24th COBEM - 2017

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2680
EFFECTS OF THE VOLUTE CASING DESIGN ON THE CHARACTERISTICS OF CENTRIFUGAL PUMP

Ruben Alexis Miranda Carrilo
Carlos Eymel Campos Rodrigues
Universidade Federal de Itajuba - *Campus* Itabira
ruben.miranda@unifei.edu.br, carloscampos@unifei.edu

Leonardo Rocha Queiroz
Matheus Silva Carneiro Fonseca
Bruna Rahd Mariano
leonardorochaqueiroz@hotmail.com, matheuscfonseca@unifei.edu.br, brunarahd@unifei.edu.br

Abstract. *A hydraulic pump is a machine that supplies or absorbs mechanical energy and transfers hydraulic energy to the fluid. Within the various components that comprise a pump, the rotor and the volute are responsible for playing an important role in the transfer of mechanical energy into pressure energy into the fluid. The present work deals with the design of the volute of a hydraulic turbo machine approximately radial generating of the centrifugal type, using water as working fluid. The volute is a component generally located around the rotor in order to decelerate the fluid by transforming part of the kinetic energy into pressure energy. The process of using the volute is to collect the fluid leaving the rotor and direct it to the turbo machinery outlet with the greatest possible uniformity and efficiency. The design of the volute structure is characterized by a gradual increase as it is directed towards exit, this increase being responsible for reducing the losses by shock and turbulence. For the dimension of the volute, certain geometrical considerations were made: the angular velocity of the rotor is constant; there is no internal partition in the volute; between rotor and volute there is no diffuser system; and the volute has a logarithmic spiral shape, excluding the tongue and the outlet nozzle. As for the flow: the analysis of the flow in the volute was made in the transverse plane, and the flow in the volute was considered permanent.*

Keywords: *Logarithmic Volute, Centrifugal Pump, Volute Design.*

1. INTRODUCTION

A hydraulic pump, according to Mataix (1986), is a machine that provides or absorbs mechanical energy and transfer it to the fluid. This component is also called hydraulic machine of operation and has a widely range of uses in industries. Among the several components that construct a pump, the rotor and the volute are the responsible for present a important role in transfer of mechanical energy into pressure energy for the fluid. Thus, they are very important for the mechanical engineering field. Both components are shown in Fig. 1 (Balarim and Targa, 2004), p.221.

The rotor is a rotating component that has blades, which prosecute the force into the liquid resulting in an acceleration of the fluid. Rotor spins at high speed in a continuous movement due to the force of the motor. That results in the suctioning of the liquid which causes a vacuum′s zone at the center and a zone of high pressure at the peripheries. Further, there are classification of rotors, which can be about its stages, whereas multiples stages is necessary for reduce the dimensions and improve the efficiency. The other classification is about if the rotor is opened or closed. Wherein, opened rotor is used for fluid that presents impurities, while the closed one is for fluids without particles. In sum, the goal of the rotor is to improve the speed of the fluid. The volute is usually a component that is located around the rotor, its function is decelerate the fluid by transforming part of the kinetic energy into the pressure energy. Thus, the role of the volute is to transport the fluid out of the rotor through a designed path until the turbo-machine outlet with greater efficiency and uniformity (Quintero, 2013). The design of the volute structure is, mainly, characterized by a gradual increase in diameter as it head for the exit. Due to this enlargement, it results in reduces the loss caused by turbulence shocks, as Guimarães (1991) said. In this paper, it will be discuss how to design and scale out a rotor and a volute of a centrifugal-type hydraulic turbine generator. Moreover, the working fluid will be water, which will be considered incompressible. In addition, because it is an approximately radial pump, the width of its blades will not be constant, it will be equally spaced with the exit edge parallel to the rotor axis ignoring possible manufacturing defects. The turbo-machine will be considered as: diffuser, rotor, and volute (Quintero, 2013), p.3.

