23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

NUMERICAL STUDY OF THE PERFORMANCE OF A TWO-STAGE CENTRIFUGAL PUMP

Rafael Dunaiski, rafael.dunaiski@gmail.com
Henrique Stel, henriqueazevedo@utfpr.edu.br
Thiago Sirino, thiagosirino@hotmail.com
Rigoberto Eleazar Melgarejo Morales, rmorales@utfpr.edu.br
Edgar Minoru Ofuchi, edgarofuchi@utfpr.edu.br
Multiphase Flow Center - NUEM
Federal University of Technology – Paraná, UTFPR
Av. Sete de Setembro 3165 – Curitiba, PR, Brazil

Abstract. *Computational Fluid Dynamics is used in this work to perform a numerical study of the performance of a two-stage centrifugal pump with a vaned diffuser and a volute. A wide range of operating conditions is assumed, including several flow rates and impeller rotating speeds. The flow inside the pump is modeled numerically using a multi-block, transient rotor-stator technique, with structured grids for all pump parts. Comparisons between experimental and numerical head curves show a good agreement, pointing to a valid numerical model. Global parameters such as head and efficiency are evaluated for each pump component. Streamlines, pressure and turbulent kinect energy fields are also showed. Results might help to understand how the pressure gain is distributed through the pump, and to address efforts in the search for a more efficient and affordable pumping system.*

Keywords: *numerical, centrifugal pump, vaned diffuser, volute, performance.*

1. INTRODUCTION

Centrifugal pumps are widely used in several applications, such as chemistry, oil and food industries, water distributions systems, etc. This type of component aims to increase the pressure of the pumped fluid. Through the addiction of kinetic energy provided by a rotating impeller, pressure increase is obtained with an increasing of the area of the impeller and by means of a volute. Further increase in pressure happens if a static diffuser is used, which is designed to provide appropriate direction to the flow that exits the impeller and to increase the cross-section area of the hydraulic channels. More than one stage is also frequently found in centrifugal pumps, cases when more than one impeller are used for a greater pressure increment. However, the combination of several impellers, vaned diffusers and a volute might introduce several complex flow characteristics that largely deviate from regular inviscid pump flow theory.

The study of the flow inside centrifugal pumps has received much attention in recent years. Several methods are applied in such studies, each being more appropriate for each situation. For instance, experimental methods can be used for evaluation of the pump performance at certain operational conditions. The main advantage of this approach is the reliability provided by a good approximation of the real application, but experiments might be very expensive and can be too complex to be worth assembling in situations where flow field details, such as velocity and pressure distributions, are needed. Another typical approach to the study of the flow in centrifugal pumps is the analytical one. Its results and application are quite straightforward, but are based on several simplifying assumptions and are available just for a few simpler applications. For more complicated situations, and where it is not possible or not viable to perform experiments, the solution might be the use of a numerical approach. It generally provides great flexibility to analyze a set of different pump geometries and operating conditions. A very useful tool to accomplish numerical results is the computational fluid dynamics (CFD).

Several works have used CFD to study the flow inside centrifugal pumps. For example, Asuaje et al. (2005) studied the influence of a volute on the pressure and velocity fields in a centrifugal pump impeller. They showed that it is necessary a minimum number of nodes in the computational mesh to obtain a numerically stable solution and that the use of different turbulence models resulted in similar results for performance parameters. Cheah et al. (2007), amongst several other works, showed that heavy recirculations might be found inside the impeller and diffuser for off-design situation. Feng et al. (2007a) conduct works about unsteady flows in centrifugal pumps, using both CFD and visualization techniques. Their qualitative comparisons between CFD and experimental visualization show that it is possible to reproduce with reliability the flow using numerical methods. In another work, Feng et al. (2007b) found out that there is a cross-dependence between the impeller and diffuser channel flows, which were also found by Amaral et al. (2009). Stel at al. (2013) studied the flow in the first stage of a two-stage centrifugal pump with vaned diffuser. They concluded that the flow becomes badly oriented for part-load conditions.

Previous works showed that it is strictly necessary to achieve a proper design of pumps. The efficiency of the system is highly affected by the flow structures inside the pump. The knowledge of how these structures change with the operational conditions is key for the development of more efficient components.

This paper aims to perform transient CFD simulations of the flow in a complete two-stage centrifugal pump, which follows the study of Dunaiski at al. (2014), who performed a similar study but only with steady state simulations. All the components of the pump are modeled and the transient flow through all of them is going to be analyzed. In this sense, flow field details will be analyzed in each part of the pump in order to investigate their sensibility to different operating conditions.

