18th Brazilian Congress of Thermal Sciences and Engineering
November 16-20, 2020 (Online)

ENC-2020-0548
NUMERICAL ANALYSIS AND ANALYTICAL PERFORMANCE OF A CENTRIFUGAL FAN THROUGH THE CONSTRUCTIVE ANGLE IMPELLER OUTLET.

Bruna Beatriz de Paiva
Reginaldo Ribeiro de Sousa
UFGD – Grande Dourados Federal University / FAEN-College of Engineering
Dourados-MS, Road MS-270/Dourados-Itahum, km 12, ZipCode: 79804-970
brubeatrizp@hotmail.com, reginaldosousa@ufgd.edu.br.

Abstract. *Several simplified mathematical models are known as design of flow machines, however, the finite element and volume method has helped to design these products, as well as to visualize the flow of the fluid and understand the losses generated by this movement. This work has as main objective to evaluate the performance of a centrifugal fan through the design parameters influenced by the constructive angle of the impeller outlet. The study takes two approaches, the analytical model developed by the Euler Equation and the finite volume method using the ANSYS Fluent Academic software. The results obtained are essential characteristics, mainly to determine the application of a flow machine: hydraulic power, head, flow and the yield curve. These parameters are adapted to three variations of the impeller's constructive exit angle, 20 °, 30 ° and 40 ° submitted to 10 variations of rotation. The results are exhibit by means of comparative graphs, which the demonstrate a behavior similar to the analytical model. It is possible to validate an interference of the constructive angle of exit, view the behavior of the fluid and find an ideal design point to the proposed model.*

Keywords: *flow machines, numerical analysis, analytical analysis, constructive angle.*

1. INTRODUCTION

The flow machines are equipment that receive this name because the fluid is not at any time confined within the machine's housing, but rather in a continuous flow through it, being subject to energy variations due to the dynamic effects of the fluid current (GERMER, 2013). They were developed by a set of in-depth concepts of thermodynamics and aerodynamics that seek to take advantage of the energy conversion contained in fluids, yielding or absorbing, for some engine or generating application respectively. Such purpose is known in equipment such as turbines, pumps and fans that currently serve from industrial to residential sectors.

All of these machines are made up of two fundamental building elements: the impeller and the volute casing (HENN, 2006). The theoretical study of the flow through the impeller is based on velocity vectors, which can be represented by a triangle at any point of the flow in the impeller (GERMER, 2013). Even though the real flows existing are very complex (three-dimensional and transient), the essential phenomena can be analyzed how a simple flow model and with the speed triangles (BRASIL, 2010). However, a computer numerical simulation applied the fluid machines has enabled not only the visualization of complex flows, but also allowed to design new types of machines, predict their behavior through tests and arrive at the construction of more efficient prototypes, with less time and effort (HENN, 2006). This is also possible due to similarity and dimensionless dimensions, that is, a model that implies being indispensable for a complete validation. The models, enlarged or reduced, must be geometric, kinematic and dynamically similar to the projected machines (Santos, 2017).

Several studies involving flow machines have verified how the amount of blades or shape of the volute influences the efficiency of such equipment, finite element method and finite volume. However, few studies have addressed analyzes on the triangles of velocity, which motivated the study. Therefore, the objective of this work is to evaluate the influent the constructive angle of exit in the performance of a centrifugal fan impeller (radial) by the method of finite volumes through the CFD tool, ANSYS Fluent, student license, comparing the results obtained with the analytical model developed by the Euler equation.

2. PROCESS, NUMERICAL ANALYSIS AND ANALYTICAL ANALYSIS

The flow machine chosen for this work is a radial fan model with an impeller of five blades curved backwards. The details of this impeller are presented in item 2.1. As already mentioned, this work makes the comparison of results between the numerical model and the analytical model. Details on such models are described in items 2.2 and 2.3 respectively.

2.1 Description of the problem

The problem analyzed in this work is composed of a fan with a radial impeller. This type of impeller allows several configurations in relation to the direction of its blades, therefore, the model is composed of 5 blades curved backwards with 5mm of thickness, the constructive angle of entry (β4) is 15° and the constructive angle of exit (β5) of the impeller varies from 20°, 30° and 40°. The dimensions of the impeller are 200 mm outside diameter and 110 mm inside diameter.

To design the blades, a triangle at the entrance and another at the exit of the impeller were built. Fig. 1 exemplifies the triangle designated by terms c4, u4 and w4 for entry, the internal diameter of the impeller and an illustration of the velocity vectors that form the velocity triangle.

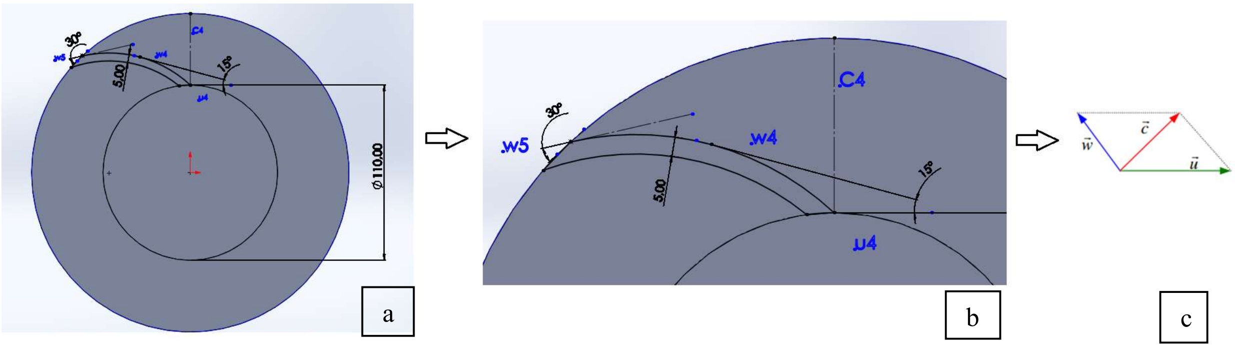

Figure 1. Impeller model (a) Constructive triangle for a blade (b) Illustration of the constructive triangle (c)

As for the geometry of the volute, the Archimedes principle was followed, which is based on a centralized internal square with 10% of the external diameter of the impeller for the construction of the spiral box (volute). This type of dimensioning is commonly applied in industrial sectors. The adopted speed of rotation varies 10 times in a range from 1600 rpm to 3400 rpm.