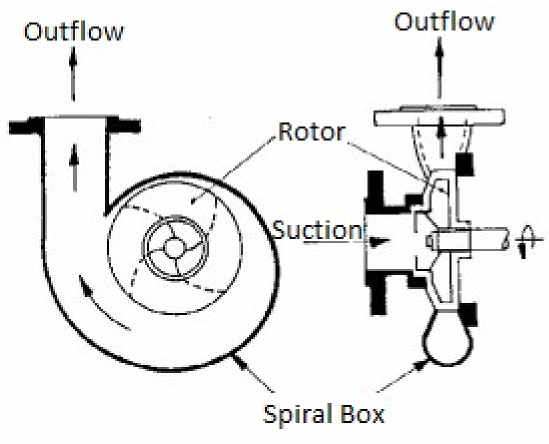


Figure 1. Components of a Hydraulic Pomp

2. PURPOSE

This work has as main purpose to design a rotor and a volute of high efficiency, based on the equation that the literature has. In addition, as a secondary objective, it is expected to develop the design of the projected elements as well as its prototype, which will be done via 3D printing.

3. THEORETICAL BACKGROUND

3.1 Classification of Turbo-machinery

Turbo-machines are equipment governed by Euler's law, which are closely related to the flow and dynamics of fluids. There are two categories of turbo-engines: Thermal Turbo-machines (those that work with fluid with variable specific mass) and Hydraulic Turbo-machines (those that work with constant specific mass fluid). In order to make it possible to better understand hydraulic machines, we distinguish two classes of these machines (Quintero, 2013), p.29:

- Motor Turbo-machinery: those who have functioning caused by the energy supply of the fluid flowing through them, that is, the fluid carries the energy required to drive the machine motion.
- Generating Turbo-machinery: are those that provide the fluid energy to its flow, the rotor movement to the fluid provides the kinetic energy required for it to flow through the pipe network pipelines.

3.2 Volute

According to Quintero (2013) "Generating radial turbo-machinery (pumps and fans) generally have a volute around the outer periphery of the rotor" p.28, which aim to receive the fluid from the rotor outlet (in the design in question, there was no use of diffuser) and thus, to conduct this fluid to the turbo-machinery outlet, uniformly. Non-uniformity may appear when the nominal flow is changed, resulting in non-uniformity of flow, radial forces on the machine axis and pressure oscillations. The volute is in the shape of a spiral or a curved funnel which increases its area as nearer to the discharge area as illustrated in Fig. 2 (Medeiros, 2009), thereby reducing the velocity of the fluid and increasing its pressure.

After the beginning of its spiral the volute grows uniformly so that a balance of the radial forces acting on the rotor along its periphery occurs. However, even due to such configuration of this component, there is an uneven distribution of the pressure therein, when the pump is operating outside its nominal point giving rise to a reaction perpendicular to the axis. Such a reaction, called radial thrust, can be avoided by using the double volute, which consists of dividing the flow into two ducts, in order to balance the reactions caused by the pressure difference along the same conduit, as shown in Fig. 3 (Medeiros, 2009).

Another benefit of using the volute is that in pumps with this system the rotor can be reduced by up to 20% of the maximum value without loss of efficiency (Brasil, 2010).

3.3 Calculating the Volute Dimensions

For the design of the volute the following calculations procedure was used. In addition, in the Figure. 4 (Quintero, 2013) is shown how the variables are distributed:

Considering the radius of rotor being equal to 111 mm, it is necessary a suitable spacing between rotor and volute,

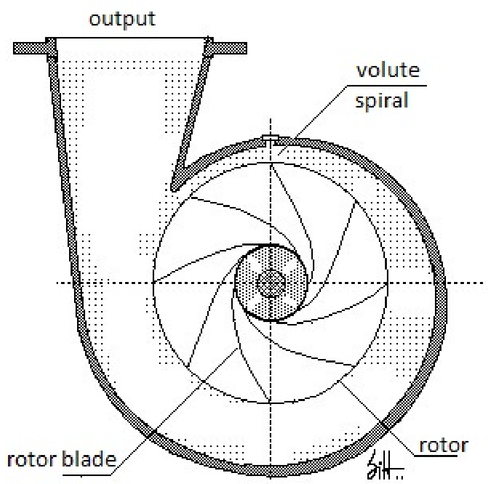


Figure 2. Spiral Volute

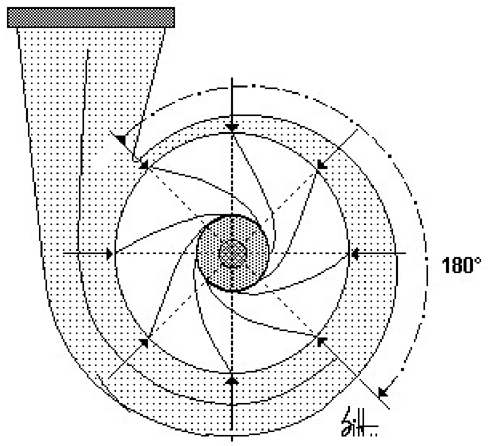


Figure 3. Double Volute

so the volute does not interfere in movement of the rotor and the fluid trajectory is secured through the volute and consequently decreases the circulation effect. This separation can be observed in Fig. 5. Therefore, the base diameter of volute respects the Eq. (1):

