Strojniški vestnik - Journal of Mechanical Engineering 63(2017)9, 529-536

DOI:10.5545/sv-jme.2017.4526
Original Scientific Paper
Received for review: 2017-04-22
Received revised form: 2017-06-16
Accepted for publication: 2017-07-14

Wall Roughness Influence on the Efficiency Characteristics of Centrifugal Pump

Andrej Lipej[1] – Simon Muhič[1,*] – Duško Mitruševski[2]
[1] Faculty of Technologies and Systems, Slovenia
[2] SM Pumps, Ljubljana, Slovenia

Many different machines are developed in industry and in terms of energy conversion, their efficiency is one of the most important parameters. A lot of theoretical, experimental and numerical analyses are done in the development process in order to obtain required characteristics. Computational fluid dynamics (CFD) analyses are a very important part of the development process. To obtain accurate results it is important to pay attention to geometry definition, usage appropriate numerical model, quality of the computational grid, realistic boundary conditions and all of the other parameters regarding fluid and solid material properties. A very important issue is usually the correct selection of the turbulence model. In most CFD analyses, only smooth surface is taken into account without considering any wall roughness. Besides the usage of different physical and mathematical models and all required parameters, the wetted surface roughness can also be one of the important origins of the numerical results inaccuracy. In the paper, the analysis of the influence of different parameters such as the sand-grain equivalent parameter is presented. The influence of y^+ on the accuracy of the flow analysis with different absolute roughness of the surfaces is also analyzed. For basic relations, the flow in simple geometries like flow over flat plate and flow in circular pipe has been analyzed. The conclusions of the preliminary research work are used in the case of efficiency prediction of centrifugal pump with rough walls. The final numerical results are compared with the experimental ones and show better agreement in comparison with the flow over smooth walls.
Keywords: pump, CFD, wall roughness, turbulence

Highlights
- *Water pumps are very important hydraulic machines with significant electricity consumption, because of the huge number of the operating machines.*
- *The accuracy of numerical prediction of energetic characteristics is very important for the development process of centrifugal pumps.*
- *One of the important parameters in CFD analysis is wall roughness, which is the main reason why in the paper an investigation of the influence of roughness on the CFD results is performed.*
- *Using correct computational grid parameters is very important for the accuracy of numerical analyses.*

0 INTRODUCTION

Fluid flow over rough walls is theoretically explained quite well. There are many research works in this field. Well known relations also exist 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, presented in Moody diagram.

For solving Navier-Stokes equations in 3D arbitrary complex geometries, it is not possible to get analytical solution, but only numerical. The similar situation is also prediction of losses, caused by different surface roughness of the walls.

In the CFD analyses, the surface roughness can be analyzed in two ways. First, the complete surface shape can be taken into account, but such approach needs very fine computational grids and in the majority of industrial application it is not an appropriate method.

Another method is the usage of special parameters, which define the surface roughness. In some commercial software, the sand-grain equivalent parameter is used to predict the surface roughness. It is known from literature that sand-grain equivalent parameter does not depend only on the roughness amplitudes, but also on the shape and frequency of the roughness [1].

An important issue in the CFD analysis is the near wall treatment with different turbulence models and using the exact value for y^+ parameter is meaningful. This parameter is very important in the analysis of smooth surfaces, however, it is even more important when the surface is not hydraulically smooth.

The arithmetic average of the roughness profile R_a is defined as (Fig. 1):

$$R_a = \frac{1}{n}\sum_{i=1}^{n}\left|y_i\right|. \tag{1}$$

Literature review about the usage of CFD analysis of rough walls shows that the number of scientific papers about this topic is quite moderate. It is possible to find some papers about the connection of turbulence models and wall roughness [2] and [3]. Some authors

*Corr. Author's Address: Faculty of Technologies and Systems, Na Loko 2, Novo mesto, Slovenia, simon.muhic@fts-nm.si

performed some research work on the basic influence of the wall roughness on the turbulent boundary layer [4] to [6]. The paper [7] presents analysis of the losses in centrifugal pump with smooth and rough walls. In the paper, some figures about the difference between experimental and numerical obtained efficiency are presented. It is also possible to find research work about the degradation of an axial-turbine stage [8] as a consequence of wall roughness using the CFD analysis. Some papers dealing with wall roughness are devoted to different topics like mine ventilation networks [9] or just numerical analysis around blades [10] as well as numerical analysis of the flow in the pipes with rough walls [11].

