The Journal of Engineering and Exact Sciences – JCEC, Vol. 04 N. 02 (2018)
journal homepage: https://jcec.ufv.br
doi: https://doi.org/10.18540/jcecvl4iss2pp0220-0224
OPEN ACCESS – ISSN: 2527-1075

CFD SIMULATION OF A CENTRIFUGAL PUMP IMPELLER WITH VARIATIONS OF NUMBER OF BLADES

SIMULAÇÃO CFD DO ROTOR DE BOMBA CENTRÍFUGA COM ALTERAÇÃO DA QUANTIDADE DE PÁS

H. M. P. ROSA[1,*], B. S. EMERICK[2]

[1]Universidade Federal de Viçosa, Departamento de Engenharia Produção de Mecânica, Viçosa, Minas Gerais, Brasil
[2]Universidade Federal de Santa Catarina, Departamento de Engenharia Mecânica, Florianópolis, Santa Catarina, Brasil

*Autor correspondente: Departamento de Engenharia de Produção e Mecânica, Universidade Federal de Viçosa, Viçosa, Minas Gerais, Brasil, Fone: +55 31 3899-4698.
Endereço de E-mail: henrique.rosa@ufv.br (H. M. P. Rosa).*

A R T I C L E I N F O

Article history:
Received: 2017-09-26
Accepted: 2018-03-24
Available online: 2018-06-30

p a l a v r a s - c h a v e
Rotor de Bomba Centrífuga
Número de Pás
Simulação Numérica

k e y w o r d s
Centrifugal Pump Impeller
Number of Blades
Numerical Simulation

A B S T R A C T

The purpose of this paper is to model and simulate the water flow into the impeller of a centrifugal pump. Using CFD techniques two impellers were modeled, differing only by the number of blades of each one (6 and 10). From the results of the simulations, the impeller's characteristic and the results were compared for each impeller. The model with 10 blades was more efficient for a real application of a centrifugal pump. All the results have shown themselves consistent with the turbomachinery theory, proving the efficiency of CFD simulation tools in the project of this kind of equipment.

R E S U M O

O propósito deste artigo é modelar e simular o escoamento de água em um rotor de uma bomba centrífuga. Utilizando técnicas de CFD, dois rotores foram modelados diferenciados unicamente pelo número de pás de cada um (6 e 10). A partir dos resultados das simulações, as características dos rotores e os resultados foram comparados para cada caso. O modelo com 10 pás foi mais eficiente para uma aplicação real de uma bomba centrífuga. Todos os resultados são consistentes com a teoria de turbomáquinas, comprovando a eficiência da ferramenta de simulação CFD no projeto deste tipo de equipamento.

1. INTRODUCTION

Centrifugal pumps play a key role in both domestic and industrial applications. Without them, the fluid transportation between different levels and big distances would be much more hard-working and the cost of this operation would be much higher. Inside a centrifugal pump, the forces developed in the liquid mass are responsible for its movement and increase of its pressure (Stepannof, 1961).

The main component of this equipment is the impeller, which is a set of blades that drive the flow passing through them. Many authors call the impeller like the "heart" of centrifugal pump, since the total energy changes happens on it. Its centrifugal action accelerates the fluid passing through it, increasing the pressure energy.

Physical characteristics of the impeller, such as number of blades, blade angle and diameter, for example, have great influence in the final performance of this equipment. The change of these parameters alter the amount of available hydraulic energy in the impeller's exit, and, consequently, the amount of available work the flow can perform (Rajendran, et al., 2012).

The theory of energy transformation into a pump impeller is based on angular momentum equation, considering one-dimensional flow. For ideal conditions and with no viscosity fluid, the impeller should have infinite number of blades for correctly guide the flow (Figure 1) and product a same pressure for each radius into impeller. Therefore, the number of blades has influence on impeller performance. In real situations, if there is many blades, the flow is better guided, but the loss due the viscous friction is higher.

