ORIGINAL ARTICLE

Numerical simulation of complex flow structures and pressure fluctuation at rotating stall conditions within a centrifugal pump

Anthony A. Alubokin¹,² | Bo Gao¹ | Zhang Ning¹ | Longlong Yan¹ | Junxiang Jiang¹ | Evans K. Quaye¹

¹School of Energy and Power Engineering, Jiangsu University, Zhenjiang, China
²School of Engineering, Bolgatanga Technical University, Sumbrungu, Ghana

Correspondence
Bo Gao, School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China.
Email: gaobo@ujs.edu.cn

Funding information
Six Talent Peaks Project in Jiangsu Province, Grant/Award Number: KTYHY-060; National Natural Science Foundation of China, Grant/Award Number: 51576090

Abstract
Rotating stall contributes to global oscillation vibration problems, accompanied by noise and possible turbomachinery damage. This study with special emphasis on the vaned diffuser investigates the unsteady pressure interaction with the stall within the pump. A low specific speed centrifugal pump (nₛ = 69), fitted with a vaned diffuser is modeled and studied. The model pump performance curve shows the characteristic positive slope at 30% of the best efficiency point flow rate; 1.0 Φₙ which is attributed to the stall phenomena. A finite volume method is employed with unsteady computations initialized utilizing shear stress transport k-ω before proceeding with DDES. Pressure fluctuation and velocity magnitude normalized values are used to investigate the evolution of stall cell generation. The root mean square (RMS) values and normalized pressure (C_p) values are elicited to gain insight into pressure pulsation within the flow domain. The distinguished “starfish” shape is observed for monitor points md1 to md20, with the RMS trend decreasing with increasing flow rate from the pump shut off. Although in the vaned diffuser flow channel, an increase in pressure fluctuation along the flow channel toward the trailing edge is observed, the vaned diffuser channel shows a similar trend. The stall cell propagates at a speed of Ω_RS = 0.078 at 0.2 Φₙ, while at 0.1 Φₙ propagates at a speed of Ω_RS = 0.087; the stall speed tends to increase approaching pump shut off. Three distinguishable stall channels are observed from the flow structure for a five vaned diffuser; entering the stall, stalled, and stall recovery stages, within the flow channels.

KEYWORDS
centrifugal pump, pressure fluctuation, stall propagation speed, vaned diffuser
1 | INTRODUCTION

Rotating stall is one of the numerous reasons that account for global oscillation vibration problems and this usually occurs when the incident angle of flow at the impeller outlet is lower than the designed incidence causing the generation of stall cells, which eventually block the flow channel. This is usually accompanied by noise and vibration, which could lead to turbomachinery damage. Its occurrence usually manifests as stall cells in some isolated channels and these would gradually propagate circumferentially along with the impeller rotation and reportedly at a fraction of the impeller rotating speed. The rotating stall is a widespread occurrence in compressors and has attracted a considerable amount of attention over the years; however, the same cannot be said for centrifugal pumps. Centrifugal pumps enjoy significant usage all over the world for many industrial applications and for their preference of providing higher operation efficiency and pump head, but like many other pump machinery, is riddled with the problem of the rotating stall when operated at part load. The stall relates to the areas of recirculation within either the impeller, diffuser channel, vanless passages, or other pump components. Their occurrence inadvertently results in a significant pressure and velocity fluctuation, which could lead to an unstable performance of the pump. It is, however, imperative to state that much of the internal unsteady flow structures within the pump are attributed to the rotor-stator interaction. At lower flow rates of the pump operation, flow instabilities ensue and often are attributed to the stalling of some channels of the diffuser for a diffuser-impeller configuration or the stalling of impeller flow channels for an impeller-volute configuration. These instabilities at lower flow rates are manifested on the pump performance curve as a positive slope, hence limiting the pump reliability over a wide range of operations. System curves influence rotating stalls depending on the characteristics of losses manifested on the performance curve. Typically, these characteristics are in contrast with the monotonically negative curve and appear as a positive slope (humped) toward pump shut-off and a saddle-shaped curve; often showing a positive curve toward the best efficiency point (BEP) on the performance curve. The humped shape is typical of centrifugal machinery and has been attributed to losses due to stalling of the flow at part load conditions. It is worthy of note that these positive slopes are strictly an indication of operation instability.

Over the decade most studies, both numerical and experimental have sought to shed more insight on the characteristics of the rotating stall; its onset, and propagation, within the pump. To show the origins of the rotating stall, Lucious and Brenner numerically investigated velocities fluctuating during rotating stall in a centrifugal pump and determined the stall frequency both in the stationary and moving frames. Zhou et al. and Zhang et al. on the contrary, relied on pressure fluctuation for the detection of rotating stall frequency. Other studies have somewhat indicated in their research a possible detection of a characteristic stall frequency at part load working conditions and further showed the rotor-stator interaction (RSI) between the diffuser throat trailing edge of the impeller flow channel. More recently Zhang et al. in a study indicated the frequency of rotating stall at lower flow rates of a centrifugal pump with a slope volute. Their study also indicated unsteady pressure pulsation at stall conditions. Gao et al. also, through experimental data also shed light on pressure fluctuation and rotating stall in a low specific speed centrifugal pump. For various diffuser-impeller configurations; be it either in compressors or pumps, both experimental and numerical methods have been relied on in efforts at understanding complex interactions at stall conditions in low specific speed centrifugal pumps.

The purpose of this study is to investigate the unsteady pressure fluctuation within a pump with special emphasis on the vaned diffuser under stalled conditions. Though recently, rotating stall occurrence is receiving some form of attention, they are most often not works exclusively dedicated to investigating such phenomena, therefore offering limited insight on the subject. To the best of the knowledge of the authors, the diffuser rotating stall in centrifugal pumps has not enjoyed considerable attention. There is no denying the fact that, the stall phenomenon is still largely understudied and the few studies are limited to specific machinery, therefore, this article, together with other studies would serve as a point of reference in furthering the understanding of rotating stalls in centrifugal pumps fitted with a vaned diffuser. This article also thoroughly discusses the unsteady evolution of stall propagation with a special emphasis on the diffuser flow channel and again at the point that the pump rotors interact with the vaned diffuser throat. This study employs velocity magnitude to distinguish stall area and unstalled areas.

