ENCIT 2018

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0651
NUMERICAL EVALUATION OF THE TWO-PHASE FLOW IN A RADIAL CENTRIFUGAL PUMP

Carlos E. R. S. C. Mendoza - mendoza.cadu@gmail.com
Rafael Dunaiski - rafael.dunaiski@gmail.com
Edgar M. Ofuchi - edgarofuchi@gmail.com
Henrique Stel - stel.henrique@gmail.com
Rigoberto E. M. Morales – rmorales@utfpr.edu.br
Multiphase Flow Research Center (NUEM), Federal University of Technology – Paraná (UTFPR)
Rua Deputado Heitor Alencar Furtado, 5000, Bloco N, CEP 81280-340, Curitiba/PR, Brazil

Abstract. *Centrifugal pumps are often used for the transportation of fluids with more than one phase. However, the presence of a gaseous phase can lead to severe performance degradation. In this work a numerical study using Ansys® CFX® to simulate the two phase flow in a radial centrifugal pump Imbil® Itap 65-330/2 is performed. Water and air were used as work fluids, and an Euler-Euler approach was employed alongside with the MUSIG model, which provides a population balance that accounts the break up and coalescence phenomena of the gas bubbles inside the pump. The results were validated through comparisons with the experimental data obtained by Cubas (2017). The numerical model was capable of predicting the pump degradation due to the presence of gas. Also, the flow patterns and the general behavior of the phases were successfully reproduced.*

Keywords: *centrifugal pump, CFD, MUSIG, two phase flow*

1. INTRODUCTION

Liquid and gas pumping is an off-design condition that affects the performance of centrifugal pumps. The presence of a gaseous phase inside the centrifugal pump degrades the head and flow rate, what is expected when the pump is operating with water only. This typical problem occurs in nuclear and petroleum industries alike. In the nuclear industry the incapacity of the pump to pump gas-liquid mixtures affects the refrigeration of the nuclear reactors causing a problem called “loss of coolant accident”. In the petroleum industry, Electric Submersible Pumps (ESPs) are used as an artificial lift technique to improve the flow rate and pressure of a given well, where gas is also present. The presence of the gas may cause a “surging effect” in the centrifugal pump. It is a severe pressure drop in the head curve causing instabilities on the pump operation.

These problems lead to several investigations over the last decades. Murakami and Minemura (1974a) studied the performance of a centrifugal pump operating under two-phase flow. The authors observed a decrease in the head curve when the gas flow rate is increased. Lea and Bearden (1982) investigated the performance of an ESP operating with air and water. They observed that the pump showed performance degradation when operating with two-phase flow. In addition, the authors tested different impeller geometries, showing that the mixed-flow impeller type had a better performance than the radial type.

Barrios (2007) and Gamboa (2008) conducted a visualization experiment to understand the two-phase flow behavior inside an ESP. They modified one stage in order to get visual access. They observed that the bubble diameters decrease from a stage to another. Also, gas pockets are formed depending on the gas and liquid flow rates. More recently Cubas (2017) presented a visualization and performance study in a radial centrifugal pump. The author identified different flow patterns inside the impeller channels.

Several experimental studies are available in the literature. Yet there is a lack of numerical studies in this area. There are two main approaches in the numerical study: Euler-Lagrange and Euler-Euler. Basically, the first solves the governing equations for each phase separately and tracks down the trajectory of the particles, whereas the second solves the phases together. Murakami and Minemura (1980) studied the two-phase flow using the Euler-Lagrange approach. They observed that the bubbles move faster when they are positioned near the suction side of the blade. In addition, the authors verified that the bubbles move due to a balance between the drag force and the pressure gradient of the liquid phase. Recently, Jimenez (2016) conducted a similar study in a radial type centrifugal pump. The author observed that the liquid flow rate, bubble diameter and rotation speed directly affect the bubble movement inside the impeller. Using the Euler-Euler approach, Caridad et al (2008) investigated the performance of an impeller of an ESP operating with water and air. The authors considered a single bubble diameter in their simulations and observed that increasing the bubble diameter from 0.1 mm to 0.5 mm the performance drops severely. Zhu and Zang (2017) conducted a numerical study using the Euler-Euler approach in an ESP with several stages. The authors compared their results with experimental data from Salehi (2012). They observed that the performance of the ESP is dependent on the bubble diameter and, at each experimental point, the authors found the appropriate bubble diameter that would reproduce the experimental result.

