Improvement of Internal Flow Performance of a Centrifugal Pump-As-Turbine (PAT) by Impeller Geometric Optimization

Jian Xu, Longyan Wang, Stephen Ntiri Asomani, Wei Luo and Rong Lu

Abstract: Rotor-stator interaction (RSI) in the centrifugal pump-as-turbine (PAT) is a significant source of high amplitude of the pressure pulsation and the flow-induced vibration, which is detrimental to the stable operation of PAT. It is therefore imperative to analyze the rotor-stator interaction, which can subsequently be used as a guideline for reducing the output of PAT noise, vibration and cavitation. In addition, it is important for a PAT to have a wide operating range preferably at maximum efficiency. In order to broaden the operating range, this work proposes a multi-condition optimization scheme based on numerical simulations to improve the performance of a centrifugal PAT. In this paper, the optimization of PAT impeller design variables ($b_2$, $\beta_1$, $\beta_2$ and $z$) was investigated to shed light upon its influence on the output efficiency and its internal flow characteristics. Thus, the aim of the study is to examine the unsteady pressure pulsation distributions within the PAT flow zones as a result of the impeller geometric optimization. The numerical results of the baseline model are validated by the experimental test for numerical accuracy of the PAT. The optimized efficiencies based on three operating conditions ($1.0Q_d$, $1.2Q_d$, and $1.4Q_d$) were maximally increased by $13.1\%$, $8.67\%$ and $10.62\%$, respectively. The numerical results show that for the distribution of PAT pressure pulsations, the RSI is the main controlling factor where the dominant frequencies were the blade passing frequency (BPF) and its harmonics. In addition, among the three selected optimum cases, the optimized case C model exhibited the highest level of pressure pulsation amplitudes, while optimized case B reported the lowest level of pressure pulsation.

Keywords: pump-as-turbine (PAT); geometric optimization; internal flow characteristics; rotor-stator interaction (RSI); blade passing frequency (BPF)

1. Introduction

The need for electrical power today is on the rise, owing to the population and economic growth. Life without electricity today is practically impossible and unpleasant because it has now become a part of our everyday life. However, getting electricity to remote areas poses a problem due to the topographical nature of these areas. Yet, most remote areas have streams and rivers with high and low head that can be difficult to generate electricity at a cheaper rate. To curb this problem, the ideal method is to apply the pump-as-turbine (PAT) technology which has the tendency to generate electricity at a cheaper rate and be more environmentally friendly. In contrast to other types of small hydraulic devices, PAT has the advantages of being small, relatively cheap and easy to produce and
maintain. Due to it numerous advantages, the PAT has been widely used in many fields of engineering, including oil and chemical industries, small- or micro-hydropower plants, et cetera. [1]. Although the use of PAT is associated with substantial benefits, there are several significant disadvantages that should not be ignored. A major drawback of PAT is the lack of guide vanes and flow control devices which limit the operating range and the ability to control the operating point efficiently [2,3]. A wide range of operations is of great significance for a PAT to perform at optimum efficiency by saving energy [4,5]. Another issue relating to PAT is the difficulty to predict the actual performance of PAT mode performance [6,7]. The efficiency of PAT compared with pump mode operation is also significantly lower at off-design conditions [8].

The current PAT research is mainly directed towards developing precise prediction of the performance of turbine mode [9] and optimizing the design [10]. Very few researches have focused on the unsteady pressure field analysis of PAT [11]. The transmission of fluid machinery is unsteady due to the impulses of the rotating blades with respect to the stator [12]. The interaction between rotor and stator produces strong pressure pulsations which affects not only the hydraulic performance but also causes excessive noise and vibration [13,14]. This in turn jeopardizes the longevity of the components. Hence, to improve the overall performance and reliability, it is very important to investigate and analyze the mechanism of the pressure pulsations in PAT.

Several research studies have investigated the unstable flow characteristics in hydraulic pumps, and the following are among them. Pei et al. [15] numerically researched a radial pump and predicted the pressure pulsations field within the entire flow channel. Numerical investigations were carried out by Spence and Aaral-Teixeira concerning the effect of four geometrical parameters on pressure pulsation in the centrifugal pump: blade tip clearance, vane configuration, hemlock radial gap and sidewall clearance [16]. As a result, increasing the distance between the rotor and the stator was found to be an effective and reasonable means to reduce the amplitude of pressure pulsation. In the pump mode, the pump-turbine flow characteristics were examined by Yao et al. [17]. The results showed that in the tube draft, runner and diffuser complex flows, such as backflow or vortex, lead to the large pulsations of pressure. Aiming at a double suction centrifugal pump, Yao et al. [18] again studied time-frequency characteristics of pressure pulsation using a fast Fourier transform (FFT) algorithm and time-frequency methods. It was found that under off-design conditions, some unsteady flow phenomena occurring inside pumps have considerable effect on the pressure pulsation. Furthermore, the authors investigated the pressure pulsations caused by rotor-stator interactions in axially split centrifugal pumps under design and off-design operating conditions. They showed that the pulsating behavior of the blade passing-through frequency was most prominent near the volute tongue, and the pressure wave propagates in both radial and circumferential directions. Tan et al. [19] showed that the volute tongue region of a centrifugal pump is relatively uniform in the design operational condition of unsteady pressure and velocity pulsations. The effect of the volute tongue on the pulsations of PAT was investigated by Yang et al. [20]. The result indicated that the amplitude of the pressure pulsation decreases as the radial clearance increases. Barrio et al. [21] also performed a computational analysis using computational fluid dynamics (CFD) codes on the unstable PAT flow. The study found that the flow only matches the geometry of the impeller in reverse mode under design conditions, and recirculating fluid regions form at part load (close to the blade discharge) and at overload (near the suction side). From the above literature, it can be established that most studies through numerical methods on the pressure pulsations were carried out in pump mode with very few in PAT mode.