2.2 Analytical analysis

The theoretical study of the flow through the impeller is based on velocity vectors, which can be represented by a triangle at any point of the flow in the impeller. The Euler equation is the basic equation for the development and study of flow machines in general, in the case of fans some considerations are adopted in order to suit the application and the fluid. This fundamental equation uses the vectorial treatment applied to the mass conservation equation in the integrated and according to Henn (2006) it is describing as:

$$M_{pá\infty} = \int r\, c_u \left(\rho \vec{c}\, .\, d\vec{A}\right) + \frac{d}{dt} \int_v r\, c_u\, \rho\, dv \tag{1}$$

Where:
$M_{pá\infty}$ = Moment of the amount of movement of the system in relation to the fixed reference point;
r = Radial distance from the axis to the particle under consideration;
c_u = Tangential component of the fluid particle velocity;
$d\vec{A}$ = Representative vector of an area element considered;
ρ = Specific gravity of the fluid;
dv = Elementary fluid volume.

As adopted premises are permanent regime, uniform velocity in the sections and unidimensional flow. Based on the velocity triangle and the continuity equations the integrals that represent the flow rate, reduces the Eq. 1 by:

$$M_{pá\infty} = \rho\, Q_r (r_5 C_{u5} - r_4 C_{u4}) \tag{2}$$

Where:
ω = Angular rotation speed of the impeller;
Q_r = Flow rate that passes through the impeller;
r_5 = Impeller outlet radius;
r_4 = Impeller inlet radius;
C_{u5} = Tangential component of absolute outlet speed;
C_{u4} = Tangential component of absolute inlet speed.

By rearranging the equations, it is possible to obtain characteristic equipment data such as hydraulic power and lifting height that defines the operational capacity of the equipment. Then, Eq. 2 is rewritten in order to represent the power supplied by the impeller, Eq. 3:

$$P_{pá\infty} = \omega M_{pá\infty} = \rho Q_r \omega (r_5 C_{u5} - r_4 C_{u4}) \tag{3}$$

Based on the geometric parameters, the velocity triangle and the construction angles, considering the entry without turbulence (α4 = 90º), we have the theoretical lifting height Eq. 4 for infinite number of blades:

$$H_{t\infty} = \frac{{u_5}^2}{g} - \frac{u_5 \, cotg\,(\beta_5)}{g\pi D_5 b_5} Q_r \tag{4}$$

Where:
$H_{t\infty}$ = Theoretical lifting height for an infinite number of blades;
u_5 = Tangential velocity;
g = Gravity acceleration;
D_5 = External diameter of the impeller;
b_5 = Channel width;
β_5 = Constructive angle of the impeller outlet.

Finally, to evaluate the hydraulic loss, the efficiency equation, Eq. 5 was adopted:

$$\eta_h = \left(\frac{H}{H_t}\right)^1 \tag{5}$$

Onde:
η_h = Hydraulic efficiency;
H = Lifting height;
H_t= Theoretical height.

2.3 Numerical analysis

The fan was modeled using the *SolidWorks 2018* software and sized as already mentioned in item 2.1. Solve the problem, CAD was imported to *ANSYS Academic 2020 R1* in order to use the Fluent environment. With the workbench platform established, the DesignModeler was used to define two regions, the volute and the impeller, as shown in Fig. 2.

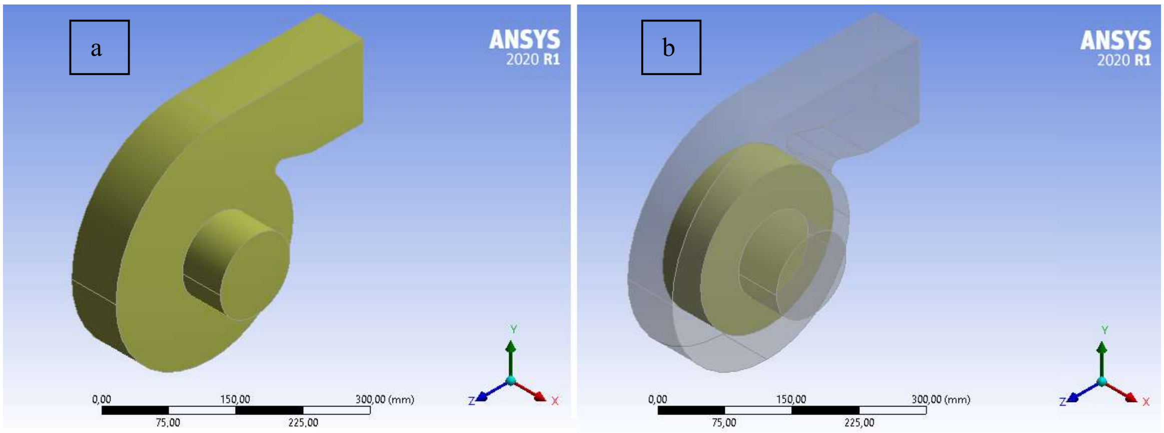

Figure 2. Domain the of volute (a) and impeller (b)

The mesh that structures the model was created separately for each domain in order to refine them according to their dimensions, since the mesh generated with the default settings was very coarse. The refinement was carried out using the sizing method function and the mesh is composed of hexahedral, tetrahedral and prismatic elements as shown in Fig. 3.