Despite the variety of works presented, there is still a lack of studies on numerical works. For example, in the Euler-Euler approach, the bubble diameter is kept constant, thus reducing its applicability to the real situation where break up and coalescence of the bubbles occur. In this sense, this work presents a numerical study of the air-water flow inside a centrifugal pump using an Euler-Euler approach with a population balance that accounts the break-up and coalescence phenomena inside a centrifugal pump.

2. GEOMETRY

The first stage of Imbil® Itap 65-330/2 is used in the numerical simulations. The first stage is composed by a rotating impeller with eight blades and a vanned diffuser with twelve blades. Figure 1 shows the geometry used in this work. Due to the periodicity of the stage, only a quarter of the geometry can be simulated, thus reducing the computational effort. At the best efficiency point (BEP) in the 1150 rpm rotational speed, the pump is designed to deliver a pressure boost of 69.3 kPa and a flow rate of 36.6 m³/h.

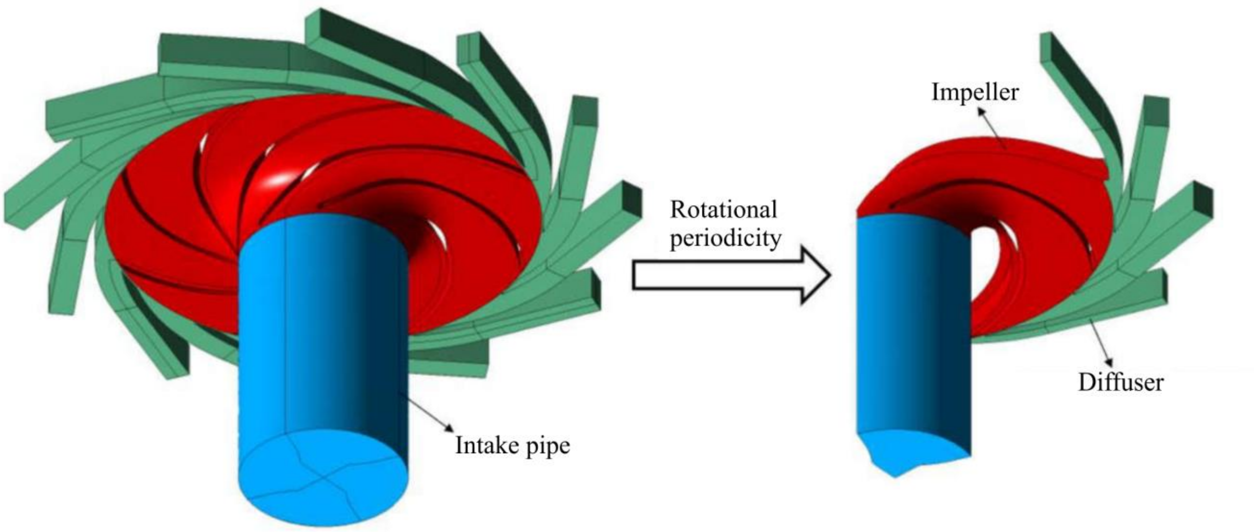

Figure 1 – Geometry of the fluid domains

3. NUMERICAL MODEL

3.1 Governing equations

The two-phase flow is modeled using the Euler-Euler model (Ishii and Hibiki, 2011). The governing equations that must be solved are the mass (1) and momentum (2) equations for each phase, as presented below.