2. MATHEMATICAL MODEL

Numerical simulations were performed through a range of rotating speeds (1150, 1000, 806 and 612 rpm) and for normalized volumetric flow rates within 0.25 and 1.50, which are the ratio between a flow rate and the design flow rate (Q/Q_{des}), between. The design flow rate is 36.5 m³/h. The operating fluid considered is water, with density ρ=997 kg/m³ and dynamic viscosity μ=8,9.10^{-4} Pa.s. Flow is assumed turbulent, transient, isothermal and incompressible, and forces due to the gravitational acceleration are neglected. For this case, mass and momentum balance equations can be written in the general conservative form as:

$$\frac{\partial}{\partial t}(\rho\phi)+\nabla\cdot(\rho\vec{V}\phi)=\nabla\cdot(\Gamma_{\phi}\nabla\phi)+S_{\phi}+P_{\phi} \tag{1}$$

where $\vec{V}$ is the velocity vector and t is the time. For the continuity equation, Eq (1) is derived using $\phi=1$ and $\Gamma_{\phi}=S_{\phi}=P_{\phi}=0$. For the momentum balance equation, $\phi=\vec{V}$ and $\Gamma_{\phi}=\mu_{eff}$ (the effective dynamic viscosity which is obtained by adding the fluid viscosity to a turbulent viscosity calculated through the use of a turbulence model). The momentum source term S_{ϕ} might be a combination of effects such as Coriolis and centrifugal field while the term P_{ϕ} stands for the pressure gradient usually written as ∇P. The turbulence model chosen was the standard $\kappa-\varepsilon$.

3. NUMERICAL MODEL

The pump used in this study is a two-stage centrifugal pump model Imbil ITA 65-330/2. A multi-block methodology is used in order to model the numerical domain of the pump. Each part of the pump is divided into one or more numerical subdomains. The Ansys-CFX software is used to perform this method. It considers each one of the subdomains separately, transferring information of the flow between them through interfaces. Two of the subdomains are rotative (both impellers) while the others are stationary. Figure 1 shows that six subdomains are considered: an intake pipe, the first impeller, the vaned diffuser, the return channel, the second impeller and the volute. Information about the geometry of the impellers and the vaned diffuser are shown in the Tab. 1.

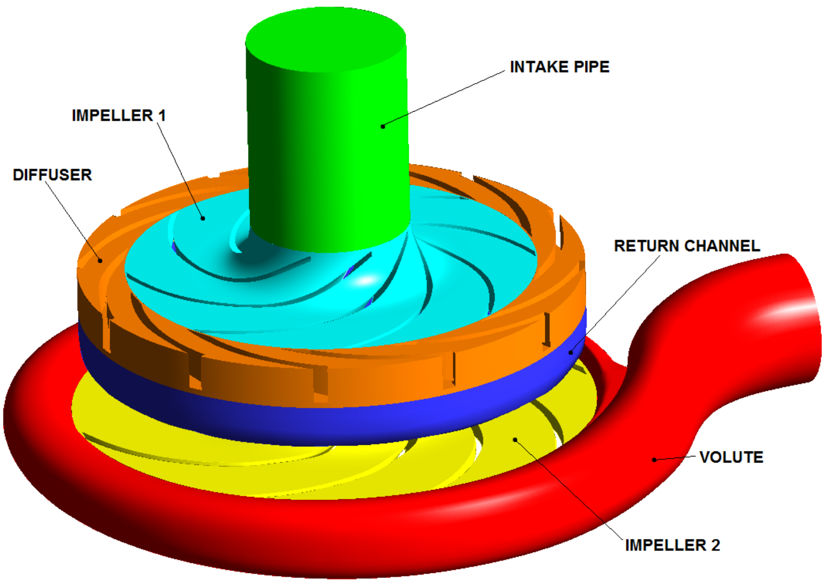

Figure 1. Numerical Domain

The boundary conditions assumed are shown in Fig. 2, along with a schematics of the subdomains modelled. At the pipe intake inlet it is set a reference static pressure of 0 Pa (respect to a regular domain pressure of 1 atm). This pipe is modeled with the objective of maintain the inlet boundary condition far from the impeller eye. At the outlet of the volute, the flow rate of each simulation is specified. No-slip condition is applied at all pipe walls.

Table 1. Specifications of the impellers and the diffuser of the pump.