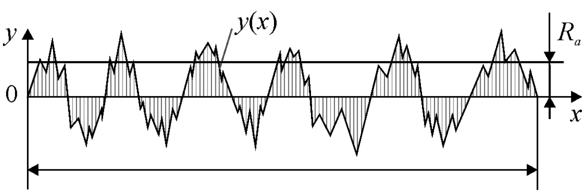

Fig. 1. *Roughness profile*

1 METHODS

Surface roughness has a significant influence on the engineering problems and leads to an increase in turbulence production near the rough walls. This also has an influence on increasing wall shear stress. Accurate prediction of near wall flows depends on the proper modelling of surface roughness [12].

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. For rough walls, the logarithmic profile exists but moves closer to the wall and the near wall treatment becomes more complex, since it now depends on two variables: the dimensionless wall distance y^+ and the mean roughness height (R_a).

The arithmetic average of absolute roughness, which must be specified like the wall boundary conditions, is presented in many commercial software with the sand-grain roughness equivalent [12]. It is important to consider that the sand-grain roughness height is not equal to the geometric roughness of the surface. Wall friction depends on the type of roughness (shape, distribution, etc.) and not only on roughness height. Therefore, determining the appropriate equivalent sand-grain roughness height is crucial.

In this paper, we use basic relations for the incompressible fluid motion with the Reynolds-Averaged Navier-Stokes system of equations.

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0,$$
$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu_t \nabla^2 \overline{u}_i + \frac{\partial \tau_{ij}}{\partial x_j}. \tag{2}$$

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

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{\kappa}\ln(y^+) + C, \tag{3}$$

where

$$y^+ = \frac{\rho \Delta y u_\tau}{\mu}, \tag{4}$$

$$u_\tau = \left(\frac{\tau_\omega}{\rho}\right)^{\frac{1}{2}}. \tag{5}$$

The above logarithmic relation for the near wall velocity is different if wall roughness is taken into account. The new logarithmic velocity profile is:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{\kappa}\ln(y^+) + B - \Delta B, \tag{6}$$

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

$$h^+ = \frac{h u_\tau}{\nu}, \tag{7}$$

and the dimensionless sand-grain roughness:

$$h_s^+ = \frac{h_s u_\tau}{\nu}. \tag{8}$$

In addition, for sand-grain roughness, ΔB is defined by:

$$\Delta B = \frac{1}{\kappa}\ln(1 + 0.3 h_s^+). \tag{9}$$

Depending on the dimensionless sand-grain roughness h_s^+, three roughness regimes are defined:

- Hydraulically smooth: $0 \le h_s^+ \le 5$,
- Transitional-roughness: $5 \le h_s^+ \le 70$,
- Fully rough flow: $h_s^+ \ge 70$.

2 CFD ANALYSIS OF ROUGH WALLS

In the paper, we also present the numerical results of flow analysis in the centrifugal pump, where wall roughness was taken into account.

Technical roughness, which has peaks and valleys of arbitrary shape and different sizes, can be presented by an equivalent sand-grain roughness [1]. In the Eq. (8), the variable h_s means the equivalent of average roughness height.

In the last ten years, many papers have dealt with the numerical analysis of the pump characteristics. The results have been obtained using different types of computational grids, different turbulent models, steady state or unsteady approach and different software. The most analyses were done by using hydraulic smooth walls.