As pumps are not designed to operate as turbines, their geometrical parameters are determined without taking the PAT mode operation into account [22]. Therefore, to improve the operational reliability and efficiency, the hydraulic design of the PAT has to be optimized to suit both operational modes. The traditional hydraulic design of the PAT however deeply depends on the designers’ experience, which restricts its development [23]. As such, to improve the efficiency and operational reliability of the PAT, there is the need to apply advanced optimization algorithms so that optimization would not solely depend on the designers’ experience. Several researchers have proposed modifications
to optimize the performance of the hydraulic characteristics. These modifications include trimming the impeller blades, adjusting the blade number, adding splitter blades, installing guide vanes and rounding impeller leading edges, et cetera. [24–26].

To save computational resources and obtain the optimal parameter combination accurately within a certain range, a surrogate model and an optimization algorithm must be applied. The application of optimization algorithm with the surrogate model (which can build a function between design variable and optimization objective) can improve pump performance efficiently. This method has been widely applied in centrifugal pumps optimization. Nourbakhsh et al. [27] used a particle swarm optimization method and non-dominated sorting genetic algorithm (NSGA) II algorithm with an artificial neural network (ANN) model to find the Pareto front of two conflict objectives of centrifugal pumps: efficiency and NPSHr. Derakhshan et al. [28] also used a global optimization method based on the ANN and the artificial bee colony algorithm along with a validated 3-D Navier–Stokes flow solver to redesign the impeller geometry and improve the performance of a pump.

Above all, the primary aim of the study is to optimize the geometrical parameters of the PAT impeller ($b_2$, $\beta_1$, $\beta_2$ and $z$) at a wide range of flow conditions to examine its influence on hydraulic performance. Secondly, the aim is to examine, analyze and predict the unsteady pressure pulsation distributions within the PAT flow zones under different operating conditions. This study proposes an optimization procedure combining the design of experiment (DOE), surrogate models and Pareto-based genetic algorithm (PBGA) with numerical simulation to improve hydraulic efficiencies and operational reliability of a PAT under $1.0Q_d$, $1.2Q_d$ and $1.4Q_d$ ($Q_d$ is the design flow rate) flow conditions. The surrogate model is applied to construct a nonlinear function between the objective functions and the design variables at design and off-design conditions. Moreover, the simulation accuracy is verified by comparison with the experimental data for the baseline PAT model, and the flow field analysis of the original design and Pareto-optimal design are compared to illustrate the performance improvement of PAT.

2. Numerical and Experimental Setup

2.1. PAT Modelling

This study selects a baseline centrifugal pump which operates both in pump and turbine modes in National Research Centre of Pumps. The detailed design parameters of the pump are as follows: 25 m$^3$/h flow rate; 8.5 m head; 1450 r/min rotating speed and 81.5 specific speed. Figure 1 shows the computational flow domain of the pump model working as a turbine. As can be seen, the flow enters from the volute, then drives the impeller to rotate and finally exits from the outlet pipe.

![Computational flow domain of the numerical model.](image)
2.2. Computational Setup

In the study, ANSYS ICEM software is applied to create hexahedral structured grids for all components of the numerical model. The impeller component is periodically meshed which is set as the rotational domain. The hub, shroud, back and front chamber parts are meshed separately, while the volute and the discharge pipe are put together as one domain. The structural grids for the three domain parts are shown in Figure 2. Particularly, the meshes close to the interfaces between the impeller and the radial volute are densified to increase the complexity of the flow fields in that region due to its flow exchange characteristics. To consider the effect of the number of grids on the calculation accuracy, the grids were defined in the near-wall flow regions and the mesh independence check was performed with the grid numbers. The total number of grids was approximately 2.9 million [29]. After the analysis, the mesh of $2.9 \times 10^6$ grids is selected for the CFD simulation study of the baseline pump model since it reaches a good balance of the accuracy and computational cost.

![Figure 2. Meshes for the different computational flow domains. (a) Hub and shroud, (b) impeller blades, and (c) volute with tongue enlargement.](image)

For the computational setup of the simulation, the same setting as in reference [15] was preferred. For the turbulence model, the standard $k$-$\omega$ STT turbulence model as shown in Equations (1)–(4) is employed. The impeller operates at 1450 r/min, and 1000 iterations are calculated in the steady state. For the unsteady simulation, eight complete revolutions are computed while the time step of 1° rotational degree is set.

\[
\frac{D}{Dt}(\rho \omega) = \nabla (\rho D_\omega \nabla \omega) + \frac{\rho \gamma G}{\nu} - \frac{2}{3} \rho \gamma \omega (\nabla \cdot \mathbf{u}) - \rho \beta \omega^2 - \rho (F_1 - 1)CD_{kw} + S_\omega \tag{1}
\]

\[
\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D_k \nabla k) + \rho G - \frac{2}{3} \rho k (\nabla \cdot \mathbf{u}) - \rho \beta^* \omega k + S_k \tag{2}
\]

where, in the equation, the expression of turbulent viscosity coefficient $\nu_t$ is expressed by:

\[
\nu_t = a_1 \frac{k}{\max(a_1 \omega b_1 F_2^3 S)} \tag{3}
\]

\[
a_{k1} = 0.85, a_{k2} = 1, \alpha_{\omega 1} = 0.5, \alpha_{\omega 1} = 0.856, \beta_1 = 0.075, \beta_2 = 0.0828, \gamma_1 = 5/9, \gamma_2 = 0.44, \beta^* = 0.09, a_1 = 0.31, b_1 = 1, c_1 = 10 \tag{4}
\]