Description	Impeller 1	Diffuser	Impeller 2
Number of blades	8	12	8
Inlet diameter	80 mm	208.6 mm	76 mm
Outlet diameter	204.2 mm	252.8 mm	260 mm
Inlet blade height	21 mm	18 mm	21 mm
Outlet blade height	12 mm	18 mm	8 mm
Inlet blade angle	22.5 deg	21 deg	23.5 deg
Outlet blade angle	36 deg	10 deg	35 deg

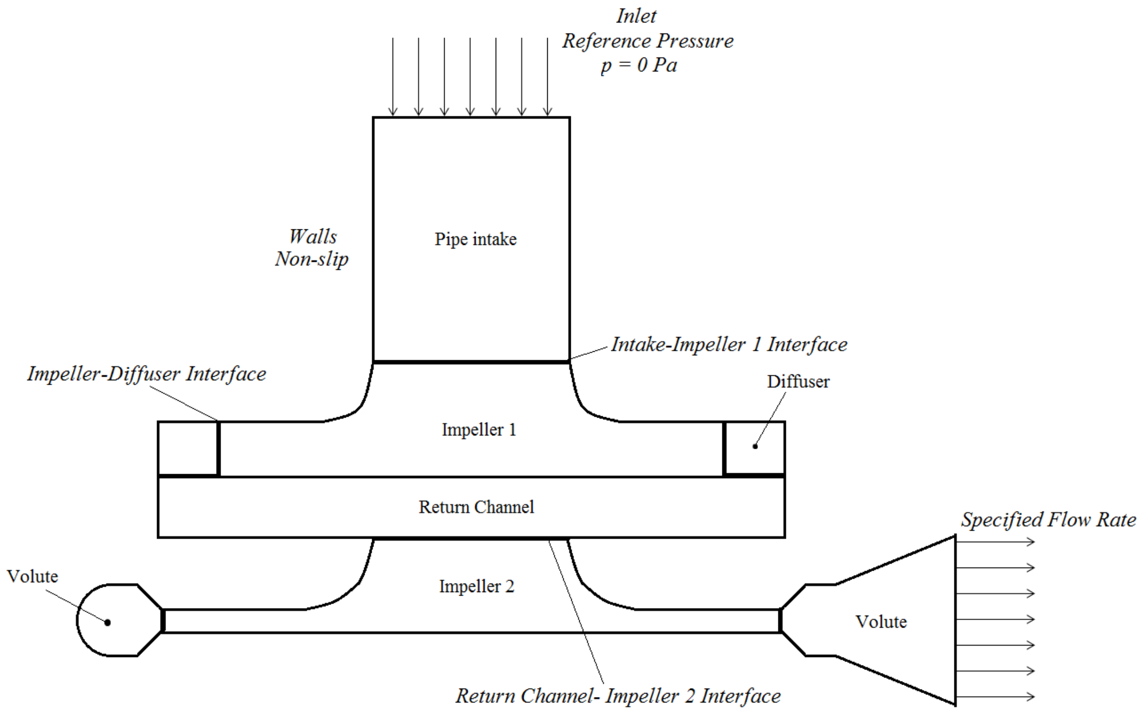

Figure 2. Schematics of the subdomains and boundary conditions adopted in the numerical simulations.

A "transient rotor-stator" approach is used to account for the impellers rotation, which is known to provide accurate results, since the mesh is rotated at each timestep. This approach takes into consideration every type of transient phenomena, resulting in a more reliable model than a stationary impeller approach would deliver. However, the transient rotor-stator boundary condition requires a flow field as initial condition, which is obtained via simulations that use the "frozen rotor" technique. The frozen rotor condition aims to treat the impeller as stationary, and source terms that consider the centrifugal and Coriolis rotational effects are directly included in the momentum balance equation (as the term in Eq. (1)). A mean flow field in the rotating domain is generated by this method, which gives a fair approximation of averaged quantities and is much less time-consuming than transient solutions (ANSYS, 2009).

The computational mesh used in the simulations was created using the softwares Ansys ICEM-CFD and Ansys Turbogrid 14.5. Several tests were carried out in order to achieve a grid that would generate reliable results with the least computational cost. The total number of nodes of the selected grid is around 2,500,000. Structured grids are used for all pump parts, with refinements in key points such as the impeller-diffuser interface and the blades tips. The simulations were carried out using the element-based finite volume method as implemented in Ansys-CFX, with a high order interpolation scheme to the advective terms. An overview of the mesh of the first impeller can be seen in Fig. 3.