$$\frac{\partial \alpha_k \rho_k^X}{\partial t} + \nabla \cdot \left(\alpha_k \rho_k^X \vec{V}_k^{X\rho} \right) = 0 \tag{1}$$

$$\frac{\partial \alpha_k \rho_k^X \vec{V}_k^{X\rho}}{\partial t} + \nabla \cdot \left(\alpha_k \rho_k^X \vec{V}_k^{X\rho} \vec{V}_k^{X\rho} \right) = -\nabla \left(\alpha_k P_k^X \right) + \nabla \cdot \left(\alpha_k \bar{\bar{T}}_k^X \right) + \alpha_k \rho_k^X \vec{g}^{X\rho} + \vec{M}_{ki} \tag{2}$$

α is the volume fraction of the phase k (liquid or gas), ρ is the density, $\vec{V}$ is velocity, P is pressure, g is the gravitational acceleration, $\bar{\bar{T}}$ is the diffusivity tensor, which takes the molecular viscosity and the turbulent viscosity into account. This latter is modeled using the two-equation k-ω Shear Stress Transport. $\vec{M}_{ki}$ accounts for the momentum exchange at the interface between the phases. The superscript X and $X\rho$ represent the phase average and the Favre averaging of the variable, respectively. The problem is considered isothermal, hence one must not solve the energy equation.

Among the different terms in the momentum equation the term $\vec{M}_{ki}$ is the result of the interactions between the phases and it is modeled as the sum of different forces acting on the bubble, which are:

$$\vec{M}_{ki} = -\alpha_k \frac{\vec{F}_D + \vec{F}_{VM} + \vec{F}_L + \vec{F}_{TD} + \vec{F}_{WL}}{\forall_p} \tag{3}$$

$\vec{F}_D$, $\vec{F}_{VM}$, $\vec{F}_L$, $\vec{F}_{TD}$ and $\vec{F}_{WL}$ are the drag, virtual mass, lift, turbulent dispersion and wall lubrication forces, respectively. $\forall_p$ is the volume of a given bubble.

The traditional bubbly flow is modeled using a single mean bubble diameter. However, the bubble diameter has a significant effect inside a centrifugal pump, as shown by Jimenez (2016). Thus, in this work, a population balance model known as MUSIG (ANSYS, 2012) is utilized to model a range of bubble diameters, which takes the effects of brake up and coalescence of the bubbles into account.

3.2 Numerical approach

ANSYS® CFX™ is used to solve the governing equations through an Element-based Finite Volume Method (EbFVM). A high resolution scheme is utilized to discretize the advection term. The pressure-velocity coupling is done through an algorithm proposed by Rie and Chow (1983) that considers the co-located grid used in this problem. ANSYS® CFX™ uses a multi-block technique that considers each component of the pump as a single domain, and these domains are connected through interfaces. This approach is commonly used in turbomachinery problems.

A non-structured tetrahedral mesh is used to discretize the pump domain. Figure 2 presents the mesh utilized. A refinement of the mesh close to the wall in the impeller and diffuser was done to improve the calculations near to the wall.