If the roughness is not taken into account, some losses in the flow are neglected or underestimated. These losses depend on the absolute size of roughness and on the velocity of the fluid. Therefore many times the comparison between the numerically obtained efficiency and experimental results show different discrepancies, depending on operating conditions.

Before we started analyzing the flow in the pump, we had to check some basic relations between computational mesh parameters, turbulence models and roughness height. In order to obtain these answers, two simple test cases were performed. First, the flow near a flat plate and second the flow in a pipe.

In Fig. 2 the velocity distribution near the flat plate wall for four different sand-grain equivalent sizes 0 µm, 10 µm, 35 µm and 100 µm is presented, for the computational grid near flat plate with the size of the first element near the wall 10 µm.

The upper graph shows the complete velocity distribution in the boundary layer, while the lower graph shows just the velocity distribution inside 100 µm space near the wall.

The flow boundary conditions and computational grid for all four results in Fig. 2 are completely the same. The difference is just the roughness height.

Similar velocity distribution is presented in Fig. 3, where different mesh size near the wall is used, with the size of the first element near the wall 40 µm.

It is known that wall roughness increases the wall shear stress and breaks up the viscosity sublayer in the turbulent flows. The consequence is also the downward shift in the near wall velocity profile (Eq. 6).

In Fig. 2 it can be seen that for the very low values of y^+ (lower than 5) the velocity near the wall is even higher for rough walls in comparison with smooth walls. In Fig. 3, where y^+ is higher than 10, the situation is the opposite. The results show the influence of y^+ on the accuracy of the near wall velocity profile.

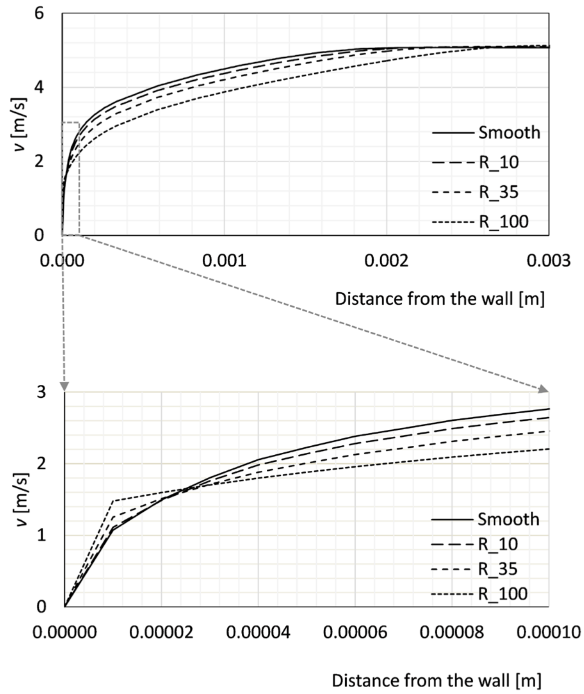

Fig. 2. *Velocity distribution in boundary layer*

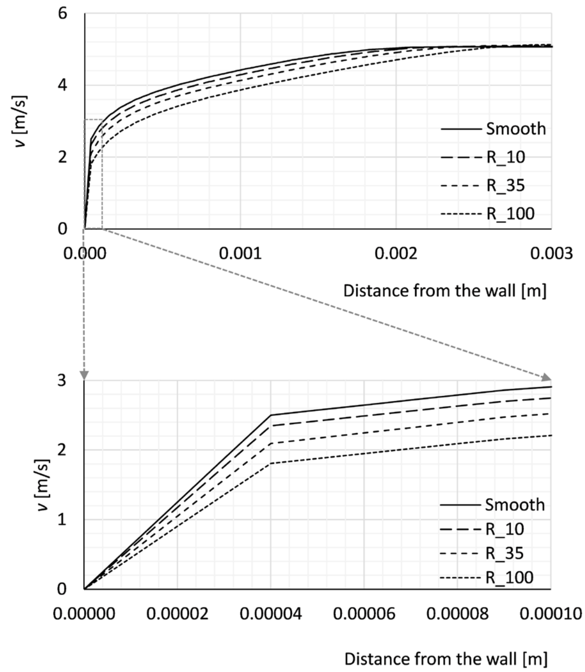

Fig. 3. *Velocity distribution in boundary layer*

The results in Figs. 2 and 3 show that too small y^+ is not suitable for the analysis of the losses, where rough walls are taken into account.

