Challenges in the Numerical Analysis of Centrifugal Pumps

Energetic, Cavitation, and Dynamic Characteristics

Andrej Lipej
Faculty of Mechanical Engineering
University of Novo Mesto
Novo Mesto, Slovenia
andrej.lipej@fs-unm.si

Abstract-Pumps are energetic devices that are present in a wide range of industries. Their installed power varies from a few tens of watts up to hundreds of megawatts. As the energy consumption is connected with global warming and climate change, the development of pumps that have good characteristics is very important. Theoretical and experimental methods have long been utilized in the development of pumps, but recently much research has been carried out with the help of Computational Fluid Dynamics (CFD) analysis. Certain numerical analyses can be of high quality and accurate, but special operational phenomena still occur, where special attention is required when performing analysis. This category includes operating regimes outside the Best Efficiency Point (BEP), the formation of inlet recirculation, cavitation, and the influence of wall roughness on all the above characteristics. The current paper presents the main problems that developers face when numerically analyzing the above-mentioned special operating regimes. Both numerical and experimental results are considered.

Keywords-pump; computational fluid dynamics; operation characteristics; cavitation

I. INTRODUCTION

Pumps are used in all branches of industry. Depending on the different requirements for the basic characteristics of the head (H) and the flow rate (Q), piston pumps [1] or centrifugal pumps [2] are used. Centrifugal pumps are developed to operate at a calculation point, but in reality, they operate over a wider range. The service life of a pump and the probability of various damages depend on its operating range [2]. Pumps have a theoretically defined recommended operating range, but it is not always feasible for this condition to be met. From the efficiency point of view, the pump is recommended to operate most of the time near the Best Efficiency Point (BEP). For accurate knowledge of the kinematics of the flow conditions in the pumps, the best option is to use Computational Fluid Dynamics (CFD). The use of numerical methods is an indispensable tool for the development of pumps. Numerical simulations of flow conditions are more or less accurate, depending on the operating range. At the same time, high accuracy of calculations and analysis of various characteristics in the widest possible range of operation must be achieved during the development. Stationary calculations are sufficient for some characteristics, but in some special cases, it is necessary to use non-stationary calculations. The current article aims to present various problems that arise when using numerical methods in the development phase of centrifugal pumps and reversible pump turbines. The problems are mainly related to instabilities in pump operation and numerical instabilities. Comparisons between the measured results and the CFD analysis results are needed to better understand the problems and validate non-standard numerical procedures. The results of model measurements for most presented cases in the paper were done based on international standards [3].

In some cases, it was necessary to analyze specific properties that are rarely discussed in the literature. One such example is the consideration of wall roughness, which may be due to different influences in the operation of the pumps [4]. To the best of our knowledge, there is no literature relevant to the link between roughness and the use of a suitable turbulent model with a proper computational grid. In such cases, certain studies need to be performed for fairly fundamental properties, such as boundary layer flow analysis [5]. The influence of wall roughness on pump operation is presented in [6]. The research in the paper was focused on issues where it is necessary to perform non-usual procedures in numerical simulations. The first example is related to the instability of H (Q) characteristics of different types of pumps. The stability of pump characteristic is the most important and complex design criterion. Some research has already been done in this area. A detailed description of the numerical simulation of a reversible pump turbine in pump mode is presented in [7]. A similar topic is addressed in [8], where more focus is given on the analysis of guide vanes geometry. There have been quite a few articles dealing with experimental research [9]. Another example is associated with unstable operation as a consequence of inlet recirculation. A numerical simulation of the causes and consequences of inlet recirculation is presented in [10]. Inlet recirculation has various consequences, including cavitation [11] and rotating stall [12], which can occur regardless of favorable standard cavitation characteristics. Input recirculation analysis in most cases requires the selection of a suitable turbulent model for nonstationary calculations, which is presented in [13].

Corresponding author: Andrej Lipej

The numerical analysis results presented in this paper are the result of the author's many years of research in the field of centrifugal pump development.