2.3. Experimental Setup

A complete schematic of the experimental set up is shown in Figure 3. The experimental set up with pressure gauge, valves, underground tank and pipes was installed. The baseline pump model introduced above is used as the PAT. The flow rate is measured using an electromagnetic
flow meter installed in the main pipe line. The inlet pressure is measured by an electronic pressure
gauge and outlet pressure is measured by a static pressure sensor with the measuring range up to
2 MPa. To measure the output power, a synchronous generator is coupled to the PAT. The torque is
acquired with a torque sensor and an auxiliary pump is also used to supply the water at desired head
and flow rate. The auxiliary pump supply water is from the underground tank to the inlet of the
pump tested. The real PAT test rig operation is shown in Figure 4. Due to the chaos of the real test
rig background, the labels of different components in Figure 4 are not marked which can be easily
identified by comparison with Figure 3.

![Figure 3](image-url)

**Figure 3.** Schematic layout of the pump-as-turbine (PAT) experimental setup: (a) motor, (b) pump inlet pipe, (c) pump outlet pipe, (d) bypass valve, (e) bypass pipe, (f) electromagnetic flow rate, (g) generator, (h) PAT, (i) inlet and outlet pressure transmit and (j) PAT outlet pipe.

![Figure 4](image-url)

**Figure 4.** Completed PAT setup for validating the baseline model.

### 3. Optimization Process

As shown in Figure 5, the optimization process is divided into four main steps. First,
the optimization objective is selected, and the design variables and variables range are determined
by a sensitivity test. Second, the sampling space is obtained by design of experiment (DOE). In this
space, the objective values according to every design variable are calculated by ANSYS CFX software.
Third, the surrogate model is constructed and the accuracy of the surrogate model is verified by error
analysis. Finally, the Pareto-optimal solutions are found by running the optimization algorithm.
3.1. Optimization Objective

To broaden the operating range of the PAT efficiency area, the efficiencies based on design point, $1.0Q_d$, best efficiency point (BEP), $1.2Q_d$ and overload point, $1.4Q_d$ are selected as the optimization objectives. The PAT efficiency is defined as:

$$\eta = \max \left( \frac{P}{\rho gQH} \right)$$  \hspace{1cm} (5)

where $\rho$ and $g$ are the flow density and acceleration of gravity, respectively. $Q$, $H$ and $P$ are the flow rate, head and mechanical power, respectively.

3.2. Design Variables and Surrogate Models

Due to the design experience and impeller geometrical parameters that have great impact on the efficiency and head [30], the blade outlet width $b_2$, blade exit angles $\beta_1$ and $\beta_2$ on the hub and shroud and blade number $z$ are chosen as the input variables, and their limits used as the optimization constraint are shown in Table 1 [31].

<table>
<thead>
<tr>
<th>Design Variables</th>
<th>Lower</th>
<th>Original</th>
<th>Upper</th>
</tr>
</thead>
<tbody>
<tr>
<td>$b_2$/mm</td>
<td>0.010</td>
<td>0.012</td>
<td>0.015</td>
</tr>
<tr>
<td>$\beta_2^{\beta}$</td>
<td>20</td>
<td>30</td>
<td>35</td>
</tr>
<tr>
<td>$\beta_1^{\beta}$</td>
<td>20</td>
<td>30</td>
<td>35</td>
</tr>
<tr>
<td>$z$</td>
<td>4</td>
<td>6</td>
<td>8</td>
</tr>
</tbody>
</table>

In order to establish the relationship between the input parameter and the output properties of PAT, the artificial neural network (ANN) is used for the surrogate modeling [32]. Figure 6 shows the general structure of the ANN and it has been adopted to fit the PAT efficiencies at the part load, BEP and overload flow conditions with four design variables of the impeller. The steps for the basic principles of the ANN were carried out in our previous study [31]. As shown in the Figure 6, the ANN
includes input layer, hidden layer and output layer. The hidden layer and output layer are connected by a weight and activation function. As a result, the trained predictive model can accurately predict the PAT efficiency.

![Artificial neural network (ANN) structure for PAT efficiency objective.](image)

**Figure 6.** Artificial neural network (ANN) structure for PAT efficiency objective.

### 3.3. Multi-Objective Optimization

A multi-objective optimization problem is different from a single objective optimization problem because of the requirement to optimize several objectives. For multi-objective problems with several conflicting objectives, there is usually an optimal solution trade-off. The advantage of a PBGA is that the method for determining multiple objectives is reduced to a single fitness measure by the creation of a number of fronts that are then sorted according to non-domination. The mathematical steps of the multi-objective optimization calculations are done with reference to [31]. The detailed setting parameters of the PBGA are listed in Table 2. The population size and number of generations will directly affect the calculation cost of the optimization process and the quality of the optimization results. Therefore, in order to avoid inbreeding and poor convergence in the optimization process, the population number and the number of generations are determined. In addition, crossover fraction and function tolerance are set to 0.85 and 10, respectively, to ensure population diversity and better convergence.