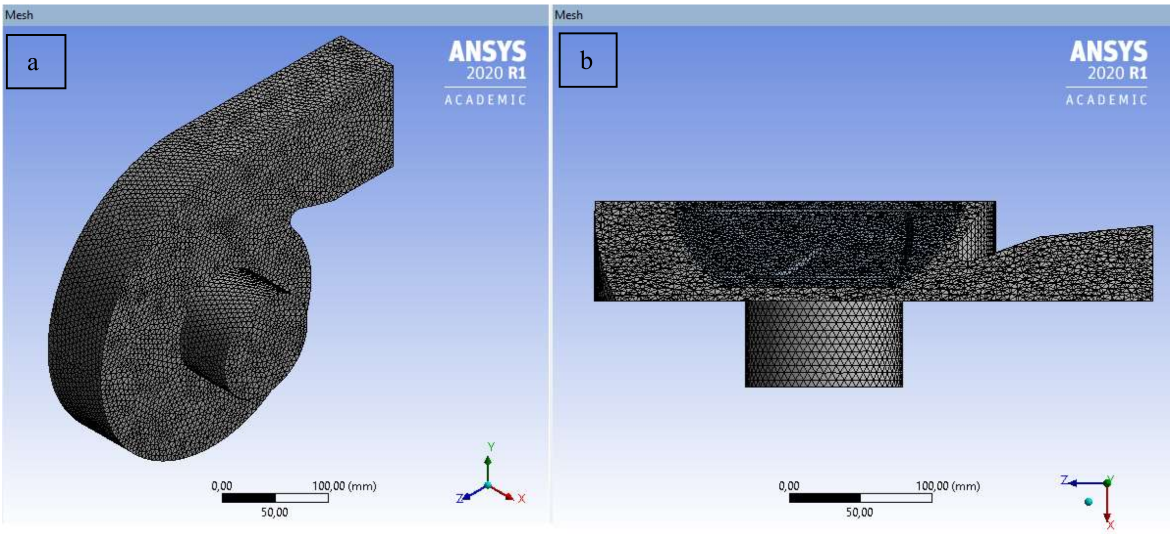

Figure 3. Isometric view (a) and top view with sectioned plan (b) of the total domain mesh.

It could have been more refined, however there is a limitation of elements by the student version. Fig. 4 presents this study comparing the elevation height of the numerical analysis of the ventilator according to the mesh refinement calculated based on Eq. 4 and the rotation of the impeller.

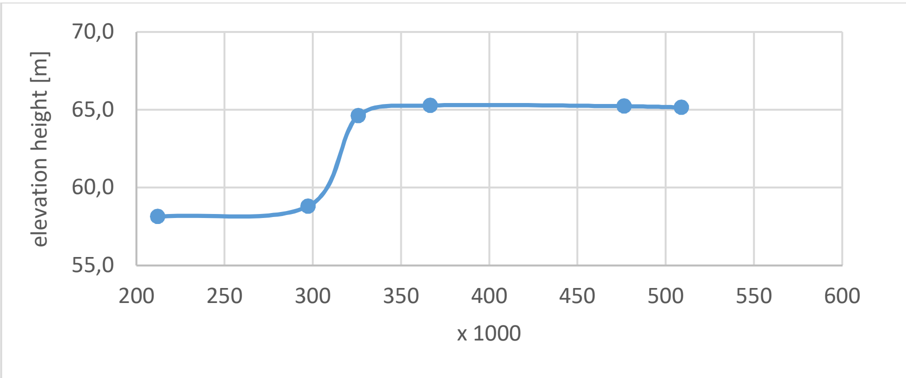

Figure 4. Mesh independence curve.

After mesh generation, the physical properties of the model were defined. As well as the premises adopted for the fundamental equation of flow machines or Euler equation, the flow occurs in a steady state, the working fluid is ideal and with mass conservation, however a finite number of blades is considered, with hydraulic and volumetric losses existing. The working fluid is air because it has low viscosity and is incompressible at speeds of a maximum of 100 [m/s] (GERMER, 2013) and with a temperature of 25 °C. The boundary conditions adopted have the gauge pressure as a reference, that is, 0 Pa both at the entrance and at the exit of the volute, where the pressure difference of the system guides the air flow. Fig. 5 shows the suction nozzle (a) and discharge of the volute (b). The angular velocity ranges from 1600 rpm to 3400 rpm.

The turbulence model adopted for resolution is the k-e model, the best known among the models offered that involves two differential transport equations, as it is robust, precise and has stability (GABBI, 2013). The solution method applied to the pressure-speed coupling corresponds to SIMPLE, the default relaxation factors and zone reference being the volute. The initialization was standard and with an absolute frame of reference.

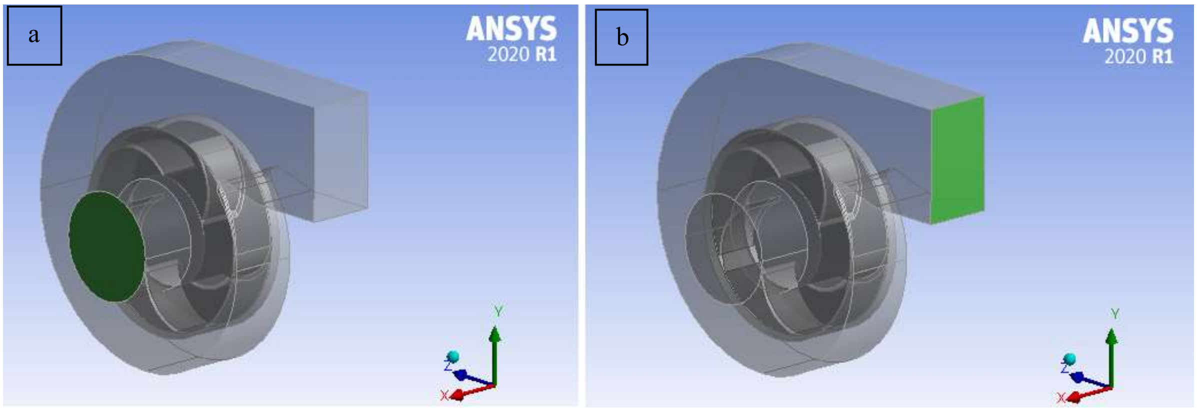

Figure 5. Air flow from the radial fan to the suction nozzle (a) and volute discharge nozzle (b)