II. INSTABILITY OF PUMP CHARACTERISTICS

Centrifugal pumps operate under different operating conditions, depending on the magnitude of the flow Q and the head H. The relationship of the two variables is given by a function known as the H-Q characteristic. Theoretically, this function decreases with increasing flow. The shape of the function is influenced by many operating parameters, so it is not trivial to identify possible instabilities in the operation of the pump. By measurements on the model or prototype, it is possible to determine very precisely the shape of the above characteristic, but without detailed measurements of the kinematics, it is not possible to determine the possible causes of unstable areas during pump operation. By using CFD it is possible to determine the exact characteristic and also predict the causes for the formation of the so-called unstable regions in the H-Q characteristic. Since the flow conditions in the pumps are very demanding, mainly due to the diffuser effect in the fluid flow through the impeller, it is necessary to be very careful when preparing computational grids, choosing a turbulent model, determining boundary conditions, and deciding on stationary or nonstationary analysis. All numerical analyses are based on the solution of Reynolds-Averaged Navier-Stokes system of equations, i.e. continuity equation and momentum equation. Since a region of instability in the H-Q characteristic occurs in areas where the flows are smaller than the QBEP, the flow conditions in this region are associated with several different causes of instability. The inlet angles and consequently the velocity triangles at the inlet to the pump indicate a mismatch of kinematic and geometric angles, which affects the flow conditions inside the impeller and also in the area in front of the impeller.

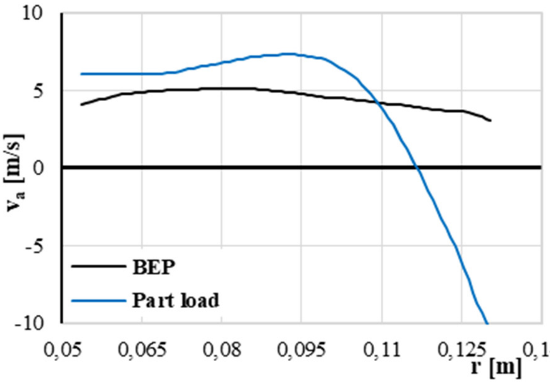

Fig. 1. Axial velocity component at the inlet of the impeller.

In the front of the impeller, the so-called inlet recirculation can occur, which continues downstream through the runner. The recirculation area can be detected accurately enough if quality computational grids are used. A sufficiently dense computational grid must be considered, one that provides orthogonality and all recommendations regarding the individual relationships between sides within elements and regarding the size of the adjacent elements.

Since the above phenomena are always associated with temporal changes, it is necessary to determine whether the stationary calculation is sufficiently accurate or whether a much more time-consuming non-stationary calculation needs to be used. In the presented case, it was found that stationary calculations are not the most appropriate, because they cannot describe all the effects of operating conditions on flow conditions. Cavitation and inlet recirculation of the centrifugal pumps have a significant impact on the operation and the service life of the pump. Partial flow recirculation directly affects the reliability of the pump and limits the operating range. Damage to the pressure side of the inlet blade rotor due to cavitation due to recirculation may occur. The hydraulic design of the pump should prevent many of the negative effects of cavitation due to recirculation in the pumps. The basic criterion for free-range recirculation is the suction Specific Speed (SS) of the pump.

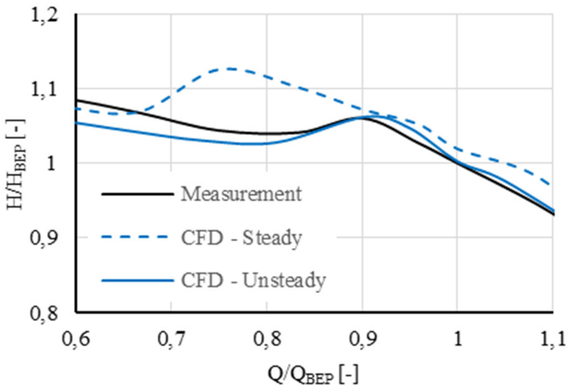

Fig. 2. Hump zone analysis. Measurement, steady and unsteady state simulation results.

Figure 2 shows the measurement and numerical analysis results in the case where the pump characteristic has a well-defined area of instability. The results of numerical analysis differ considerably if we perform a stationary or nonstationary calculation. A comparison of the results shows that in this case, it is very important to use a non-stationary calculation that predicts the characteristics of the pump with sufficient accuracy, which is also confirmed by the comparison with the measurement results. In the case of efficiency analysis, there are no such pronounced differences between stationary and non-stationary calculations, but a slightly better result can still be observed compared to the measurement results if a non-stationary calculation is used (Figure 3). Since the numerical and experimental results agree quite well in stationary calculations, such results can mislead us and it is possible to obtain a bad result for the H(Q) characteristic. If the kinematics of the flow at the inlet to the impeller is monitored in parallel, it can be roughly determined when non-stationary calculations should be started.