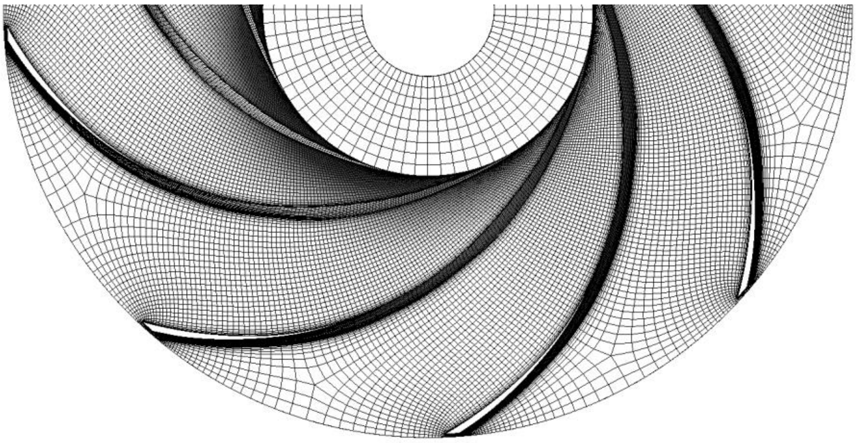

Figure 2. Schematics of the subdomain and boundary conditions adopted in the numerical simulations.

4. RESULTS

4.1 Validation of the results

The numerical results obtained through the simulations were compared with the experimental data of Amaral (2007), who measured performance parameters for the same pump used in this work. The comparison is made for a range of normalized volumetric flow rates and rotational speeds and it is shown in Fig. 3.

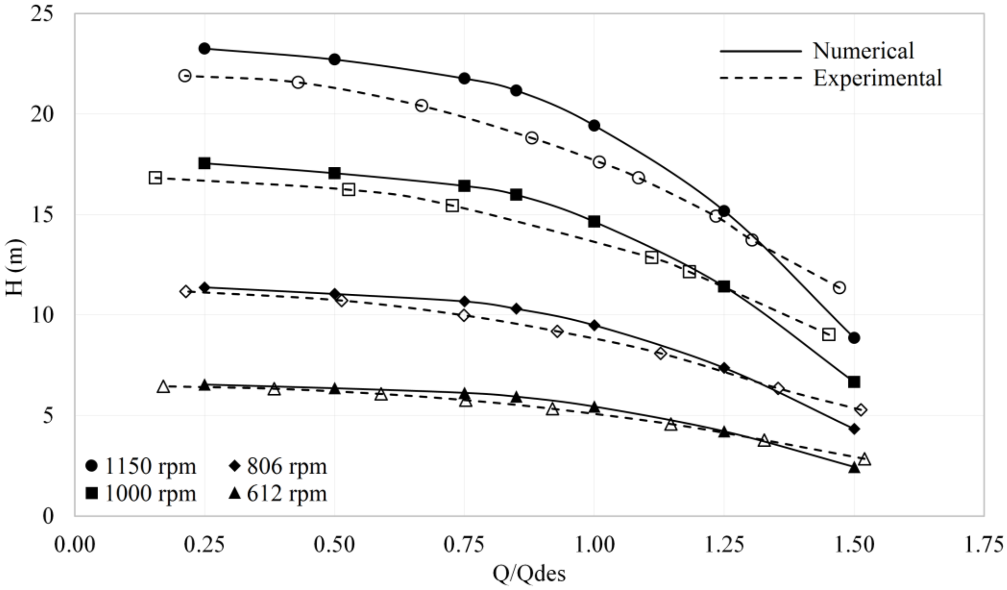

Figure 3. Comparison of numerical and experimental head values for several flow rates and rotational speeds.

It can be seen that there is a good agreement between the results obtained by both methods. The greatest deviations are observed for high flow rates, but overall differences are rather small. The average percentage variance observed for all points is around 6.8%. Figure 4 presents a comparison between numerical and experimental values for the pump hydraulic efficiency. It is generally expected that the numerical values should be greater than the experimental ones, since effects like disk friction, mechanical and volumetric losses of the real model are neglected in the numerical model. Although the values cannot be directly compared, the tendency of the curves seems to be the same and the best efficiency point (BEP) is close for both sets of data. In the numerical set of data it is observed that the BEP is not the same as the design flow rate extracted from the manufacturer's chart. Rather, it is dislocated to a value close to 85% of the design flow rate. This kind of situation can happen if the manufacturer's data is not presented with precision, which is the more probable cause, or if the experimental setup that generated such data is not well adjusted.

