Numerical Investigation of the Clocking Effect on the Centrifugal Pump as Turbine

W Jiang1, Z L Liu1, Y C Wang1, J G Yang1 and G Y Hou1,2

1College of Water Resources and Architectural Engineering, Northwest A&F University, Yang Lin, Shaan Xi 712100, China

E-mail: gyhou415@nwafu.edu.cn

Abstract. The clocking effect is widely recognized to have an important impact on the function of rotating machinery. The effect of the clocking position on the flow field and the hydraulic performance in a pump as turbine (PAT) with vane diffusers has been investigated and analyzed numerically. The results show the hydraulic performance of the PAT decreasing gradually as the diffuser blade near the tongue. Under optimal operating conditions, the maximum difference in efficiency for different guide vane positions is 2.2%. When the position of the tongue is equidistant from the two diffuser blades, the energy loss of the volute and the diffuser is minimized. And the flow field results prove that the timing position affects the flow field in the volute and diffuser to a certain extent, but has little effect on the impeller. In addition, this may provide ideas for the installation of PAT with guide vanes, and the diaphragm should be as far away as possible from the two adjacent diffuser blades.

1. Introduction

Hydraulic turbines have the capability to extract energy from high-pressure liquids and convert it into mechanical energy. In industrial practice, the pressure and flow of high-pressure liquids vary widely, and accordingly, many kinds of hydraulic turbines are available for different applications. One of the most common types of hydraulic turbines for power production is the so called pump as turbine (PAT), including single-stage and multi-stage structures. PATs have many attractive features such as the construction simplicity, the low cost and the performance reliability. For these reasons, PATs are largely adopted in many industrial processes, small hydropower installations and other fields of energy-saving technologies.

Recently, many researchers have attempted to predict the performance of PATs by adopting both numerical simulations and experimental approaches, either based on pump mode performance [1-3] or pump geometric parameters [4-5] applied various Computational Fluid Dynamics (CFD) codes to research unsteady turbulent flows in PAT. Abazariyan et al. [6] proposed a correlation coefficient by simulating the effect of viscosity on the hydraulic performance of PAT to evaluate the relationship between efficiency and flow coefficient. Morros et al. [7], Fecarotta et al. [8] and Barrio et al. [9] performed numerical simulations to explore the transient flow characteristics in PATs. Fernández. J. et al. [10] and Arulmurugu, A. et al. [11] adopted a computational approach to investigate the radial thrusts in PAT, and the work highlighted that the radial force in PATs is generally larger than that in pump mode. Jain et al. [12] adopted an experimental approach aiming to predict the turbine performance on the basis of available values related to a pump performance, obtaining the performance curves of a pump as turbine at different specific speeds. Shinhama et al. [13] performed numerical simulations and
experiments to investigate the performance of pumps as turbines and the flow field up- and down-stream with respect to the impeller.

Most of the works available in literature focus on the hydraulic optimization of PAT. For example, Roth, S. et al. [14] and Sun H. et al. [15] analyzed several geometric set-up of the impeller in order to identify strategies suitable to enhance the performance of PAT. The results suggest that the modification applied at the periphery of the impeller blades, known as impeller rounding, was the most beneficial, leading to an efficiency rise in the range of 1.5%-2.0%. Yang. S. S.et al. [16] studied the influence of the trimming of the impeller on the hydraulic performance of PATs. Wang et al. [17] achieved the purpose of improving the efficiency and cavitation performance of PAT by improving the structure of the blade.

