NUMERICAL SIMULATION ON A MIXED-FLOW PUMP OPERATING IN A TURBINE MODE

Peter Hlbočan*, Michal Varchola*

The paper deals with a modeling of a flow in hydraulic parts of a turbo machine operating in a pump and in a turbine mode. The results are Q-H (Q-Y) performance characteristics of the pump operating in both modes. An important piece of knowledge is a shift of the pump's and the turbine's BEP and the change of specific speeds of the turbo machine in a pump and in a turbine mode, which emerges from it. The goal of this paper was to find a shift of pump's and turbine's BEP and to obtain information about hydraulic efficiency of the pump operating in a pump and turbine mode. Anyway the aim of CFD simulation is to evaluate Q-H (Q-Y) characteristic from the point of view of a stability of Q-H curve. The analysis of flow patterns in the impeller and in the vane diffuser in a pump and turbine mode is a valuable contribution to the topic of a hydraulic interaction between an impeller and a vane diffuser.

Keywords: *reversible machine, turbine impeller, pump impeller, performance characteristic*

1. Introduction

An application of a centrifugal pump in a turbine mode is nowadays used very often. A single stage spiral pump is the most frequently used reversible turbomachine but axial and mixed – flow pumps with a vane diffuser are employed in a turbine mode less commonly. Historically there are known several cases of a reversibility of pumps or turbines since the mid of the 20th century [2]. The effectiveness of an operation in both modes as well as differences between optimal performance parameters are the ultimate issues while using a turbomachine in a reverse operation mode. This fact has a significant meaning in a reverse operation mode of a turbomachine and when looking for an interaction between hydraulic design of a turbine and a pump as well. Unfortunately there are not many experimental findings related to these issues in wide range of specific speeds. Findings about performance parameters of a pump in a turbine mode are also important by reason of employing of series-manufactured centrifugal pumps in a turbine operation mode in small hydropower plants. Therefore experimental findings and comparison of performance parameters of a pump in a pump and in a turbine operation mode with a possibility for eventual use in small hydropower plants are very important and they are important by the reason of investigating a relationship between optimal parameters of a turbomachine in a pump and in a turbine mode as well. An expensive physical experiment might be replaced by a CFD simulation in many cases. A CFD simulation of a few pumps in a pump and in a turbine operation mode was performed in this

* Ing. P. Hlbočan, prof. Ing. M. Varchola, CSc., Institute of Chemical and Hydraulic Machines and Equipment, Faculty of Mechanical Engineering, Slovak University of Technology in Bratislava

contribution. The aim was to look for relationships between optimal parameters in both operation modes. The research was performed on the pumps with the vane diffusers.

2. Performance parameters of a pump in a turbine mode

For a flow-rate of an ideal reversible turbomachine stands: $|Q_\mathrm{P}| = |Q_\mathrm{T}|$, $|n_\mathrm{P}| = |n_\mathrm{T}|$, $Y_\mathrm{tP} = Y_\mathrm{tT}$. A real specific energy of a pump is corrected by hydraulic losses which are included in a hydraulic efficiency of a pump η_hP and a turbine η_hT:

$$\text{In a case of a pump} \qquad Y_\mathrm{P} = Y_\mathrm{tP}\,\eta_\mathrm{hP}\ , \tag{1}$$

$$\text{In a case of a turbine} \qquad g\,H = Y_\mathrm{T} = \frac{Y_\mathrm{tT}}{\eta_\mathrm{hT}}\ . \tag{2}$$

If theoretical specific energy of a pump and a turbine is equal than stands:

$$Y_\mathrm{P} = Y_\mathrm{T}\,\eta_\mathrm{hP}\,\eta_\mathrm{hT}\ ,$$

if revolutions of an impeller of a pump and a turbine are same, than $Y_\mathrm{P} < Y_\mathrm{T}$, if we assume that $Y \sim u^2$, than

$$\frac{u_\mathrm{T}}{u_\mathrm{P}} \approx \sqrt{\frac{1}{\eta_\mathrm{hP}\,\eta_\mathrm{hT}}} > 1\ . \tag{3}$$