2 | NUMERICAL CONSIDERATIONS AND MODELING

Over the decades with the emergence of various eddy-viscosity-centered RANS models, academics found out that these were inadequate in explaining the turbulent interaction and flow physics when it came to accurately investigating complex separation regions of flow structure. This shortfall is reduced if not eliminated when resorting to LES models to investigate turbulence; they are effective in resolving the anisotropic, large-scale flows. However, using LES to investigate turbulent flow, comes at a high cost of
computation resources and time. Hybrid methods combining both the RANS; to reduce the computational cost, and the LES for specific areas of interest within the computational domain seemed to be the solution, but also posed some challenges in terms of selecting regions to run LES and assigning separate computational conditions to RANS and LES.\textsuperscript{19,20}

The detached eddy simulation (DES) model and its derivatives\textsuperscript{19,21–23} address the constraints mentioned above the model allows for LES solvers to be applied to specific areas of unsteady separated flows while transitioning into RANS solvers for near-wall and steady flow regions.\textsuperscript{24} Delayed DES (DDES) a modification of the DES caters for the inadequacies, resolves grid induced separations\textsuperscript{21} and proves to be effective.\textsuperscript{12,25,26} In this study, the numerical approach DDES is used and all numerical computations were run on the commercial Ansys fluent 19 code.

2.1 | Computational domain and mesh generation

This model pump studied is a low specific speed centrifugal pump with the specific speed of \( n_s = 69 \), Table 1 shows the design specification of the model pump. The computational domain is comprised of five components: the inlet, a shrouded impeller, a closed vaned diffuser, a volute, and an outlet. The inlet duct is four times the impeller inlet diameter (\( D_1 \)); this would ensure a fully developed turbulent flow at the impeller inlet. Figure 1 shows the computational domain.

| Impeller | Diffuser |
|----------|----------|
| **Inlet diameter** | **Outlet diameter** |
| \( D_1 \) (mm) | 80 | \( D_3 \) (mm) | 260 |
| **Outlet diameter** | **Diffuser inlet width** |
| \( D_2 \) (mm) | 250 | \( b_3 \) (mm) | 21 |
| **Blade outlet width** | **Diffuser inlet angle** |
| \( b_2 \) (mm) | 15 | | 16.8° |
| **Inlet blade angle** | **Diffuser outlet angle on the pressure side** |
| 34.3° | | 17° |
| **Outlet blade angle** | **Diffuser outlet angle on the suction side** |
| 30° | | 17° |
| **Blade wrap angle** | **Diffuser outlet width** |
| 115° | \( b_4 \) (mm) | 20 |
| **Design operating point** | **Volute** |
| 1.0 \( \Phi_N \) | | |
| **Design flow rate** | **Diffuser wrap angle** |
| \( Q_d \) (m\(^3\)/h) | 55 | | 105° |
| **Design head** | **Inlet diameter** |
| \( H_d \) (m) | 22 | \( D_5 \) (mm) | 340 |
| **Rotating speed** | **Specific speed** |
| \( n_d \) (r/min) | 1450 | \( n_s \) | 69 |

The computational mesh is generated with Ansys-ICEM 19, five meshes were generated and the best three; distinguished by the degree of refinement, are evaluated and the most suitable adopted for the study. All computational grids are comprised of structured hexahedral mesh elements. The finer the mesh the better the chances of achieving solution convergence. A fine hexahedral mesh is much better than and medium refined hexahedral mesh as this is also much better than a coarsely refined mesh. Structured mesh provides for easy connectivity between nodes and elements, faster to achieve convergence, and better utilizes the memory, the drawback to this mesh is that angles and curves pose a problem and compromise quality at those regions. Unstructured mesh, on the other hand, is suitable for complex geometries with lots of angles and curves as the elements and nodes vary; are mostly tetrahedral or triangular. A hybrid of the two serves the purpose of improving the refinement of an area of interest and easily meshing complex areas within the geometry.\textsuperscript{27} The computational mesh is refined according to the cell height and characterized by the pump Head, the Orthogonal quality, and \( y^+ \) values at the nominal flow rate, as shown in Table 2. The grid independence shows that an increase in the node size and refinement based on the cell height \( h \) did not result in any significant change in pump efficiency, average \( y^+ \) for the impeller blade wall, and diffuser vanes. It is, of course, important to note how these individual mesh variants respond to flow rate and flow head; as captured in Figure 3A,B, there is a profound
agreement between the design flow head for meshes designated E and C, with almost a seamless agreement between the two for varying flow rate. However, for predictions for pump efficiency, mesh E and D show the best agreement with the design expected efficiency (83%) and both display a good agreement for varying flow rates.

The final computational mesh (mesh E) adopts a first layer height \( h \) of 0.1 mm, a volute average \( y^+ \) value; 39, that certifies the log law region requirement of \( y^+ < 30 \) while those of averaged wall \( y^+ \) values of the impeller and diffuser are 5.89 and 1.7, respectively. These are values approaching the laminar region \( y^+ = 1 \) where \( y^+ = u^+ \).\(^{11} \) Figure 2 shows the computational grid for the flow domain of the pump.

The hydraulic efficiency and head coefficients were obtained as follows:

\[
\eta_N = \frac{W}{\rho QU^2}, \quad (1)
\]
\[
\psi_N = \frac{gH_d}{U^2}. \quad (2)
\]

### 2.2 Turbulent model and boundary conditions

All computations were initialized with a steady-state simulation; thus, the Reynolds averaged Navier-Stokes relations (RANS) of the shear stress transport (SST) \( k-\omega \) turbulence model is used. The unsteady calculation for this study is carried out by the DDES method. DDES has proved promising in resolving flow separation and turbulence, as shown from the findings of these references.\(^{25,28,29} \)

A trade-off between DDES and SST models:

From Figure 4, the superiority of DDES to resolve the finer details of flow over SST is demonstrated. At \( t = 0.8280 \) s,
DDES can capture sufficiently complex flow structures within the diffuser channel whereas SST captures only those in the impeller to a certain degree.