Another criterion evaluated was the number of necessary iterations. We chose to check the velocity of inlet and outlet in the fan volute using the graph generated by the software itself, as shown in Fig. 6. The black line represents the velocity of inlet, while the red line represents the velocity of outlet in the volute. For low velocities the model showed convergence of around 100 iterations, only 200 iterations were accepted as a guarantee and as shown, with the increase in rotation, the convergence occurred more quickly, the quantity adopted being sufficient.

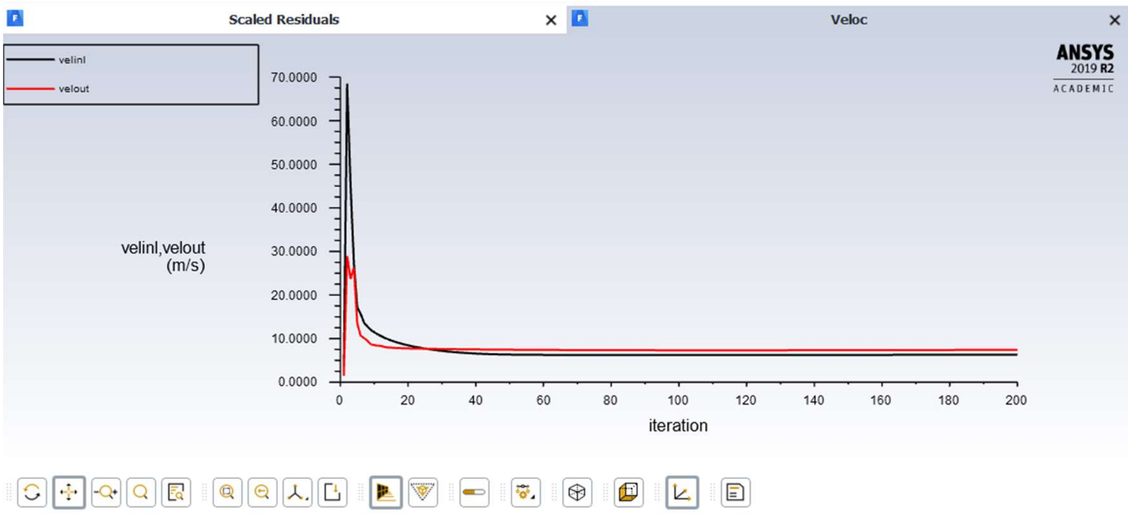

Figure 6. Inlet and outlet velocity graph for β5 of 20° and 2600 rpm.

By setup configurations we chose: 3D, double precision, showing the mesh after processing and in the machine configuration, only 1 processor was used, resulting in computational time around 20 minutes. To survey the height, hydraulic power and flow parameters, the volute inlet / outlet mass flow was extracted from the software and a treatment was carried out regarding the measurement units, in parallel, the tangential velocity (u5) also provided by the simulation, which is subject only to numerical error. As a simplification for solving the model, the values found were applied to the same equations presented in the previous topic, the Euler equation.

3. RESULTS

Based on the conditions of the previous ones, the following figures are the comparative graphs between the analytical and numerical results of the model. They indicate as theoretical height and hydraulic power curves for the 3 constructive angles of output (20°, 30° and 40°) where each one presents a behavior following the desired curve. Note that the proposed centrifugal fan has low hydraulic loss and a fixed volume loss.

To evaluate the graphs, it should be noted that the compared values are chosen in two ways: the analytical model considers the theoretical parameters for an infinite number of blades, governed by valid premises involving only one impeller geometry. As for the numerical model, the data is influenced by the volute, that is, there are internal hydraulic and volumetric losses during the flow of the fluid.

As shown in Fig. 7, the increase in the impeller velocity causes the theoretical height values to be closer and the opposite happens in Fig. 11, that for lower velocities the closest results are close. This is due to a displacement of the boundary layer caused by the geometric shape of the internal contours of the blades that alter the flow direction. With a finite number of blades, the channels become wider and the fluid has more difficulty to be guided, however, it is compensated by increasing the angular velocity, since the fluid velocity is proportional to the impeller rotation velocity.

Another point evaluated is the characteristic curves of the flow machine. Fig. 13, 14 and 15 show the curves with a similar drop behavior, but with volumetric losses in the system. However, for high velocities as shown in Fig. 13, the gain in the elevation height of the numerical analysis is greater than that of the analytical analysis, which implies an over-dimensioning of the machine, that is, when applying the values selected in Eq. 5 o filter a yield equal to or greater than 100% which does not make it real. The expected result would be the high elevation values of the numerical analysis for rotations equal to or equal to values below the analytical analysis precisely because the numerical model is subject to volumetric losses in the volute.

Finally, in order to visualize all these parameters that generate hydraulic losses without a fan, Fig. 16 (a), (b), (c) and (d), respectively, show the behavior of the velocity contour vectors, the pressure in the volute region and close to the blades. In addition, it is possible to see the details of vortices in the lower region between the impeller and the volute, that is, regions of fluid recirculation that generate internal volumetric losses. Fig. 17 (a), (b) and (c) shows the behavior of the fluid for the purchased angles.