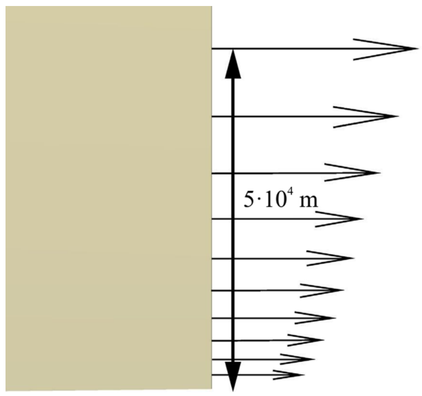

Fig. 4. *Velocity vectors in boundary layer*

The second test case of the flow analysis was the flow in the circular pipe (Fig. 4). The length of the pipe is 1.5 m and the diameter is 0.05 m. The results of this test show the influence of the computational mesh size near the wall on the accuracy of the results. The obtained CFD results were compared with the theoretical results, calculated using theoretical Moody friction factor (Eq. 10). In Fig. 5 a comparison of ratio between theoretical and numerical losses for different roughness heights and different size of dimensionless parameter y^+ is presented.

Theoretical results were obtained by the following equations:

$$\xi_{th} = \lambda \frac{L}{D} \frac{\rho u^2}{2}, \tag{10}$$

$$\lambda = \frac{1.325}{\left[\ln\left(\frac{h}{3.7D} + \frac{5.74}{\mathrm{Re}^{0.9}} \right) \right]^2}, \tag{11}$$

for $5000 \leq \mathrm{Re} \leq 10^8$ and $10^{-6} \leq h/D \leq 10^{-2}$.

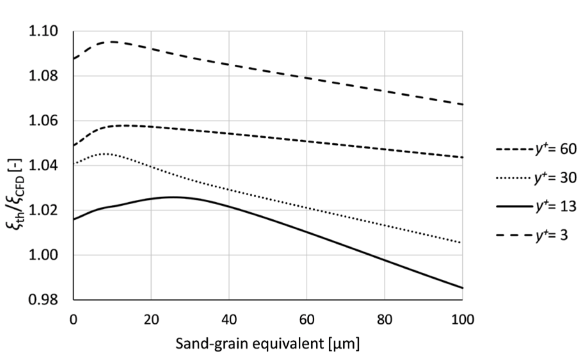

Fig. 5. *Comparison of numerical and theoretical prediction of losses of the flow in pipe*

The above results show that the influence of the y^+ is significant and we can conclude that a very small y^+ over predicts the losses in circular pipe. Very high values give better results but still over predict the losses.

3 CFD ANALYSIS OF THE FLOW IN PUMP

Comparison of the centrifugal pump efficiency obtained by measurement and numerical results, where only smooth walls were taken into account, shows that efficiency difference is not a constant for different operating regimes, but is much higher at full load than at part load. The reasons for such results can be different. We know that real walls are rough (Fig. 6) thus taking wall roughness into account in the numerical analysis can probably improve the results.