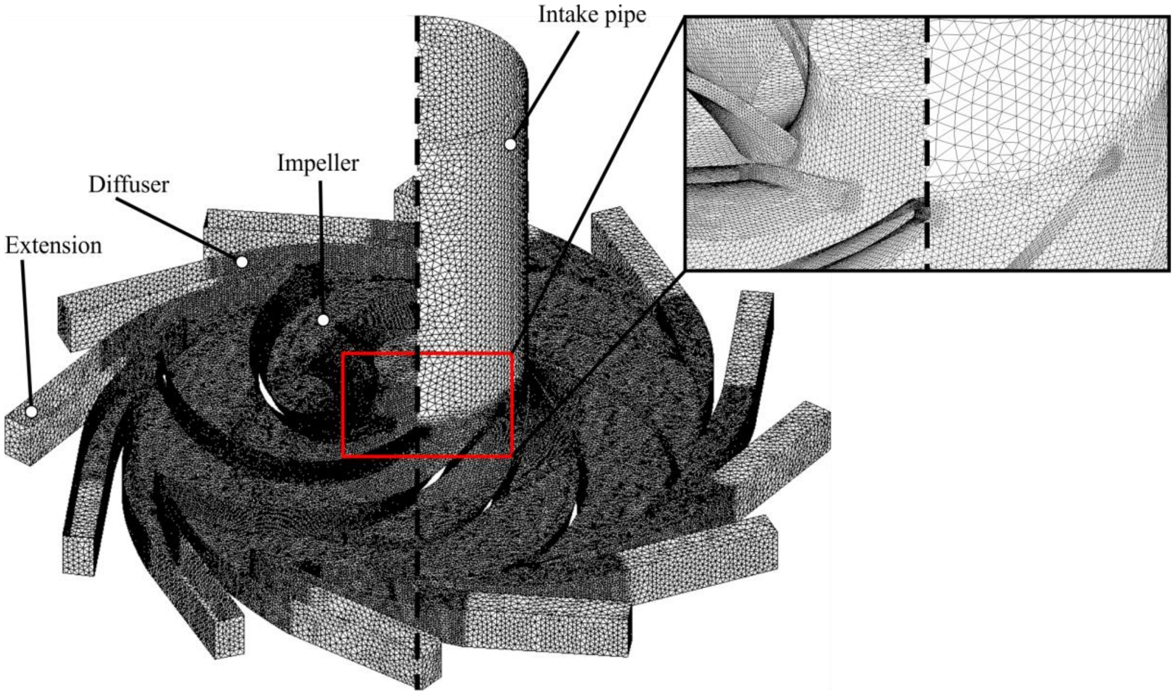

Figure 2 – Computational mesh used for the numerical model

The boundary conditions applied to the problem are shown in Figure 3. A non-slip condition is considered on all walls. A mass flow rate is imposed at the inlet, for both liquid an gas phases. For the liquid phase the mass flow rate is distributed with a velocity profile, while for the gas a constant profile is used. The volume fraction of each phase is also given at the inlet, as well as the bubble diameter. For all cases a constant bubble diameter is set (d_B = 3.5 mm), although it is possible to input a distribution of bubble diameters. At the outlet, an opening condition is used, which allows the entrainment as well as the exit of the phases. This brings more stability to the numerical solution.

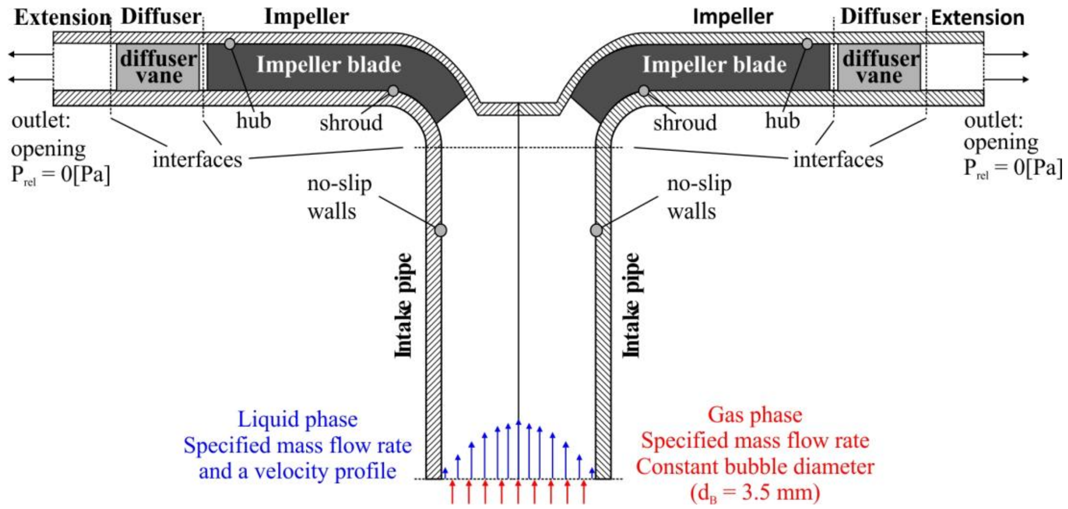

Figure 3 – Boundary conditions applied to the numerical model

4. RESULTS

The numerical results herein obtained were validated with comparisons against the experimental values obtained by Cubas (2017) for the same pump herein studied.