**Table 2.** Setting parameters of the Pareto-based genetic algorithm (PBGA).

<table>
<thead>
<tr>
<th>Design Variables</th>
<th>Lower</th>
</tr>
</thead>
<tbody>
<tr>
<td>Population size</td>
<td>100</td>
</tr>
<tr>
<td>Number of generations</td>
<td>1000</td>
</tr>
<tr>
<td>Pareto-front population</td>
<td>0.8</td>
</tr>
<tr>
<td>Crossover fraction</td>
<td>0.85</td>
</tr>
<tr>
<td>Function tolerance</td>
<td>10</td>
</tr>
</tbody>
</table>

### 4. Results and Discussion

#### 4.1. Experimental Validation

The optimization of a PAT relies on the numerical simulation method which starts from the baseline PAT model. Hence, the accuracy of CFD simulation is critical to the reliability of the optimization results. In this section, the experimental test result of the baseline PAT model is presented before introducing the optimization outcomes. The hydraulic PAT characteristics curves of the CFD and the experimental test are plotted at different flow conditions. The numerical calculations in this research are carried out in line with the experimental test results for validation. Figure 7 shows characteristics of the efficiency and head curves. Here, the efficiencies of the PAT increase to reach the best efficiency point (BEP) at flow rate of 30 m$^3$/h and begin to decline afterwards under both CFD and experimental measurements as established in Figure 7a. At the best efficiency point, the CFD and the experimental setup recorded hydraulic efficiency of 75.35% and 74.15%, respectively. The efficiencies of the CFD data compared with their respective experimental data increase about 1.6%, respectively. Thus, the CFD results record a slightly higher efficiency than the experiments and this can account for the flow
leakages and other high turbulent effects that exist in the flow under these operating points. This can also be attributed to the mechanical losses ignored during the numerical calculations.

Figure 7. Performance characteristic curve of the tested PAT and computational fluid dynamics (CFD): (a) efficiency (b) head.

Figure 7b shows the pressure head characteristics curve. Generally, it can be observed that with increasing flow rate, the head of the CFD and experimental test increased. In turbine mode the head increased steadily as the flow rate increased from part-load to overload operating condition.

4.2. Analysis of Sensitivity Test

The sensitivity test analysis on design variables of the geometry is very important to optimization procedures. These tests are applied to establish the level of influence between the design variables and the objective function. In this study, Isight software is employed for the sensitivity analysis. Figure 8 shows the effect of the main geometry parameters of the impeller on PAT efficiency under flow conditions (1.0\(Q_d\), 1.2\(Q_d\), 1.4\(Q_d\)). As shown in Figure 8, the selected geometric parameters which are the blade exit angles, blade inlet width and blade number (\(\beta_2\), \(\beta_1\), \(b_2\) and \(z\)) have minimal impact on PAT efficiency under design point 1.0\(Q_d\). However, the biggest impact on the PAT efficiency was established under overload flow condition 1.4\(Q_d\).

Figure 8. Correlation of the input and output variables.

Figure 8 presents a correlation between the input and outputs values. The correlations values are ranked from zero (0) to positive one (+1). If the value is closer to (+1) it indicates a high correlation between the input and output, and if the value is closer to (0) it indicates a weak correlation. The negative
and positive signs shows whether the inputs and the outputs variables are negatively or positively correlated. At the design operating point $1.0Q_d$, the design variable $\beta_2$ has the highest correlation value of 0.52, followed by the blade width $b_2$ which has a negative correlation value of $-0.17$. The blade number had the least correlation. At the best efficiency point (BEP), $\beta_2$ recorded a positive correlation value of 0.62, followed by the blade number $z$ which recorded a positive correlation value of 0.14. The blade width $b_2$ had the least correlation value of 0.2. Similarly, in the overload operation $1.4Q_d$, the design variable $\beta_2$ recorded the highest correlation value of 0.58, followed by the blade width $b_2$ with a positive correlation value of 0.36. The blade angle $\beta_1$ also recorded the least correlation value of 0.07 as established in Figure 8. It has been established that the four main geometry parameters of the impeller ($\beta_2$, $\beta_1$, $b_2$ and $z$) plays an important role under $1.0Q_d$, $1.2Q_d$ and $1.4Q_d$ for PAT efficiency. Hence, these four geometry parameters were selected as design variables.

Figure 9 shows a bar graph that ranks the design variables (inputs) over the objective function (outputs). The significance of these graphs is to establish which of the design variables have the most influence on the objective function. The blue and red colors in Figure 10 indicate a positive and negative relationship between the input and the output variables, respectively. It can be seen that at design point $1.0Q_d$ that the order of influence of the parameters on the PAT efficiency are, $\beta_2 > z > \beta_1 > b_2$; at the BEP $1.2Q_d$ the order is $\beta_2 > \beta_1 > b_2 > z$; at the overload operating condition $1.4Q_d$ the order is $\beta_2 > b_2 > \beta_1 > z$. It implies that $\beta_2$ has a dominant influence on the single objective of maximizing the PAT efficiency.

![Figure 9](image-url)

**Figure 9.** Ranks of the design variables over the PAT efficiency at (a) $1.0Q_d$, (b) $1.2Q_d$ and (c) $1.4Q_d$.

### 4.3. Analysis of Multi-Objective Optimization

The prediction functions between design variables and objective functions were built by using an ANN model. The Pareto-based genetic algorithm (PBGA) was used to obtain 1000 groups of Pareto-optimal solutions as shown in Figure 10. Three representative points A, B and C were selected to verify the accuracy of PBGA prediction. The comparison of the efficiencies for the original baseline model and the three optimized cases at the design point, best efficiency point (BEP) and overload flow conditions is presented in Table 3. Evidently, the optimized cases have a great improvement in the PAT efficiency compared to the baseline model.
Figure 10. Pareto-frontiers from ANN.

Table 3. Comparison of baseline model and the optimized cases.