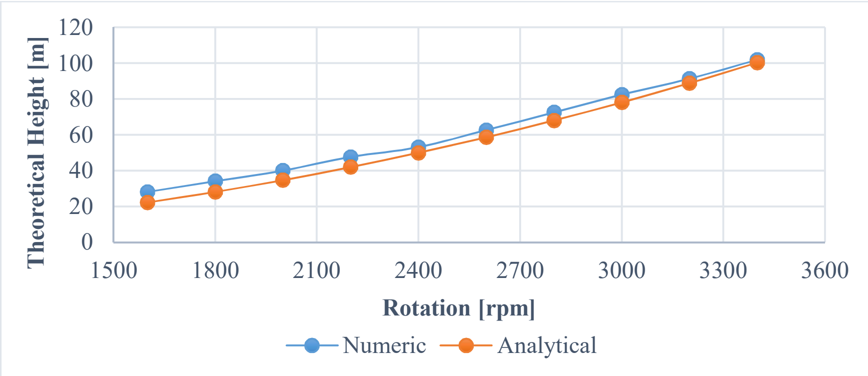

Figure 7. Theoretical height x rotation graph for output angle β5 20°.

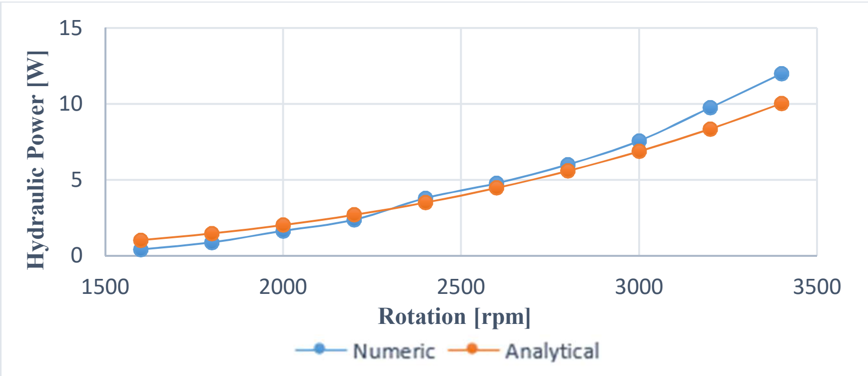

Figure 8. Hydraulic power x rotation graph for outlet angle β5 20°.

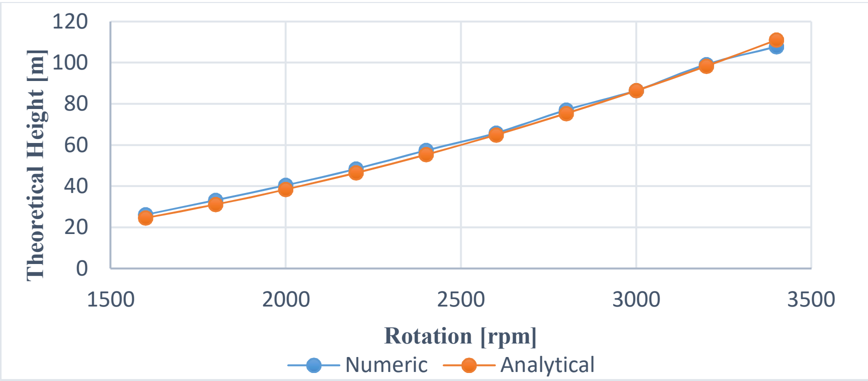

Figure 9. Theoretical height x rotation graph for output angle β5 30°.

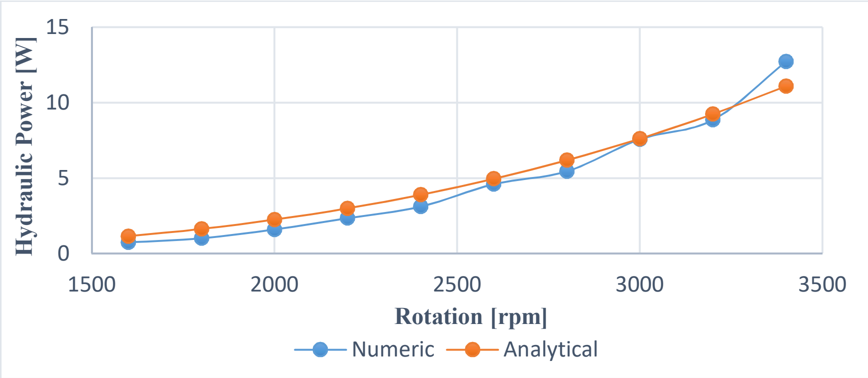

Figure 10. Hydraulic power x rotation graph for outlet angle β5 30°.

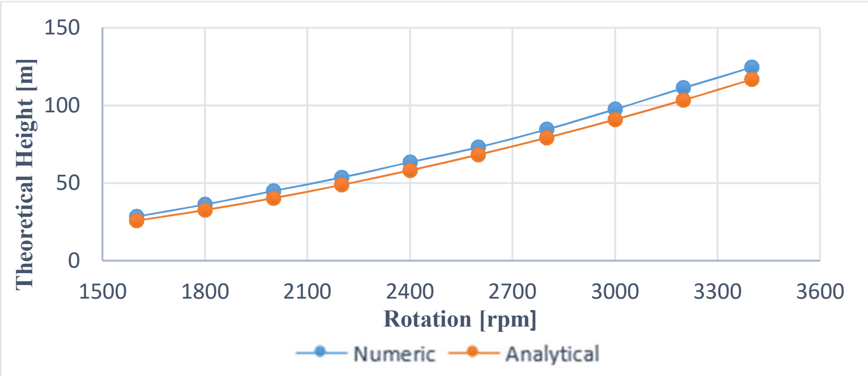

Figure 11. Theoretical height x rotation graph for output angle β5 40°.