$$r_{ev} = 1.05 \times r_5 \tag{1}$$

However, the above equation couldn't be used on calculus, due to the geometry of the projected part be larger than

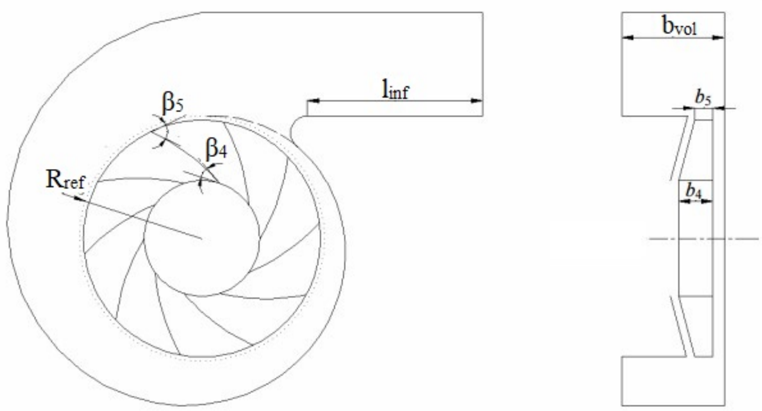


Figure 4. Cross-Section of a Volute

original volute. To provide the assembly between the designed equipment and engine was adopted a radius of 61 mm.

$$r_{ev} = 61mm \tag{2}$$

The blocking factor $B_f v$ represents the difference between the real and ideal dimensions, directly influencing mass flow. For the calculation, the maximum blocking factor was chosen, due to manufacturing tolerance is 0.1.

$$B_{fv} = 0.1 \tag{3}$$

According to Carrillo (2010) the tongue thickness (t_{lv}) equal to 5 mm is ideal to the size of the pump studied and is advisable to begin at Ψ_{sv} $5\,^{\circ}$ starting on the base circle. The radius of this circle is $r_l v$. This location is perfect because of the mean streamline. Because the large difference between the rotor and the ideal tongue, another location e thickness for the tongue was adopted. This part is set at Ψ_{sv} $0\,^{\circ}$ with thickness of 2.56mm. Further, it can be seen at Figure 6 (Carrillo, 2010).

Furthermore, the re-circulation mass should be considered. As can be observed on Fig. 6 there are different zones of velocity in the volute, this fact leads to fluid re-circulation. Carrillo (2010) indicates that an increase of 5% of mass flow has to be considered, offsetting the mass re-circulation. As shown in Eq. (4) and Eq. (5).

$$m_r = \%ofre - circulation \times m \tag{4}$$

$$m_r = 1.1080 \times 0.05 = 0.0555\frac{kg}{s} \tag{5}$$

For calculations, the flow mass considered is the flow mass desirable plus re-circulation mass (5% of desirable mass), represented in Eq. (6) and Eq. (7). In result of that Fig. 7 was obtained by equation Carrillo (2010).

$$m_v = m_r + m \tag{6}$$

$$m_v = 0.0554 + 1.1080 = 1.1634\frac{kg}{s} \tag{7}$$

3.4 Circular Section

The Eq. (8) and Eq. (9) result in volute entry area value A_t and b_e represents thickness the of rotor exit.