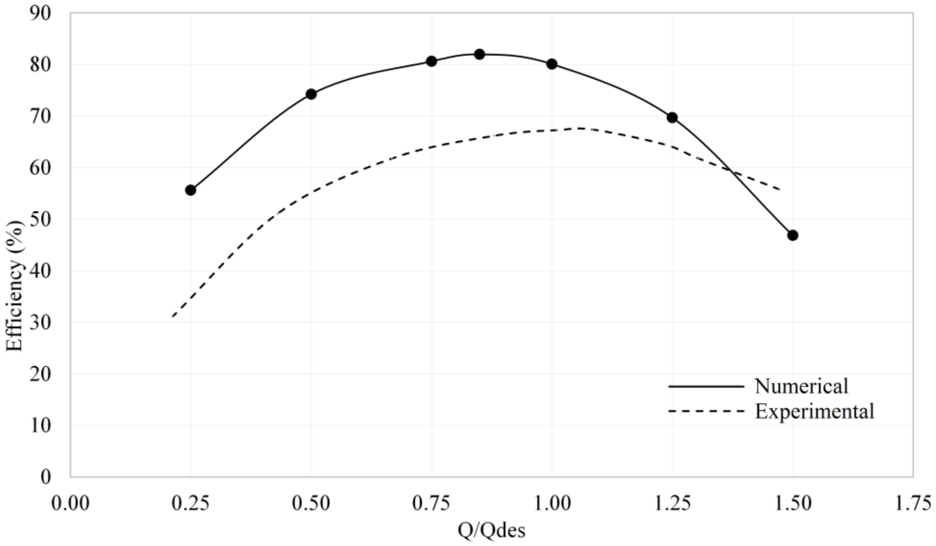

Figure 4. Comparison between numerical and experimental hydraulic efficiencies for 1150 rpm

4.2 Flow Field

Figure 5 shows streamlines of the flow field placed on a plane positioned midspan of the impeller channel passage inside the first impeller and the diffuser. The streamlines colors represent a normalized velocity, which uses as a normalization parameter the tangential speed of the first (U_1) or second (U_2) impellers at 1150 rpm, depending on the component analyzed. All the flow fields depicted from this section onwards are an averaged field obtained from a full rotation of the impellers, thus discarding that any instantaneous phenomenon is being taken into account. Four load conditions with the same rotational speed (1150 rpm) were compared. Both situations, the numerical BEP and the design flow rate, are presented. In addition, part-load and overload situations are shown. It can be seen that for part-load conditions heavy recirculations take place inside both the impeller and the diffuser channels, with the effect being more pronounced at the diffuser. These recirculations are responsible for the greater degradation observed in such situations. In a near BEP situation, the flow seems to be more organized, and the streamlines follow the shape of the blades. The flow inside the diffuser is very organized as well and recirculations are not observed. For overload conditions, although the streamlines seems aligned to the blades shape, there is degradation due to high friction losses. High velocity regions are seen at the suction sides of the diffuser vanes. This happens because the flow enters the diffuser with a high incidence angle, which leads to a non-uniform velocity distribution and an increase in pump losses.

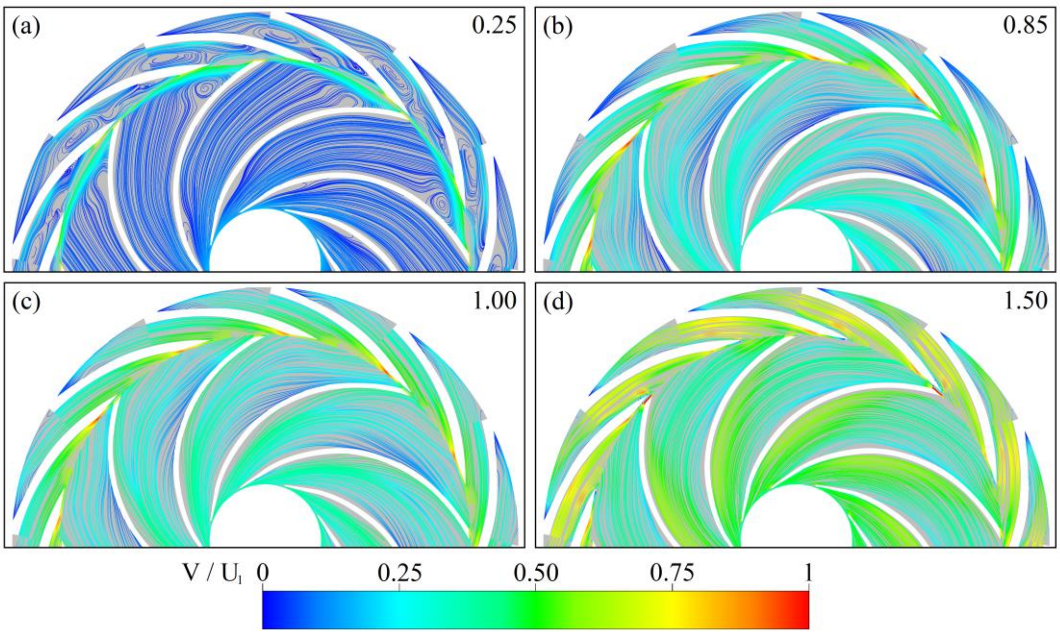

Figure 5. Streamlines inside the first impeller and the diffuser for four normalizade flow rates and 1150 rpm.