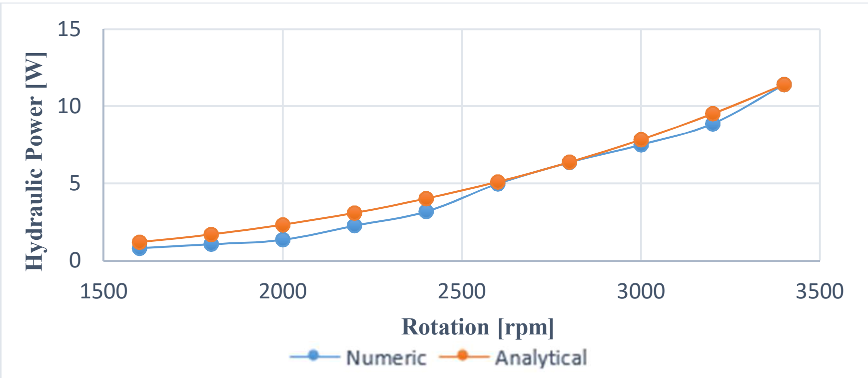

Figure 12. Hydraulic power x rotation graph for outlet angle β5 40°.

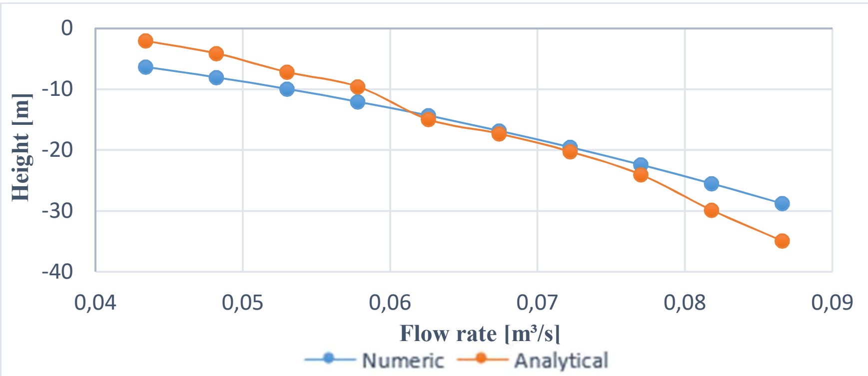

Figure 13. Graph of the impeller characteristic curve for outlet angle β5 20°.

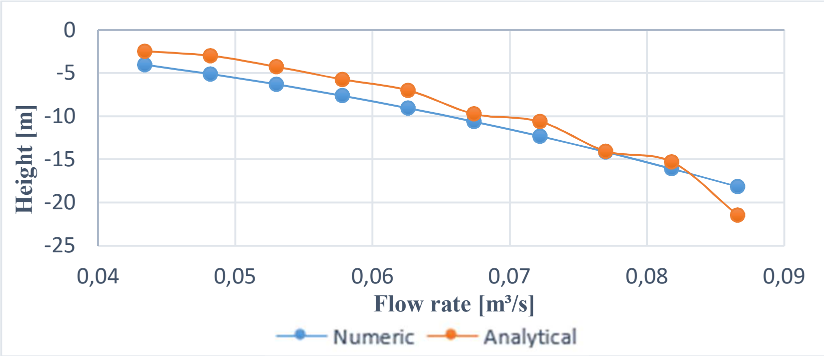

Figure 14. Graph of the impeller characteristic curve for outlet angle β5 30°.

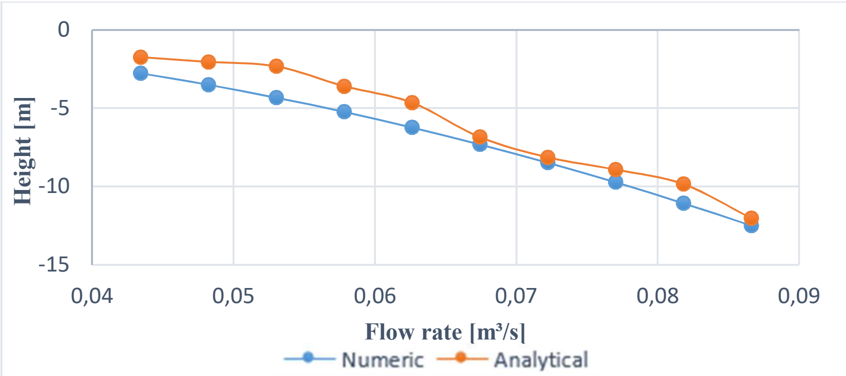

Figure 15. Graph of the impeller characteristic curve for outlet angle β5 40°.

