

Wall roughness influence on the NPSH characteristics of centrifugal pumps

Andrej Lipej¹, Simon Muhič¹, Duško Mitruševski²

¹University of Novo mesto, Faculty of Mechanical Engineering, Slovenia

²SM-Pumps, Slovenia

Abstract. Pumps are very important and very widespread energetic machines. They are used in all branches of industry. Pumps are also a major consumer of electricity and therefore their development is even more important. From theoretical through experimental development in the last decades, a quantity of research has focused on numerical analysis. These are used to optimize all kinds of characteristics, from energy and cavitation, to dynamic characteristics. At the beginning of Computational Fluid Dynamics (CFD) analyses were related to the physical model (potential flow, Euler equation, Navier-Stokes equations). Subsequently, a lot of research was carried out in the field of various turbulent models. With the launch of high performance multi-processor computers, more and more time-dependent problems have been analysed. All the above mentioned research was carried out with numerical calculations on smooth surfaces. The fact is that in reality there is always some roughness on wet surfaces, depending on the material, the technology of production of certain parts of machines and finishing surface treatment. In the article, we give preliminary results of numerical analyses of centrifugal pumps, where the roughness of the walls was taken into account. The analyses were completed for different types of pumps, with different roughnesses and for different characteristics. First the effect of the roughness on the efficiency is shown, further the cavitation characteristics were analysed. The basic idea for research comes from the fact that there are still many cases where the matching of the results of the measurements and the results of numerical analyses are not perfect. One of the possible reasons can be the roughness of the walls.

1 Introduction

In last decades Computational Fluid Dynamics (CFD) has played an important role in the centrifugal pumps characteristics improvement. Relative results of the characteristics curves for different hydraulic shapes can be predicted very accurately. Usually the prediction of the absolute values of various characteristics is still a challenge. In the paper, we present an example where results of steady state and unsteady CFD analysis were obtained with wall roughness taken into account. In usual CFD analysis it is not possible to take the real roughness into account, but sand-grain equivalent. It is known that sand grain equivalent parameter does not depend only on the roughness amplitude, but also on the shape and frequency of the roughness [1]. Fluid flow over rough walls is theoretically explained quite well. There are many research works in this field [2].

The results in this article refer to a wider overview of the results of the benchmarking and CFD analyses that have been carried out and published recently. Trends of deviations are known and presented in many papers. We wanted to show that by the wall roughness taking into account the, numerical results in certain cases can better match experimental results.

Many papers about numerical and experimental analysis of NPSH characteristics have been published in the last fifteen years [3], [4] and [5]. Investigations of roughness influence on NPSH characteristics

cannot be found often in literature. Pump performance of a low specific speed centrifugal pump in cavitating flow conditions has been analysed in the research work of Limbach [6], with a state-of-the-art CFD method and cavitation model for both, design and off-design conditions with varying surface roughnesses.

2 CFD Analysis

Surface roughness has an important influence on the engineering problems and leads to an increase in turbulence production near the rough walls. This has also an influence on increasing wall shear stress. Accurate prediction of near wall flows depends on the proper modelling of surface roughness. The near wall treatment, which is used in ANSYS CFX-Solver (Scalable Wall Functions, Automatic Wall Treatment) is appropriate when walls are considered as hydraulically smooth. The logarithmic profile exists for rough walls but it is moved closer to the wall and the near wall treatment becomes more complex which now depends on two variables: the dimensionless wall distance y^+ and the mean roughness height (R_a).

2.1 Roughness analysis

Basic relations for the incompressible fluid motion, the Reynolds-Averaged Navier-Stokes system of equations were used in this paper.

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu_t \nabla^2 \bar{u}_i + \frac{\partial \tau_{ij}}{\partial x_j} \quad (1)$$

Usual logarithmic relation for the near wall velocity is presented by the equation:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{k} \ln(y^+) + C \quad (2)$$

where

$$y^+ = \frac{\rho \Delta y u_\tau}{\mu} \quad (3)$$

$$u_\tau = \left(\frac{\tau_\omega}{\rho} \right)^{1/2} \quad (4)$$

Above logarithmic relation for the near wall velocity is different, if wall roughness is taking into account. The new logarithmic velocity profile is:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{k} \ln(y^+) + B - \Delta B \quad (5)$$

Where $B = 5.2$. The ΔB is a function of the dimensionless roughness height, h^+ , defined as

$$h^+ = \frac{h u_\tau}{\nu} \quad (6)$$

and the dimensionless sand-grain roughness,

$$h_s^+ = \frac{h_s u_\tau}{\nu} \quad (7)$$

in addition, for sand-grain roughness [7], ΔB is defined by:

$$\Delta B = \frac{1}{k} \ln(1 + 0.3 h_s^+). \quad (8)$$

Three regimes are defined depending on the dimensionless sand-grain roughness:

Hydraulically smooth: $0 \leq h_s^+ \leq 5$,

Transitional-roughness: $5 \leq h_s^+ \leq 70$,

Fully rough flow: $h_s^+ \geq 70$.