2.2.1 | Governing equation

The continuity equation and the unsteady Navier-Stokes equation can be spatially filtered, resulting in the governing equation for DDES. This is expressed to satisfy incompressible flows as described herein as an incompressible form.\textsuperscript{18,22,30} This is a build-up on the $k$-$\omega$ SST RANS\textsuperscript{12,28,31} and is summarized as follows:

\begin{align}
\frac{\partial \rho k}{\partial t} + \nabla (\rho U_k) = & \nabla \left[ \mu + \frac{\mu_t}{\sigma_k} \right] \nabla k + P_k \\
& - \rho \sqrt{k \lambda_{DDES}}. \quad (3)
\end{align}

\begin{align}
\frac{\partial \rho k}{\partial t} + \nabla (\rho U_k) = & \nabla \left[ \mu + \frac{\mu_t}{\sigma_k} \right] \nabla k \\
& + 2(1 - F)\rho \frac{\nabla k \nabla \omega}{\sigma_{\omega}^2} - \rho \beta_2 \omega^2 \\
& + \alpha_3 \frac{\omega}{k} P_k. \quad (4)
\end{align}
The turbulent eddy viscosity is computed as:

$$\mu = L_{\text{RANS}} \sqrt{k}.$$ 

Such that,

$$L_{\text{RANS}} = \frac{\alpha_t \sqrt{k}}{\max(\alpha_t \omega \cdot F_i S)}.$$ 

The SST incorporated functions; $F$ and $F_1$, could be obtained in detail from Gritskevich et al.\textsuperscript{12}

To obtain the length function ($L$) is as follows:

$$l_{\text{DDES}} = l_{\text{RANS}} - f_d \max(0, l_{\text{RANS}} - l_{\text{LES}}),$$

$$l_{\text{LES}} = C_{\text{DES}} H_{\text{max}},$$

$$l_{\text{RANS}} = \frac{\sqrt{k}}{C_{\mu} \omega},$$

$$C_{\text{DES}} = FC_{\text{DES}1} + (1 - F)C_{\text{DES}2}.$$ 

$L_{\text{max}}$ is the maximum edge length of the local grid, The empirical blending relation $f_d$ is written as:

$$f_d = 1 - \tan \left( (C_{\text{dir}} \delta) \right),$$

$$r_d = \frac{v_l + v}{k^2 d_w \sqrt{0.5(\zeta^2 + \Omega^2)}}.$$ 

The constants in the above equation could be verified from Ref.\textsuperscript{12} Where the $s$ is the value of strain rate tensor, and $\Omega$ is the value of the vorticity tensor.

2.2.2 | Boundary conditions

The boundary conditions chosen for this study is a combination of the Velocity specification method (velocity magnitude) in the absolute reference frame and normal to the boundary for the inlet. In contrast, a pressure outlet (total pressure) is set constant in the absolute backflow reference frame and normal to the boundary for the outlet. This has proven to speed up convergence and is much stable than other boundary combinations.

As mentioned in Section 2.3, steady calculation, SST $k$-$\omega$ model is selected. The outlet pressure ($P_o$) is set constant at approximately $1 \times 10^5$ Pa, whereas, the nominal flow rate velocity and percentages of nominal flow rates ($1.0 \Phi_N$) are (30%, 20%, and 10%) set each at a time to run a computation. The solution method used is the SIMPLE algorithm, and a second-order upwind spatial discretization, set for momentum. For the transient calculation, on the other hand, the DDES model is used, the spatial discretization for momentum is set as bounded central differencing. Discretization for turbulent kinetic energy and dissipation rate are set to second-order upwind. The transient formulation is set to bounded second-order implicit. During the steady-state and transient calculations, the cell zone condition was set as moving frame motion and mesh motions respectively with a wall shear condition of no-slip and a standard wall roughness condition.

2.2.3 | Time step and computational resources

At the nominal flow rate ($1.0 \Phi_N$), the mesh element size ($\Delta x$) and at a constant velocity ($u$) a time step ($\Delta t$) of $1.149 \times 10^{-4}$ which represent one degree ($\theta$) of the impeller rotation. Five revolutions are test-run to ensure solution convergence, given an averaged cell convective courant number of (0.4) as this meets the minimum requirement of the CFL number depending on the time chosen, should be less than 1.0.\textsuperscript{32–34}
Numerical computations were performed on a Dell PowerEdge R940 server, which has a total of 80 CPUs; it takes about 45 CPUs to complete a periodic revolution of the impeller in a time of about 19 h. The time step is set as $\Delta t = 1.149 \times 10^{-4}$ s, totaling 360-time steps per impeller revolution. The residual convergence criteria for each time step are reduced to $1.0 \times 10^{-5}$ and the maximum number of iterations per time step was limited to 60. As a result, the total period of the simulation was taken as 25 revolutions.

2.3 | Numerical validation

As indicated earlier the numerical simulation gives the hydraulic head coefficient, which is devoid of a lot of the losses; mechanical, volumetric, frictional, and so on, experienced in the experimental set-up determination of pump head. As flow velocity decreases at low flow rates, the experimental error at these flow rates of interest is high, see Figure 5. Suffice to say that the numerical simulation accuracy is further guaranteed by the mesh reliability test earlier conducted; showing no significant change with increased refinement.

Performance experiments were replicated five times to ensure the reliability of the results. The experiments were conducted in a closed-loop scheme and compliant with the ISO standards (ISO9906) for testing pump machinery, using the root-sum-square (RSS) method.

$$\begin{align*}
U_e &= \pm \sqrt{(e_s^2 + e_r^2)}, \\
&= \pm \sqrt{\left(\frac{Q}{N}\right)^2 + \left(\frac{N}{Q}\right)^2}.
\end{align*}$$

where $U_e$ is the uncertainty of sample data. While $e_s$ and $e_r$ are systematic and random errors respectively. Table 3 presents the uncertainty score for the laboratory experiments with systematic error estimated from instrument calibration and precision. Per ISO9906, the uncertainty is within an acceptable range. It is noteworthy, to indicate that the overall efficiency uncertainty is not given in this study due to a malfunction in the torque meter.