The so called clocking effect has a deep inspiration to the function of rotating machinery, which refers to changing the relative circumferential position between the rotor rows (or the two stator rows) in the same frame of the reference. Saren, V. E. et al. [18] first discovered the phenomenon of the diffuser position impact in a turbine. Since then, the majority of the available literature on the issue has been focused on the application of the approach to compressors and pumps [19-22], while only little attention has been paid to its potential adoption with PAT. The change of the stator diffuser position on the aerodynamic performance of compressors is investigated, revealing a potential benefit in terms of efficiency equal to 0.3%-0.7% [23-25]. Many scholars have studied the clock effect on the kinetic energy and pressure pulsation of compressors. [26-28]. The effect of the diffuser positions on the pressure fluctuation and hydraulic performance in pumps were analyzed by Jiang Wei et al. [29]. Tan Lei et al. [30] and Tan Minggao et al. [31] showing that the performance of pumps resulted improved for locations of the volute tongue approximately equidistant from the two diffuser vanes. Wang et al. [32] believe that the position of the guide vane has an influence on the main frequency of pressure pulsation, and is more affected by stator-rotor interference.

Although a large amount of studies has been dedicated to the clocking effect with regard to compressors and pumps, the clocking effect in PAT is a topic not fully explored yet and more research in this direction is strongly advisable. The aim of the present work is to investigate the impact of the diffuser position on PAT by using a CFD approach.

2. Computational model and methods

In order to explore the relation of the diffuser positions on the function of PAT, a computational model, consisting of a single-stage experimental centrifugal pump with vane diffuser, was adopted. The characteristic parameters include the discharge Q=40 m³/h, hydraulic head and rotational speed are 60m and 2900 rpm respectively; the specific speed N_s is assumed equal to 52 m/s.

The entire domain in the CFD model includes five section, as shown in Figure 1. In order to obtain a relatively accurate results, the lengths of outlet section and inlet section were expanded to five folds their respective diameters.

![Figure 1. Computational fluid domains](image-url)

In the pump mode, the impeller exhibits a clockwise circulation to suck the liquid into the entrance
without pre-swirl. The absolute velocity of the liquid raises in terms of tangential component, and also the static pressure increases due to the centrifugal force effect. In the PAT model, the impeller is subject to an anticlockwise rotation. The liquid in the diffuser forces the blade to put the impeller into rotation generating shaft power as output.

The structured hexahedral mesh in the commercial software ICEM CFD 17.1 had been used to mesh the computing domain, as shown in Figure 2. According to the experimental results, under the optimal conditions, head and efficiency of PAT are 60.5 and 60.8% respectively. Therefore, four grid schemes are selected in this paper as shown in Table 1, the error of scheme 3 and scheme 4 is less than 2%, but considering the calculation amount, scheme 3 is selected as the final scheme.

### Table 1. Verification of grid independence.

| Scheme | grid number($10^6$) | $H$ | $H$-error | $\eta$ | $\eta$-error |
|--------|---------------------|-----|-----------|--------|--------------|
| 1      | 1.22                | 64.12 | 0.06 | 58.40 | -0.04 |
| 2      | 3.61                | 62.82 | 0.04 | 59.20 | -0.03 |
| 3      | 5.66                | 61.75 | 0.02 | 60.40 | -0.01 |
| 4      | 7.45                | 61.48 | 0.02 | 61.20 | 0.01 |

The total grid number is about $5.66 \times 10^6$. The $y^+$ in this model is between 30 and 50.

The commercial software ANSYS-CFX 17.1 with SST turbulence model is applied in the numerical simulation for PAT with guide vane. The pressure is as the inlet and the mass flow as the outlet. And the all physical surfaces of the PAT adopt the no-slip boundary condition. The frozen-rotor and the transient-rotor-stator methods in steady and unsteady calculations are applied for the interfaces between the rotating and stationary components for, respectively. The initial condition of transient simulation is set to the steady-state simulation with a time step of $5.74 \times 10^{-5}$s.

