for solutions. In the Turbogrid module it allows uniformity of the sections by means of uniform meshes.

2.2 Fluid Domain Configuration

The setting domain specifies the angular velocity of the impeller at 3450 rpm and rotates along the Z axis as shown Figure 3. We selected the turbulence model in

SHEAR STRESSTRANSPORT, to represent predictions of the onset and amount of flow separation under adverse pressure gradients, including the transport effects on the turbulence viscosity formulation, the fluid temperature, is 18°C.

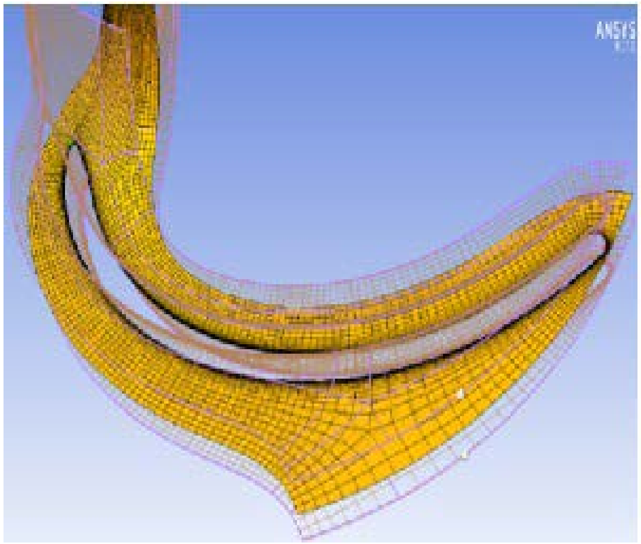

Figure 2. Impeller meshing.

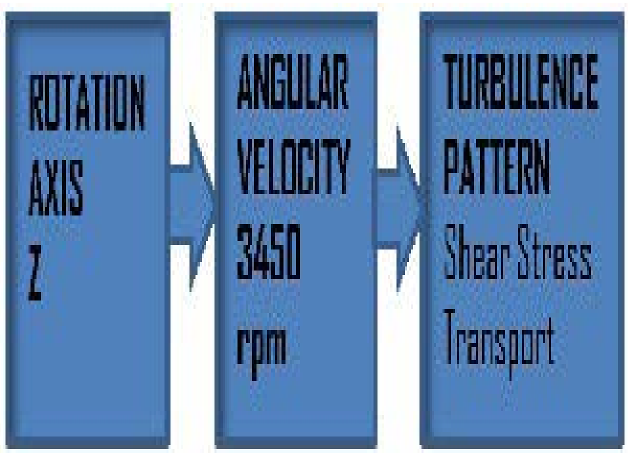

Figure 3. Domain configuration.

2.3 Edge Condition

Input edge condition. The data is taken from the centrifugal pump module of the University of the Armed Forces, where the value of the flow rate in normal operation is 77.9 GPM, the suction diameter of one and a half inches, with these data the fluid velocity is calculated, the input limit condition is 4.31 m/s. as shown in Figure 4.

Output edge condition. The discharge pressure is a data from the pump module of the University of the Armed Forces with a value of 42.75 KPa.

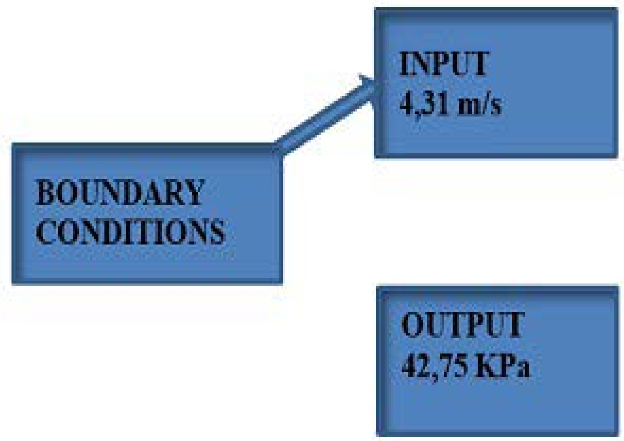

Figure 4. Edge condition.

2.4 Convergence Criteria

As shown Figure 5, the following convergence criteria can be mentioned: The number of iterations will be the first convergence criterion, the "Solver" will run to find the numerical solution of the equations of government (continuity and momentum), and 1000 iterations are established.

The second convergence criterion requires an accuracy of the flow lines and pressure contours. The rms value is specified in 1e-4 because it is an engineering level value[14].

The third convergence criterion is the "Time Scale Control", which is the value of the time scale, allows the equations to be solved; a value of 1 / Ω is set, where omega (Ω) is the angular velocity (3450 RPM), transformed into radians per second.

The fourth convergence criterion is the "Conservation Target", which guarantees the accuracy of the results and the conservation of the equations by entering a value of 1%.