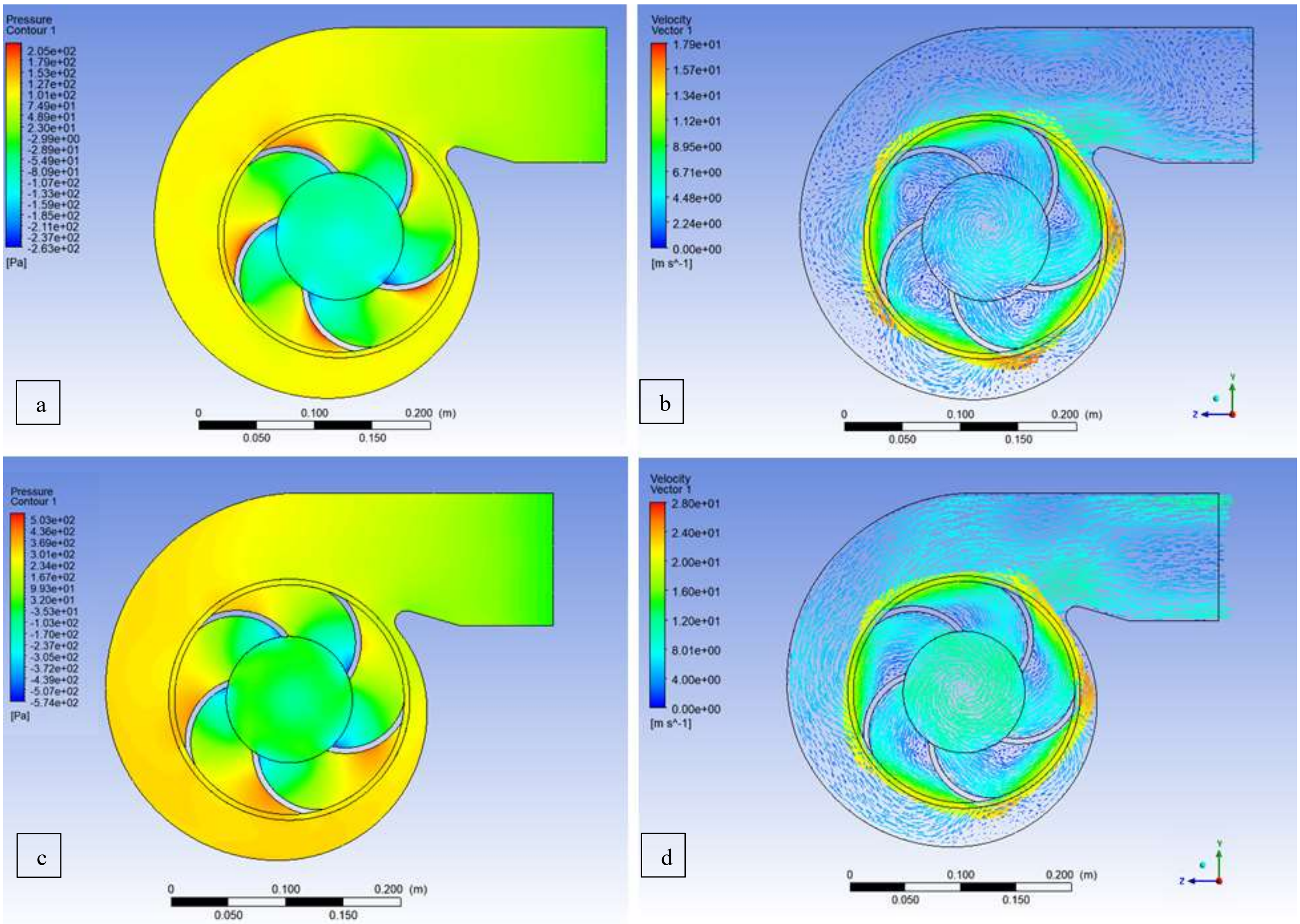

Figure 16. Pressure field and velocity vectors of the centrifugal fan for the constructive angle β5 = 20 ° for 1600 rpm in (a) and (b) and 2400 rpm in (c) and (d) respectively.