<table>
<thead>
<tr>
<th>Cases</th>
<th>b2</th>
<th>beta2</th>
<th>beta1</th>
<th>z</th>
<th>1.0Qd</th>
<th>1.2Qd</th>
<th>1.4Qd</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.012</td>
<td>30</td>
<td>30</td>
<td>6</td>
<td>73.9184</td>
<td>78.8454</td>
<td>78.5914</td>
</tr>
<tr>
<td>A</td>
<td>0.014</td>
<td>33.52</td>
<td>23.49</td>
<td>8</td>
<td>81.3734</td>
<td>84.8116</td>
<td>85.9235</td>
</tr>
<tr>
<td>B</td>
<td>0.013</td>
<td>23.06</td>
<td>28.09</td>
<td>8</td>
<td>83.5815</td>
<td>85.6957</td>
<td>86.9401</td>
</tr>
<tr>
<td>C</td>
<td>0.013</td>
<td>23.88</td>
<td>29.26</td>
<td>8</td>
<td>76.3896</td>
<td>79.4484</td>
<td>80.741</td>
</tr>
</tbody>
</table>

Figure 11 presents a comparison of the PAT model and the optimized cases A, B and C at different flow conditions. Table 4 indicates the improvement of the efficiencies for different optimized PAT cases with respect to the baseline model. At design point, optimized case B performed better with an increase of 13.1% in efficiency, while optimized cases A and C increased by 10.1% and 3.34%, respectively. At the best efficiency point (BEP), optimized case B increases by 8.67% in efficiency, while optimized cases A and C increased by 10.1% and 3.34%, respectively. At overload condition, optimized case B’s increase in efficiency is 10.62% while the optimized cases A and C’s increases are 9.32% and 2.74%, respectively.

![Graph showing comparison of PAT efficiencies for different case scenarios.](image-url)

Figure 11. Comparison of PAT efficiencies for different case scenarios.
4.4. Analysis of Internal Flow Characteristics

To investigate how the impeller design variables ($\beta_2$, $\beta_1$, $b_2$ and $z$) impact the PAT internal flow field, the unsteady CFD simulations of the optimized PAT models are carried out at the flow rate of 25 m$^3$/h and rotational speed of 1450 r/min. The resultant optimized PAT flow characteristics are presented and analyzed.

4.4.1. Static Pressure Distribution

(a) Volute Flow Passage

For the present optimized cases A, B and C, the turbulent pressure distribution contours under different flow conditions in the PAT volute are presented in Figure 12. It can be observed that the static pressure within the flow domain declines from the volute inlet to the outlet of the PAT. A relatively high pressure region is observed at the inlet of the volute in all the optimized cases. As the flow progresses through the volute flow channel, the static pressure decreases due to an increase in velocity as a result of the gradual reduction of the cross-sectional area of the volute. The pressure distribution in the volute flow channel is not periodic due to the asymmetric shape of the volute and the interaction between the impeller and the volute.

The flow at the volute tongue region exhibits higher pressure gradients, for all three optimized impeller cases. In the regions close to the volute tongue under design point, BEP and the overload exhibited high pressure fluctuation intensity in all the optimized impeller cases A, B and C. It can also be observed that as the flow condition increases from design point to overload, the magnitude of the pressure fluctuation intensity increases in unison. It is worth noting that pressure distribution within the volute casing is dependent on flow rate. Under all the flow conditions, the BEP ($1.2Q_d$) exhibits a slightly smoother flow in all the optimized cases.

However, by comparing pressure fluctuation intensity in the optimized cases A, B, and C; optimized case C, exhibited high-pressure fluctuation intensity under all flow conditions, indicating high operational instability. This can be attributed to poor parameter combination between the volute and the optimized impeller and rotor-stator matching.

(b) Impeller Flow Passage

The pressure fluctuation intensity distribution within the impeller flow zones are shown in Figure 13. The pressure fluctuation intensity shows a continuous decrease from the blade leading edge (BLE) to the blade trailing edge (BTE) of the impeller. It is observed that static pressure on the pressure side (PS) of the blade is more intense than the suction side (SS). Thus, the impeller is pushed to rotate by the pressure difference between the surfaces. The pressure difference from the pressure side to the suction side of the impeller blade decreases from the blade leading edge (BLE) to blade trailing edge (BTE).

The minimum value of the static pressure inside the impeller is located at the trailing edge of the blades on the suction side. The decreasing static pressure from the BLE to the BTE of the impeller is due to the continuous transmission of energy from the fluid to the impeller blades. The pressure fluctuation intensity within the optimized cases increased as the flow conditions increased from design point ($1.0Q_d$) to overload point ($1.4Q_d$). Comparatively, optimized case C generated the higher pressure fluctuation intensity indicating likelihood of high vibration and noise; this phenomenon can be attributed to bad parameter combination of the design variables or impeller-volute matching. However, optimized case

Table 4. Improvement of optimized cases with respect to the baseline model.

<table>
<thead>
<tr>
<th>Cases</th>
<th>Design Point ($1.0Q_d$)</th>
<th>BEP ($1.2Q_d$)</th>
<th>Overload Point ($1.4Q_d$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>10.1%</td>
<td>7.56%</td>
<td>9.32%</td>
</tr>
<tr>
<td>B</td>
<td>13.1%</td>
<td>8.67%</td>
<td>10.62%</td>
</tr>
<tr>
<td>C</td>
<td>3.34%</td>
<td>0.76%</td>
<td>2.74%</td>
</tr>
</tbody>
</table>
A exhibited moderate pressure fluctuation intensity compared with optimized case B, indicating that the design variables for optimized case A have a high tendency of improving PAT performance.