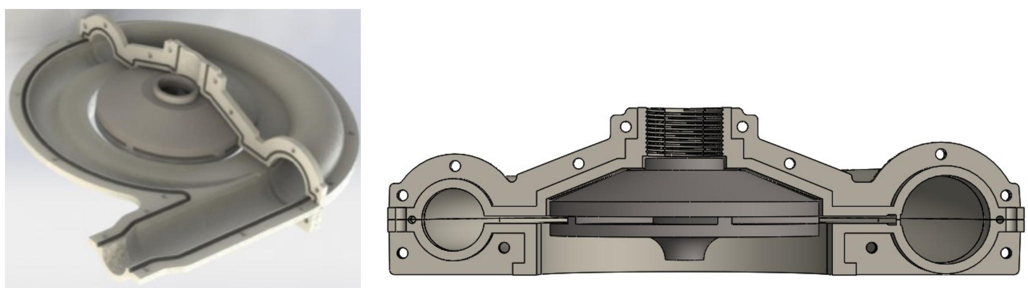
$$A_t = 2 \times \pi \times r_{ev} \times b_{ev} \times (1 - B_{fv}) \tag{8}$$

$$A_t = 0.0010m^2 = 1052.6744mm^2 \tag{9}$$

Thereby, meridian velocity in the entry of volute and radius of the exit r_t are obtained, respectively by Eq. (10) and Eq. (11).

$$C_{msv} = C_\theta = \frac{m_v}{\rho_{water} \times A_t} = 1.1072\frac{m}{s} \tag{10}$$

$$r_t = \sqrt{\frac{A_t}{\pi}} = 18.3051mm \tag{11}$$



(a) Volute view with rotor and upper right part withdrawn

(b) Volute section cut with mounted rotor

Figure 5. Volute with mounted rotor

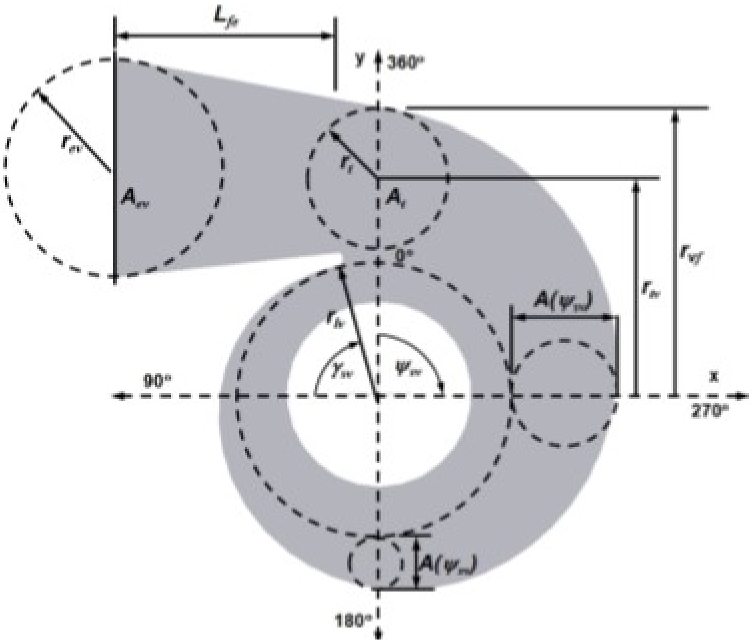


Figure 6. Representation of the Elements of Construction of the Volute

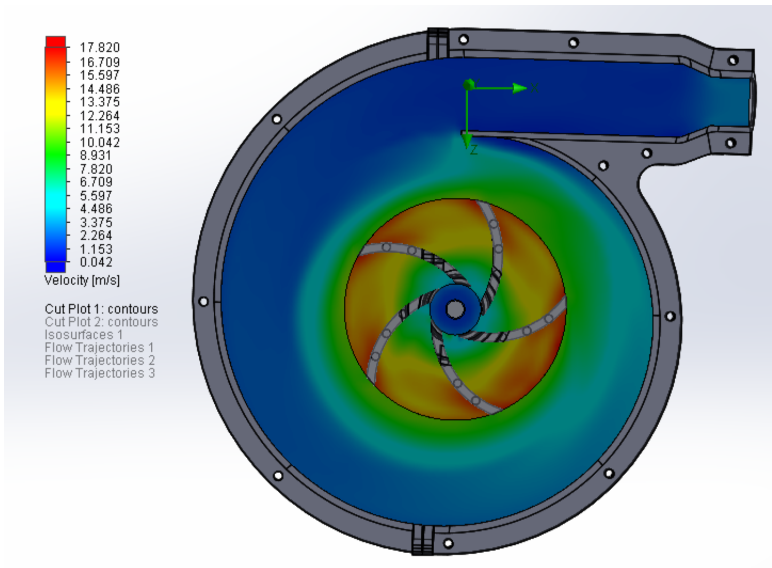


Figure 7. Simulation of flow on designed volute

The base circle of tongue is calculated through the Eq. (12). A factor Δr_{lv} of 1.3856 was considered in the calculations.