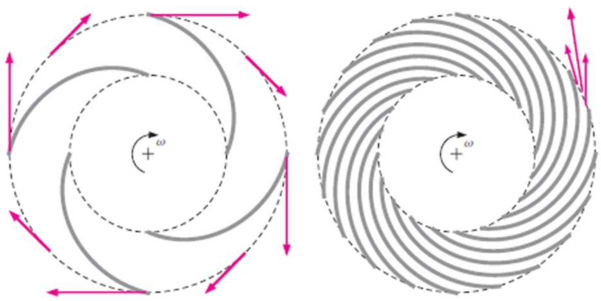

Figure 1 - Impeller with few blades and impeller with many blades.

Fluid driving with higher performance and lower energy losses is requisite of great relevance in the design of systems using centrifugal pumps, because it leads to fewer expenses with maintenance, implantation of the system and energy costs (Turton, 1995). Many changes in the project are made before its physical implantation, but the costs of prototyping sometimes prevent its improvement.

In this context, over the past years, Computational Fluid Dynamics (CFD) has stood out as a new approach for problems involving fluid flow. Numeric simulations have a great potential to predict the performance of a specific equipment in a determined system before its implantation (Yedidiah, 2014). Therefore, changes in the design of a plant are much easier and less expensive if the designers already know what can be improved before building any prototype (Nataraj and Singh, 2013).

The use of CFD tools to design these complex machines has only been made possible due to the increasing availability of computing resources. These evolved side-by-side with an increase in accuracy of numerical methods, and CFD shifted from a pure academic research tool into the competitive industrial companies design offices (Pascoal, et. al, 2009; Voorde, et. al, 2003).

In this paper are described the model and simulation, using CFD techniques, of two centrifugal pump impellers, one with six blades and other with ten blades. From the results of the simulations, the impeller's characteristic curves were built and the results were compared for each impeller.

2. COMPUTATIONAL SIMULATION

2.1 Equations

In the averaging of steady-state incompressible flows, the conservation equations can be solved based on the average Reynolds values or the time-averaging approach; however, the most common method of modeling turbulent flows is the time-averaging method. Using this approach for the case of incompressible flows, the general forms of the governing equations could be expressed as relations (1)–(5). As we have a rotating control volume, these equations employ the relative motion, therefore the relative velocity of fluid W.

Since the pumped fluid is incompressible and the flow is in a steady state, the continuity equation has the following form:

$$\vec{\nabla}.\overrightarrow{W} = 0 \qquad (1)$$

The equation of conservation of momentum is expressed as Equation 2. Where ρ is the density of fluid, P is pressure, SM, the source term, and τ, the shear stress tensor.

$$\rho(\overrightarrow{W}.\vec{\nabla})\overrightarrow{W} = -\vec{\nabla}P + \vec{\nabla}\tau + S_M \qquad (2)$$

For flows in a rotating frame of reference, rotating at a constant angular velocity ω, additional sources of momentum are required to account for the effects of the Coriolis force and the centrifugal force, written as Equations 3 and 4 respectively:

$$S_{Cor} = -\rho\vec{\omega}x\overrightarrow{W} \qquad (3)$$

$$S_{Cfg} = -\rho\vec{\omega}x(\vec{\omega}x\vec{r}) \qquad (4)$$

So that, the additional sources of momentum is written as Equation 5:

$$S_M = -\rho\vec{\omega}x\overrightarrow{W} - \rho\vec{\omega}x(\vec{\omega}x\vec{r}) \qquad (5)$$

Where r is the location vector.

To solve problems using CFD different solution methods are employed, the most used are the Finite volume Method. This method model three-dimensional flow. The approximated solutions are obtained through conservation balance in volume control, differing from the methods of Finite Difference and Finite Element, which work with grid points, not being conservative in a discrete level. Thus, ANSYS CFD uses the finite volume method to assure conservation of mass, momentum and energy. The studied domain is divided into several sub-regions, called control volume.

2.2 Computational Model

In order to allow the performance of the CFD simulation, the first step was the creation of the geometry that represents the fluid passing through the impeller. After, the mesh for the model was created and the boundary conditions were inserted to perform the simulation in the correct conditions.

After the simulation was completed, a new geometry has been created, changing only the number of blades of the impeller, in order to analyze the influence of this physical change in the performance of this equipment.