The turbulent model k-ω SST is recommended for stationary calculations and the Scale-Adaptive Simulation (SAS-SST) for non-stationary analysis.

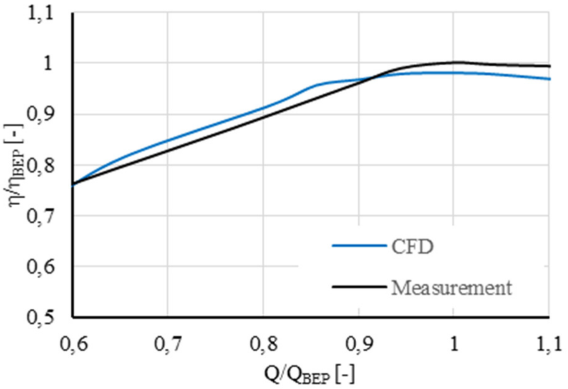

Fig. 3. Efficiency comparison.

III. NUMERICAL INSTABILITY

The numerical simulations according to different operating conditions can basically be divided into 3 main areas. The first area is near the BEP. The two others are at part load and at full load. Large differences can be observed between these 3 results if continuous monitoring of the efficiency convergence during the calculation is running. Due to the favorable flow conditions at the impeller inlet, the convergence is very stable in the optimal mode, which can be seen well in the upper curve in Figure 4. The other 2 curves represent the operating range at partial and at maximum flows. At partial flows, a small fluctuation in the efficiency of about 2% is observed and at maximum flows the instability of convergence is more than ± 15%. In particular, this must be taken into account when it is necessary to accurately predict numerically the efficiency characteristic in the wider operating range of the pump. By simultaneously monitoring the convergence of different variables, it can be determined when it is necessary to use non-stationary instead of stationary calculation. In any case, this is not always the same for all types and specific rotating pump speeds, so continuous monitoring can greatly simplify and shorten the development time of a new pump. From these results, it can be concluded that standard convergence monitoring with residual monitoring can lead to erroneous conclusions. Therefore, it is advisable to always monitor the convergence of individual variables that are important for the calculation.

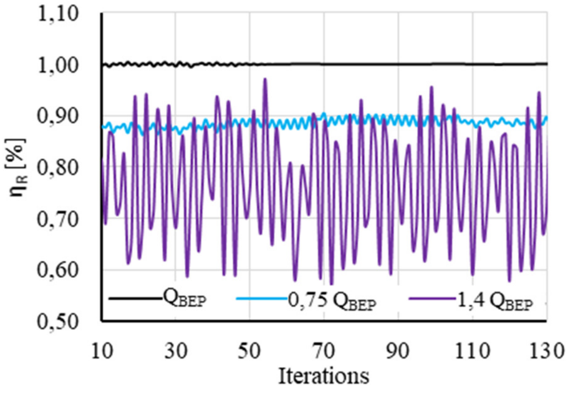

Fig. 4. Efficiency comparison.

IV. INLET RECIRCULATION IN PUMPS

Inlet recirculation (Figure 5) in the pumps can cause many side effects with harmful consequences such as vibration, noise, and erosion damage. The probability of damage depends on the specific rotational speed, cavitation characteristics, and the suction energy level at the inlet.

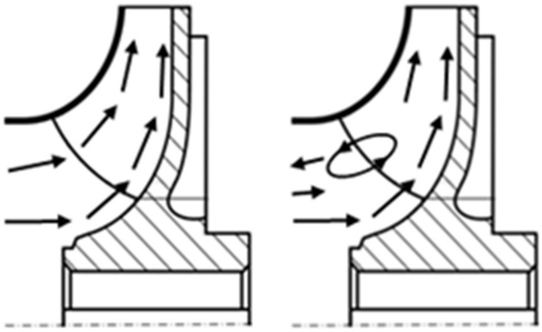

Fig. 5. Inlet recirculation.