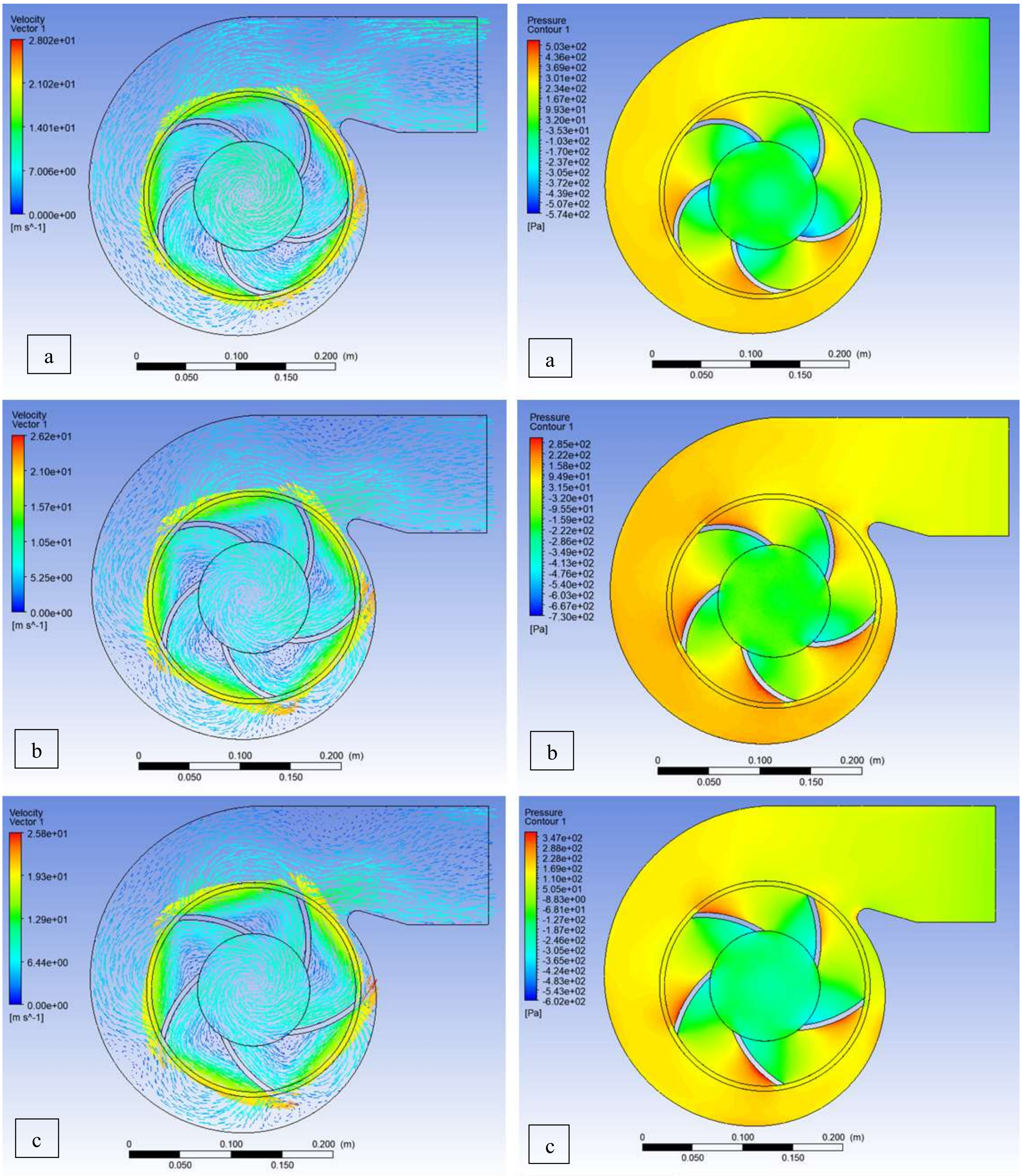

Figure 17. Pressure field and velocity vectors for 2400 rpm rotation for β5 = 20 ° (a), β5 = 30 ° (b) and β5 = 40 ° (c) respectively.

4. CONCLUSION

The present work shows that the constructive angles of output influence at efficiency of the rotor together with the operational condition of the machine, in this case, the angular velocity. As a student version software is used, the convergence of the mesh found, around 500.000 (five hundred thousand) elements, is considered satisfactory, applied to a simulation of 200 iterations and reaching a residual error between $10e^{-2}$ to $10e^{-4}$, therefore, a solution without great computational effort.

Through a general analysis, Fig. 9 conveys the idea that the constructive angle β5 = 30 ° is the ideal exit angle for the project, as it presents an almost linear behavior to the analytical for the theoretical height. However, the behavior of the hydraulic power in Fig. 8, Fig. 10 and Fig. 12 indicates that more factors influence at the flow rate and that there is an ideal fan rotation range that would be between 2400 to 2800 rpm, known as the nominal point or design point. It is worth emphasizing the difference between the lifting height and the theoretical lifting height since the first refers to the height delivered by the machine, involving hydraulic and volumetric losses of the system, while the second is the height delivered only by the impeller, respectively.

In a comparison of the visual results of the pressure and velocities fields, it can be seen in Fig. 17 the increase in the pressure field at the outlet of the blades, mainly in the lower part of the volume as the value of the construction angle increases. The velocity vectors highlight that the flow does not circumvent the surface of the blades, generating vortices and fluid recirculation inside the volute and in areas close to the discharge mouth, resulting in mandatory volumetric losses, which may require that this model of fan has less for low rotations in the paddle settings.

In short, the results validate the premises adopted for the analytical solution of the Euler Equation, even if at great simplifications in relation to the real models, since the comparison is made between the theoretical analytical parameters for an infinite number of blades, that is, considering only a geometry of the impeller with a numerical model considering the entire flow machine as the control volume. According to the literature, the behavior for a finite number of blades implies the loss of energy in the areas where the fluid leaves the impeller as it was visualized, to overcome the friction surfaces and give the energy delivered, therefore the theoretical height and the theoretical flow impeller are greater than the fan height and flow.

5. REFERENCES

Aureliano, F. d'S, Guedes, L. C. V., 2019. "Computational fluid dynamics (CFD): Behavioral study and optimization of the blades number of a radial fan". In 29th International Conference on Flexible Automation and Intelligent Manufacturing, - FAIM 2019. Limerick, Ireland.

Brazil, A. N. Termo Hydraulic Flow Machines, 2010. Mar. 2020. <http://ftp.demec.ufpr.br/disciplinas/tm120/apostila_mh/capitulo2.pdf>

Gabbi, R. Mathematical Modeling of Turbulent Flow in Axisymetric Channel with "Bluff-Body", 2013. Postgraduate, Regional University of the Northwest of the State of Rio Grande do Sul, Ijuí, Brazil. May. 2020 <https://bibliodigital.unijui.edu.br:8443/xmlui/handle/123456789/1712>

Germer, E. Flow Machines. Curitiba: Utfpr-ct, 2013. Feb. 2020 <Apostila-MaqFluxo-EG.pdf>.

Gomes Junior, C. Improvements to a centrifugal fan through numerical simulation. 2010. 21 f. Monograph, Federal University of Rio Grande do Sul, Porto Alegre, 2010. Feb. 2020 <https://www.lume.ufrgs.br/bitstream/handle/10183/25884/000754800.pdf>.

Heen, E. A. L. Fluid machines. 2nd ed - Santa Maria: Ed. da UFSM, 2006. 474p. pp. 25-70.

Kothe, L.B., Luz J. L. R., Vecina, T.D.J., 2016ª. "Design for Optimizing a Centrifugal Fan Through Computational Fluid Dynamics (CFD)". At the Technical Scientific Congress of Engineering and Agronomy - CONTECC 2016. Foz de Iguaçu, Brazil.

Oliveira Junior, J. A. A. Design of a Centrifugal Fan with Back Curved Blades Aided by CFD. 2004. 29 f. Monograph, Federal University of Rio Grande do Sul, Porto Alegre, Brazil. Feb. 2020 <http://www.geste.mecanica.ufrgs.br/pss/diploct/Joao_Am%C3%A9rico.PDF>.

Santos, F. L. Numerical Analysis Of Centrifugal Fan, 2017. Postgraduate, Regional University Of Northwestern State Of Rio Grande Do Sul, Panambi, Brazil. Mar. 2020 <https://bibliodigital.unijui.edu.br:8443/xmlui/handle/123456789/4508>.

Silva, J. S., Vitor, D. G., Lopes, R.P., 2013. "Construction of Centrifugal Fans for Agricultural Use", Brasília, Brazil.

6. RESPONSIBILITY NOTICE

The authors Bruna Beatriz de Paiva and Reginaldo Ribeiro de Sousa are the only responsible for the printed material included in this paper.