3 | RESULTS

3.1 | Internal flow structure in the rotor and stator domains

3.1.1 | Tangential and radial velocity distribution

It is crucial to investigate the velocity distribution of the flow within the impeller at the outlet and at the vane-diffuser channels leading and trailing edges for channel 1. The tangential velocity of the core flow is proportional to the propagation velocity of the stall cells. From the velocity triangle; Figure 6, the radial velocity ($c_m$) component at the impeller outlet transitions into the diffuser channel, while the peripheral velocity ($U_i$) is equivalent to the relative tangential velocity; designated $c_{1v}$, for the impeller flow domain. For the diffuser, on the other hand, the relative tangential velocity component ($c_v$) is the tangential component of the absolute velocity ($w$).

As illustrated in Figure 7, the variation for the radial velocity at part load flow. It is observed that the average radial velocity decreases with decreasing flow rate; $1.0 \Phi_N$ to $0.1 \Phi_N$, accompanied by the generation of reverse flow at the lower flow rates relative to the very low flow rates. Again, perturbation increases with decreasing flow rate as a result of the flow becoming more three-dimensional. The backflow of fluid into the
impeller channel occurs prominently at the core flow zone and is more intense toward the suction side of the impeller blade.

Backflow generated within the impeller tends to limit the quantity of flow into the diffuser channel, resulting in low velocity around the diffuser inlet, which is indicative of backflow within the diffuser flow channel. Posa et al. were able to adduce backflow prominence at the suction side of the flow channel at 77% of the impeller shroud span, this was achieved by using the hydraulic flow angle and the transverse velocity standardized by the relative velocity at the impeller outlet. Ulrik et al. employed transverse and radial velocity to capture reverse flow from the time-series data collected, it is worthy of note that this does not give the precise location of these flows. Relative tangential velocity remains fairly unchanged at the suction side closest to the impeller blade, but rather shows a general decrease as flow rate decreases from the design point flow rate. The flow tends to jet out into the vanless gap at high velocity at the suction side whereas, on the other side of the blade; the pressure side, a wake flow is

**FIGURE 6** Velocity triangle showing radial and tangential components

**FIGURE 7** Radial & Rel. tangential velocity profiles velocity profile at impeller outlet segment
observed. In the lower flow rates, reverse flow is observed with flow returning from the suction side toward the pressure side and is more rigorous at flow rates 0.1 and 0.3 $\Phi_N$.

3.1.2 | Relative tangential velocity and radial velocity in vaned-diffuser channel

The tangential and radial velocity components at the diffuser inlet and outlet show its velocity distribution. Flow from the impeller enters the diffuser channel tangentially as a result of the rotation of the impeller the fluid in the vaneless gap between the impeller and the diffuser aids this entry at the vaned diffuser throat. From Figure 8, it is observed that the incoming flow first strikes the diffuser vane at the pressure side and then is deflected toward the suction side, this process involves the dissipation of kinetic energy into static pressure energy, this is much more prominent at the lower flow rates from 0.3 to 0.1 $\Phi_N$, this induces flow separation and recirculation of flow which could lead to the stalling of the channel relative to flow in the channel at the design point flow rate. At the outlet of the vaned diffuser channel, this does not occur, rather a much more steady flow into the volute is observed with little perturbation close to the shroud and hub chambers of the pump.

3.1.3 | Comparison between one-dimensional (1D) and DDES radial velocities

The 1D analysis of the radial velocities in the section of the impeller, diffuser inlet, and outlet. Figure 9 shows the arithmetic mean radial velocity obtained for the DDES numerical simulation for channel 1 at a period of 1.0341 s and in 1D analysis.

For the impeller outlet; DDES shows a relatively high radial velocity for the impeller outlet and begins to increase from 0.2 to 0.3 $\Phi_N$ and does the same marginally to the nominal flow of 1.0 $\Phi_N$ compared with 1D radial velocity. Although the DDES radial velocity for the diffuser inlet is taken at the throat of the vane entrance, the radial velocities are low compared with 1D analysis for 0.1 $\Phi_N$ through to 0.3 $\Phi_N$ except for the 1.0 $\Phi_N$. This could be attributed to the DDES section, not corresponding to the 1D section for the diffuser throat. The observed trend gives credence to the concept of investigating the impact of the vane diffuser throat on the diffuser stall, discussed in later sections.

At lower flow rates, the flow tends to be more three-dimensional as earlier intimated, and subjected to turbulence.
with the development of a lot of secondary flow structures. The flow at these conditions is largely affected by the effects of both the centrifugal and Coriolis forces. The Russby number ($Ro_{ax}$) is the ratio of the centrifugal effect to the Coriolis force effect. At part load flows, $Ro_{ax} > 1$, as a result, secondary flow structures develop, and much of the flow moves toward the shroud or the front chamber of the pump.$^{39}$ On the backdrop of the above, the plane closest to the shroud is resorted to in the next section to examine the internal flow structure at the very low flow rates where the pump operates under stall conditions; at a plane of 0.2 of the shroud side, that is 38.5 mm for impeller and 34.5 mm for diffuser across the axis of rotation.

### 3.1.4 Velocities in suction pipe and impeller

Figure 10A shows the flow within the impeller and suction pipe flow channels. The velocity in the suction channel is the velocity in the stationary frame whereas that of the diffuser is relative velocity. The vectors in Figure 10B are presented in the coordinate frame in the $yz$-plane and at a stalled phase of $t_f N = 20$ for 0.2 $\Phi_N$.

Flow separation and turbulence increase as the flow rate decreases to the pump shut off. As large flow separation zones appear, as a consequence of flow velocity deceleration and decreasing flow attack angles, the flow-through is confined to the pressure side of the channels with relatively less recirculation and stalling. The suction pipe similarly has flow separation, most likely because of the interaction of the return flow along the shroud and hub walls into the impeller eye. The recirculation of flow starts in the area of the suction pipe closest to the impeller eye and develops toward the inlet of the pipe relative to decreasing flowrate from 1.0 $\Phi_N$. The boundary layer effect greatly contributes to the trend,$^{4}$ as thickening of the boundary layer increases with decreasing flow rate while the wall shear stress reduces along the blade walls, at a threshold value of eddy viscosity equals zero is reached where the flow separation ensues, inducing recirculation and return flow in the impeller.