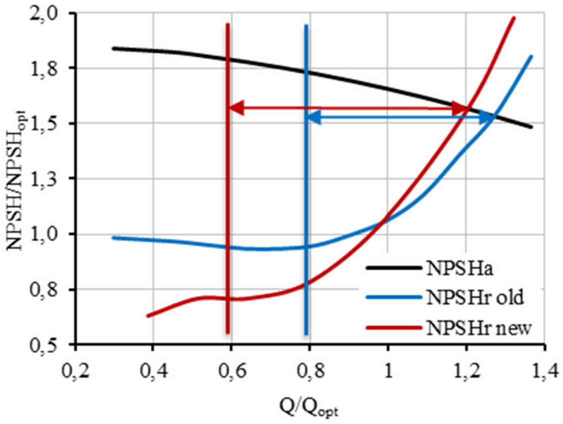

Fig. 6. NPSH characteristics of pump with safe range of operation.

By improving the characteristics associated with the inlet recirculation, the area where the pump operates without the risk of cavitation is increased. The maximum flow that determines non-cavitation operation is determined where the $NPSH_a$ and $NPSH_r$ curves intersect. However, by reducing the inlet vortices, it can be determined what the minimum flow must be such that there is no danger of cavitation due to the inlet recirculation. The results of the safe area of the pump, in which there will be no cavitation are presented in Figure 6. With the optimized geometry, in the case shown above, the NPSH characteristic is slightly worse, but the area at part load is significantly increased.

The flow conditions at the inlet vary greatly depending on the operating regime of the pump. Figure 7 shows two situations - without and with inlet recirculation. Under unfavorable conditions at the inlet, this is also transferred to the space between impeller blades, where various vortices are observed (Figure 8). A novelty in the numerical analysis of the inlet recirculation is the cognition that in terms of cavitation characteristics it is not enough to analyze the NPSH parameters but the parameters related to the inlet recirculation must be also considered. Only in this way it is possible to obtain a correct assumption about the interval of the operating area without cavitation.

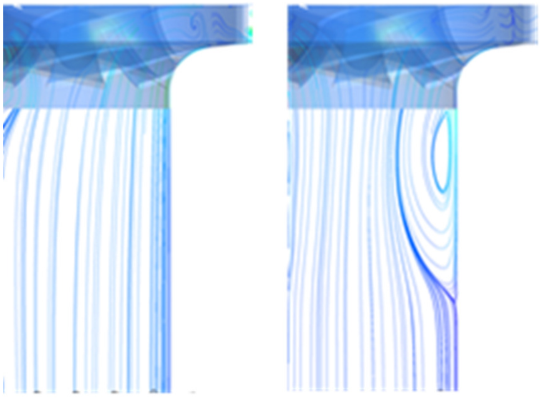

Fig. 7. Flow conditions in front of impeller – without (left) and with inlet recirculation (right).

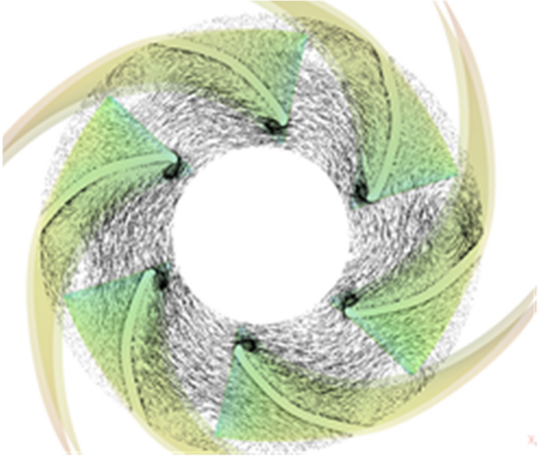

Fig. 8. Flow inside the impeller – vortices.

V. WALL ROUGHNESS

The pumps are made of different materials and with different technological processes. As a result, wetted surfaces have different roughness values. In most cases, this fact does not greatly affect the performance characteristics, but there are cases where the surface roughness of the rotor blades can affect the performance characteristics. Since numerical flow analysis (CFD) has been widely used in the development of pumps in recent times, it is necessary to determine how accurately it is possible to numerically analyze the influence of wall roughness on the energy and cavitation characteristics of pumps. The first step in the numerical analysis of the flow along rough surfaces is to determine which turbulent model enables satisfactory results and how the characteristics of the computational grids affect these results (Figure 9). Figure 9 shows the difference between the theoretical and the numerical results when using different sizes of the non-dimensional parameter y^+. It is known that the influence of roughness is closely related to the modeling of the fluid in the boundary layer and therefore such a preliminary analysis is very welcome. From the obtained results, it was seen that the values of y+ significantly influence the final calculation results. To analyze the use of appropriate parameters of computational grids, a numerical analysis of 2 simple examples was performed, where it was possible to compare the numerical with the theoretical results. The need to consider roughness in numerical simulations can be seen in Figure 10, where a comparison of numerical and experimental results for two smooth and rough walls is shown. These findings were used in the analysis of various pumps, where it was confirmed that even with numerical analyzes, where surface roughness is considered, it is possible to obtain useful results in the analysis of energy and cavitation characteristics.