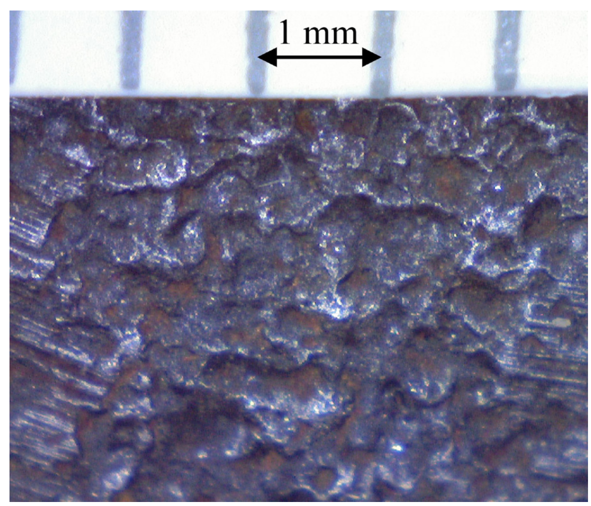

Fig. 6. *Microscope view of pump impeller surface made of cast iron*

Because of the above-mentioned differences, the efficiency analysis for different wall roughness and for at least five flow rates from part load to full load regime (Table 1) was performed.

Table 1. *Operating points and roughness height*

Flow rate = Q/Q_{BEP}	Roughness [μm]
0.73	0
0.87	50
1.00	100
1.13	
1.27	

A numerical analysis of the pump was done for five flow rates and for each operating point with three different roughness heights (Table 1). In Table 1 the flow rates are presented relatively depending on best efficiency point (BEP) flow rate.

The numerical analysis results of the flow in the pipe gave us the basic recommendations about the required computational grid parameters and overall quality for the accurate analysis of the flow in the centrifugal pump (Fig. 7). The specific speed the analysed pump is n_q=24. The main geometric parameters of the pump are:

- inlet diameter 0.16 m,
- outlet diameter 0.33 m,
- outlet width 0.031 m.

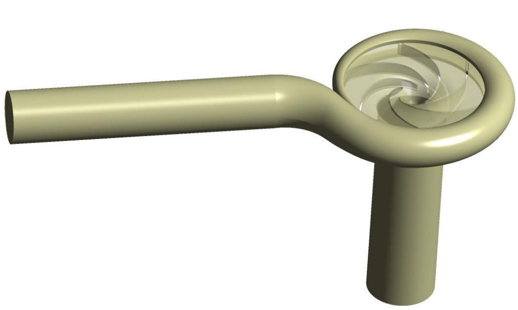

Fig. 7. *Computational domain of the pump*

CFD analyses were done with computational grids with about 13.5 million elements (Fig. 8). The y^+ depends on the size of the first element and on the size of the speed. Because the numerical analysis was done for different flow rates, it is not possible to have the same value of y^+ for all operating regimes just with one computational grid.

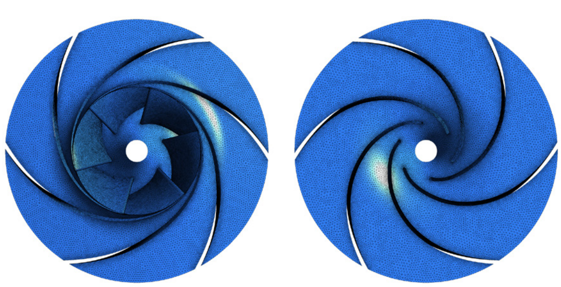

Fig. 8. *Computational grid of pump impeller*

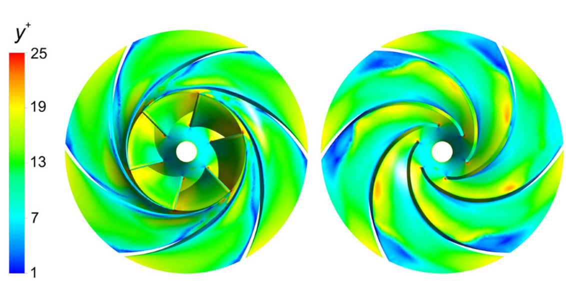

Fig. 9. *Distribution of y+ for BEP*

In our case, we had one computational grid for all flow rates and y^+ distribution on the impeller blades is presented in Fig. 9 for BEP. For the flow rates smaller than BEP, the y^+ is smaller, for the full load operating regime the y^+ is bigger than in the Fig. 9.