Figure 6 shows streamlines of the flow field placed now on a plane positioned midspan of the impeller channel passage inside the second impeller and the volute. It is easy to notice that the second impeller follows the same tendency set by the flow field in the first impeller. This shows that the impellers were designed so that for the same flow rate they behave in a similar fashion, with large recirculation zones for part-load conditions and a more geometry-oriented flow for high flow rates. It can be noticed, however, that small vortices are observed close to the impeller in all cases, including the highest flow rate shown, but they might be caused by the inability of the return channel in delivering a purely axial flow to the second impeller. Also, for part-load and near BEP situations no recirculations are seen inside the volute. However, for overload conditions, heavy recirculations are observed at the exit of the volute. This is another important factor for the reduction of the performance for greater flow rates.

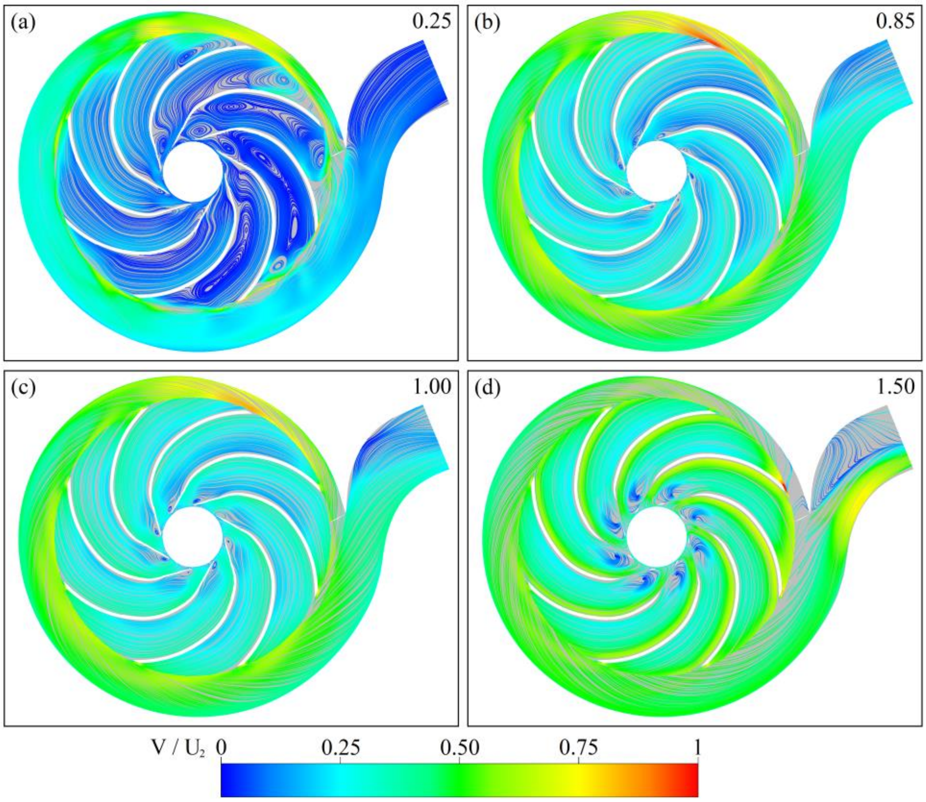

Figure 6. Streamlines inside the second impeller and the volute for four normalized flow rates and 1150 rpm.

Another comparison is shown in Fig. 8, this time depicting the pressure field across the whole pump. It can be seen that although almost all parts are responsible for pressure gains through the pump, a pressure loss indeed occur inside the return channel as afore mentioned. However, this component is necessary to rearrange the flow to the second impeller, reducing losses inside the second stage.

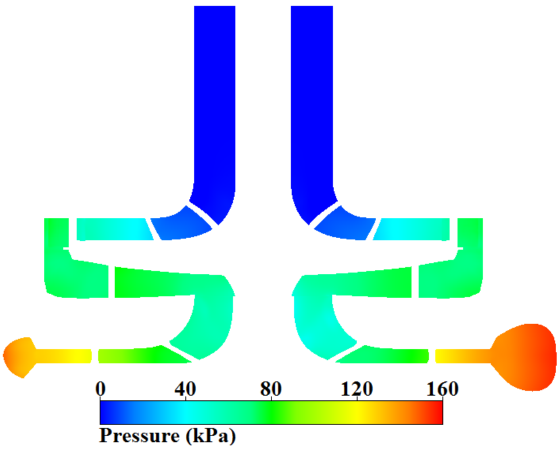

Figure 8. Pressure field in a pump cross section plane pump with a normalized flow rate of 0.85 and 1000 rpm.