2.5 CFD Analysis under Cavitation Regime

To determine the behavior of the impeller in cavitation, data from the test bench were taken by choking the ball valve located in the suction pipe (V1) at 45° [16], recording a drop in discharge pressure. As shown in Figure 6, in the main Ansys Worbech screen we duplicate the module project and proceed to designate it with CAVITATION. In the new module we adjust the parameters obtained in the test bench; we create the water vapor and place its properties. In the cavitation model the fluid saturation pressure is entered, in this case the module registers a temperature value of 18° C, the saturation pressure is 2064 Pa[16]. The edge condition at the input is designated and the variables are designated, the value of' 0' for steam and '1' for water. The edge conditions are adjusted with the module data when working under cavitation. In the outlet condition, the discharge pressure drops to 19.31 KPa and 46.6 GPM.

Data obtained from pumping system data acquisition.

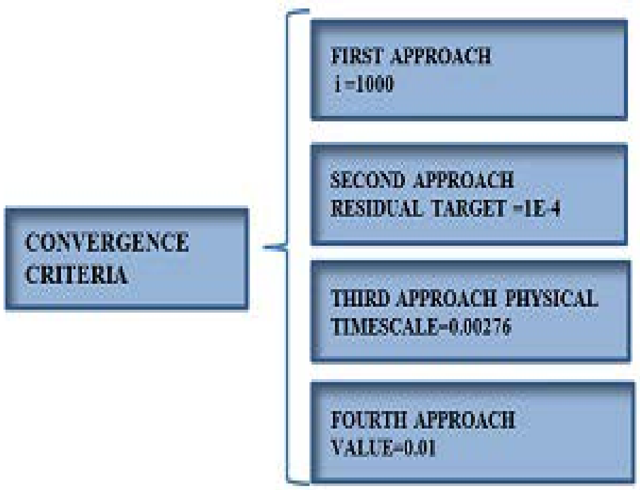

Figure 5. Convergence criteria.

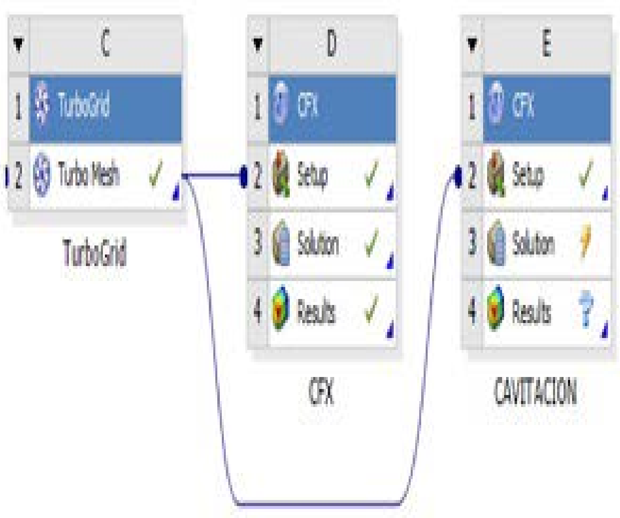

Figure 6. Cavitation module.

3. Result and Discussion

The presence of the cavitation phenomenon is evident in the blue zones, the upper walls of Figure 7 show that they are subject to saturation pressure 2064 Pa and that area affects the impeller in a total way.

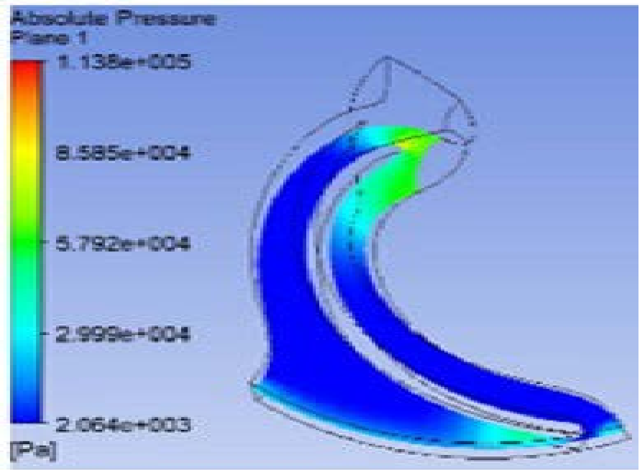

Figure 7. Upper wall in cavitation regime.

The passage of the fluid from the liquid to the vapor is recorded and therefore the formation and implosion of bubbles begin their harmful passage in the eye of the impeller and propagate through the blade as seen in Figure 8.