### 3.1.5 Return flow

At lower flow rates the angles of attack at the impeller inlet increase, and so also does the relative velocity, this results in the large mass flow moving toward the pressure side of the impeller blade and as a consequence, the fluid detaches from the suction side of the blade allowing large flow separation to occur.$^{3,40}$ Quite a significant quantity of these separations develops into large recirculation zones or vortices due to the pressure difference between the core of these suction side generated recirculation and the surrounding fluid. These generated vortices rotate in the direction of the impeller and could grow in magnitude to stall the flow channel. These stall cells generated in the impeller will rotate in the opposite direction of the impeller rotation due to the difference in angular velocity of the stall cell and the impeller. It is incisive to note that these generated vortices do not grow in magnitude to the extent that an impeller channel is stalled, therefore the vaned diffuser which is introduced to convert the velocity momentum into static pressure energy does so to prevent the full effect of these vortices completely stalling the impeller channel as seen in Figures 10A and 18, the number and magnitude of these recirculations, return and or reverse flow zones increase as the flow rate approaches pump shut off. As earlier intimated this flow detachment occurs at the suction side whereas an intense stagnation is observed at the pressure side, the pressure gradient generated between the two results in a return of flow or flow reversal or backflow in the impeller. The vortices reduce the mass flow out of the impeller, with flow jetting out at the pressure side of the impeller outlet.

A significant return flow generated in the impeller that has been attributed to the occurrence of secondary flow structures;$^{41,42}$ vortices/coherent structures at the impeller outlet and rotating stall in the diffuser other than is initiation from flow separation induced at the impeller blade suction surface is that resulting from return flow by Sinha et al.$^{43}$ Usually, the impeller blade tip clearance most often induces tip-leakage inflow within a pump operating under low flow
rates; resulting in an overflow from the pressure side leading edge of the blade to the suction side of the blade. Similarly in a shrouded impeller, the clearance between the trailing edge of the blade tip and the diffuser is also a source of return flow induction as observed in centrifugal compressors. Backflow usually manifests in the flow channel when flow turns back from one flow channel into the next flow channel at the trailing edge of the impeller blade, resulting in a velocity distortion. The distortion in the velocity creates a low-pressure area with an accompanied vortex that may grow to results in the blockage of the flow channel. Figure 18 shows return flow as induced by the mixing of flow in the vanless gap; thus the peripheral flow and the flow from the pressure side of the impeller. The fluid jetting out of the pressure side

**FIGURE 10** (A) Flow within the suction pipe and impeller and (B) meridional view at 0.2 $\Phi_N$
of the impeller is not observed flowing into the suction side of the same impeller vane, neither at the mid-plane nor at the shroud side as reported for centrifugal compressors with diffusers during stall conditions and axial flow pumps when tip-leakage occurs.

The recirculation generated in the suction pipe is mostly due to the return flow along the shroud wall; as these may persist till the impeller eye, interacting with the flow from the suction duct. That which is attributed to the return flow along the side chambers is not captured in this study, though it accounts largely for vortex generation and recirculation at the impeller eye. The reason being the computational domain is devoid of the casing; thus no side chambers were included.

The next sections of the manuscript will expatiate on the evolution of diffuser, employing turbulent intensity and velocity magnitude. Also, the origins of the stall in the diffuser will be adduced bearing in mind the backflow and vortices generated in the impeller at a lower flow rate.

### 3.2 The evolution of stall in the vane diffuser

The evolution of the rotating stall in the model pump is dynamic and its profound effect is prominent within the vane diffuser. Throughout the operation of the pump at $0.2\Phi_N$, the phase evolution of stalling of flow channel 1 is illustrated in Figure 11. At the initial phase at period T (0.0414 s), the flow into the channel is limited as a result of the lowering of the angle of attack (AoA) of the incoming fluid from the impeller and the vaneless gap between the impeller and vane diffuser. The incoming flow is high in kinetic energy and impacts the pressure side of the vane diffuser blade this is then deflected toward the mid-channel and toward the suction side. This is accompanied by the dissipation of energy, which is converted into static pressure energy. Perturbation of the flow induces flow separation at both the suction and pressure sides of channel 1 at the inlet of the vaned diffuser.

The phase of pump rotation considered in this study are $t = 0.2484, 0.4554, 0.6210, 0.8280,$ and 0.9936 s and these are standardized with the blade passing frequency ($f_N$) to yield: $t_{f_N} = 6, t_{f_N} = 11, t_{f_N} = 15, t_{f_N} = 20, t_{f_N} = 24,$ respectively.

#### 3.2.1 Turbulent intensity and velocity magnitude

Turbulent intensity at the diffuser inlet greatly influences the stalling of the channel downstream from Figure 13B it is seen that the stalling of channel 1 is characterized with low turbulent intensity for $t_{f_N} = 20$ relative to the other phases of flow, this is sequentially followed by the recovery phase $t_{f_N} = 24$ with the next least turbulence. In contrast, the velocity magnitude at the same white probe line, does not follow the trend sequentially in terms of the phase of flow. At the flow recovery phase, the velocity magnitude is lowest at the pressure side; the least at the pressure side for all five phases but picks up toward the suction side of the diffuser vane; it records the highest velocity magnitude at the suction zone. At the highest flow phase, the flow is fairly distributed along the white probe line, the flow is highest at the pressure end of the vanes and then takes a downward trajectory toward the middle and finely increases again toward the suction side of the vanes.

The fluid flow through channel 1 some five cycles later ($t_{f_N} = 11$), presents quite a different structure relative to the stalling of the flow channel. The proffered explanation offered to this structure accounts for the channel experiencing the most fluid flow through channel 1. It suffices to argue that, at this phase, the flow is more radial in the diffuser. It again could be inferred that; at this phase, the flow AoA is higher relative to that at the stall phase. The averaged flow $Q^*(Q/Q_s)$; which is a function of the diffuser channel area and the averaged flow velocity, relative to the stalled channel's is $Q_s = 1.88 \text{ m}^3/\text{h}$, see Table 4. $Q_s$ is the discharge at a particular phase. This is the point at which the flow is at its maximum and shows the highest averaged turbulent intensity; refer to Figure 14. For the point of extraction for the white probe line, see Figure 12. The diminishing in size and collapse of stall cells allows for the free flow of fluid down the channel. At this point, the flow is characterized by an intense tangential flow velocity across the flow channel. The turbulence reduces the thickening of the boundary layer while increasing the thinning of the shear effect of the boundary layer. Hence there is less attachment of fluid to the walls of the channel due to flow separation at the walls.