$$r_{lv} = \frac{\Delta r_{lv}}{1000} \times r_{ev} = 0.08452m = 84.52160mm \tag{12}$$

Thus, the distance between the centre of volute and the begin of the exit nozzle r_{tv} can be calculated (Eq. (13)):

$$r_{tv} = r_{lv} + \frac{t_{lv}}{1000} + r_t = 0.10783m = 107.83mm \tag{13}$$

3.5 Circular Section Considerations

The first step on mean streamline determination is define the begin of it, making possible calculate the increase angle $beta_{vLCM}$ (Eq. (16)).

$$r_{lv} = r_{vLCM} = 84.52160mm \tag{14}$$

$$\beta_{vLCM,rad} = tan^{-1} \times (\frac{ln\frac{r_{tv}}{r_{lv}}}{2\pi}) = 0.03874rad \tag{15}$$

$$\beta_{vLCM,\circ C} = \beta_{vLCM,rad} \times \frac{180}{\pi} = 2.21951^{\circ} \tag{16}$$

The design of LCM is obtained through Eq. (21), variation Ψ_{sv}. As Ψ_{sv} increases the r_{vlcm} increases exponentially, resulting in the characteristic format. Converting the polar coordinates into rectangular Eq.(16), the mean streamline behaviour is easily observed in Fig. 8.

$$tan\beta_{vLCM,rad} = tan\beta_{vLCM} \times \Psi_{sv,rad} \tag{17}$$

$$r_v(\Psi_{sv}) = r_{lv} \times e^{\beta_{vLCM,rad} \times \Psi_{sv,rad}} \tag{18}$$

$$X_{LCM} = r_v \times sin\Psi_{sv,rad} \tag{19}$$

$$Y_{LCM} = r_v \times cos\Psi_{sv,rad} \tag{20}$$

(21)

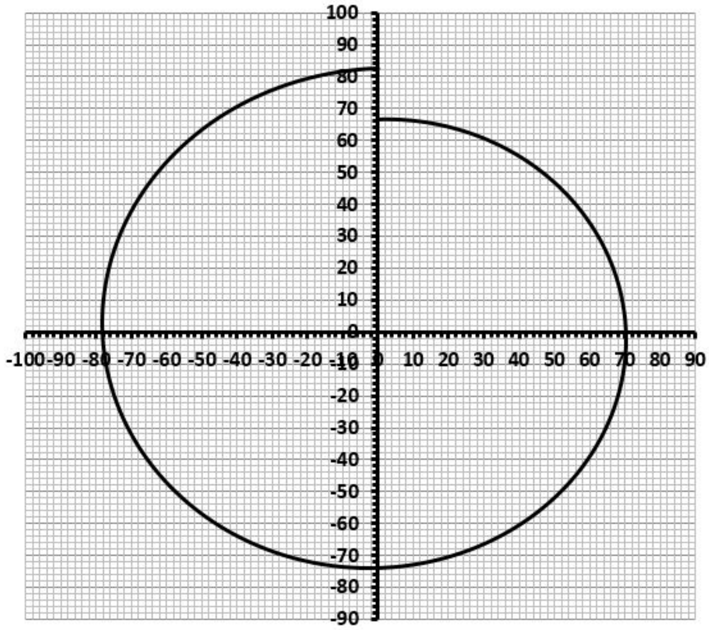


Figure 8. Log Spiral Volute of BC-LCM

The distance between the LCM and base circle o volute represents the radius of the section r_v (Ψ_{sv}), as represented in Fig. 6 knowing the values of the radius is possible calculate the velocity and flow in each section.

$$r_{sv} = 1000 \times \sqrt{\frac{A_{sv}}{\pi}} \tag{22}$$

$$m_{sv}(\psi_{sv}) = m_v \times \frac{\Psi_{sv}}{360} \tag{23}$$

$$Q_{sv}(\Psi_{sv}) = \frac{m_{sm}}{\rho_{water}} \tag{24}$$

$$A_{sv}(\Psi_{sv}) = \frac{Q_{sv}}{C_{sv}} \tag{25}$$

(26)

Figure 9 demonstrates the variation of the radius according to increase of the Azimuth angle. Representing the LCM in the designed volute

Figure 10 shows the 3D model of the designed volute. As can be observed, there are four distinct parts that compose the entire volute. This is due to the fact that the entire part is larger than the maximum print size.