Figure 4 shows a comparison between the pressure gain of the first impeller and diffuser of the pump, normalized with the pressure gain at the design flow rate, for numerical and experimental values at a rotational speed of 500 rpm and at a fixed air mass flow rate of 1.4 kg/h. Also, the pressure gain for single phase flow is plotted.

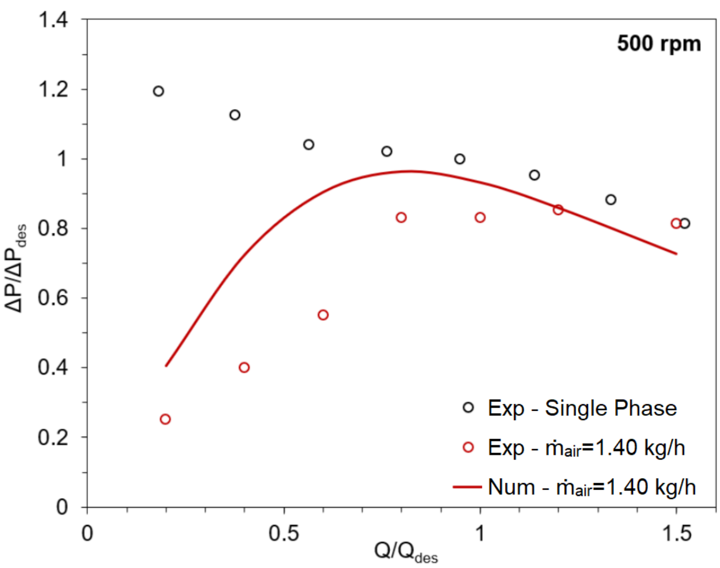

Figure 4 – Comparison between experimental and numerical results for 500 rpm and air mass flow rate of 1.40 kg/h

It can be observed that the numerical values follow the experimental ones quite well, with small differences at higher flow rates. However, for flow rates smaller than $0.75Q_{des}$ the numerical model tends to overestimate the experimental results. Still, despite the differences observed between the experimental and numerical data sets, the model is capable of reproducing the surging, agreeing well with the experimental flow rate where the pump starts showing considerable amounts of degradation due to the presence of the gas phase.

In order to evaluate the numerical model, and assert its capabilities, flow fields obtained numerically are compared with images acquired from the experiment. Figure 5 shows the nomenclature of the points henceforth analyzed.

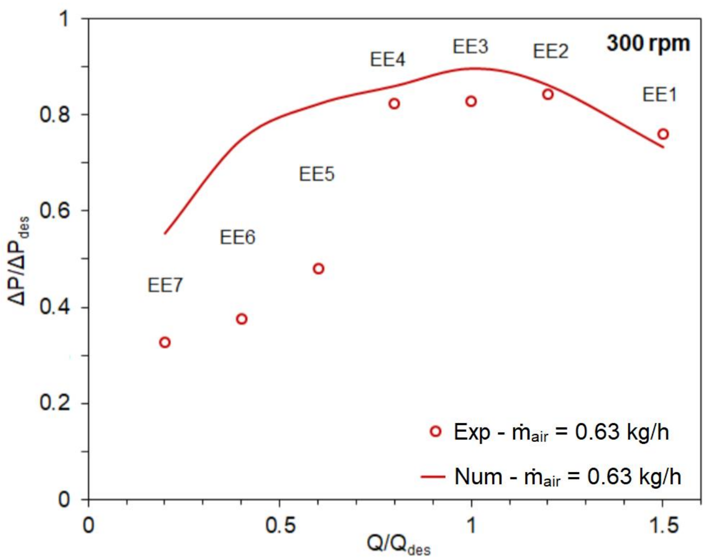

Figure 5 – Nomenclature for the points studied. Obtained for a condition of 300 rpm and air mass flow rate of 0.63 kg/h

The numerical fields presented here were obtained in a plane midspan between the hub and the shroud of the impeller. The right half of Fig 6 presents the gas volumetric fraction field, alongside with a liquid phase velocity field and the velocity vectors for both phases. The white circles on the gas fraction field represent the diameter of the gas bubbles in that location. The left half is a picture of the flow in the same simulated conditions, as obtained by Cubas (2017). Both cases represented have high liquid flow rates, hence less gas is expected to accumulate inside the pump for these conditions.