In Fig. 9, a chart comparing the pressure gain provided by each component is displayed, with the total head delivered by the complete pump. It can be seen that the main components that provide the pressure gain in the pump are the impellers. The degradation seen in both components with the increase of the flow rate is linear, and more noticeable on the second impeller, which has a greater diameter. The remaining components present a more steady behavior in low flow rates and a more abrupt drop in the pressure gain with the increase of the flow rate, contributing to a greater pressure drop on higher values. Both the volute and the diffuser contribute with pressure gain for low flow rates, but eventually act as pressure loss components for high flow rates. Although this behavior might seen unwanted, without these components the loss of pressured observed would be greater due to the increase of heavy recirculations. It can be seen that for none of the flow rates evaluated the pressure gain delivered by the return channel is positive.

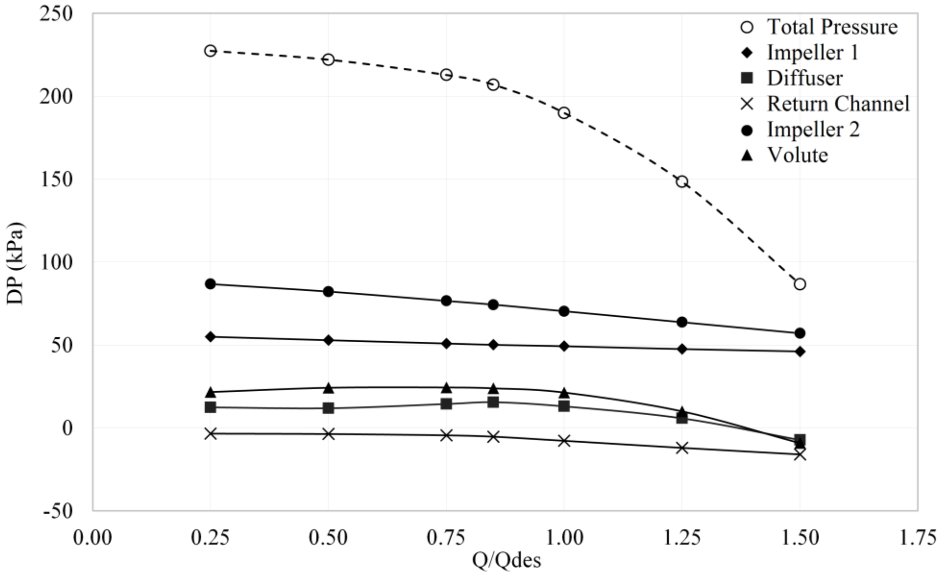

Figure 9. Comparison between pressure gain provide by each pump component for a normalized flow rate of 0.85 and 1150 rpm

Figure 10 shows the contour plots of the turbulence kinectic energy (TKE) in planes located midspan of both the first and second impellers, as well as the volute. This figure shows that the TKE levels in both impellers increase significantly for part-load conditions when compared to operation at the best efficiency point. However, this pattern is more noticeable in the regions near the exit of the channels, location where the interfaces between components induces a turbulent behavior. For BEP situations, high TKE levels remain confined mainly near the suction side of the blades and at the entrance of the second impeller, which might happen mainly due to the recirculations observed in this region.