Geometry: The first physical model was an impeller with 6 blades (Figure 2) in the software SolidWorks®, and the second model has differed of the first one only in the number of blades, ten blades (Figure 3). The blade models were obtained from a calculation procedure described by Henn (2012). The nominal point of this specific impeller is a flow rate of 24 m³/h and a total head of 8 m. The angular velocity of the impeller is 1750 rpm.

The other design specifications are listed below:

D4 (Inlet diameter) = 56,5 mm;

D5 (Outlet diameter) = 148,9 mm;

β5 (Outlet blade angle) = 25º;

b4 (inlet width) = 21 mm;

b5 (outlet width) = 10 mm;

z (number of blades) = 6 or 10;

Blades' thickness = 3 mm.

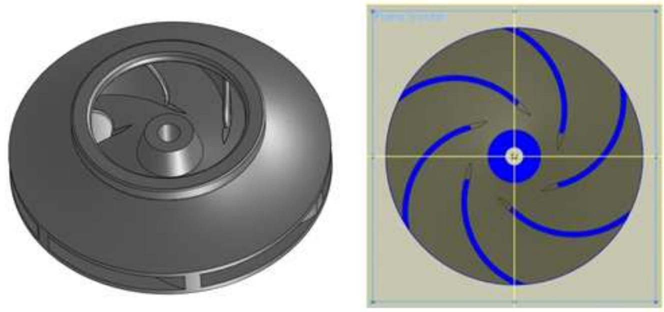

Figure 2 - Impeller with 6 blades.

The CFD simulations occurs in fluid domains only (Figure 4). Therefore, the created geometry must represent the fluid passing through the impeller, and not the impeller itself. Because of that, the "Combine" tool was used in SolidWorks to generate a geometry that represents the fluid passing through it (Figure 4).

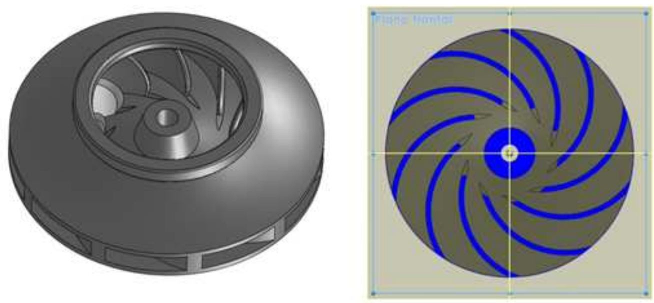

Figure 3 - Impeller with 10 blades.

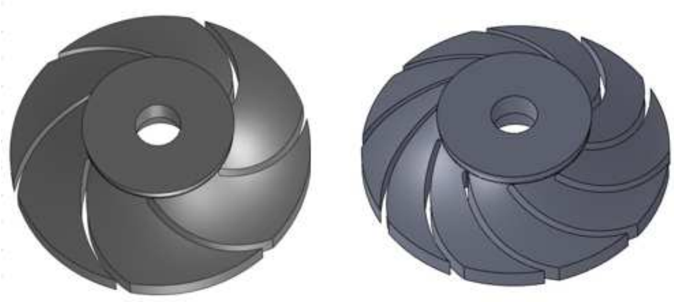

Figure 4 – Geometry representing the fluid passing through both impellers.

Mesh: After to do the geometries the next step is the generation of a grid (mesh) that will represent the control volume of the fluid flow.

To perform CFD simulations, the softwares use meshes, which are based in the numeric discretization of the fluid in a finite set of elements (control volumes), and nodes, where all the flow equations are numerically solved in each iteration of the simulation, until the convergence criteria is achieved. With this approach, all the simulation variables (pressure, velocity, temperature, etc.) can be determined in each point of the domain.

To perform the mesh generation, the software used was Meshing®, inside ANSYS® platform. In this software, it is possible to generate automatically a simple mesh or to specify parameters (such as type and size of elements, for example) to generate a more accurate and efficient one. When generating a mesh, the designer must be careful so the mesh is accurate enough to perform the simulation in a trustful way. However, if the mesh is too refined, the amount of elements will be much high, resulting in higher computational and time costs. Therefore, the designer must establish one point where the grid is enough refined, but without unnecessary refinement.