From this it is possible to state that an optimal circumferential velocity in a pump operation mode is less than in a turbine mode or a specific energy in a BEP in a turbine operation mode is greater than in a pump operation mode assuming the same rotating of an impeller. A ratio of circumferential velocities and specific speeds of a turbomachine changes according to a change in efficiencies in a pump and in a turbine mode as well. It depends on an absolute value of a hydraulic efficiency. Therefore it is possible to write:

$$\frac{u_\mathrm{T}}{u_\mathrm{P}} = (1.1\text{--}1.5)\ , \tag{4}$$

resp. assuming a same size of a turbomachine:

$$\frac{Q_\mathrm{T}}{Q_\mathrm{P}} = (1.1\text{--}1.5)\ . \tag{5}$$

Than for a specific energy in BEP stands:

$$\frac{Y_\mathrm{T}}{Y_\mathrm{P}} = \frac{H_\mathrm{T}}{H_\mathrm{P}} = (1.1\text{--}1.5)^2 = (1.21\text{--}2.25)\ . \tag{6}$$

The issue of the concrete values is an object of a CFD simulation.

3. CFD analysis and pumps used in the research

The courses of the performance curves of two different pumps with different specific speeds were analyzed. The first pump (Fig. 1a) was proposed with the mixed-flow vane diffuser and its nominal flow-rate is 85 l/s, head 14.58 m at 1800 rpm. The specific speeds of this pump are $n_\mathrm{s} = 257$. The second pump (Fig. 1b) was equipped with the axial vane diffuser and its performance parameters are: flow-rate 300 l/s, head 27.6 m at 1850 rpm. The specific speeds are $n_s = 306$. Beyond that there were available the experimental results [2], which served also for the validation of the results of the CFD simulation from the point of view of nature of the course of head and efficiency curves.

The CFD simulation in the pump in a pump as well as in a turbine mode was implemented with the intention to obtain courses of the performance curves $(Q - H(Q - Y), Q - \eta_{\mathrm{H}})$. The topic of a CFD simulation in centrifugal pumps has been analyzed by many authors e.g. Gülich [5]. The recommendations mentioned in this work were implemented into the CFD model, which was applied on this research. The complete domains of the impeller, the diffuser and straight parts of the suction and the discharge duct were included into the computational domain of the pump in a pump operation mode as well as in a turbine mode (Fig. 2). It is necessary to say that the zones 'Inlet' and 'Outlet' were interchanged between each other in the case of the simulation in a turbine operation mode. The computational grid consisted of approximately 2 500 000 hexahedral elements. The flow in a pump as well as in a turbine mode was assumed as unsteady and therefore the fully transient simulation was performed. After implementation of this approach into the simulation the flow patterns resp. values of velocities, the static pressure and other quantities which characterize the flow in the domain were obtained in the each time step. These data were used for assessment of time the averaged values of the quantities characterizing the flow and consequently the performance curves were estimated according to these time averaged data.

The static pressure at the outlet and the mass-flow rate at the inlet were set as the boundary conditions in the simulation of both – a pump and a turbine operation mode. The general grid interface (GGI) was implemented at the interface between the rotor and the stator parts of the computational domain. The instantaneous relative position of the rotor and the stator was assumed in every time step. A big advantage of this approach to CFD simulation is fact that non-stationary interactions between the rotor and the stator can be simulated. These transient phenomena have significant influence on the courses of performance curves.