### 3. Numerical method validations

The plastic injection was used to manufacture the impeller, the volute and the diffuser, as shown in Figure 3. With regard to the pump model, the performance was tested experimentally to verify the accuracy of the numerical simulation and a comparison between the two sets of results was performed. Although the hydraulic performance of the pump estimated numerically results in some cases larger than the experimental value (probably due to mechanical losses caused by mechanical seal, leakages and bearing through the balancing holes), the simulation results have similar results to the experiments, especially near the discharge ($Q=37 \text{ m}^3/\text{h}$–$48 \text{ m}^3/\text{h}$). For instance, for $Q=40 \text{ m}^3/\text{h}$, the error of the head is 5.1%, and 1.8% for the efficiency, indicating that the turbulence model and grid discretization may be suitable for the simulation in the pump model.
The accuracy of numerical results of the pump model can also be reflected by pressure pulsation. The comparison between the pressure fluctuations obtained by the numerical model and those measured experimentally for \( Q = 40 \text{ m}^3/\text{h} \) has been analyzed (see Figure 4 and Figure 5). Similar periodic pressure pulsations can be detected in both cases and detailed observations show that this pressure pulsation is caused by the Rotor–stator interaction. Figure 4 indicates that the simulation results and experimental measurements have the same trend regarding point P1, whereas the consistency between the two outputs results poor relatively to point P2, as shown in Figure 5. The flow frequency at P1 and P2 coincides with the blade-passing frequency \( (f_{BPF}) \) in both the simulation and experimental analysis. In addition, the fluctuation amplitudes of the pressure computed numerically at \( f_{BPF} \) are higher than that measured experimentally.
4. Numerical results

4.1. Hydraulic performance
Figure 6 shows the installation positions between the volute tongue and the diffuser vane in the PAT model. In the numerical simulation, the diffuser vane rotates counterclockwise every 12° to change the corresponding position of the diffuser vane. A total of 6 clocking positions are obtained refer to Table 2.

Table 2. Different cases computed by the numerical simulations

| Plan  | Plan 2 | Plan 3 | Plan 4 | Plan 5 | Plan 6 |
|-------|--------|--------|--------|--------|--------|
| angle /° | 0      | 12     | 24     | 36     | 48     | 60     |
| installation location /° | 41  | 29     | 17     | 5      | 65     | 53     |

Figure 6. Installation angles

Figure 7. Simulations results for the hydraulic performance of both pump and PAT assuming different clocking positions

(a) Comparison of the efficiency of pump  
(b) Comparison of the efficiency of PAT

Figure 8. Comparison of the efficiency for both pump and PAT under various diffuser positions
Figure 7 reveals the results of the simulations for the hydraulic performance of both pump and PAT. The analysis highlights that the diffuser position has some influence on the hydraulic efficiency of both models: in the pump model, when the tongue is situated in the middle between the two diffuser blades, the efficiency and head reaches the maximum, whereas their minimum is registered for positions of the vane near the tongue. Moreover, the difference between the maximum and minimum efficiency values increases with the flow rate (as shown in Figure 8). The conclusions obtained on the basis of the numerical simulations performed results consistent with the findings of previous studies (Jiang Wei et al. 2016). In the PAT model, the values of the hydraulic performance obtained for $\varphi=17^\circ$, $\varphi=29^\circ$ and $41^\circ$ are higher than those computed for $\varphi=5^\circ$, $\varphi=53^\circ$ and $65^\circ$ at the same flow rate. Furthermore, the efficiency reaches a maximum at $\varphi=17^\circ$, but it shows a local minimum for $\varphi=65^\circ$. And the difference between the values of the efficiency computed at $\varphi=17^\circ$ and $\varphi=65^\circ$ increases with the flow rate. Indeed, such gap results equal to 1.8%, 2.2% and 2.3% at $Q=48\text{m}^3/\text{h}$, $Q=56\text{m}^3/\text{h}$ and $64\text{m}^3/\text{h}$, respectively. In order to fully explain the change of the diffuser position on the performance of the PAT, the unsteady flow field has been analyzed in detail, as discussed in the following section.

4.2. Unsteady flow in PAT

In order to evaluate the performance of the diffuser or the volute in the PAT, the total pressure loss in these components can be defined as follows:

$$H_{\text{vloss}}, H_{\text{dloss}} = \frac{P_{\text{in}} - P_{\text{out}}}{\rho g}$$

where $H_{\text{dloss}}$ and $H_{\text{vloss}}$ are the loss in diffuser and volute respectively. $P_{\text{in}}$ and $P_{\text{out}}$ are the area weighted total pressure at the inlet and outlet. $\rho$ is the average density.