2.2 Cavitation analysis

In homogeneous multiphase flow, a common flow field is shared by all fluids, as well as other relevant fields such as temperature and turbulence. This allows some simplifications to be made to the multi-fluid model resulting in the homogeneous model. For the given transport process, the homogeneous model assumes that the transported quantities (with the exception of volume fraction) for that process are the same for all phases, that is,

$$\varphi_\alpha = \varphi \quad 1 \leq \alpha \leq N_p \quad (9)$$

Because transported quantities are shared in homogeneous multiphase flow, it is sufficient to solve the shared fields using bulk transport equations rather than solving individual phasic transport equations. The bulk transport equations can be derived by summing the individual phasic transport equations over all phases to give a single transport equation for:

$$\frac{\partial}{\partial t}(\rho\varphi) + \nabla(\rho U\varphi - \Gamma\nabla\varphi) = S \quad (10)$$

$$\rho = \sum_{\alpha=1}^{N_p} r_\alpha \rho_\alpha \quad (11)$$

$$U = \frac{1}{\rho} \sum_{\alpha=1}^{N_p} r_\alpha \rho_\alpha U_\alpha \quad (12)$$

$$\Gamma = \sum_{\alpha=1}^{N_p} r_\alpha \Gamma_\alpha \quad (13)$$

3 Test cases for CFD analysis

The geometries and computational domains of the study cases analysed in this paper are presented in figure 1. All geometries consist of complete pump with inlet pipe, impeller, spiral casing and outlet pipe. In case of double suction pump there is also suction chamber. In all cases the roughness is taken into account only inside the impeller.

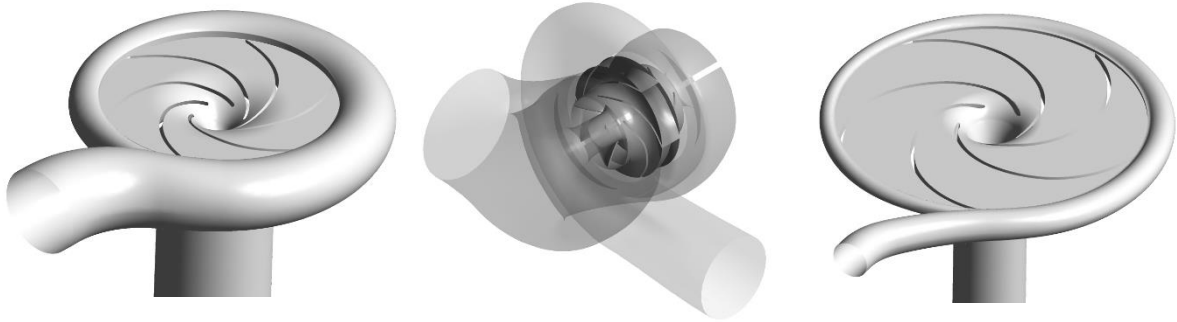


Figure 1. Geometries for analysed pumps

First two test cases were chosen because of different types of pumps and because there are some experimental results of energetic and cavitation characteristics for various wall roughnesses. The third example is presented because of the different applicability including the usage of fluid with very high viscosity.

Using appropriate computational grid parameters for flow analysis over rough walls is very important for the accuracy of numerical analysis. Near wall treatment with appropriate turbulence models and using the exact value for y^+ parameter (Figure 2) is a significant issue. This parameter is meaningful for the analysis of the smooth surfaces, but it is even more important when the wall roughness is taken into account.