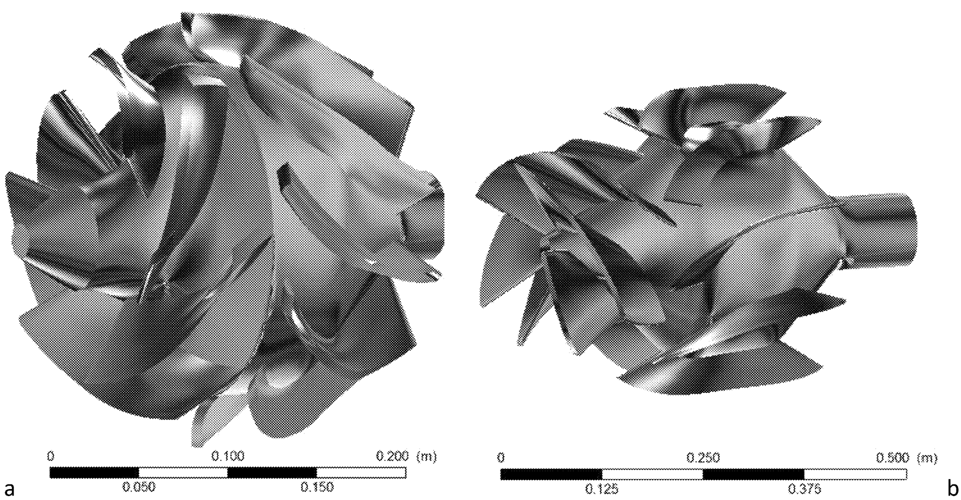

Fig.1: The geometry of the examined mixed-flow pumps (a – with the mixed-flow diffuser, b – with the axial diffuser)

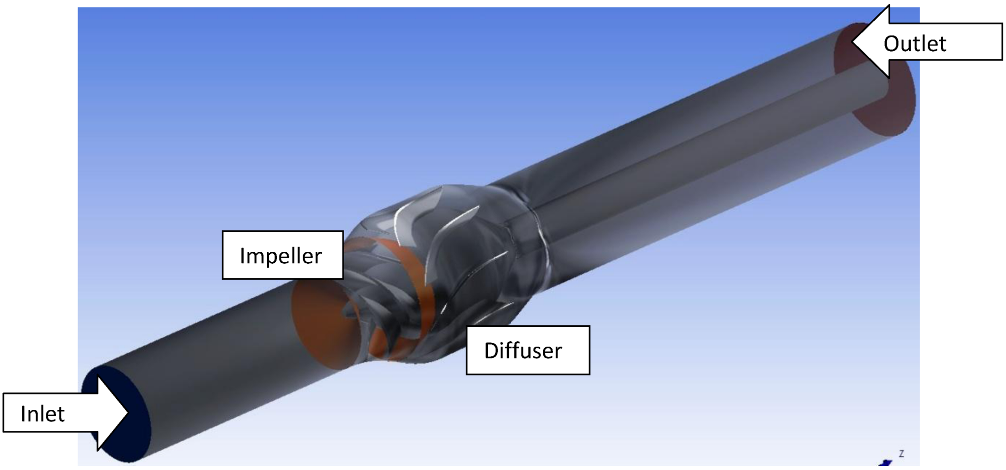

Fig.2: The computational domain

The second order discretization was implemented into the CFD model. The chosen value of the time step was selected so it corresponded to the 2° rotating of the rotor. The turbulence phenomena were modeled according to the Menter's SST (Shear Stress Transport) model. The near wall modeling was performed by automatic wall function. This approach is nowadays very common and used when simulating a flow in a turbomachinary. The detailed description of the turbulence model resp. automatic wall function is the object of several works e.g. [6].

A head in a pump operation mode was assessed according to equation (7).

$$H = \frac{p_{\mathrm{TotOut}} - p_{\mathrm{TotIn}}}{g\,\varrho} \; . \tag{7}$$

Symbols p_{TotOut} and p_{TotIn} express the total pressure at an inlet and at an outlet of the pump. A symbol ϱ represents a density of pumped liquid. For the equation of a total pressure stands (8).

$$p_{\mathrm{Tot}} = p_{\mathrm{Stat}} + \varrho\,\frac{v^2}{2} \; , \tag{8}$$

where p_{Stat} is a static pressure at a certain point in the computational domain, v is a total velocity at a certain point. A value of a total pressure at a given cross-section of a computational domain is assessed as a mass-flow averaged value at a given cross-section. A total pressure is calculated according to the equation (9).

$$p_{\mathrm{Tot}} = \frac{\sum \dot{m}_{(i)}\, p_{\mathrm{Tot}(i)}}{\sum \dot{m}_{(i)}} \; , \tag{9}$$

where $\dot{m}$ represents a local flow of mass and $p_{\mathrm{Tot}(i)}$ represents a local value of a total pressure at a certain point.