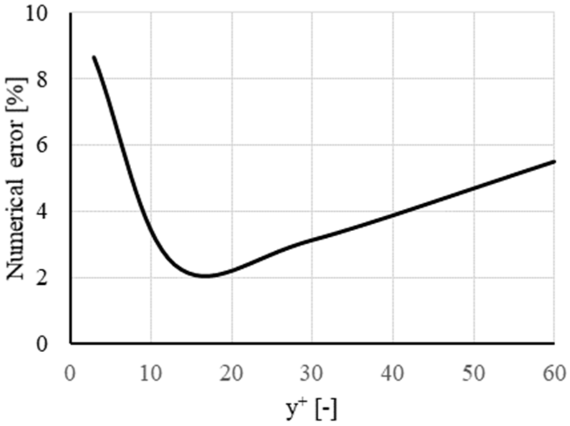

Fig. 9. Influence of y^+ on the numerical analysis accuracy.

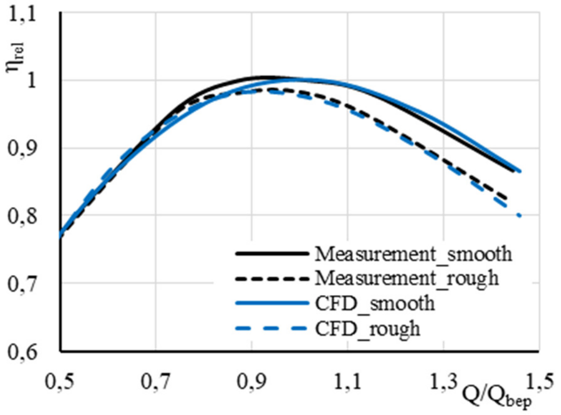

Fig. 10. Comparison of numerical and experimental results – smooth walls and rough walls.

In numerical analysis, the so-called sand grain equivalent should be considered instead of the classical roughness data. Another problem is the appropriate definition of computational networks in the roughness range. For the appropriate selection of the y+ parameter, a method based on a comparison with the theoretical results was presented, which differs from the classical comparison between measurements and numerical analysis.

VI. CAVITATION

Cavitation is a physical phenomenon that occurs during the operation of various energy and working machines. This can be a particular problem with centrifugal pumps, as cavitation characteristics are very important when choosing a suitable machine in a particular system. Since cavitation also affects many other characteristics, such as operation stability, maintenance costs, and the service life of pumps, it is necessary to pay close attention to this phenomenon during the development of new machines. As engineers increasingly use CFD analysis, it is necessary to know the main pitfalls that arise in the numerical predictions of all the operating characteristics of centrifugal pumps. Qualitative numerical prediction of cavitation is performed almost routinely, which means that numerical simulations can predict the position, the size of the area, and the intensity of cavitation, but it is not always possible to accurately obtain quantitative results of certain characteristics as a result of cavitation. Determining the critical cavitation coefficient can be an even more challenging task. Figures 11 and 12 show the location and intensity of the cavitation cloud, for two different values of suction height. It can be observed that the sizes of the steam clouds vary, as at higher suction heights (Figure 12) the risk of cavitation formation is higher. In both images, it can also be observed that the cavitation intensity varies from individual impeller blades, although the geometry of the blades is identical.

Fig. 11. Cavitation inside the impeller – low suction head.

This means that there are no uniform conditions at the pump inlet or that the situation changes greatly over time. The consequence of the above-mentioned observations is the fact that it is not always possible to obtain concretely useful results using stationary calculations, but we have to use non-stationary calculations. These require significantly longer computational times, thus extending the entire development cycle. The influence of wall roughness on cavitation characteristics was also considered during the research. When predicting the efficiency by considering roughness, the obtained numerical results were quite satisfactory. A larger problem arises in the numerical simulation of cavitation considering the roughness of the walls. Figure 13 shows the results of numerical analysis of cavitation at 3 different suction heights for smooth and rough walls.