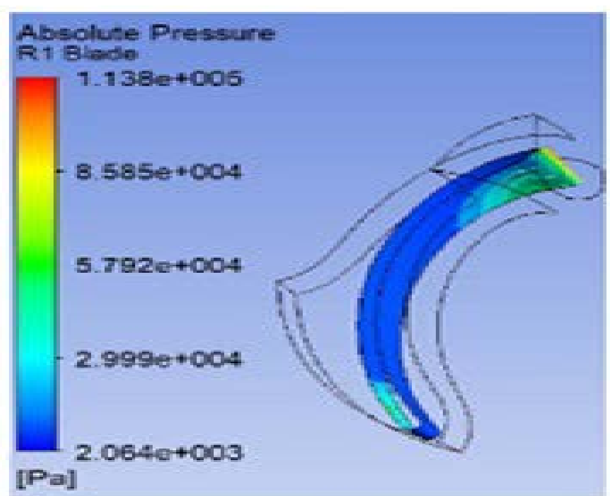

Figure 8. Blade in cavitation regime.

The type of cavitation on the impeller eye caused by a vacuum pressure in the pipe suction has effect by evidencing the erosion of this zone which will propagate throughout the blade and upper and lower walls of the impeller.

The first stage of cavitation occurs with a slight red bite in the area near the eye of the impeller, this phenomenon known as slight erosion as seen in Figure 9.

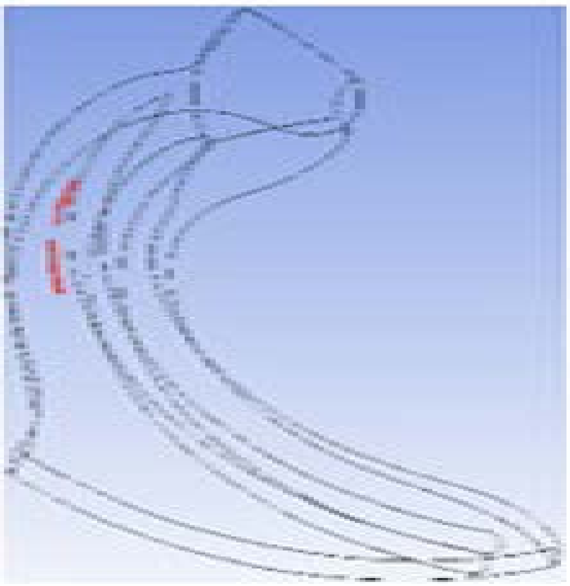

Figure 9. Slight cavitation.

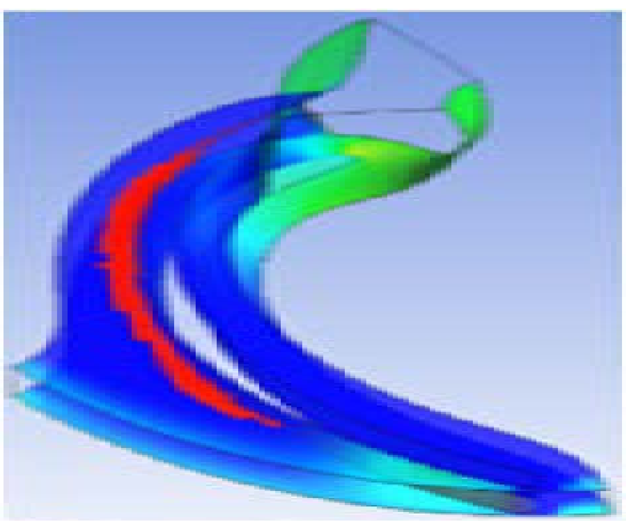

Figure 10. Average cavitation.

After of first stage; it begins to wear out the material of the impeller, failing to perform a proper maintenance will proceed to the second stage detailed in Figure 10, known as average erosion, where the wear begins to propagate towards the tajamar, forming pits in its path.

Figure 11 indicates the scenario where the wear is increasing through the blade and causing an advanced erosion or known as total damage.

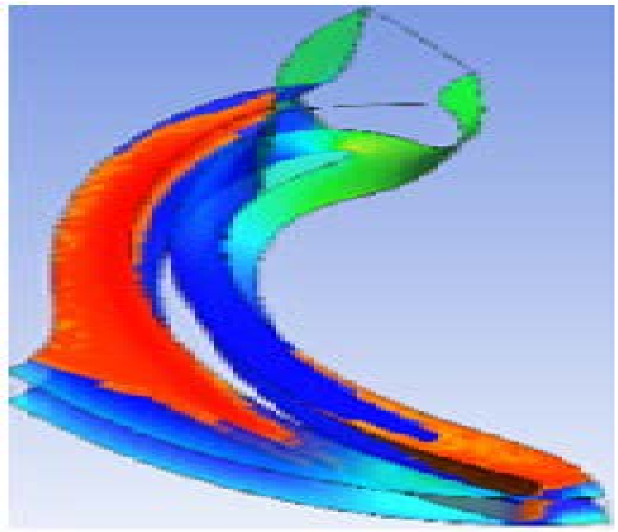

Figure 11. Advanced cavitation.