A torque of a runner M_{k} was assessed as a time-averaged value as well. Equation (5) was implemented to calculate a hydraulic efficiency. (A flow-rate Q and rotor rotating n were specified as boundary conditions of the CFD simulation.)

$$\eta_{\mathrm{hP}} = \frac{\varrho\,Q\,g\,H}{M_{\mathrm{k}}\,2\pi\,n} \; . \tag{10}$$

A head in a turbine operation mode was estimated according to the similar equation as a head in a pump mode. For a head in a turbine mode stands (11).

$$H = \frac{p_{\mathrm{TotIn}} - p_{\mathrm{TotOut}}}{g\,\varrho} \; . \tag{11}$$

For a hydraulic efficiency in a turbine mode stands (12).

$$\eta_{\mathrm{hT}} = \frac{M_{\mathrm{k}}\, 2\pi\, n}{\varrho\, Q\, g\, H} \; . \tag{12}$$

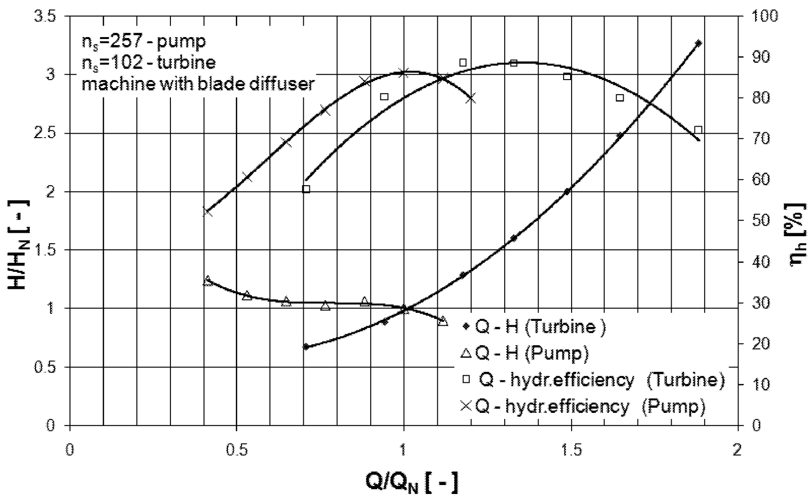

Fig.3: The performance characteristics of the pump $Q = 85\,l/s$, $H = 14.6\,m$, $n = 1800\,rpm$ in a pump and in a turbine operation mode – CFD analysis

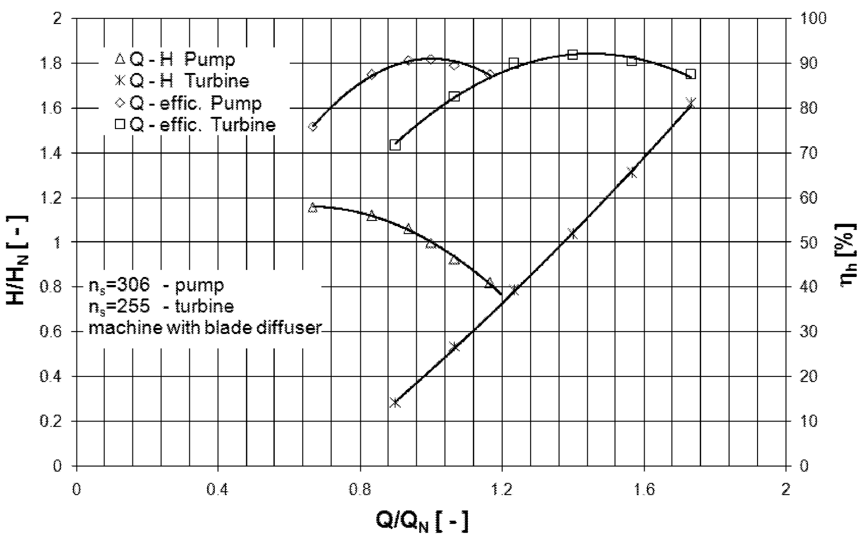

Fig.4: The performance characteristics of the pump $Q = 300\,l/s$, $H = 27.6\,m$, $n = 1850\,rpm$ in a pump and in a turbine operation mode – CFD analysis