At $t_{f_N} = 20$ the flow begins to stall flow separation vortices begin to merge and the stall cells grow in size. Turbulence reduces relative to the previous phase. This leads to increased thickening of the boundary layer along the channel and fluid bound to the suction of the vanes begins to separate and recirculation zone set in close to the suction wall of the vane.

Channel 1 completely stalls in phase $t_{f_N} = 20$, as this exhibits the lowest flow velocity and turbulent intensity at the white probe line. At this instant, the flow channel is blocked at the inlet by the recirculating flow and this essentially constricts any outflow from the leading edge toward the trailing edge of the flow channel. A reduction of turbulent intensity at the inlet is synonymous with a less kinetic energy of the fluid. This inadvertently increases the
boundary layer effect at the leading edge resulting in intense flow separation, and eventually an even more intense recirculation at the inlet of the vane. These events lead to the blocking of flow through the channel. At this point \( Q^* \) is equal to 1. The averaged relative velocity after the probe line significantly records very low flow velocities close to zero, shown in Figure 13A.

The flow then begins to recover in the stalled channel as the stall cells begin to grow smaller and eventually decay at \( t f_N = 24 \). Turbulence intensity increases at the diffuser inlet; this increases the 3-dimensional effect of the flow and the bearing effect of the Coriolis and centrifugal forces in the flow downstream the diffuser channel. These result in an increase in the flow velocities and turbulence downstream. Again, there is a reduction of the boundary layer thickness and an increase in the thinning of the shear boundary layer.
downstream both reducing the effect of boundary layer separation of flow.

3.3 | Origins of diffuser stall

A 3D vortex at the instants representing the phase of rotating stall evolution with the vane diffuser channel is displayed in Figure 15. The vortical structures give an appreciation of the turbulent shear boundary effect at the vaneless gap between the impeller and the diffuser. Also, complex coherent structures are generated as a result of the interaction of return flow and the jetting outflow at the impeller outlet. For the identification of vortical structures at the phase evolution of rotating stall, the Q criterion; which is the second invariant of the velocity tensor and expressed as follows:

$$Q = \frac{1}{2} [||\Omega||^2 - ||S||^2],$$

where Ω and S are the counter-symmetric and symmetric components of the velocity gradient tensor $\nabla u$.

From the phase evolution of vortex; coherent structures, in the vanless gap as shown in Figure 15. The vortex is an indication of the secondary flow structures at the diffuser impeller vaneless gap, they seem to give insight as to the origin of the stall cells formation. The interaction between the incoming flow from the impeller and the peripheral flow along the rotating axis generates coherent structures or vortical structures that give an indication of the onset of the stall in the vaned diffuser. It is observed that all except the vortex in channel 1 in the stall channel at instant $t_{fN} = 20$ are streamwise swirls, a noticeable coherent structure from the interaction of return flow and jetting outflow at the impeller outlet. It is distinctively characteristic, in that, it spans from the hub side to the shroud side of the flow. It is worth noting that this none streamwise vortex which represents a rotating stall, alternates and is seen from one channel to the other, depending on the stall phase of a diffuser flow channel. As stated in Section 3.1, the above argument is contingent on the return flow in the impeller. Figure 18 shows the limited outflow of fluid from the

\[\text{FIGURE 13} \quad \text{Velocity magnitude and turbulent intensity for white probe line}\]

\[\text{FIGURE 14} \quad \text{Velocity magnitude along black probe line}\]

\[\text{TABLE 4} \quad \text{Diffuser channel flow (Q*) relative to the stalled channel}\]

| Phase | $t_{fN} = 1$ | $t_{fN} = 5$ | $t_{fN} = 9$ | $t_{fN} = 14$ | $t_{fN} = 18$ |
|-------|--------------|--------------|--------------|--------------|--------------|
| $Q_t/Q_s$ | 1.878936    | 1.885803    | 1.592884    | 1.1568712    | 1.568712    |
impeller channel outlet into the diffuser throat which affects the incidence angle at the diffuser inlet. A lot of the complex flow structures; the vortex generation, is initiated by the interaction of the return flow at the impeller outlet.

It is distinctively characteristic, in that, it spans from the hub side to the shroud side of the flow. It is worth noting that this none streamwise vortex which represents a rotating stall, alternates and is seen from one channel to the other, depending on the stall phase of a diffuser flow channel.

To buttress the influence of the vane diffuser throat, Figure 16 shows the instants at which the impeller trailing edge passes the leading edge of the vane diffuser at its throat for the instant \( t_{fN} = 20 \) at every 19-degree impeller revolution angle. Flow in the impeller jets out at the trailing edge and the pressure side of the blade Bld.1, consequently, two long vortex streets; \( \beta \), are observed at \( t = t_o \). These then evolve to become shorter and bigger at \( t = t_o + 19\Delta t \), meanwhile in the diffuser channel at the inlet, both clockwise and counter-clockwise dominant streets; \( \alpha \), are observed. As the impeller blade aligns with the vane diffuser inlet, the \( \beta \) streets are broken and shed into the diffuser and along with the streamwise flow in the vaneless gap between the diffuser and the impeller. At the same time, the \( \alpha \) counter-clockwise streets become more prominent and dominate the vane inlet, while the clockwise \( \alpha \) diminishes. This dominant increase in size possibly accounts for the stalling of the channel. Figure 17 shows the turbulent kinetic energy (TKE) at the diffuser throat at mid-span; the green lines indicate the suction side of the vane leading edge whereas \( L \) is defined as \( L = \varphi/360^\circ \). Herein, angle \( \varphi \) is introduced. TKE reduces at the vane pressure surface while high values are recorded for stall flow conditions.

**FIGURE 15** Vortices revealing the start point of diffuser stall

**FIGURE 16** Merging and shedding of vortex streets
0.2 $\Phi_N$. The entire flow is turbulent at these stages, as opposed to the nominal flow conditions where near-wall turbulence is observed at the suction and pressure sides of the vane. The stalled channel exhibits high TKE at the throat of vane suction for vane diffuse channel 1.