Figure 9 shows the comparison of the unsteady loss in the volute under various diffuser set-ups.

Figure 10. Instantaneous mean value of volute under various diffuser positions

Figure 9 shows the comparison of the unsteady loss in the volute under various diffuser positions. The results decorate that the fluctuation of the total pressure loss follows a periodic trend due to the
interaction between the impeller and the tongue. The energy loss in the volute is mainly affected by the absolute velocity, due to the special structure of the volute, the direction of the velocity in the volute is independent of the increase in flow rate. The hydraulic loss generated by the velocity grows along with the flow rate due to the related increase of the velocity. Moreover, in the volute section, the clocking effect seems to affect to some extent the total pressure loss, and the total energy loss increases when the diffuser blade is located near the tongue. For $\phi=17^\circ$, the pressure loss reaches a minimum in the volute, but it come to a maximum for $\phi=65^\circ$. Furthermore, the difference between the loss registered for $\phi=17^\circ$ and $\phi=65^\circ$ increases with the flow rate, resulting equal to 0.4m, 0.6m, 0.7m at $Q=48\text{m}^3$/h, $Q=56\text{m}^3$/h and $64\text{m}^3$/h, respectively (Figure 10).

When the diffuser vane gets closer to the volute tongue, the velocity in the volute increases and the area of the high velocity results wider (as shown in Figure 11), especially in the Section 6 and Section 8, this suggests that the loss produced by the velocity increases. The velocity in the volute calculated at $\phi=17^\circ$, $\phi=29^\circ$ and $41^\circ$, results lower than that estimated for $\phi=5^\circ$, $\phi=53^\circ$ and $65^\circ$. With regard to the high-velocity region, this results located at Section 7 and Section 8 as $\phi=17^\circ$, $\phi=29^\circ$ and $41^\circ$, but it extends to Section 6 for $\phi=5^\circ$, $\phi=53^\circ$ and $65^\circ$. The volute tongue causes the flow in the component to split into three parts (as shown in Figure 12): a fluid flow, from Section 8 to Section 1, going into the diffuser, two other components of the flow direct into the diffuser immediately and flow reverse from the tongue to Section 1. When the diffuser blade is located near the tongue ($\phi=5^\circ$, $\phi=53^\circ$ and $65^\circ$), the majority of the fluid moves along with the flow section. This phenomenon can lead to higher velocity values and hydraulic losses in the volute, especially in Section 6 and Section 8. In addition to this, a small portion of high-velocity fluid flowed from Section 8 to Section 1 can collide with that directed from the tongue to Section 1, resulting in larger loss near the tongue. For $\phi=17^\circ$, $\phi=29^\circ$ and $41^\circ$, the velocity in the volute and the impact loss in the near Section 1 is lower than that for $\phi=5^\circ$, $\phi=53^\circ$ and $65^\circ$ due to the presence of a portion of the fluid flowing into the diffuser immediately, indicating that the hydraulic loss in the volute can be smaller otherwise.

![Figure 11. Velocity in the volute under various diffuser positions at Q=56 m3/h](image-url)
Figure 12. Velocity streamline in the near tongue under various diffuser positions at Q=56 m$^3$/h

Figure 13 shows the unsteady total pressure loss in the diffuser for various diffuser positions. It is pointed out that the total pressure loss fluctuates periodically, due to rotor-stator interaction, under various discharges. The maximum value of the loss increases from 14 to 22.5 when the flow rate increases from 48 m$^3$/h to 64 m$^3$/h. Moreover, the loss in the diffuser is nearly double compared to that in the volute under the same operating conditions. The reason of this can be the presence of fluid entering the diffuser passage that, after converting the pressure energy into kinetic energy in the volute, presents higher velocity causing larger energy losses in the diffuser. A further explanation can be related to the strong non-uniformity of the flow field in the diffuser, due to the complex structure of the diffuser and the influence of the impeller blade trailing edge. Furthermore, the fluctuation of the loss in volute results is weaker than that in the diffuser, indicating that the unsteady energy loss in the diffuser is affected by the volute (upstream region) and the impeller (downstream region).