Fig. 12. Cavitation inside the impeller – high suction head.

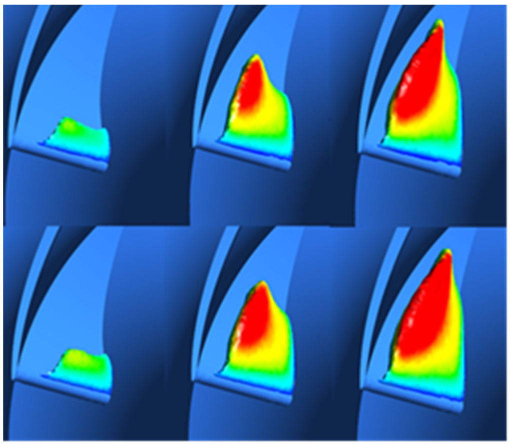

Fig. 13. Influence of roughness on the cavitation intensity – smooth wall (top), rough wall (bottom).

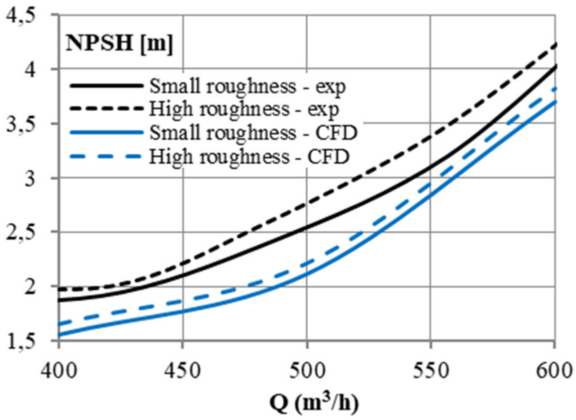

Fig. 14. Comparison of NPSH characteristics for two different roughness – numerical analysis and experimental results.

In the case of rough walls, a slightly larger size of the cavitation cloud is observed, which is not observed in all similar situations. In some cases, the ground plan size of the cloud remained the same or even decreased, but the thickness of the cavitation cloud increased. In the Figure, the color scale indicates the thickness of the cavitation cloud. Roughness also affects the NPSH characteristic. Figure 14 shows a comparison between the measurement results and the numerical results. The absolute results do not match very well, but the right trends of considering roughness in the numerical calculations are observed on the graph. In linking cavitation and roughness, it was found that it is important to consider the thickness of the cavitation area and not only the size of the cavitation clouds.

VII. CONCLUSION

The pump is one of the most important working machines in the industry. Pumps are major energy consumers and therefore special attention needs to be paid to their development phase. The pumps should operate to as high as possible efficiency without significant energy losses. They must also operate reliably with the lowest possible operating cost and their service life must be reasonably long. As numerical methods are increasingly used in the development of pumps, development engineers are facing various problems. The current article presents some of the problems that arise in the development phase when CFD analysis is used in the design. For accurate instability analysis in pump characteristics, it is recommended to analyze the kinematics of the flow at the pump inlet. When the first input vortices are observed, nonstationary calculations must be used. The k-ω SST turbulent model is recommended for stationary calculations and the SAS-SST for non-stationary analysis. Where operating conditions are far from optimum, numerical instabilities occur. It can be concluded that standard convergence monitoring with residual monitoring cannot lead to the right conclusions. Therefore, it is recommended to monitor the convergence of several variables that are important for the qualitative result in numerical analysis.

The often-overlooked consideration of wall roughness in numerical calculations has proven to be an important factor to consider at the development stage if the properties of the materials and the surface treatment technology are known. Detailed analysis revealed which computational grids are suitable for the numerical simulation of rough surfaces. Instead of the classical roughness parameters, the so-called sand grain equivalent should be considered in CFD analysis. For the appropriate selection of the y+ parameter, a method based on the comparison with the theoretical results should be used, which differs from the classical comparison between the experimental and the numerical results. The results show that the roughness affects the efficiency and the NPSH characteristics of the pumps. The surface roughness increases the wall shear stress in the turbulent boundary layer. Sometimes the numerical results match very well with the experimental ones but in many cases the difference is still relatively big. The comparison of the numerical and experimental results shows a slight improvement of the results if wall roughness is considered. In linking cavitation and roughness, it was found that it is important to consider the thickness of the cavitation area and not only the size of the cavitation clouds. In fact, all the above phenomena are intertwined and interdependent, so in most cases, consideration and analysis of all the physical phenomena presented in the article should be done during the development phase of a new centrifugal pump. By presenting the results of individual numerical analyses and some recommendations on how to perform them most optimally, we are far from covering all the problems that arise in numerical analysis of centrifugal pumps. However, it can be concluded that any contribution that enables development engineers to work more successfully in the development process is very welcome.