4. The results of CFD simulation

At first the results of CFD simulation vere processed according to the equations stated abow. In the next step they were recalculated into the form of the non-dimensional performance characteristics. Then the hydraulic efficiences in a pump and in a turbine mode were assessed according to stated equations as well. The final shapes of the performance curves were obtained by data approximation through polynomial functions. The main purpose of the CFD simulation was to identify what is the ratio between the optimal flow-rates of the turbomachine in a pump and in a turbine mode. Assessing of the hydraulic efficiences in both modes was second important purpose of the CFD analysis. The legitimacy of the application of a pump in a turbine mode is given predominantly by acceptable energy effectivness of the turbomachine. The performance curves obtained by CFD analysis are in Fig. 3 and 4. Detailed CFD analysis of both geometries confirmed prospective courses. At first it was the shift of the BEP in a turbine mode in comparison with a pump mode for the pumps with different specific speeds. By assessing of a ratio between the optimal flow rates of both operation modes it is possible to say that the BEP is realy shifted toward the greater flow-rate and the optimal flow-rate of a pump in a turbine mode is approximately 1.35 times of the optimal flow rate in a pump mode. This value is in a good agreement with the relation (5). The results show that they are dependent on the value of the hydraulic efficiency in a pump mode as well as in a turbine mode. The absolute value of the hydraulic efficiency of the turbomachine is a relatively surprising result of the CFD simulation. It is possible to state that the efficiency in a pump an in a turbine mode is nearly same. But we have to say that boundary conditions implemented in the CFD simulation of both operation modes were basicly same, but they can not be kept in a real application of a pump in a turbine mode. This fact is shown in a relation between specific energy resp. head in both operation modes. The results of the CFD simulation are in a good agreement with prospective values, which correspodents to the tolerance stated in (6).

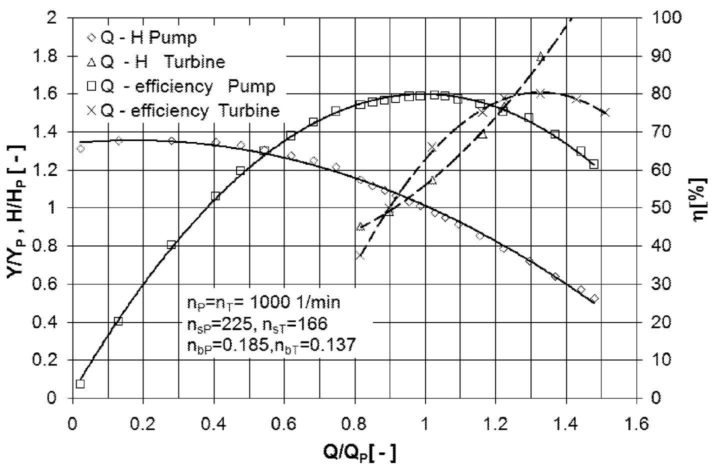

Fig.5: The performance characteristics of the volute pump $Q = 75\,l/s$ $H = 12\,m$, $n = 1500\,rpm$ in a pump and in a turbine operation mode – experiment [3]

The experimental measurements of the mixed-flow pump with a volute and specific speeds $n_s = 225$ in a turbine mode (Fig. 5) show very good agreement with the shift of the optimal flow-rate and head with the results of the CFD simulation. The good agreement with the

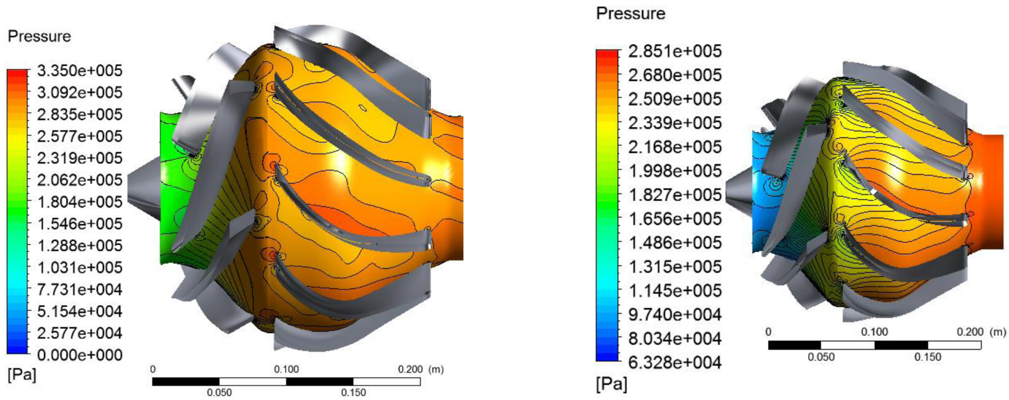

Fig.6: The distribution of the pressure in the blade channels of the pump $n_s = 257$ in a pump (left) and in a turbine (right) operation mode near by the BEP