Figure 13. $H_{\text{vloss}}$ of diffuser under various diffuser installations
In addition to this, the clocking effect appears to have a obviously impact in the diffuser, as shown in Figure 13. Indeed, the loss in the diffuser increases for positions of the vane near the tongue: the total pressure loss registered at $\varphi=5^\circ$, $\varphi=53^\circ$ and $65^\circ$, is much higher than that computed for $\varphi=17^\circ$, $\varphi=29^\circ$ and $41^\circ$. Furthermore, the loss registers a maximum value at $\varphi=53^\circ$and a minimum at $\varphi=29^\circ$. In addition to this, the difference between the energy loss estimated at $\varphi=53^\circ$and $\varphi=29^\circ$ increases with the flow rate, being equal to 1.3m, 2.1m, 2.7m at $Q=48m^3/h$, $Q=56 m^3/h$ and $64 m^3/h$, respectively (as displayed in Figure 14). The results obtained suggest that the diffuser vane position effect in the diffuser has a much wider influence on the hydraulic loss of the PAT than that occurring in the volute. Moreover, the difference between the values of loss computed at $\varphi=17^\circ$ and $\varphi=65^\circ$ in the diffuser can equal more than twice that the values in the volute at the unchanging discharge: this indicates that the impact of the diffuser effect on the performance of PAT is mainly dominated by the performance of the diffuser.

Figure 14. Instantaneous mean value of diffuser under various diffuser positions

The regions where the maximum variation the total pressure gradient is registered are correspond to the diffuser inlet and outlet, the diffuser vane trailing edge and the leading edge (see Figure 15), indicating that a conspicuous portion of the energy loss occurs in these components. The clocking position mainly affects the total pressure of the diffuser inlet, especially for the diffuser passage located near the tongue. Moreover, the total pressure decreases gradually and with a strongly non-uniform trend in the diffuser passage located near the tongue, when the diffuser vane is located near the tongue. The fluid located at the near tongue flows along the diffuser vane into the diffuser passage immediately when the middle position of the diffuser flow passage is close to the tongue, in this case, the impact loss in the diffuser inlet results smaller. However, the incidence loss increases When the diffuser blade rotates to the tongue position due to the presence of two high-velocity fluids, which flowing from the tongue to Section 1 and from Section 8 to Section 1 respectively, collide at the diffuser inlet (as shown in Figure 16). The resulting vortex is clearly visible in Figure 16 in proximity of the diffuser inlet. These results support the hypothesis that the incidence loss occurring in the diffuser inlet plays a major role on the diffuser performance under all the clocking positions considered.

The performance of impeller is an important factor that affects PAT performance, as defined as follows:

\[
H_{imp} = \frac{P_{\text{out}} - P_{\text{min}}}{\rho g}
\]  

Where, $P_{\text{out}}$ is the total pressure at the outlet of the impeller, $P_{\text{min}}$ the total pressure at the inlet of the impeller, and $H_{imp}$ is the pressure drop of the impeller.
Figure 15. Total pressure in the diffuser under various diffuser positions at Q=56 m³/h

Figure 16. Velocity vector in the diffuser under various diffuser positions at Q=56 m³/h