At the BEP, y^+ is between 10 and 30 in most areas, but on the part of pressure side of the impeller blades, the y^+ is smaller than 10 and on the part of suction side it is bigger than 30.

The quality of computational grid was provided with average aspect ratio around 100 at the near-wall elements and with the expansion ratio of 1.2. The size of elements outside the boundary layer was defined depending on the local flow properties, for each part of the pump.

The fluid in the CFD analysis was water at 25 °C, with density 997 kgm^{-3} and dynamic viscosity 8.899 kgm^{-1}s^{-1}.

The results of time dependent efficiency distribution for the two operating regimes are presented in Figs. 10 and 11. Relative efficiency is defined as a ratio between efficiency of each operating regime and maximal efficiency obtained with measurements. In Fig. 10 the results for the smallest flow rate (0.055 m^{3}s^{-1}) are presented and in this case the difference between the average results obtained using smooth walls and different heights of roughness is quite small, less than five percent.

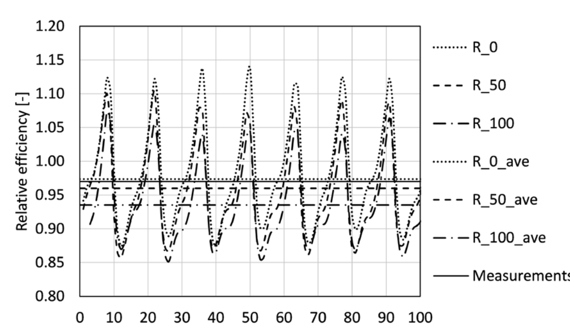

Fig. 10. *Pump efficiency distribution for different values of wall roughness for small flow rate*

If we compare the average results at part load for different heights of the roughness and for smooth wall with the results of measurements, the conclusion is that only the efficiency for smooth walls is noteworthy higher than the measured efficiency. From measurements of the pump wall roughness the approximate value of the R_a value was obtained, which is around 5 μm and with the algorithm in the paper **[1]** this value can be calculated to the sand-grain roughness coefficient. For the investigated pump, the sand-grain roughness equivalent is approximately 50 µm.

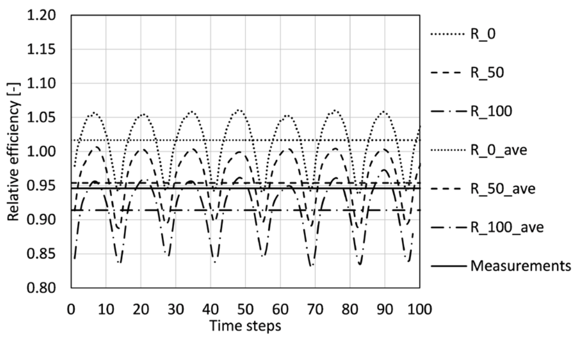

Fig. 11. *Pump efficiency distribution for different values of wall roughness for high flow rate*

In Fig. 11 the results for the flow rate bigger than the best efficiency point (0.095 m^3s^{-1}) are presented and in this case the difference between the results obtained using smooth walls and different heights of roughness is much bigger, more than ten percent.

At this operating point, roughness influence is slightly different. Efficiency for smooth walls and for 50 µm roughness is higher than measured efficiency. The efficiency is smaller only for the 100 µm roughness.

In Figs. 10 and 11 only the results for smooth walls and two values of the roughness are presented, because the computational time for unsteady analysis was quite long and we did not make calculations for the same number of time steps for all values of the roughness.

It is not possible to take all flow parameters into account, but from the obtained results (Fig. 12), we can conclude that wall roughness is an important parameter in accurate numerical analyses, especially when absolute accuracy is important.