Figure 12. Pressure distribution within the volute of the optimized cases A, B and C under flow conditions of: (a) 1.0Q_d (b) 1.2Q_d and (c) 1.4Q_d.
(c) Outlet Pipe Flow Passage

Pressure fluctuation distribution contours at the outlet pipe of each optimized case have been presented in Figure 14. It can be observed that the pressure fluctuation increases radially and decreases axially from the outlet pipe entrance. This is mainly because the flow at the impeller outlet shows a higher tangential flow velocity component than the radial, which ends up retarding the axial flow towards the downstream flow zones. However, as the flow gets away from the impeller outlet region, the impeller rotational speed influence on the flow field decreases. These occurrences weaken the tangential flow velocity and enhance the axial flow velocity. Optimized cases A and B exhibited very low pressure fluctuations at the outlet pipe as compared to optimized case C.

![Pressure contours](image)

Figure 12. Pressure distribution within the volute of the optimized cases A, B and C under flow conditions of: (a) 1.0Qd (b) 1.2Qd and (c) 1.4Qd.

4.4.2. Time Domain History of Pressure Fluctuation Coefficient

To investigate the pressure fluctuation characteristics within the PAT flow zones quantitatively, five pressure monitoring points were placed in the volute and the impeller flow channel. As shown in Figure 15, two pressure monitoring points were placed in the outlet pipe. The recorded pressure fluctuation data were processed using fast Fourier transform (FFT). The pressure fluctuation coefficient is plotted at the design point, BEP and overload flow conditions at different monitoring points. For time dependent history, a non-dimensional coefficient, \( C_p \), is employed with the expression below.

\[
C_p = \frac{(P_{mon} - P_{ref})}{0.5pU_2^2}
\]
where, the referred pressure $P_{ref}$ is the average static pressure for a complete rotation period of the impeller and $P_{mon}$ is the pressure at the monitoring points. In the figure, “V” denotes the volute cross-sectional area, which reduces from “V1” to “V5” as depicted in Figure 15. “Imp” denotes the impeller low zone. “Imp1”, “Imp2” and “Imp3” cover the impeller inlet width. “Imp4” is situated at the mid-section of the impeller while “Imp5” monitors the impeller outlet zone. “D” monitors the outlet pipe flow channel.

**Figure 14.** Pressure distribution within the outlet pipe of optimized cases A, B and C under flow conditions of: (a) $1.0Q_d$, (b) $1.2Q_d$ and (c) $1.4Q_d$. 
Figure 15. Monitoring points in the impeller, volute and outlet pipe channel.

(a) Volute Flow Passage

Figure 16 demonstrates the time domain history of pressure fluctuation coefficient, $C_p$, in the volute flow channel for all the optimized impeller cases A, B and C. The time-domain curves represent the periodic variations in pressure coefficient amplitude harmonics with respect to time step. It should be noted that the time step here is used to represent impeller position. During the impeller rotation of $360^\circ$, eight peaks and eight valleys were seen, which represent the eight blades of the impeller passing within a shaft revolution. When the impeller rotates, the blades sweep over the monitoring points, and they alternate between the high and low pressure zone.

In the volute flow passage, the pressure fluctuation at monitoring points “V1”–“V5” had quite similar trends in all the optimized cases A, B and C. Within the volute flow passage, monitoring points “V1” and “V5” recorded the highest amplitude. These high pressure fluctuation occurrences may be due to the geometrical position of the monitoring points, which are situated around the volute tongue region. It has been observed that as the flow condition increase from design point ($1.0Q_d$), best efficiency point (BEP) ($1.2Q_d$) and to overload point ($1.4Q_d$), the peak-to-peak pressure fluctuation amplitude appears to increase along.

Optimized case C recorded a higher amplitude of $C_p = 15$ and above, followed by optimized cases A and B with amplitude values of $C_p = 13$ and $C_p = 12$ under flow condition of $1.4Q_d$, respectively. Comparatively, monitoring points “V4” and “V3” recorded the weakest amplitude in all the optimized cases with the exception of optimized case C. Above all, the pressure fluctuation in the optimized case C is the strongest. Thus, optimized case C has the tendency to induce vibration and noise.

(b) Impeller Flow Passage

The time-domain pressure fluctuation for one impeller revolution at the five monitoring points (Imp1, Imp2, Imp3, Imp4 and Imp5) within the impeller flow channel is depicted in Figure 17. During impeller rotation of $360^\circ$, eight peaks and eight valleys are seen, which represent the eight blades of the impeller passing within a shaft revolution. This confirms that the pressure fluctuation intensity in the impeller flow channel depends on the interaction between the rotating impeller and the stationary
volute. The average value of the pressure fluctuation intensity decreases rapidly along the impeller flow channel from the BLE to the BTL. This occurrence is due to the energy transfer from the high pressure fluid to the mechanical shaft.

Figure 16. Time-domain of pressure fluctuations at different monitoring points within the volute of the three optimized impeller cases at the locations of: (a) V1, (b) V2, (c) V3, (d) V4 and (e) V5.

For all the optimized cases A, B and C, the pressure fluctuation at monitoring point “Imp3” recorded the most intensive pressure fluctuation among the five monitoring points located within the impeller flow zone. The high-pressure fluctuation of these regions may be due to the geometrical position of the monitoring points. Thus, at the volute outlet, where the flow still possesses a high percentage of its momentum hits the blade leading edge (BLE) surface, this impact results in different complex flow structure formation.