On the basis of the results shown in Figure 17, it can be highlighted that the performance curves of the impeller present a good periodicity, and the peak of the performance increase with the discharge for all the clocking positions analyzed. Furthermore, the fluctuation of the performance appears more intense than that registered in the volute and diffuser at the same flow rate, indicating that the transient flow in the impeller may be affected by the interaction between the impeller blade and tongue or the diffuser blade. On the other hand, under the same flow rate, the pulsation and amplitude at different clocking positions are similar, and the diffuser position has little change on the performance of the runner. Moreover, the differences under various diffuser positions does not beyond 0.15 m (as shown in Figure 18), suggesting that the change of the diffuser effect on the loss of the PAT is not strongly affected by the performance of impellers. Similarly, the circumferential velocity in the impeller decreases gradually when the fluid flows from the impeller inlet to the impeller outlet (as shown in Figure 19), illustrating that the fluid forces the impeller into rotation through the blades, generating shaft power. In addition to
this, the magnitude of the circumferential velocity results similar for all the different diffuser set-ups analyzed, as well as the variation regions of the circumferential velocity. These results are a further proof that the clocking effect has a very limited improve on the flow field of the impeller.

$$Q = 48 \text{ m}^3/\text{h}$$

$$Q = 56 \text{ m}^3/\text{h}$$

$$Q = 64 \text{ m}^3/\text{h}$$

**Figure 17.** Himp under various diffuser installations

**Figure 18.** The instantaneous Himp under various diffuser positions

**Figure 19.** Circumferential velocity contour under various diffuser positions at Q=56 m³/h

5. Conclusions

In the present study, a numerical investigation on the impact of the diffuser position with regard to the hydraulic performance and the flow field of a pump as turbine was carried out. To accomplish the purpose of the research the model of a single stage pump as turbine (PAT) with vaned diffuser was implemented and analyzed.

The results obtained indicate that the diffuser position effect has the capability to influence the
operation of the PAT. When the diffuser blade is close to the volute tongue, the efficiency and head of PAT gradually increase. Thus, the clocking effect in PATs has not negligible consequences on the performance of the overall system and should be carefully considered. The major factors affecting the operation of the PAT under different diffuser positions result related to the performance in the volute and diffuser, but are less affected by the impeller. When the location of the tongue is equidistant from the two diffuser vanes, the hydraulic loss in the volute and diffuser reaches a minimum. Conversely, a maximum of the energy loss is registered for the diffuser blades approaching the tongue. In addition, the energy loss in the diffuser results more than double in comparison of that in the volute, which indicates that the PAT performance exhibits clocking effects that may be determined by the performance of the diffuser. Finally, the simulations registered a similar behavior and values for the operation of the impeller under various diffuser positions.

With regards to the flow field, the clocking effect resulted to have a significant impact in both volute and diffuser. When the diffuser blade approaches the tongue, the velocity in the volute increases and the area of the high velocity expands to a wider region. Meantime, the total pressure decreases gradually with a strong non-uniform trend in the diffuser passage located near the tongue. Similarly, the hydraulic loss (caused by the absolute velocity) and the impact loss in the volute registered higher values than those associated with locations of the tongue equidistant from the two diffusers vanes. The clocking position mainly affects the total pressure of the diffuser, especially for the diffuser passage located near the tongue. The distribution of the circumferential velocity in the impeller result approximately invariant to the different diffuser set-ups considered, indicating that the diffuser effect has little influence on the flow field in the impeller.

Finally, according to the analysis results of performance and the vector, the optimal installation position of the guide vane of a PAT was proposed, so that the tongue should be located in the middle of the guide vane flow channel.

Nomenclature

- $Q$: The discharge [m³/h]
- $H$: Head [m]
- $H_{\text{loss}}, H_{\text{vloss}}$: The energy loss [m]
- $n$: The rotating speed [rpm]
- $C_p$: The periodic pressure component
- $Z$: The number of blades
- $\rho$: The medium density [kg/m³]
- $\eta$: The pump efficiency [%]
- $b_2$: The outlet width of the impeller [mm]
- $b_1$: The inlet width of the diffuser [mm]
- $b_4$: The outlet width of the diffuser [mm]
- $b_5$: The inlet width of the volute [mm]
- $\phi$: The installation angle of the diffuser vane [°]
- $D_2$: The outlet diameter of the impeller [mm]
- $D_1$: The inlet diameter of the diffuser [mm]
- $D_4$: The outlet diameter of the diffuser [mm]
- $D_5$: The inlet diameter of the volute [mm]
- $N_s$: Specific speed [rpm, m³/s, m]
- $P_{\text{tin}}$: The transient area weighted average pressure [Pa]
- $P_{\text{tout}}$: The transient area weighted average pressure [Pa]