For both models, in areas such as the inlet and outlet of the impeller, refinements occurred. In these areas, "sizings" were made, to determine the size of the cells across the selected face. In the software Meshing®, Named Selections had to be created in the faces where boundary conditions were present. Therefore, "Inlet", "Outlet", and "Wall" named selections were created. Both models' meshes were generated with tetrahedal elements. The resultant meshes are shown in Fig (5) and (6):

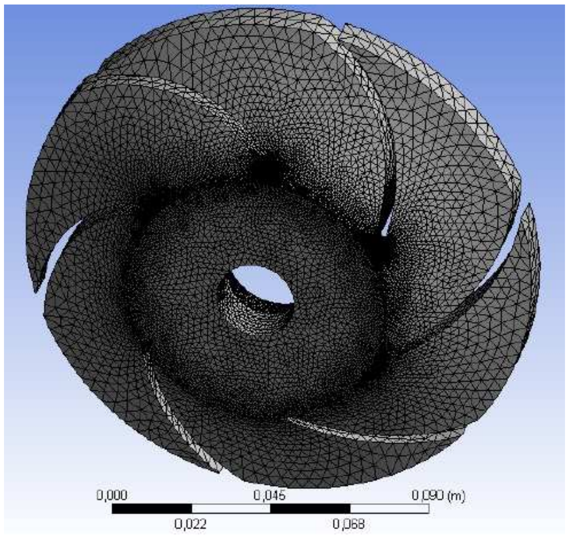

Figure 5 – Mesh in the 6 blades impeller.

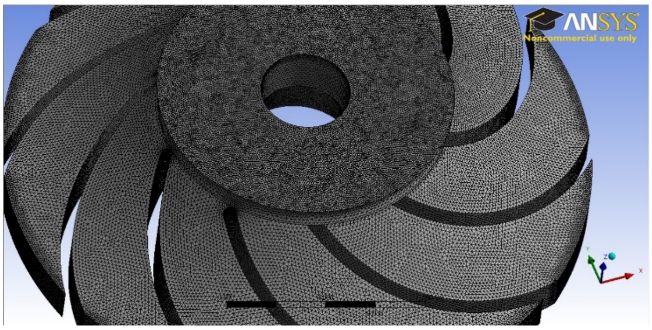

Figure 6 – Mesh in the 10 blades impeller.

Boundary conditions: In both simulations, boundary conditions were exactly same. Therefore, any changes in the results are only because of the physic differences between the impellers. The fluid domain was defined as water at 25ºC, without any heat changes, and the rotational speed was -1750 rpm across the +Z axis.

In the entrance of the impeller, an "Inlet" condition was imposed, what means that the flow only goes inside the domain. The static pressure in this boundary was exactly equal to the atmospheric (101325 Pa).

In the exit of the impeller, an "Outlet" condition was imposed, meaning that in this boundary, the flow goes only outside the domain. In this boundary, one necessary condition was the mass flow passing through the outlet. The value of the mass flow was the variable of the simulation. Each value results on a pressure difference value.

In the walls, the conditions imposed were "No slip wall", which imposes that the first layer of fluid near the wall has no movement, and "Smooth Wall", that excludes the effects of the wall roughness.

2.3 Simulation

After the insertion of all the boundary conditions, the models were simulated, varying the values of the mass flow rate in the outlet of both impellers. For each mass flow, one pressure drop was obtained. Two different kinds of pressure were analyzed, the static and total pressure. The static pressure difference does not take in the velocity effects in the gain of pressure, while the total pressure difference does.

For the simulations, the disk friction losses and the flow leakage were not considered. Consequently, the hydraulic head produced represents the specific work transmitted by the impeller blades. This is calculated by Equation 6:

$$H_{blade} = \frac{\Delta P}{\rho g} \tag{6}$$

Where H_{blade} is the impeller blade head, [meters], ΔP is the total pressure gain, [Pa] ρ is the density of the water (997,0kg/m3), and g is the gravity acceleration (9,80665m/s2).