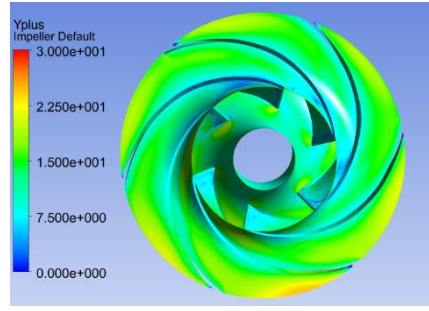


Figure 2. Distribution of y^+ on double suction impeller blades

4 CFD results

4.1 Analysis of the n_q 24 pump

For the pump with $n_q = 24$, ($Q_{opt} = 0.075 \text{ m}^3/\text{s}$, $H_{opt} = 18 \text{ m}$) numerical analysis was done for different roughness heights and five flow rates.

$$n_q = n \frac{Q^{1/2}}{H^{3/4}} \quad (14)$$

CFD analyses were done with the computational grids that consisted of 13.5 million elements. Numerical analysis was done for different flow rates. But to fulfill basic requirements for mesh quality, there should have been different computational grids for each operating point. But in our case the same grid for all operating regimes and y^+ parameter was used that is not optimal for all calculations. In figure 3 are presented the results for unsteady analysis of the pump efficiency for two different roughnesses. Calculation with rough walls provides better results if the equivalent roughness is taken into account. The difference between numerical and experimental results depend on flow rate, roughness height and quality of computational grid.

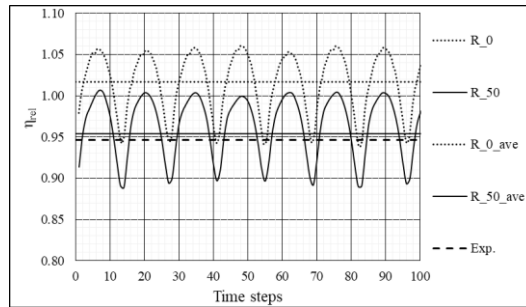


Figure 3. Efficiency distribution for unsteady analysis – smooth and rough walls

In figure 4 the comparison of numerical and experimental results of complete efficiency curve is obtained using two different roughnesses. Experimental results are presented for two wall qualities; basic roughness and two times higher.

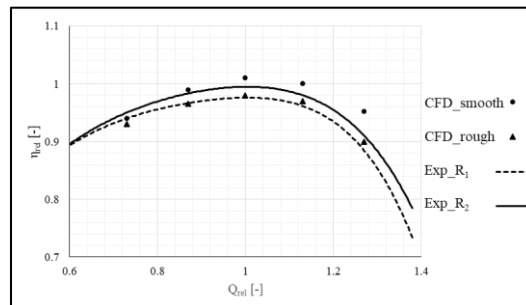


Figure 4. Comparison of experimental and numerical results – two wall roughnesses

In numerical analysis higher value of efficiency is obtained using smooth walls. Lower efficiency was obtained with sand-grain equivalent of 50 μm . We have done some preliminary investigations regarding relations between computational mesh parameters, turbulence models and roughness height. Preliminary investigations were carried out to obtain as accurate results as possible with wall roughness in the pump taken into account. Two simple test cases have been done. First, the flow near flat plate and second, the flow in a pipe [8] were analysed. Theoretical and numerical results were compared. In these analyses a well-known relations in non-dimensional form of the Darcy-Weisbach friction factor λ , Reynolds number Re and relative roughness for fully developed flow in a circular pipe were taken into account. The results show that a very small y^+ over predict the losses. Very high values give better results but still over predict the losses. In this analysis, it was attempted to keep y^+ between 10 and 20.

4.2 Double suction pump

For double suction pump ($Q_{\text{opt}} = 0.222 \text{ m}^3/\text{s}$, $H_{\text{opt}} = 75 \text{ m}$) measurements were done for two different roughness heights. The difference between experimental and numerical results for different roughness are presented in the figure 5.

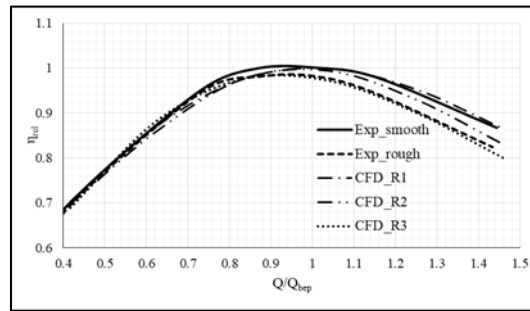


Figure 5. Experimental and numerical results – different wall roughness ($R1 < R2 < R3$)

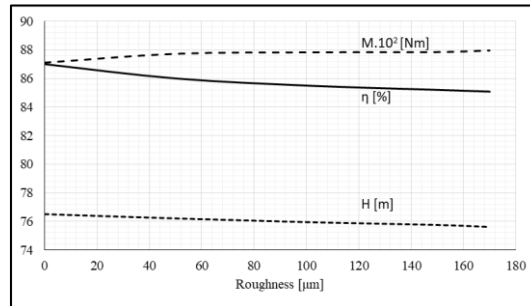


Figure 6. Torque, efficiency and head distribution for different roughness

It is shown that the difference of efficiency depends on flow rate and is bigger at full load. For H-Q characteristics the difference is almost constant.