Acknowledgements

This research was financially supported by National Science Foundation (51809218, 51509209), China Postdoctoral Fund (2018M643744), the scientific research foundation on water engineering of Shaanxi
Province (2018slkj-8, 2017slkj-5).

References
[1] Derakhshan S and Nourbakhsh A 2008 Experimental study of characteristic curves of centrifugal pumps working as turbines in different specific speeds Exp. Therm Fluid Sci 32(3) 800-7.
[2] Páscoa J C, Silva F J, Pinheiro J S and Martins D J 2010 Accuracy details in realistic cfd modeling of an industrial centrifugal pump in direct and reverse modes Journal of Thermal Science 19(6) 491-9.
[3] Vasanthakumar P, Krishnaraj J, Karthikayan S, Vinoth T and Arun Sankar S K 2017 Investigation on reverse characteristics of centrifugal pump in turbine mode: a comparative study by an experimentation and simulation Materials Today: Proceedings 4(2) 693-700.
[4] Singh P and Nestmann F 2010 An optimization routine on a prediction and selection model for the turbine operation of centrifugal pumps Exp. Therm Fluid Sci. 34(2) 152-64.
[5] González J, Oro J M F, Argüelles-Diaz and Santolaria C 2009 Flow analysis for a double suction centrifugal machine in the pump and turbine operation modes Int. J Numer. Meth. Fl. 61(2), 220-36.
[6] Abazariyan S, Rafae R and Derakhshan S 2018, Experimental study of viscosity effects on a pump as turbine performance Renewable Energy 127 539-47.
[7] Morros C S, Oro J M F, Díaz K M A 2011 Numerical modelling and flow analysis of a centrifugal pump running as a turbine: unsteady flow structures and its effects on the global performance Int. J Numer. Meth. Fl. 65(5) 542-62.
[8] Fecarotta O, Carravetta A and Ramos H M 2011 CFD and comparisons for a pump as turbine: mesh reliability and performance concerns Int. J. Energy Environ. 2(1) 1590-5.
[9] Barrio R, Fernández J, Blanco E, Parrondo J and Marcos A 2012 Performance characteristics and internal flow patterns in a reverse-running pump-turbine ARCHIVE Proceedings of the Institution of Mechanical Engineers Part C Journal of Mechanical Engineering Science 226(3) 695-708.
[10] Fernández J, Blanco E, Parrondo J, Stickland M T and Scanlon T J 2004 Performance of a centrifugal pump running in inverse mode Proceedings of the Institution of Mechanical Engineers Part A Journal of Power & Energy 218(4) 265-71.
[11] Arulmurugu A and Vasanthakumar P 2015 Numerical investigation of centrifugal pump as turbine Advances and Applications in Fluid Mechanics 17(2) 147-63.
[12] Jain S V and Patel, R N. 2014 Investigations on pump running in turbine mode: a review of the state-of-the-art Renewable & Sustainable Energy Reviews 30(2) 841-68.
[13] Shinohama H, Fukutomii J, Nakase Y, CHIN Y.,Kuwauchi T and Miyauchi S 1999 Study on reverse running pump turbine Nihon Kikai Gakkai Ronbunshu B Hen/transactions of the Japan Society of Mechanical Engineers Part B 65(638) 3399-405.
[14] Roth S, Hasmatuchii V, Botero F, Mohamed F and Avellan F 2011 Influence of the pump-turbine guide vanes vibrations on the pressure fluctuations in the rotor-stator vaneless gap Journal of the Royal Naval Medical Service 96(2) 108-16.
[15] Sun H, Xiao R, Liu W and Wang F 2013 Analysis of s characteristics and pressure pulsations in a pump-turbine with misaligned guide vanes. Journal of Fluids Engineering 135(5) 511011.
[16] Yang S S, Kong F Y, Jiang W M and Xiao Y 2012 Effects of impeller trimming influencing pump as turbine Computers & Fluids 67 72-8.
[17] Wang T, Wang C, Kong F, Gou Q and Yang S 2017 Theoretical, experimental, and numerical study of special impeller used in turbine mode of centrifugal pump as turbine Energy 130 473-85.
[18] Saren V E, Savin N M, Smirnov S A, Krupa V G and Yudin V A 2016 Hydrodynamic interaction of axial turbomachine cascades Journal of Engineering Mathematics 55(4) 39-59.
[19] Pavesi G, Cavazzini G, and Ardizzon G 2016 Numerical analysis of the transient behaviour of a variable speed pump-turbine during a pumping power reduction scenario Energies 9(7) 534.
[20] Wang Z, Zhu B, Wang X and Qin D 2017 Pressure fluctuations in the s-shaped region of a reversible pump-turbine Energies 10(1) 96.