4. Conclusion

The energy performance parameters are affected by the presence of cavitation by obtaining a decrease in the discharge height or TDH from 6.4 mca to 2.9 mca, decrease in pumping capacity from 77.9 GPM to 46.6 GPM, all these reflected in its efficiency since it is the ratio between the useful power that depends on the height of discharge and flow and the power of activation that the energy entering the pump that product of the cavitation causes a behavior of Vibration in the shaft-rotor assembly. For this reason, there is a decrease in the efficiency of 34.46% in normal operating mode and in cavitation a value of 13.96%.

In order to analyze the phenomenon of cavitation in the eye of the pumping system impeller, the ball valve V1 was closed an angle of 45°, located in the suction pipe to create an empty pressure exceeding the vapor pressure of the pumped fluid. Experimentally recording the pressure reduction at discharge from 42.75 KPa to 19.31 KPa.

The cavitation model of Ansys 17.2, the steam module is created, with a temperature of the fluid present in the experiment of 18° C and discharge pressure of 19.31 KPa; after obtaining the convergence of the equations, it was possible to observe the pressure distribution of 2064 Pa in the blades that indicate the formation and subsequent implosion of vapor bubbles, besides zones of red color in the eye of the impeller that are known as light erosion, Medium and advanced, causing a wear on the surface of the same.

The proposed methodology of linear application analysis has been shown to be adequate to describe the behavior of the heart of a pumping system before and after a process that allowed us to observe the great weakness of the process under these conditions, showing conclusively the affectation of the performance of a centrifugal pump.

5. References

1. Raghavendra SM. CFD simulation of centrifugal pump impeller using ANSYS-CFX. International Journal of Innovative Research in Science, Engineering and Technology. 2014; 3(8):1–9.
2. Ushashri P. Computational analysis on performance of a centrifugal pump impeller. Proceedings of the 37th National and 4th International Conference on Fluid Mechanics and Fluid Power; Chennai, India. 2010.
3. Purushothaman K. Analysis of a centrifugal pump impeller using ANSYS- CFX. International Journal of Engineering Research and Technology. 2012; 1(3):1–7.
4. Shah S. CFD based flow analysis of centrifugal pump. Proceedings of the 37th National and 4th International Conference on Fluid Mechanics and Fluid Power; Chennai. 2010.
5. Bacharoudis EC. Parametric study of a centrifugal pump impeller by varying the outlet blade angle. The Open Mechanical Engineering Journal. 2008; 2:75–83. Crossref
6. Nova FV. Cavitation modelling of a centrifugal pump impeller. 22nd International Congress of Mechanical Engineering; Sao Paulo, Brazil. 2013. p. 1–12.
7. González PJ. Modelización numérica del flujo no estacionario en bombas centrífugas: efectos dinámicos de la interacción entre rodete y voluta. Universidad de Oviedo. Departamento de Energía; 2000.
8. Yuka IGA. Numerical study of sheet cavitation break-off phenomenon on a cascade hydrofoil; 2001. p. 1–8.
9. Klever A. Desarrollar un modelo CFD para el análisis del comportamiento del fluido en tuberías del banco de pérdidas de turbomaquinaria hidráulica y laboratorio. Escuela Superior Politécnica de Chimborazo; 2016. p. 1–45.
10. Luis DF, Yamal MI, Rios ALG, Eduardo L. Construcción y pueta en marcha de un banco de cavitación para bombas centrífugas peque-as. 2006; 1(30):1–6.
11. Munson BR, Donald FY, Okiishi TH. Fundamentos de Mecánica de Fluídos. Mexico: Limusa-Wiley; 1999. p. 1–43.
12. Dear JP, Field JE. A study of the collapse of arrays of cavities. Journal of Fluid Mechanics. 1988; 190: 409–25. Crossref
13. Egusquizaa E, Farhat M, Avellan F, Coussirat M, Escaler X. Detection of cavitation in hidraulic turbines. Mechanical Systems and Signal Processing. 2006; 20:983–1007. Crossref
14. Arnaldo DDT, Gruber ACV, Nehomar MN. Análisis numérico del campo de flujo a través del dominio fluidizado impulsor-voluta en una bomba centrífuga. Revista Ingenierıa UC. 2013; 20(2):1–13.
15. Vargas BS. Estudio del fenómeno de cavitación en una bomba centrífuga de 1 HP y su influencia en los parámetros de rendimiento energético en la facultad de ingeniería civil y mecánica. Universidad Técnica de Ambato. Facultad de Ingeniería Civil y Mecánica. Carrera de Ingeniería Mecánica; 2014. p. 67–68.
16. ANSYS. Introduction to Multi Zone Meshing. ANSYS Meshing Application Introduction; 2009.