From the analyses of the efficiency, head and torque for different roughness heights can be seen that the gradient of the efficiency and torque are bigger when the roughness is small and very low for higher roughness heights (Figure 6).

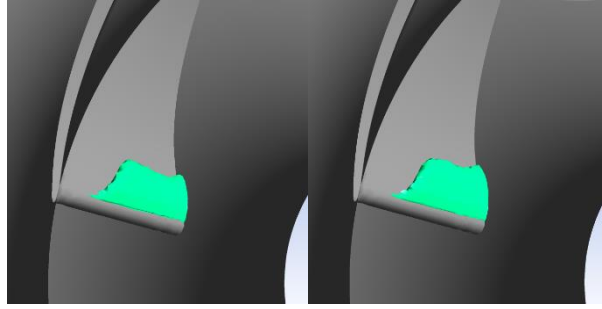


Figure 7. Size of cavitation cloud for smooth (left) and rough (right) walls – inlet pressure is 120 kPa

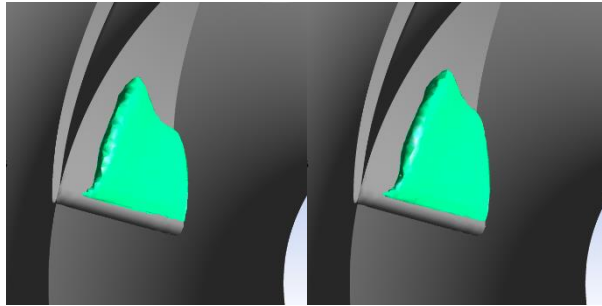


Figure 8. Size of cavitation cloud for smooth (left) and rough (right) walls – inlet pressure is 80 kPa

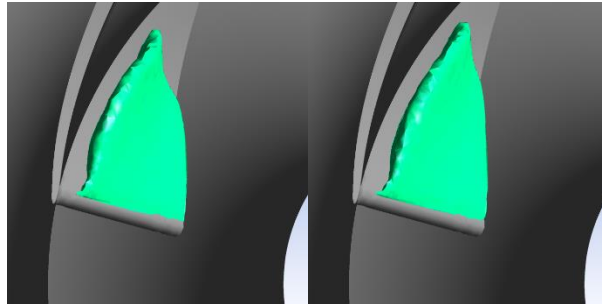


Figure 9. Size of cavitation cloud for smooth (left) and rough (right) walls – inlet pressure is 60 kPa

In the figures 7 to 9 the difference of cavitation cloud size for three different inlet absolute pressures and two roughnesses is presented. Left figures present results with smooth walls and right figures present results for rough walls. A slightly bigger cavitation cloud can be noticed in the right figures, where the roughness is higher. This result also influences the NPSH characteristics but very moderately.

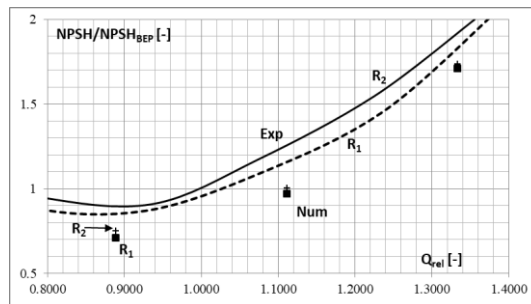


Figure 10. Comparison of experimental and numerical prediction of $NPSH_r$ for different roughness ($R_1 < R_2$)

In figure 5 and figure 10 experimental results are presented for basic roughness R_b and 3.4 times bigger roughness. In figure 10 there are also numerical results obtained by basic sand-grain equivalent and 3.4 times bigger coefficient. The difference between both numerical results is very low.

4.3 Analysis of the nq 15 pump

Numerical analysis of very low specific speed pump was done for different roughness. As a fluid we have used a water and the fluid with viscosity 60 cP. We would like to obtain the influence of the roughness when the pump is used for higher viscous fluid. The influence of the roughness is worth mentioning only with low viscosity. At high viscosity value, the influence of the wall roughness is negligible (Figure 11).