[21] Kim J H, Cho B M, Kim S, Kim J W, Suh J W, Choi Y S, Kanemoto T and Kim J H 2014 Design technique to improve the energy efficiency of a counter-rotating type pump-turbine Renewable Energy 101 647-59.

[22] Yao Y, Xiao Y, Zhu W, Zhai L, An Soo H and Wang Z 2014 Numerical analysis of a model pump-turbine internal flow behavior in pump hump district IOP Conference Series: Earth and Environmental Science 22(3) 032040.

[23] Pejovic S, Karney B W and Gajic A 2011 Analysis of pump-turbine s instability and reverse waterhammer incidents in hydropower systems Journal of Trauma 70(4) 970-7.

[24] Liu J, Liu S, Sun Y, Jiao L, Wu Y and Wang L 2013 Three-dimensional flow simulation of transient power interruption process of a prototype pump-turbine at pump mode Journal of Mechanical Science & Technology 27(5) 1305-12.

[25] Zeng W, Yang J and Guo W 2015 Runaway instability of pump-turbines in s-shaped regions considering water compressibility Journal of Fluids Engineering 137(5) 051401.

[26] Rzadkowski R, Gnesin V, Kolodyazhnaya and L Szczepanik R 2011 Stator clocking in first stage of so-3 compressor Advances in Vibration Engineering 10(2) 119-31.

[27] Schennach O, Pecnik R, Woisetschläger J, Göttlich E and Heitmeir F 2007 Influence of vane clocking on the unsteady flowfield in a one and a half stage transonic turbine Journal of Turbomachinery 130(3) 538-44.

[28] KÖ nig S, Stoffel B and Taher Schobeiri M 2009 Experimental investigation of the clocking effect in a 1.5stage axial turbine-part i: time-averaged results Journal of Turbomachinery 131(2) 021003-14.

[29] Jiang W, Li G, Liu P F and Fu L 2016 Numerical investigation of influence of the clocking effect on the unsteady pressure fluctuations and radial forces in the centrifugal pump with vaned diffuser International Communications in Heat & Mass Transfer 71 164-71.

[30] Tan M, He N, Liu H, Wu X and Ding J 2016 Experimental test on impeller clocking effect in a multistage centrifugal pump Advances in Mechanical Engineering 8(4).

[31] Qu W, Tan L and Cao S, Wang Y and Xu Y 2016 Numerical investigation of clocking effect on a centrifugal pump with inlet guide vanes Engineering Computations 33(2) 465-81.

[32] Wang W, Pei J, Yuan S and Yin T 2017 Experimental investigation on clocking effect of vaned diffuser on performance characteristics and pressure pulsations in a centrifugal pump Experimental Thermal and Fluid Science 90 286-98.