Optimized case C recorded the highest amplitude of $C_p = 15–16$ with a flow condition of $1.4Q_d$. Optimized cases A and B had a quite similar trend, with optimized case A, recording the least amplitude of $C_p = 11.2$. A similar trend was also observed at monitoring point “Imp2” which is located at the mid-section of the blade leading edge (BLE). Monitoring point “Imp5”, which is located at the impeller trailing edge, rendered the lowest pressure coefficients, which are lower than $C_p = 8$, especially at
the design flow condition, in all the optimized cases. This makes the impeller eye highly susceptible to cavitation. It can be seen that the amplitude of pressure pulsation increases with increasing flow conditions. Among all the optimized cases, optimized case C exhibited the most intense pressure fluctuation compared to optimized cases A and B.

Figure 17. Time-domain of pressure fluctuations at different monitoring points within the impeller of the three optimized impeller cases at the locations of: (a) Imp1, (b) Imp2, (c) Imp3, (d) Imp4 and (e) Imp5.

(c) Outlet Pipe Flow Passage

Figure 18 shows the pressure fluctuation in the outlet pipe. In one complete revolution; the flow at “D1” and “D2” in all three optimized cases fluctuates eight times, which equals the blade number. This confirms that the pressure fluctuation in the outlet pipe is due to the interaction between the rotating impeller and the stationary volute. Pressure fluctuation distribution at monitoring points “D1” and “D2” are selected to study the pressure fluctuation intensity in the discharge pipe. The value of the pressure fluctuation increases from the center of the outlet pipe to its near wall as indicated in Figure 16.
At monitoring point “D1”, optimized case C recorded the lowest pressure fluctuation value of $C_p = 5.8$, under all the flow conditions. However, optimized cases A and B exhibited a similar trend with $C_p = 6.1$ under all the flow conditions. Optimized cases A, B and C at monitoring point “D2” exhibited a similar trend of low-pressure fluctuation under all the flow conditions.

4.4.3. Frequency Domain History

(a) Volute Flow Passage

Figure 19 shows fast Fourier transform (FFT)-based frequency domain pressure pulsations at points “V1”–“V5” for the three optimized cases. Note that the impeller rotational speed is 1450 r/min, which indicates that the impeller rotating frequency $f_n$ is 24.167 Hz, leading to the blade passing frequency (BPF) of 193.33 Hz as there are eight impeller blades. The frequencies excited by pressure fluctuation happened at 193.33 Hz ($8 \times f_n$), 386.6 Hz ($16 \times f_n$) and 773.3 Hz ($32 \times f_n$).

For the three optimized cases, the blade passing frequency (BPF) was generally found to be the main frequency. The pressure fluctuation amplitudes continuously increase from monitoring point “V3” all the way to “V1”, with the exception of “V2”, in all the optimized impeller cases. Monitoring point “V5” recorded the highest pressure coefficient amplitude on the blade passing frequency harmonics. It recorded a $C_p$ value of about 0.96 with optimized case A under $1.4Q_d$ flow condition, followed by optimized case B with a $C_p$ value of 0.84 under flow condition of $1.4Q_d$. However, the monitoring point “V2” and “V3” recorded the weakest pressure coefficient amplitude. At monitoring point “V2” of optimized case C, a $C_p$ value of 0.18 was attained under flow condition of $1.4Q_d$, followed by optimized case B, with a $C_p$ value of 0.17 under flow condition of $1.4Q_d$.

(b) Impeller Flow Passage

Figure 20 shows the properties in the impeller flow passage. The pressure fluctuation amplitudes continue to decrease from the blade leading edge (BLE) to the blade trailing edge (BTE) as depicted in the figure. The monitoring point “Imp4” recorded the peak pressure coefficient amplitude of the blade passing frequency harmonics. At monitoring point “Imp4”, optimized case B, recorded a high
A $C_p$ value of 0.5 under 1.4$Q_d$ flow condition, followed by optimized cases A and B underflow condition of 1.4$Q_d$ and 1.2$Q_d$, respectively, recording a $C_p$ value of 0.42.

Figure 19. Frequency domain history of pressure fluctuation in volute flow passage at the locations of: (a) V1, (b) V2, (c) V3, (d) V4 and (e) V5.

Monitoring point “Imp1”, denotes the suction side (SS) of the impeller blade. Optimized case B recorded a high $C_p$ value of 0.4, followed by optimized case A, with a $C_p$ value of 0.27 under flow condition of 1.0$Q_d$, respectively. Monitoring point, “Imp2” denoting the mid-section of the blade inlet width ($b_2$) recorded a $C_p$ value of 0.43 with optimized case A, under 1.2$Q_d$ flow condition, followed by optimized case B, with a $C_p$ value of 0.38 under 1.4$Q_d$ flow condition. Monitoring point “Imp3”, representing the pressure side (PS) of the impeller blade, recorded a high $C_p$ value of 0.44 with optimized case A under 1.2$Q_d$ flow condition, followed by optimized case B, under flow condition of 1.4$Q_d$ recorded a $C_p$ value of 0.4. Monitoring point “Imp5”, denoting the impeller eye, recorded a comparatively lower $C_p$ value of 0.16 with optimized case C under flow condition of 1.0$Q_d$ and 1.4$Q_d$, respectively.
followed by optimized case B with a \( C_p \) value of 0.84 under flow condition of 1.4 \( Q_d \). However, the monitoring point “V2” and “V3” recorded the weakest pressure coefficient amplitude. At monitoring point “V2” of optimized case C, a \( C_p \) value of 0.18 was attained under flow condition of 1.4 \( Q_d \), followed by optimized case B, with a \( C_p \) value of 0.17 under flow condition of 1.4 \( Q_d \).