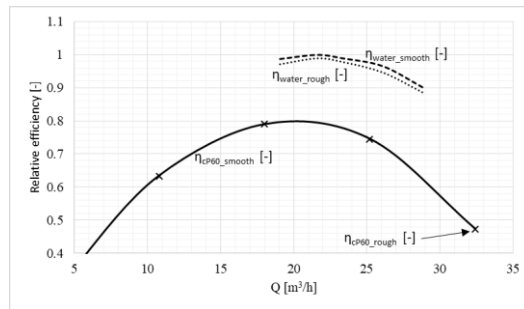


Figure 11. Efficiency distribution for smooth and rough walls for two different viscosities



Figure 12. Cavitation intensity for smooth walls and different cavitation coefficient



Figure 13. Cavitation intensity for rough walls and different cavitation coefficient

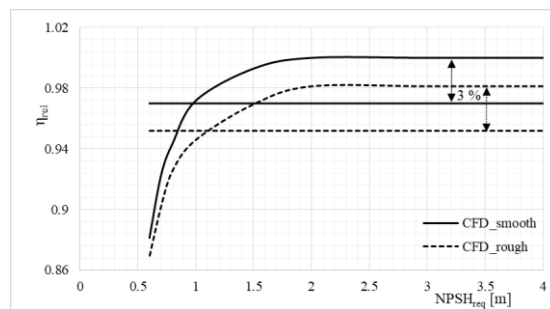


Figure 14. Efficiency drop for different roughness

For low specific speed pump the $NPSH_{req}$ is very low and the transition from cavitation inception to fully developed cavitation is fast. The flow inside the impeller is not completely axisymmetric and steady state sliding interfaces ‘frozen rotor’ or ‘stage’ do not give proper results. That is why in this case unsteady analysis was done. For best efficiency point intensity of cavitation is presented in Figure 12 and Figure 13 for different roughness. The difference is barely visible but in the case of efficiency drop, the difference is noticeable (Figure 14).

5 Conclusions

Numerical prediction of energetic and cavitation characteristics of different hydraulic machines can be obtained using different CFD codes. Sometimes numerical results match very well with the experimental ones but in many cases the difference is relatively big.

Wall roughness is present in the majority of the industrial flow analyses, but it is not taken into account in most of the CFD analyses. Numerical analysis of the flow over rough walls can be performed in different ways. One possibility is to analyse exact geometry of the wall roughness which is very demanding and time consuming. Another option is using equivalent parameters for roughness prediction which is much more economic from the computational point of view. This is the reason why many CFD codes use different coefficients like sand-grain equivalent coefficient.

The influence of the wall roughness on the efficiency and NPSH characteristics is presented in this paper. The surface roughness increases the wall shear stress in the turbulent boundary layer. This is the reason why it is necessary to pay attention to the near walls flow.

The comparison of the numerical and experimental results shows a slight improvement of the results if wall roughness is taking into account. For the efficiency curve obtained without cavitation, the difference between smooth and rough walls increase from part load to full load. At the highest flow rate the difference can be a few percent. In the case of numerical analysis of cavitation, a wall roughness does not have so significant influence.

It is also very important to use transient analysis, because the flow in pumps is usually unsteady. There are still several open questions about the influence of computational grids quality and height of the roughness, about turbulent models and cavitation models. The future research work will cover all of the above open issues.

References

- [1] Adams, T., Grant C. and Watson H. 2012 A Simple Algorithm to Relate Measured Surface Roughness to Equivalent Sand-grain Roughness. *International Journal of Mechanical Engineering and Mechatronics*. **1**(1).
- [2] Aupoix, B. 2014 Wall Roughness Modelling with k-w STT Model. *10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements*.
- [3] Coutier-Delgosha, O. 2005 Numerical Simulation of Turbopump Inducer Cavitating Behavior. *International Journal of Rotating Machinery*. (2), 135–142.
- [4] Kaya, M. and Ayder E. 2017 Prediction of cavitation performance of radial flow pumps. *Journal of Applied Fluid Mechanics*. **10**(5), 1397–1408.
- [5] Krumenacker, L., Fortes-Patella R. and Archer, A. 2014 Numerical estimation of cavitation intensity. *IOP Conference Series: Earth and Environmental Science*. **22**.
- [6] Limbach, P., Müller, T., Blume, M. and Skoda, R. 2016 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. ISROMAC, Hawaii, Honolulu.
- [7] ANSYS, CFX ANSYS CFX-solver theory guide 2017 ANSYS CFX.
- [8] Lipej, A., Muhič, S. and Mitruševski, D. 2017 Wall roughness influence on the efficiency characteristics of centrifugal pump. *Strojniski Vestnik/Journal of Mechanical Engineering*. **63**(9), 529–536.