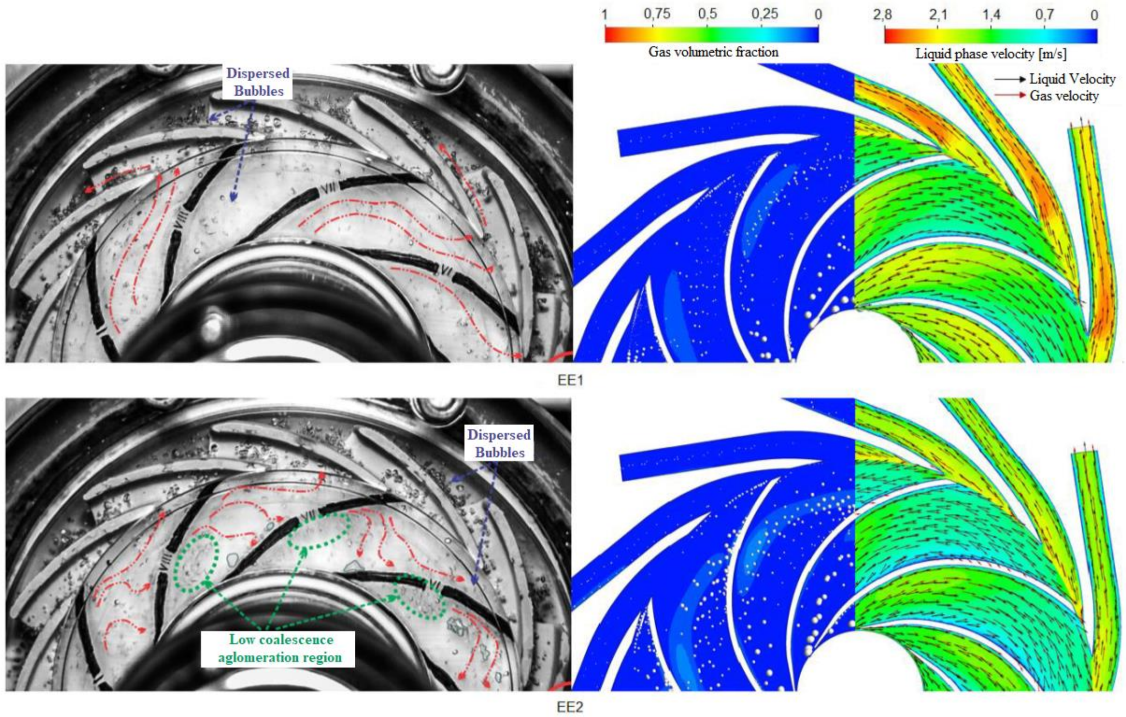

Figure 6 – Comparison between the numerical and experimental results for the EE1 and EE2 points

For the EE1 case, it can be seen that the gas particles are initially located at the pressure face of the impeller blade. As the gas goes through the channel the gas is slightly displaced to the suction face of the blade, before easily exits the impeller. Drag force is predominant on this case, since the velocities of the liquid and gas phases are very similar. Also, the movement of the gas bubbles is very similar to the ones seen in the experimental images. In the EE2 case, it can be seen that there is a greater discrepancy between the numerical and experimental fields. While the experimental case presents some points where coalescence starts to appear, the numerical field is still very similar to the one observed at a higher flow rate. However, the general deviation trend between the gas and liquid phase fields is already represented in the numerical case.

Figure 7 represents the flow field compared with experimental results, but now for a relatively small flow rate. This time, the mean diameter of the gas bubbles dispersed in the liquid phase is depicted. Also, for a more visual comparison, white spheres represent the diameter of the bubbles. The white areas represent gas pockets, where the numerical model identified a presence of gas of nearly 100%.