Figure 11 shows the exploded view of the final assembly. It is possible to identify the volute, the rotor, the volute sealing system, the set screws and a pump specimen that is normally used.

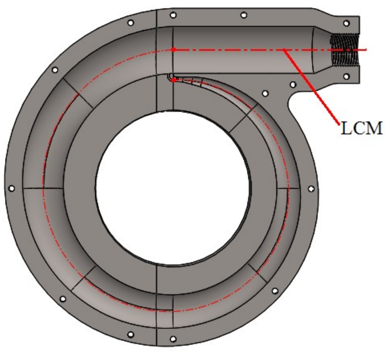
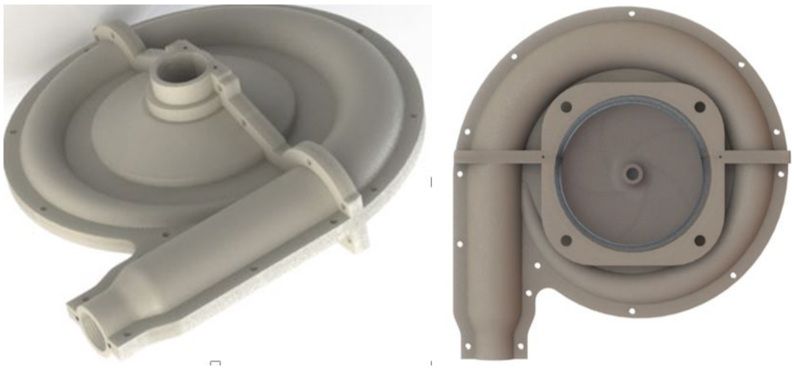


Figure 9. Top view of the volute with representation of the LCM



(a) Top view (b) Botton view

Figure 10. Mounted volute

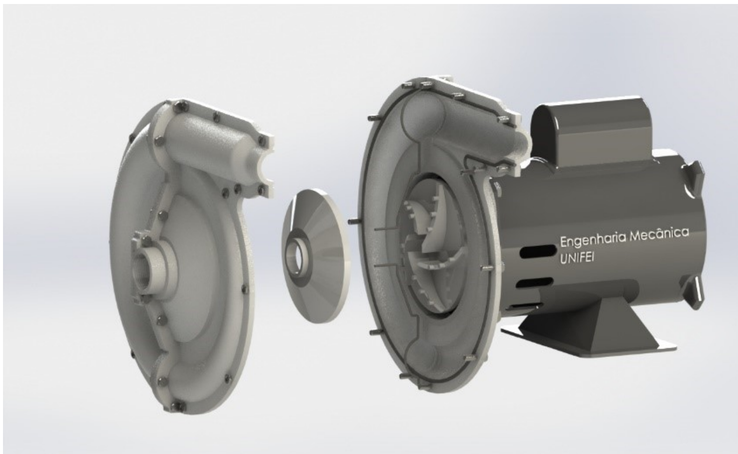


Figure 11. Exploded view of the final set

4. COMPUTATIONAL ANALYSIS OF THE ROTOR-VOLUTE ASSEMBLY

The design parameters required for the computational simulation are the same as those previously calculated for the rotor and volute. These results were used to generate the 3D geometry of the pieces in Solidworks 2016. Once the rotor and volute geometry were generated, they are transferred to the Flow Simulation tool for the construction of the computational mesh used in the simulation and inclusion of the boundary conditions in the system. Considering the low processing power used for the simulation, a coarser mesh was used. In addition, several factors were ignored to speed simulation time. In this way, it is recommended as a future proposal, a more advanced simulation with greater processing power to acquire more satisfactory data.

4.1 Boundary Conditions

Figure 12 shows the location where the boundary conditions were employed.

(a) Wall Boundary: The "Real Wall" option was considered in view of the adhesion principle and the relative velocity of the fluid in contact with the wall equal to zero.

(b) Inlet Boundary: The condition of total pressure, total temperature and flow rate was used

(c) Outlet Boundary: The ambient pressure condition was used.