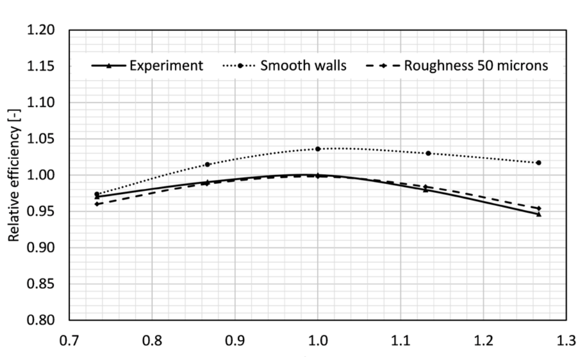

Fig. 12. *Comparison of pump efficiency characteristics between experiment and numerical analysis*

Comparison between numerical and experimental results is presented in Fig. 12. Results are presented for different flow rates between 0.7 and 1.3 Q_{BEP}. For small flow rates, all results are very close and the difference between experimental results and results for rough wall is very small.

The measurements of the efficiency were done using different instruments:

- flow rate - electromagnetic flow meters,
- head – pressure gauge,
- power – wattmeter,
- rotational speed – digital rpm device.

The measurements uncertainty was in accordance with the international standard ISO 9906. The relative uncertainties of the used instruments for each variable are:

- flow rate ±1.5 %,
- head ±1 %,
- power ±1 %,
- rotational speed ±0.35 %.

The different situation is for the operating point right of the best efficiency point, where the difference between experimental results and results for smooth wall is around seven percent. Taking wall roughness into account, the numerical results approach the experimental results, but the difference is still around one percent, approximately the same order of magnitude as at a part load operating regime.

The comparison in the Fig. 12 is presented for only one sand-grain equivalent roughness size 50 µm, which is supposedly the closest to the real value and was obtained at roughness measurements of the impeller surface and calculated to the appropriate sand-grain equivalent. The results considering wall roughness are better in comparison with the results for smooth walls, but there are still some discrepancies, which are probably also caused by different parameter y^+, since the computational grid for all calculations was the same. There are also other possible reasons for the inaccuracy of the numerical results, which were not taken into account.

4 CONCLUSIONS

The numerical prediction of energetic 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 situation is different. That is why the researchers paid a lot of attention to the quality of computational grid, appropriate turbulence models, accurate boundary conditions and other necessary flow and fluid parameters.

Nevertheless, consideration of the wall roughness usually remained forgotten. Wall roughness is present at the majority of the industrial flow analyses. One of the possible reasons why the wall roughness is neglected is the required quality of computational grid or some problems with turbulence models.

The numerical analysis of the flow over rough walls can be performed in two ways. First with exact geometrical modelling of the wall roughness, and second with using special equivalent parameters for roughness prediction. From the computational point of view, the first method is very demanding and time consuming. That is why many CFD codes use different coefficient, like sand-grain equivalent coefficient.

Because there is not a lot of research work on this topic, this paper presents some relations between the roughness coefficient and the usage of numerical methods for accurate prediction of losses in centrifugal pumps. In particular, the focus was on the size of the computational grids near the walls.

The paper also presents the influence of the dimensionless parameter y^+ on the accuracy of numerical results with comparison of theoretical and numerical results. It is shown that very small values of y^+ give a big difference between theoretical and numerical results. The usage of appropriate y^+ is necessary, because the surface roughness increases the wall shear stress in the turbulent boundary layer.

The second part of the paper deals with the numerical prediction of unsteady pump efficiency for different values of wall roughness. The results show that the influence of the wall roughness is not negligible thus it is very important to consider it. The main problems are the exact prediction of the sand-grain roughness equivalent and using the appropriate y^+ coefficient for all calculations.

Further investigations will analyze different shapes of wall roughness in terms of finding the accurate interface between real wall roughness and sand-grain roughness equivalent.