REFERENCES

[1] M. Elashmawy, "3D-CFD Simulation of Confined Cross-Flow Injection Process Using Single Piston Pump," *Engineering, Technology & Applied Science Research*, vol. 7, no. 6, pp. 2308–2312, Dec. 2017, https://doi.org/10.48084/etasr.1561.

[2] D. Florjančič, *Trouble-shooting: handbook for centrifugal pumps: sources, diagnoses, recommendations*. Ljubljana, Slovenia: Turboinštitut, 2008.

[3] *IEC 60193 - Hydraulic turbines, storage pumps and pump-turbines – Model acceptance tests*. IEC, 2009.

[4] F. B. Mainier, A. M. Coelho, and E. F. Barros, "Corrosivity Evaluation of Copper-Nickel Alloy (90/10) in Pumps Used in Offshore Platforms for Seawater Pumping," *Engineering, Technology & Applied Science Research*, vol. 9, no. 5, pp. 4636–4639, Oct. 2019, https://doi.org/10.48084/etasr.3016.

[5] F. B. Mainier, A. M. Coelho, and E. F. Barros, "Velocity Boundary Layer Analysis of a Flat Plate Heat Exchanger in Laminar Flow: A Case Study," *Engineering, Technology & Applied Science Research*, vol. 2, no. 6, pp. 310-314, Dec. 2012, https://doi.org/10.48084/etasr.251.

[6] A. Lipej, S. Muhič, and D. Mitruševski, "Wall Roughness Influence on the Efficiency Characteristics of Centrifugal Pump," *Journal of Mechanical Engineering*, vol. 63, no. 9, pp. 529–536, 2017, https://doi.org/10.5545/sv-jme.2017.4526.

[7] U. Ješe, "Numerical Study of Pump-Turbine Instabilities: Pumping Mode off - Design Conditions," Ph.D. dissertation, Université Grenoble Alpes, Grenoble, France, 2015.

[8] J. Jia, J. F. Zhang, Y. F. Qu, H. K. Cai, and S. B. Chen, "Study on hump characteristics of pump turbine with different guide vane exit angles," *IOP Conference Series: Earth and Environmental Science*, vol. 240, Mar. 2019, Art. no. 072038, https://doi.org/10.1088/1755-1315/240/7/072038.

[9] J. Yang, G. Pavesi, S. Yuan, G. Cavazzini, and G. Ardizzon, "Experimental Characterization of a Pump–Turbine in Pump Mode at Hump Instability Region," *Journal of Fluids Engineering*, vol. 137, no. 5, May 2015, https://doi.org/10.1115/1.4029572.

[10] A. Lipej and D. Mitrusevski, "Numerical Prediction of Inlet Recirculation in Pumps," *International Journal of Fluid Machinery and Systems*, vol. 9, no. 3, pp. 277–286, Sep. 2016, https://doi.org/10.5293/IJFMS.2016.9.3.277.

[11] P. Limbach, T. Müller, M. Blume, and R. Skoda, "Numerical and experimental investigation of the cavitating flow in a low specific speed centrifugal pump and assessment of the influence of surface roughness on head prediction," presented at the International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, HI, USA, Apr. 2016.

[12] J. Vesely, L. Pulpitel, and P. Troubil, "Model research of rotating stall on pump-turbines," presented at the Hydro 2005, Villach, Austria, 2005.

[13] C. Wang, Y. X. Zhang, J. Y. Zhang, Z. W. Li, and A. Xu, "Comparative research on various turbulence closure models for unsteady cavitation flows in the ultra-low specific speed centrifugal pump," *IOP Conference Series: Earth and Environmental Science*, vol. 240, Mar. 2019, Art. no. 062038, https://doi.org/10.1088/1755-1315/240/6/062038.