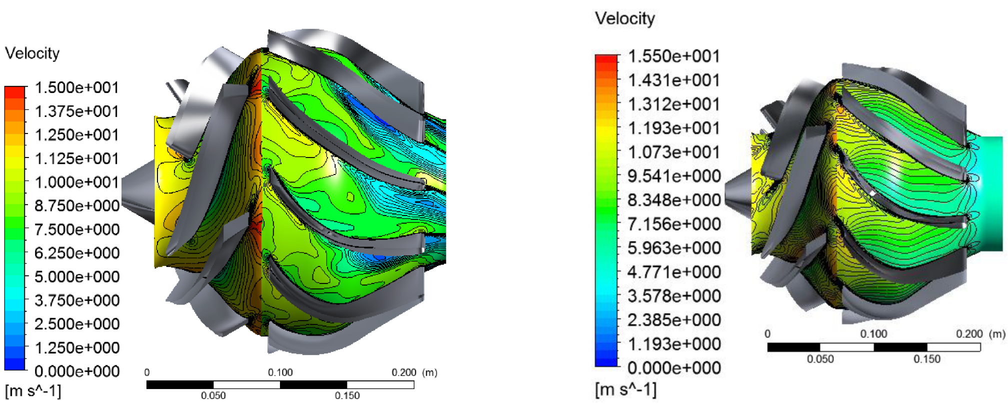

Fig.7: The distribution of the relative velocity in the blade channels of the pump $n_s = 257$ in a pump (left) and in a turbine (right) operation mode near by the BEP

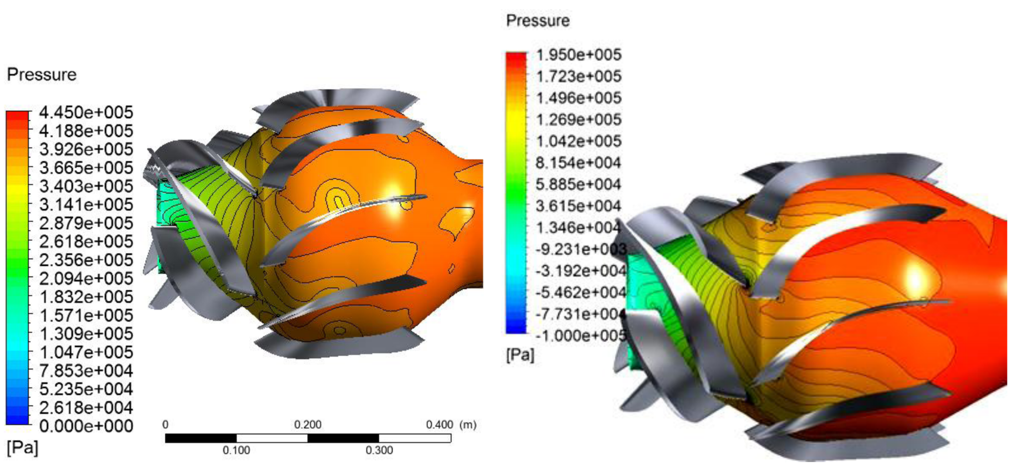

Fig.8: The distribution of the pressure in the blade channels of the pump $n_s = 306$ in a pump (left) and in a turbine (right) operation mode near by the BEP