(b) Impeller Flow Passage

Figure 20 shows the properties in the impeller flow passage. The pressure fluctuation amplitudes continue to decrease from the blade leading edge (B LE) to the blade trailing edge (BTE) as depicted in the figure. The monitoring point “Imp4” recorded the peak pressure coefficient amplitude of the blade passing frequency harmonics. At monitoring point “Imp4”, optimized case B, recorded a high \( C_p \) value of 0.5 under 1.4 \( Q_d \) flow condition, followed by optimized cases A and B under flow condition 1.4 \( Q_d \) and 1.2 \( Q_d \), respectively, recording a \( C_p \) value of 0.42.

Figure 20. Frequency domain history of pressure fluctuation in impeller flow passage at the locations of: (a) Imp1, (b) Imp2, (c) Imp3, (d) Imp4 and (e) Imp5.

(c) Outlet Pipe Flow Passage

Figure 21 shows the properties in the monitoring points of the flow passage of the outlet pipe “D1” and “D2”. The pressure fluctuation decreases as the flow moves away from the impeller eye region. At monitoring point “D1”, optimized case C recorded high \( C_p \) values of 0.16 and 0.14 under flow condition 1.4 \( Q_d \) and 1.0 \( Q_d \), respectively. Similarly, at monitoring point “D2”, optimized case C recorded \( C_p \) values of 0.11 and 0.10 under flow condition 1.4 \( Q_d \) and 1.2 \( Q_d \). It can also be noticed that for optimized case C, outlet pipe pressure pulsation was the highest, whereas optimized cases A and B exhibited the lowest pressure pulsation amplitudes. It is established that the pressure fluctuation
within the outlet pipe is the spread of the pressure fluctuation caused by the rotor-stator interaction within the volute and the impeller.

![Graphical representation of pressure pulsation amplitudes](image)

Figure 21. Frequency domain history of pressure fluctuation in outlet pipe flow passage at the locations of: (a) D1 and (b) D2.

5. Conclusions

The selected centrifugal PAT is distinguished by the problem of low operating range and high pressure pulsations. Thus, it remains a major challenge to expand the operating range and reduce the pressure pulsation during operation of the PAT. In order to maximize the PAT efficiency under multi working conditions, the design variables of the PAT impeller blade have been optimized in this paper to broaden its operational range. Furthermore, the intensity of the pressure pulsation was examined with numerical methods throughout the entire PAT flow passage. Below are some of the key findings and recommendations that can be taken into account during the PAT design optimization to boost PAT performance and operating reliability.

1. In this paper, the three-objective optimization problem is successfully solved and 3D Pareto solutions are achieved. There is a big performance improvement in the internal flow characteristic for all three optimized cases. Among which, case B exhibits the best performance under all flow conditions. At the design point (1.0Qd), best efficiency point (1.2Qd) and overload point (1.4Qd) flow conditions, the efficiency of case B increases by 13.1%, 8.67% and 10.62%, respectively. This implies that larger blade exit angles on the hub (β2) and smaller blade exit angles on the shroud (β1) have a combinatorial effect on PAT efficiency improvement.

2. The analysis of the time and frequency domain of the unstable pressure distribution within PAT shows that the high frequency pulsation is reduced by design variables of optimized cases A and B. However, case C exhibits the highest pressure pulsations and the lowest efficiency, while case B presents the lowest level of pressure pulsations and the highest efficiency. Therefore, high frequency pulsation directly affects PAT noise and stability, resulting in increased energy loss and reduced efficiency.

3. The pressure pulsation main frequencies are usually found at the impeller rotation frequency and also at different monitoring points within the entire PAT flow passage. The rotor-stator interaction (RSI) therefore constitutes the main factor affecting the characteristics of the pressure pulsations within the PAT flow passage. In the three PAT components (volute, impeller and outlet pipes), the blade passing frequency (BPF) is found to be the main frequency.

4. An investigation of the unsteady pressure field indicates that the unsteady pressure pulsation could spread downstream. Therefore, the two forms of pressure pulsations are the impeller and
the outlet pipe flow passage. The low-frequency pressure pulsation is also found to decrease rapidly along the PAT flow channel. Thus, the unstable pressure pulsation inside the outlet pipe results mainly from the distribution of the unsteady pressure within the volute.

5. The RSI produces the highest pressure pulsation distribution within the volute tongue region for the three optimized cases. Thus, volute tongue modification is a better approach to reducing the intensity of the pressure pulsations in the volute and the entire PAT.

6. For future research directions, it is recommended that more accurate and also computationally heavy CFD simulation models such as detached eddy simulation [33] should be applied to the numerical prediction of PAT performance. Moreover, it is worthwhile trying to convert the multi-objective optimization problem into a single-objective one [34], so that the optimization process can be simplified and hence increase the likelihood of reaching a global optimal solution.

**Author Contributions:** Methodology, J.X.; Writing—original draft, S.N.A.; Writing—review & editing, L.W., W.L. and R.L. All authors have read and agreed to the published version of the manuscript.

**Funding:** The authors would like to express their great gratitude for the financial support from the independent project of Institute of Fluid Engineering Equipment, Jiangsu University. The funding from the Natural Science Foundation of Jiangsu Province (Grant No. BK20180879), the High-level Talent Research Foundation of Jiangsu University (Grant No. 18JDG012) and the Construction of Dominant Disciplines in Colleges and Universities and Universities in Jiangsu (PAPD) are equally acknowledged.

**Conflicts of Interest:** The authors declare no conflict of interest.

**References**