3. RESULTS AND DISCUSSIONS

3.1 Impellers' Curves

From the simulation of both impellers, the results obtained are shown in the Tables 1 and 2 below:

Table 1 – Results for the 6 blades impeller.

Six Blades Impeller			
Mass Flow Rate (Kg/s)	**Volume Flow Rate (m³/h)**	**Total Pressure Difference (Pa)**	**H_{blade} (m)**
0,009	0,0325	158307	16,19
2,772	10,01	155666	15,92
5,540	20,00	141448	14,47
6,650	24,01	135630	13,87
8,317	30,03	130181	13,31
11,089	40,04	114240	11,68

Table 2 – Results for the 10 blades impeller.

Ten Blades Impeller			
Mass Flow Rate / Kg/s	**Volume Flow Rate / m³/h**	**Total Pressure Difference / Pa**	**H_{blade} / m**
0,008	0,030	155509	15,91
2,772	10,01	168097	17,19
5,000	18,05	161056	16,47
6,670	24,08	146747	15,01
8,317	30,03	137906	14,10
11,089	40,04	103750	10,61

With these data, it is possible to obtain the impellers' curves, shown in Figure 7:

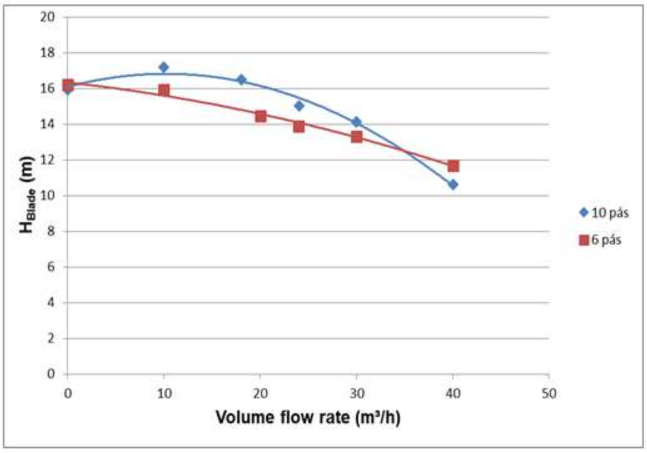

Figure 7 – Impellers' Curves.

From that, it is possible to identify that the model with 10 blades has shown to perform a higher impeller head. That occur over all range of volume flow rate, except for high flow rate (bigger than 35m3/h). For nominal flow rate (24m3/h) the impeller head for 10 blades is 8,2% higher than 6 blades.

By analyzing the curves singly, we observe that the shapes of them are coherent with curves found on the catalog of manufacturer.

For zero flow rate (shut-off point), the impeller head is practically coincident for both impellers. It is according theory of centrifugal pumps that say the head on shut-off depends only outlet diameter (D5) and angular velocity. Therefore, the number of blades does not have any influence on shut-off.

3.2 Pressure Distribution

The Figure 8 and 9 show the static pressure distribution inside 6 and 10 blades impeller for 6,67kg/s mass flow rate (nominal rate). These figures show that 10 blades present a more uniform pressure distribution than 6 blades. It is consequence from effect of better flow guidance, that causes better velocity distribution, what consequently result in better pressure distribution.

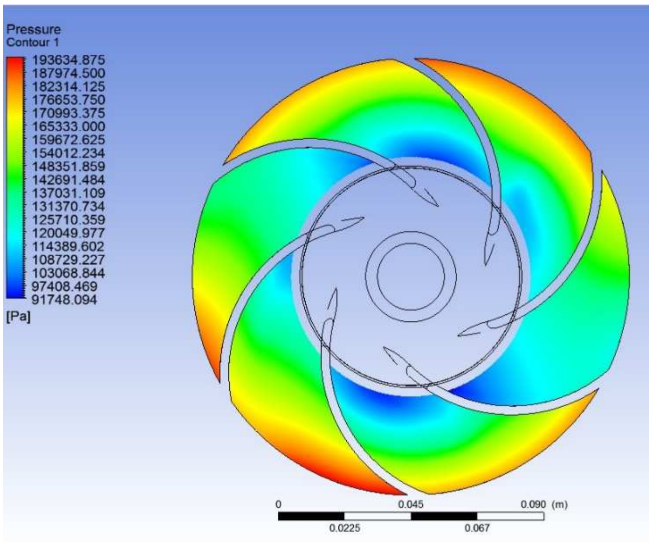

Figure 8 – Pressure distribution on six blades rotor.