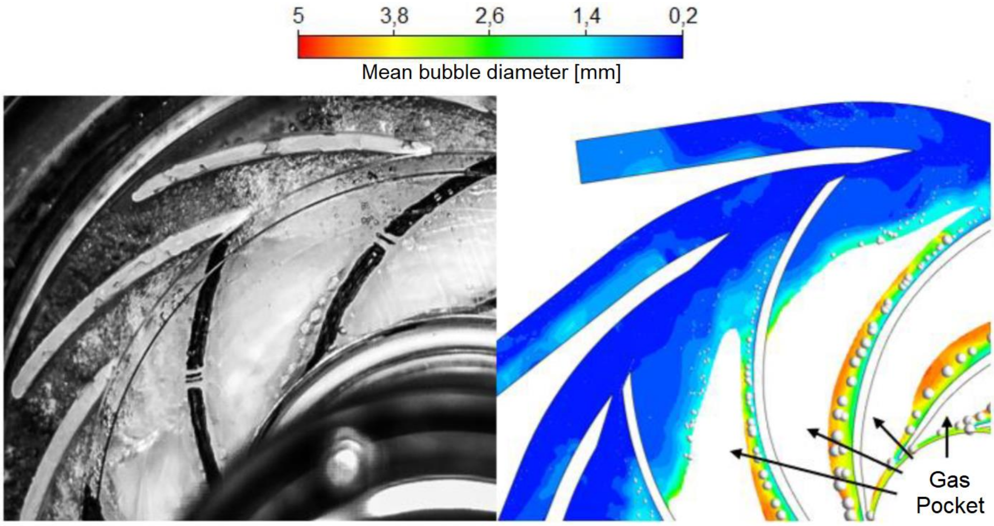

Figure 7 – Comparison between the numerical and experimental results for the EE7 point

This point (EE7) is in a flow rate where the surging has already occurred. The pressure provided by the pump has started to drop, due to the presence of gas pockets. The gas pockets occupy most of the channel and it is adjacent to the suction face of the blade. A liquid film is formed near to the pressure side, where bubbles of considerable size are located. This behavior can also be observed in the experimental image, and the bubbles have approximately the same size for both cases, when compared to the ones at the exit of the channel. This indicates that the MUSIG model used to model the bubble diameter is accurate enough to reproduce the experimental setup.

Another analysis that can be performed via CFD − and a very difficult one to be performed using experimental methods − is the influence of the several forces that can act on a bubble and influence its path inside the impeller. Those forces are of fundamental importance understanding how the phases interact and how those interactions affect the performance of the pump. This study is depicted in Fig. 8, which represents a near-BEP situation. The forces are evaluated over a streamline.

One can observe that the most predominant forces in the flow are the drag and the pressure gradient. The sum of the forces pushes the bubbles towards the pressure face of the blades as seen in previous flow fields. This study is also relevant when it comes to providing information for CFD users of what forces are more significant, and must therefore be modelled with extra care.