### 3.4 Pressure pulsations

This section deals with the fluctuation of pressure in both the stator and the rotor of the model pump at stalled flow conditions. Pressure signals obtained are normalized into a dimensionless entity, relying on Equation (13).

$$C_P = \frac{\Delta P}{0.5 \rho U_2^2}. \quad (13)$$

In a bid to amply appreciate the magnitude of pressure fluctuations at the blade passing frequency ($f_{BPF}$) relative to part load flow rates, the FFT method via the Hanning window is relied upon to transform standardized pressure signals ($C_P$) to frequency domain signals. The frequency-domain data are illustrated in Figure 19 for impeller revolution, as observed, the (RSI) plays a pivotal role in the amplitude of frequency signals. Undoubtedly, the RSI and as well as the vaned diffuser vane number result in a significant drop in the $f_{BPF}$ within the impeller flow channel adverse to the same impeller without a vaned diffuse as reported in the team’s previous work\textsuperscript{12,26} with an impeller rotating frequency; $f_N$, of 24.2 Hz. The impeller monitoring points $f_{BPF}$ occur at 120.8 Hz which is five times the $f_N$ whereas the $f_{BPF}$ in the diffuser channel occurs at 145 Hz, which is six times the $f_N$. It is inferred that the number of stator vanes has an impact on the $f_{BPF}$ within the impeller flow channel and the same could be said about the $f_{BPF}$ in the diffuser relative to the number of impeller blades, a further testament to the far-reaching effect of the RSI between the impeller and stationary part of the flow domain.

The amplitude at the $f_{BPF}$ in the impeller outlet and diffuser flow channel decrease from the pump shut off toward the nominal flow rate. However, point ms5 shows higher amplitudes than in d1 except at the BEP flow rate,
which is expected, due to the relatively stable flow regime. At 0.1 $\Phi_N$ the pulsation frequency is the strongest for ms5, $C_p = 0.263$ and a harmonic of 3$f_{BPF}$; similarly, the strongest d1 signal at $f_{BPF}$ for 0.1 $\Phi_N$ is $C_p = 0.132$, with harmonics of 2$f_{BPF}$, 6$f_{BPF}$. This is a peculiar trend with pumps operating at part load, as instability heights at lower flow rates and consequently strong RSI.

The root mean square (RMS) values are chosen to show the evolution of pressure magnitude within both the rotating impeller and the stationary diffuser domains, these are normalized dimensionless values arrived at with Equation 13.5

$$\text{RMS}(\varphi) = \sqrt{\frac{\varphi_1^2 + \varphi_2^2 + \varphi_3^2 + \cdots + \varphi_N^2}{N}}$$  \hspace{1cm} (14)

The Amplitude ($\varphi$) expressed from the pressure coefficient $C_p$ at a frequency range of 0–1500 Hz defines the RMS values at different calculation moments. Figure 20 gives insight into the pressure pulsation within both the impeller and the diffuser flow channels, it is noticed that from flow rates near pump shut-off, pressure fluctuation decreases as the flow rates approach the nominal flow rate value for the rotating flow domain. A somewhat similar trend is observed in the stator; save for points d2 and d3 which are distinct, except for a spike in point d2 at flow rate 0.2 $\Phi_N$ attributable to intense flow separation and stalling of the flow channel.

Pressure fluctuation increases along the flow channel toward the trailing edges of the blades, where flow separation and backflow is observed, whereas in the stator flow channels, from the throat of the diffuser to the exit of flow at the vanes trailing edge, a reverse of the argument stated above is observe; pressure fluctuation is highest at the throat and inlet of the diffuser, attributed
3.5 | Pressure fluctuation in the stationary domain

This section considers pressure pulsation within the impeller-vaned diffuser gap, the vaned diffuser-impeller gap, and the vaned diffuser flow channel. The effect of stall channels on the pulsation magnitude is also addressed.

The propagation of the rotating stall cell is affected tremendously by the gap between the impeller blade trailing edge and the diffuser leading edge. Several studies report the effect of the aforementioned gap, Qin et al. employed a 2D inviscid approach to ascertaining this effect, and they found that the pressure coefficient decreased with an increase in the radial gap. A study by Sinha et al. showed that the radial gap between the impeller and the diffuser attributed quite a lot to the onset of the stall. The circumferential flow or the high-speed leakage resulted in the confinement of stall to certain zones within the diffuser channel. Figure 21A presents pressure pulsation magnitude as RMS for point md 1 to md 20 at the impeller-vaned diffuser gap at an interval of $\theta = 18^\circ$. The precise location of the monitoring points is indicated in Figure 22.

The general observation is that pressure pulsation magnitude decreases from pump shut off to nominal flow rate; $1.0 \Phi_N$, with a minimum RMS value of 0.00078 at md 13; 234° clockwise from the volute cut-water viewed in the $\epsilon$ coordinate frame, for flow rate $1.0 \Phi_N$. The highest peak occurs at 0.1 $\Phi_N$ with an RMS value of 0.0075 at md 7, 126° clockwise. The characteristic “starfish” shape which is prominently seen for the part-load flow rates, is an affirmation of the RSI; the vaned diffuser’s effect on the pressure pulsation. It will suffice to argue that, because of the impeller-vaned diffuser gap; which for this model pump is an optimal value (5 mm) selected during design, the circumferential flow and the jet-wake flows from the impeller trailing edge interact, causing dissipation of kinetic energy, resulting in a static pressure rise. This rise then amplifies the stall signals at stall conditions.

The pressure amplitude; $\varphi$, at the $f_{BPF}$ depicted in Figure 21B, shows the signal strength for part load flow rates. Summarily observing Figure 21B shows 0.1 $\Phi_N$ has higher amplitudes; (0.039 at $\theta = 45^\circ$), at $f_{BPF}$ followed by
0.2 \( \Phi_N \) (0.034 at \( \theta = 45^\circ \)) and then 0.3 \( \Phi_N \) (0.023 at \( \theta = 270^\circ \)). Diffuser channel 1 is stalled in both 0.1 \( \Phi_N \) and 0.2 \( \Phi_N \) which could account for the conspicuously high amplitudes at \( \theta = 36-54^\circ \). The observation could be attributed to the stalled channel forcing a backflow from the diffuser channel into the impeller-vaned diffuser gap.

3.6 | Pressure fluctuation in the vaned diffuser channel

The section looks at pressure fluctuation at the \( f_{BPF} \) for monitoring points within the vaned diffuser flow channel in the midplane in the \( z \)-axis and narrowing in on flow rate 0.2 \( \Phi_N \).