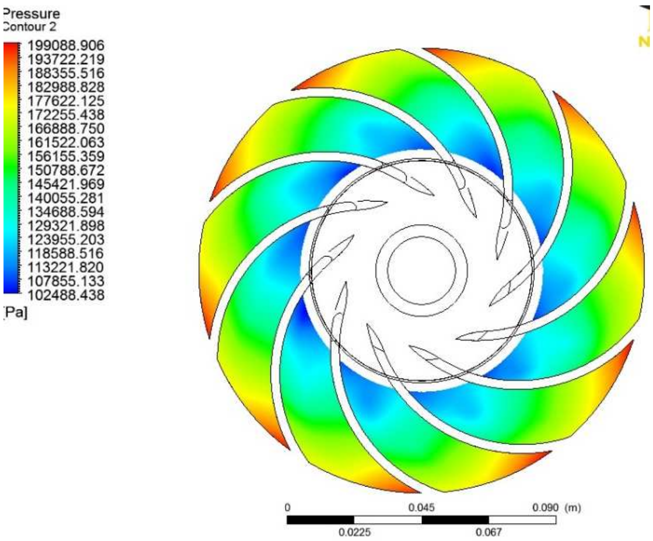

Figure 9 – Pressure distribution on ten blades rotor.

4. CONCLUSIONS

In this paper, were realized multiple simulations using the CFD approach. The main goal was to analyze and determine, through the impellers characteristic curves, which of two configurations of impeller, with 6 and 10 blades, presented larger impeller head.

The comparison between the characteristic curves in the impellers has shown that the model with 10 blades presented a higher impeller head for the usual range of volume flow rates for this particular impeller. For nominal flow rate the impeller blade is 8,2% higher than 6 blades. This indicates that for this impeller the effect of better flow guidance had a positive influence and bigger than the effect of viscous friction.

The characteristics curves and pressure distribution showed the results of simulation were coherent with the theory of centrifugal pumps and manufacture's catalog.

Finally, the efficiency of CFD simulation tools was proved as a part of the design of centrifugal pumps, because it allows the designers to predict the performance of the equipment in a specific system. Knowing the performance, the designers can make changes in the project to increase the efficiency, decreasing costs and increasing productivity of the plant.

REFERENCES

NATARAJ, M.; SINGH, R. Analyzing pump impeller for performance evaluation using RSM and CFD. **Desalination and Water Treatment**, v. 52, p. 6822-6831, 2014.

PASCOA, J.C.; MENDES, A.C.; GATO, L.M.C. A fast iterative inverse method for turbomachinery blade design. **Mechanics Reserch Communications**, v. 36, n. 5, p. 630-637, 2009.

RAJENDRAN, S.; PURUSHOTHAMAN, K. Analysis of a centrifugal pump impeller using ANSYS-CFX. **International Journal of Engineering Research & Technology (IJERT)**, v.1, n. 3,2012.

STEPANNOF, A.J. Pompes centrifuges et pompes hélices. Paris: Dunod, 1961.

TURTON, R.K. Principles of turbomachinery. 2nd ed., London: Champman & Hall, 1995.

VOORDE, J.V.; VIERENDEELS, J.; DICK, E. A force-based grid manipulator for ALE calculations in a lobe pump. **Journal of Thermal Science**, v. 12, n. 4, p. 318-322, 2003.

YEDIDIAH, S. A study in the use of CFD in the design of centrifugal pumps. **Engineering Applications of Computational Fluid Mechanics**, v. 2, n. 3, p. 331-343, 2008.