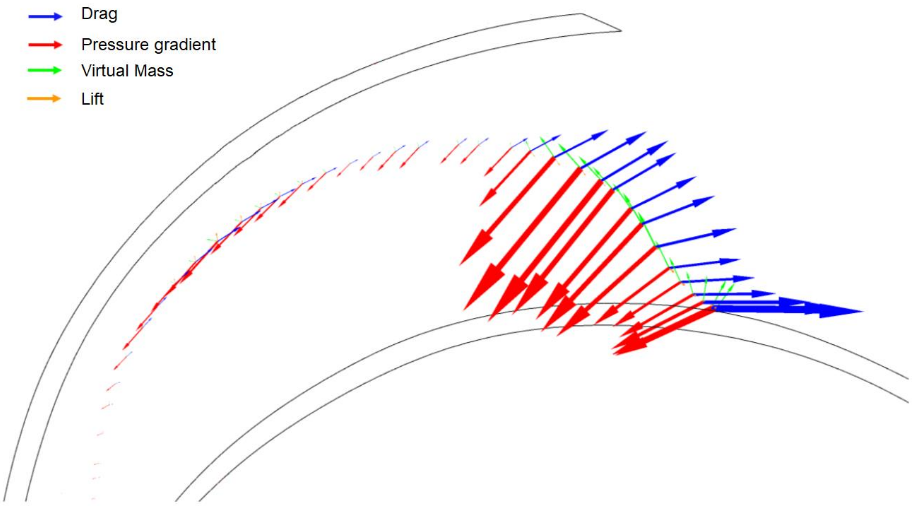

Figure 8 – Forces acting over a gas streamline for 300 rpm and Q/Q_{des}=0.8

5. CONCLUSIONS

In this work a numerical study of the two-phase flow in a radial centrifugal pump was carried out. Results were compared and validated with experimental results. A methodology considering coalescence and the breakup of gas bubbles dispersed in the continuous liquid phase was applied, as well as a variation of size between them. Bubble size was found to be of fundamental importance in the correct characterization of the flow, especially because one of the key forces involved in the flow (drag) is highly dependent on their diameter. Since it is very difficult to characterize bubble diameter throughout the domain beforehand, having a model that takes such effects into account might be regarded as an advantage. Although a comparison between the numerical and experimental results showed some differences, a qualitative analysis showed that the numerical model is capable of reproducing complex phenomena and providing useful and valuable insight about the flow. A more detailed analysis of the forces acting on the bubbles may be developed in a future occasion, thus leading to more reliable and efficient CFD models.

6. REFERENCES

ANSYS, 2012. ANSYSs Academic Research, Release 14.5, Help System, CFX Documentation. ANSYS,Inc..

Barrios, L., 2007. Visualization and Modeling of Multiphase Performance inside an Electrical Submersible Pump. Ph.D. thesis, The University of Tulsa, Oklahoma.

Caridad, J., Asuaje M., Kenyery F., Tremante A. and Aguillón O., 2008. "Characterization of a Centrifugal Pump Impeller Under Two-Phase Flow Conditions". *Journal of Petroleum Science and Engineering*, Vol. 63, p. 18-22

Cubas, J. M. C., 2017. *Estudo Experimental do Escoamento Bifásico Ar-Água em uma Bomba Centrífuga Radial*. MSc dissertation, UTFPR, Curitiba.

Gamboa, J., 2008. *Prediction of the Transition in Two-Phase Performance of an Electrical Submersible Pump*. Ph.D. thesis, The University of Tulsa, Oklahoma.

Ishii, M. and Hibiki, T., 2011. *Thermo-Fluid Dynamics of Two-Phase Flow*. Springer, New York, 2nd edition.

Jiménez, F. A., 2016. *Análise Numérica da Dinâmica de uma Bolha Isolada de Gás no Rotor de uma Bomba Centrífuga*. MSc dissertation, UTFPR, Curitiba.

Lea, J. F. and Bearden, J. L., 1982. "Effects of Gaseous Fluids on Submersible Pump Performance". *Journal of Petroleum Technology*, 2922-2930.

Murakami, M. A. and Minemura, K., 1974a. " Effects of Entrained Air on the Performance of a Centrifugal Pump". *Bulletin of the JSME*. p. 1047-1055.

Minemura, K. and Murakami, M. A., 1980. "Theoretical Study on Air Bubble Motion in a Centrifugal Pump impeller. *ASME Journal of Fluids Engineering*, Vol. 102, p. 446-453.

Rhie,C.M. and Chow, W. L., 1983. "Numerical study of the turbulent flow past an air foil with trailing edge separation". *AIAA Journal*. Vol. 21, p. 1525–1532.

Salehi, E., 2012. *Experimental Studies on the Effect of Number of Stages on Electrical Submersible Pump Two-Phase Flow Performance*. Ms.C. dissertation, The University of Tulsa, Oklahoma.

Zhu, J. and Zhang, H., 2017. "Numerical Study on Electrical-Submersible-Pump Two-Phase Performance and Bubble-Size Modeling". *SPE Production & Operations*.

Lee, Y.B., 2003. *Studies on the growth of the frost layer based on heat and mass transfer through porous media*. Ph.D. thesis, Seoul National University, Seoul.

7. RESPONSIBILITY NOTICE

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