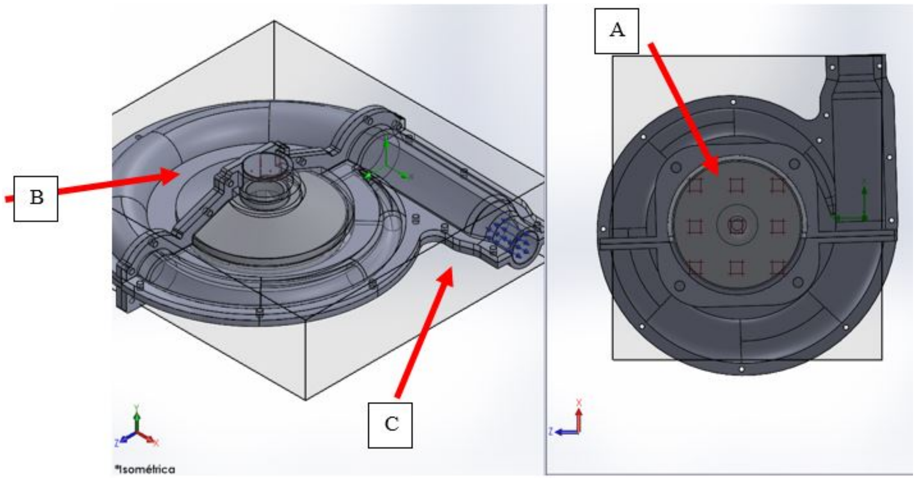


Figure 12. Boundary Conditions

For the case analyzed, a hexadecimal mesh was created throughout the domain, which is the only mesh used by SOLIDWORKS 2016.Figure 12 shows the mesh used in the computational simulation.

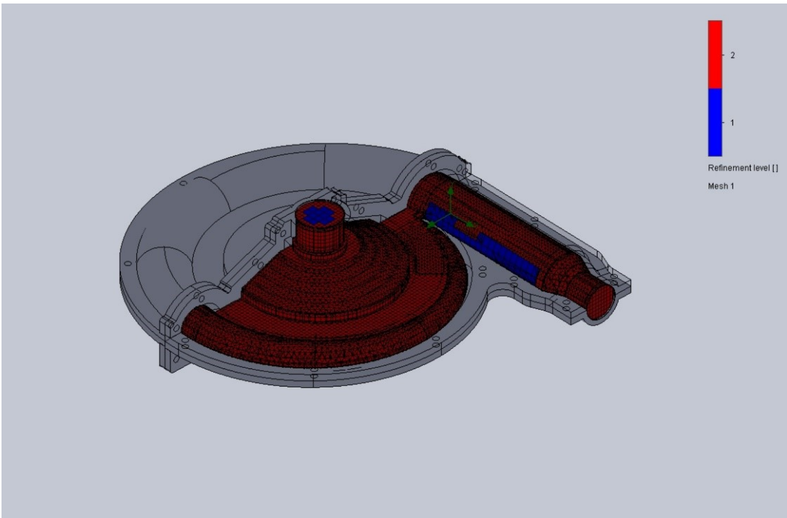


Figure 13. Created Mesh

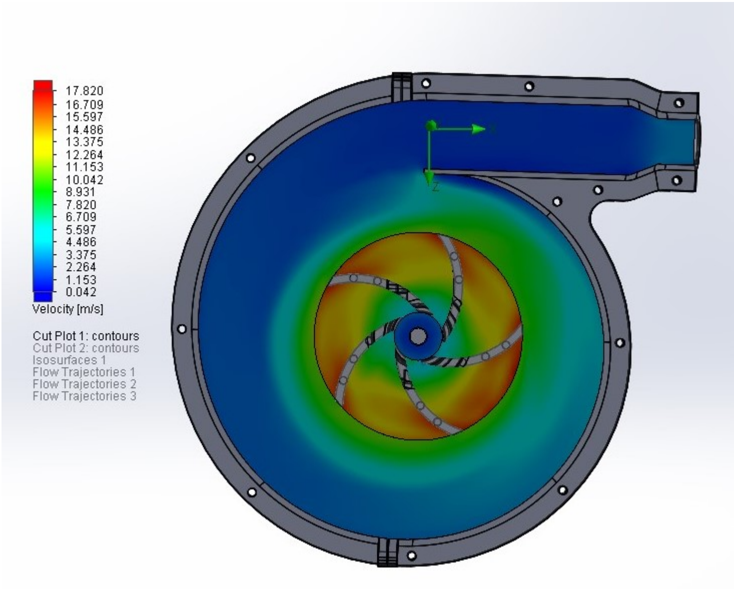
Table 1 describes the number of nodes and elements used in the numerical simulation for each component of the set.