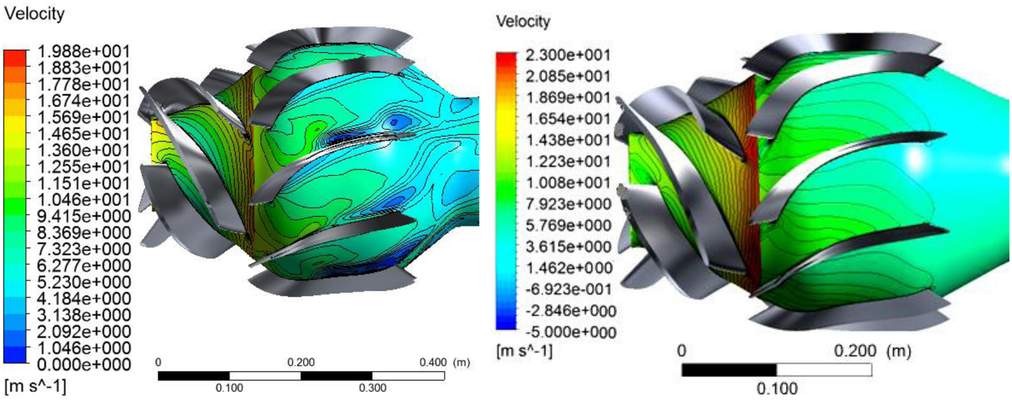

Fig.9: The distribution of the relative velocity in the blade channels of the pump $n_s = 306$ *in a pump (left) and in a turbine (right) operation mode near by the BEP*

assumptions is affirmed by the course of the main kinematic quantities and pressure obtained by CFD simulation. Fig. 6 shows the distribution of the pressure in the flow path in the pump (left part of the figure) and in the turbine operation mode (right part of the figure). Fig. 7 shows the distribution of the relative velocity in the both operation modes. The distributions of the pressure and relative velocity in the flow path of the second machine ($n_s = 306$ in the pump mode and $n_s = 255$ in the turbine mode) are shown in the Fig. 8 and Fig. 9. From the pictures arise the conclusion that the turbomachine has good efficiency in a pump as well as in a turbine mode.

5. Conclusion

The detailed CFD analysis of the mixed-flow pump in pump and in a turbine mode showed a good agreement between performance parameters while comparing experimental results in a pump and in a turbine operation mode. From this analysis in the form of performance characteristics and hydraulic efficiency these conclusions might be stated.

a. The experimental results and the results of CFD simulation demonstrate the same shift of the optimal flow-rate in a pump and in a turbine mode.

b. The specific energy of the machines is greater in a turbine operation mode than in a pump operation mode. Specific energy in the both operation modes differes in the value which is in the tolerrance of the relation (6). The values depent on the values of hydraulic efficiency. The higher efficiency of a turbomachine results smaller difference between specific energy (head) in a pump and in a turbine operation mode.

c. The experimental results shown in [7] which are valid for far smaller specific speeds than investigated pumps have shown that ratios between parameters in a pump and in a turbine mode correspondent to relations (5) and (6).

d. Mixed-flow pumps with a vane diffuser might be employed in a turbine operation mode without any troubles.

e. The hydraulic efficiency of the pump in a pump and in a turbine mode is practically the same.

f. The results of CFD simulation of the parameters of the mixed-flow pump can replace an expensive experiment with a good accuracy and they might be used by consideration about the application of a mixed-flow pump in a turbine operation mode.

References

[1] Varchola M., Sikhart R.: Porovnanie charakteristík diagonálneho stroja v čerpadlovej a turbínovej prevádzke, Zborník abstraktov, HYDROTURBO 2010, Patince 2010

[2] Bláha J., Melichár J., Mosler P.: Použití hydrodynamických čerpadel jako turbín, Energetika 5/2011

[3] Varchola M., Sikhart R., Gančo M., Strýček O.: Výskum diagonálneho čerpadla 150 BQZ v turbínovej prevádzke, HZ95/95 SjF STU

[4] Bláha J., Brada K.: Hydraulické stroje. Technický pruvodce 70, Česká matice technická SNTL 1992

[5] Güllich J.F.: Centrifugal Pumps, Springer Berlin Heidelberg New York, 2008

[6] Menter F.R., Langtry R., Hansen T.: CFD Simulation of turbomachinary flows – verification, validation and modeling, Otterfing Germany, 2004

[7] Lobanoff V.S., Ross R.R. : Centrifugal Pumps – Design and Application, Copyright 1985, 1995 by Butterworth-Heinemann, Gulf publishing Company, Houston TX

Received in editor's office: November 6, 2012
Approved for publishing: February 12, 2013