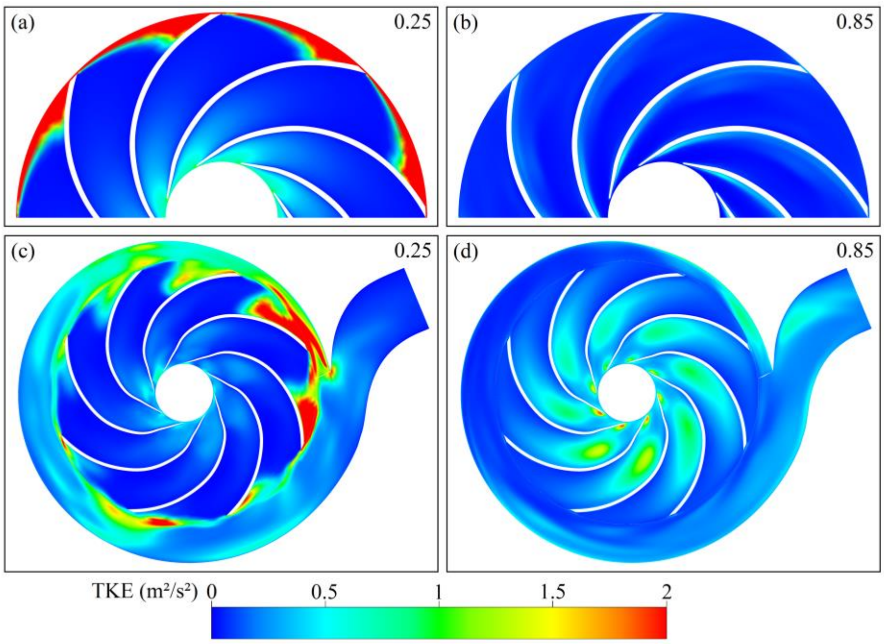

Figure 11. Turbulent Kinect Energy field in a cross section plane through the pump with a normalized flow rate of 0.85 and 1150 rpm.

The results above show some important aspects of the behavior of a centrifugal pump with two impellers. Not just the pump geometry can be sometimes optimized with some simple observations of the flow structure for both part-load and over-load operational conditions, but field engineers can also have good predictions of the system behavior when the operational condition vary due to several uncontrolled reasons.

5. CONCLUSIONS

This work presented a numerical and qualitative study of the flow in a two-stage centrifugal pump. All the components of the pump were modeled and it was possible to see how they contribute individually to the head provided by the pump. More discussions can be added, and more profound analysis can be made. Still, the results obtained show a good agreement between the numerical model and experimental results. Both global and local parameters were analyzed using the results from the numerical simulations. It was shown that for the volute, as much as for the impellers, recirculations might occur at flow rates different than the BEP. For other components, separation flow might occur even near BEP and for higher flow rates. Thus, it is important to understand how the flow behaves in such situations. It was also noticed that the return channel, which is necessary to better organize the flow that enters the second stage, is actually a pressure loss device itself. It was possible to show that both impellers behave in a similar fashion for the same flow rate, even though the second impeller shows small recirculation zones for all flow rates. Also, it was observed that high TKE levels occur on regions near the interfaces between impellers and other components. Finally, it could be noticed that the second impeller provides greater head than the first one. As a continuation of this work, it is intended to enhance the numerical model, for example using other turbulence models. Some important issues also not covered in this paper are subject for future works, such as numerical simulation of multiphase flows and the influence of viscosity on the flow in all pump parts.

6. REFERENCES

Amaral, G.D.L., 2007. Modelagem do Escoamento Monofásico em Bomba Centrífuga Submersa Operando com Fluidos Viscosos. M.Sc. Thesis, Universidade Estadual de Campinas, Campinas, SP, Brazil.

Amaral, G., Estevam, V. and França, F. A., 2009. "On the Influence of Viscosity on ESP Performance". SPE *Production & Operations*, 24(2), pp. 303–311.

ANSYS. ANSYS CFX-Solver Modeling Guide. *ANSYS CFX Release 12.1*, 2009.

Asuaje, M., Farid, B., Kouidri, S., Kenyery, F. and Rey, R., 2005. "Numerical Modelization of the Flow in Centrifugal Pump: Volute Influence in Velocity and Pressure Fields". *International Journal of Rotating Machinery*, 2005(1), pp. 244–255.

Cheah, K. W., Lee, T. S., Winoto, S. H. and Zhao, Z. M., 2007. "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions". *International Journal of Rotating Machinery*, 2007(1), pp: 1–8.

Dunaiski, R., Stel, H., Morales, R.E.M., Sirino, T., 2014. "Numerical Analysis of the Fluid Flow in a Two-Stage Centrifugal Pump". ". In *Proceedings of the 15th Brazilian Congress of Thermal Science and Engineering – ENCIT, 2014.* Belém, Brazil.

Feng, J., Benra, F. K. and Dohmen, H. J., 2007a "Qualitative Comparison between Numerical and Experimental Results of Unsteady Flow in a Radial Diffuser Pump". *Journal of Visualization*, Vol.10, n.4, pp. 349-357

Feng, J., Benra, F. K. and Dohmen, H. J., 2007a "Numerical Investigation on Pressure Fluctuations for Different Configurations of Vaned Diffuser Pumps". *International Journal of Rotating Machinery*, pp. 1–10.

Stel, H., Amaral, G.D.L., Negrão, C.O.R., Chiva, S., Estavam, V., Morales, R.E.M., 2013 "Numerical Analysis of the Fluid Flow in the First Stage of a Two-Stage Centrifugal Pump with a Vaned Diffuser". *Journal of Fluids Engineering*, Vol. 135, pp. 071104-1–071104-9.

7. RESPONSIBILITY NOTICE

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