Table 1. Simulate Parameters

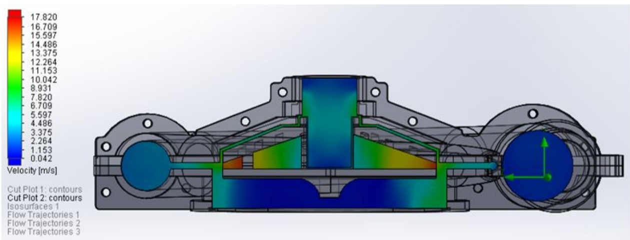
Version	Flow Simulation
File type	FLD
Interaction	334
Physic time	0s
CPU time	1209s
Cells	104645
Fluid Cell	51196
Solid Cell	53449
Partial Cell	29513

4.2 Speed Analysis

Figure 14 shows the velocity distribution in the rotor and volute. It can be seen that the speed of the fluid in the rotor increases along the radius and shows a slight variation in the output. This variation becomes greater when the flow begins to traverse the volute until it leaves at point C with a much smaller velocity.



(a) Top view



(b) Southern view

Figure 14. Speed Distribution

5. CONCLUSION

We realize that several considerations must be made in order to determine the initial parameters of a design of a rotor and volute for centrifugal pumps. The considerations that the literature provides the directions of the calculation to be initiated, yet the experience of the designers is the one that allows the correct interpretation of the results, as well as exerts influence in the decision to continue along a path or to abandon it and to continue in other. Many challenges are encountered during manipulation of the variables involved in the calculation base, the relations that one exercises over the other, the iterative process that treats the error conditioned to the calculation process, in fact, many points that end up interrupting for some time the fluency of the project, but that in a way, allow the enrichment of the work as well as the increase of knowledge added to those that are part of this constructive process. The equation of exponential rise were used to determine the volute geometry. Through a simulation realized with the module FlowSimulation of Solidworks, the behaviour of the fluid was analyzed.

(a) To good distribution of mass (main function of the volute), the velocity trough all the sections has to be the same. As demonstrated, the Mach velocity along of all sections of volute remains constant.

(b) Generally, the recirculation mass zone is located almost at exit of volute. Meanwhile, because of not ideally location of tongue, this zone has moved to the spiral's begin.

(c) The exponential rise demonstrated to be an efficient method to design a volute. However, considerations has to be carefully adopted, to avoid recirculation and decrease in flow

6. REFERENCES

Balarim, C.R. and Targa, L.A., 2004. “Custo de bombas centrifugas funcioando como turbinas em microcentrais hidreletricas”. 01 Jun. 2016 <http://beluco.net/citations/v24n01a25.pdf>.

Brasil, A.N., 2010. “Bombas: Classificacao e descricao”. 30 Jun. 2016 <http://sistemas.eel.usp.br/docentes/arquivos/5817712/LOQ4015/capitulo3 bombasclassificacaoedescricao.pdf>.

Carrillo, R.A.M., 2010. *Projeto e analise de desempenho de Turbinas Radiais*. Ph.D. thesis, Universidade Federal de Itajuba, Itajuba.

Guimarães, L.B., 1991. “Máquinas hidraulicas”. 01 Jun. 2016 <http://ftp.demec.ufpr.br/disciplinas/TMEC036/Apostila20de20Maquinas20Hidraulicas20-20V19.pdf>.

Mataix, C., 1986. *Mecanica de Fluidos y maquinas hidraulicas*. Ediciones Del Castillo, Madrid, 2nd edition.

Medeiros, C.F.d., 2009. “Abastecimento de água”. *Universidade Federal de Campina Grande–UFCG. Campina Grande, PB*.

Quintero, Y.F.G., 2013. *Análise da interação Rotor-Voluta de turbomáquinas centrífugas por meio do escoamento potencial levando em consideração a variação da geometria e espaçamento das pás do rotor.* Ph.D. thesis, Universidade Federal de Itajubá, Itajubá.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.