5 NOMENCLATURES

B Constant, [-]
C Log-layer constant, [-]
D Diameter, [m]
h Roughness, [m]
h^+ Dimensionless roughness height, [-]
h_s^+ Dimensionless sand-grain roughness, [-]
L Length, [m]
n_q Specific speed, [rpm]
Q Flow rate, [m^{3}s^{-1}]
Q_{opt} Optimal flow rate, [m^{3}s^{-1}]
p Pressure, [Pa]
R_a Arithmetic average of roughness, [m]
u^+ Near wall velocity, [ms^{-1}]
u Velocity, [ms^{-1}]
U_t Velocity at a distance Δy, [ms^{-1}]
u_τ Friction velocity, [ms^{-1}]
Δy Distance from the wall, [m]
y^+ Dimensionless distance from the wall, [-]
ΔB Shift, [-]
κ Von Karman constant, [-]
λ Friction factor, [-]
μ Dynamic viscosity, [kgm^{-1}s^{-1}]
ν Kinematic viscosity, [m^{2}s^{-1}]
ν_t Turbulent kinematic viscosity, [m^{2}s^{-1}]
ξ Losses, [Pa]
ρ Density, [kgm^3]
τ_ω Wall shear stress, [Pa]

6 REFERENCES

[1] Adams, T., Grant, C., Watson, H. (2012). A simple algorithm to relate measured surface roughness to equivalent sand-grain roughness. *International Journal of Mechanical Engineering and Mechatronics*, vol. 1, no. 2, p. 66-71, **DOI:10.11159/ijmem.2012.008**.

[2] Aupoix, B. (2014). Wall Roughness Modelling with k-ω SST Model, *10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements*, Marbella.

[3] Aupoix, B., Spalart, P.R. (2003). Extensions of the Spalart-Allmaras Turbulence Model to Account for Wall Roughness. *International Journal of Heat and Fluid Flow*, vol. 24, no. 4, p. 454-462, **DOI:10.1016/S0142-727X(03)00043-2**.

[4] Lin, C., Moeng, C., Sullivan, P.P., McWilliams, J.C. (1998). The effect of surface roughness on flow structures in a neutrally stratified planetary boundary layer flow. *Physics of Fluids*, vol. 9, no. 11, **DOI:10.1063/1.869439**.

[5] Akinlade, O.G. (2005). *Effects of Surface Roughness on the Flow Characteristics in a Turbulent Boundary Layer*, PhD Thesis. Department of Mechanical Engineering, University of Saskatchewan, Saskatoon.

[6] George, J., De Simone, A., Iaccarino, G., Jimenez, J. (2010). Modeling roughness effects in turbulent boundary layers by elliptic relaxation, *Center for Turbulence Research, Proceedings of the Summer Program*, p. 119-128.

[7] Juckelandt, K., Bleeck, S., Wurm, F.-H. (2015). Analysis of losses in centrifugal pumps with low specific speed with smooth and rough walls, *Proceedings of 11th European Conference on Turbomachinery Fluid dynamics & Thermodynamics*, p. 1-10.

[8] Kang, Y.-S., Yoo, J.-C., Kang, S.-H. (2006). Numerical prediction of roughness effects on the performance degradation of an axial-turbine stage. *Journal of Mechanical Science and*

Technology, vol. 20. no. 7, p. 1077-1088, DOI:10.1007/BF02916007.

[9] Levin, L.Y., Semin, M.A., Klyukin, Y.A. (2014). Estimation of wall roughness functions acceptability in CFD simulation of mine ventilation networks. *Proceedings of XLII International Summer School–Conference*, p. 25-32.

[10] Dutta, R., Nicolle, J., Giroux, A.M., Piomelli, U. (2016). Evaluation of turbulence models on roughened turbine blades. *28th IAHR Symposium on Hydraulic Machinery and Systems, Grenoble*, DOI:10.1088/1755-1315/49/6/062007.

[11] Vijiapurapu, S., Cui, J. (2010). Performance of turbulence models for flows through rough pipes. *Applied Mathematical Modelling*, vol. 34, no. 4, p. 1458-1466, DOI:10.1016/j.apm.2009.08.029.

[12] ANSYS CFX Solver Theory Guide (2016). Release 17.0., ANSYS, Inc., Canonsburg.