Figure 23 depicts the \( f_{BPF} \) amplitude (\( \varphi \)) of pressure pulsation at diffuser monitoring points and part-load conditions. A cursory look reveals that along the vaned diffuser flow channel, pressure signal amplitude decreases with an increase in flow rate close to pump shut off. At 0.1 \( \Phi_N \) the highest amplitudes are observed at monitoring points located at the leading edge and gradually decrease toward the trailing edge of the vaned diffuser channel. However, a different trajectory is observed for the points in the flow channel and those at the trailing edge. There is a general increase of signal amplitude for flow channel points from 0.1 to 0.2 \( \Phi_N \) for all four channels except channel 5, which differs in trend. Most points at the outlet of the vaned diffuser show a marginal rise in pressure fluctuation signals with the obvious exception of the fifth channel.

First, it is posited that the RSI between the throat of the vaned diffuser and rotating impeller, the increasing of the incidence at the outlet of the impeller as opposed to lower angle of attack at the vaned diffuser inlet, is the suggested reason for this observation. The complex secondary flow structure generated at the impeller-vaned diffuser gap results in the rushing of fluid into the pressure side of the vaned diffuser inlet and followed by a vigorous deflection to the suction side of the same channel initiating flow separation and stall cells.

Second, at part load approaching pump shut off, intense flow separation is observed, channel 2 for flow rate 0.1 \( \Phi_N \) is stalled as there is virtually no difference in signals strength (\( \varphi \)) at vaned diffuser channel point d5 and same flow channel outlet monitoring point d6, this implies that stall cells appear and eventually blocks the channel. Similarly, channel 1 at the same flow rate 0.1 \( \Phi_N \); not just as much as channel 2 is stalled, there is a marginal pressure recovery between the points d2 and d3, as the stall cells slightly amplify the \( f_{BPF} \).

3.6.1 | Diffuser-volute gap

At the outlet of the vaned diffuser; the gap between the vaned diffuser trailing edge and the volute inlet has 20 monitoring points to observe the fluctuation of pressure.
at stall conditions. Figure 24 shows the evolution of pressure pulsation magnitude at stalled part load flow. RMS ($\varphi$) magnitude reveals a fairly distributed pattern; rising gradually halfway along the circumference to point dv8 at a change rate of approximately 75% more than that of point dv1 for flow rates 0.2 to 1.0 $\Phi_N$, whereas at flow rate 0.1 $\Phi_N$ the percentage change is 91% from point dv1 to dv8. This then gradually decreases for all part load flow rates to point dv11. There is yet another hike in pressure amplitude for all flow rates at point dv12 and gradually reduces toward the throat of the volute. There is an established pattern of alternating increasing magnitude of RMS for 0.3 to 0.1 $\Phi_N$ and then to nominal flow rate 1.0 $\Phi_N$, this is attributed to the fact that at part load conditions, the intense flow separation at the impeller-vaned diffuser gap couple with the stalling of flow at the inlet and within the flow channels, result in non or marginal pressure recovery at the diffuser outlet, hence the absence of obvious pulsations. Additionally, this alternation corresponds to high and low diffuser outlet velocity.

Signal strength (amplitude; $\varphi$) at the $f_{BPF}$ for all 20 monitoring points trend differently, with flow rate 0.2 $\Phi_N$ showing higher peaks from point dv10 through to dv12, followed by flow rates 0.3 $\Phi_N$ and then 0.1 $\Phi_N$ presenting the least of peak heights; significantly low pulsation at the diffuser-volute gap. The values for point dv11 to point dv15 at 0.2 $\Phi_N$ are nearly four times higher than those for 0.1 and 0.3 $\Phi_N$, this invariably gives credence to the turbulent intensity at these zones at lower flow rates. The highest value of $\varphi$ at the $f_{BPF}$ for flow rate 0.2 $\Phi_N$ is recorded at dv12 with a percentage change 99% from the amplitude at dv1. Relative smaller pressure amplitude values are seen for flow rate 0.1 $\Phi_N$, its strongest signal by far occurs at point dv8; $\varphi = 0.00001$. At 0.1 $\Phi_N$ the flow channels have very low inflow velocity and barely take in fluid, resulting in very little or no pressure recovery at the diffuser outlet. This could account for the very low pulsation amplitudes at 0.1 $\Phi_N$ arising from the marginal pressure gradient between the two zones, see Figure 24B.

3.7 | Stall position and speed

The static pressure ($P$) for an impeller revolution $T = 0.0414$ s, the position of the high highest and the lowest pressure pulsation are used to adduce the relative position of a particular stall cell at a period. As intimated by Berten et al. rotating stall signal is amplified when pressure pulsates and vice versa. The estimation of the relative position of the stall cell is therefore based on the lowest trough and the highest spike of static pressure. Figure 25 gives a fair idea of the position of a particular stall cell as a function of the impeller revolution.

The notations are defined as follows: the $\Delta \theta$ represents the angle of propagation of the stall cell in the pressure domain while $\Delta t$ is the period for which this occurs in the pressure domain.

The relative propagation speed prediction used by Gyarmathy et al. is adopted as follows:

$$\Omega_{RS} = \frac{\omega_{RS}}{\omega_{rot}} = \frac{1}{1 + C} = \frac{\sin^2 \alpha}{\sin^2 \alpha + \gamma \sin^2 \beta}$$

where $C = \frac{\sin^2 \beta}{\sin^2 \alpha}$ and $\gamma = \frac{M_{diff}}{M_{impeller}}$.

The stall speed ratio $\Omega_{RS}$ for flow rate 0.2 $\Phi_N$ is estimated, granted the following parameter; $M_{diff} = 0.742$.
kg and $M_{imp} = 0.776$ kg; these are volume integrals mass ($M$) for both the diffuser and impeller respectively. The absolute impeller blade outlet angle from the tangential direction; $\beta$ is $30^\circ$ while that at the diffuser blade inlet $\alpha$ is $6^\circ$. The ratio of inertia ($C$) of fluid in the impeller relative to that in the diffuser while considering the flow within the vaneless gap and ($\gamma$) the mass ratio; $C = 11.95$ while stall ratio $\Omega_{RS} = 0.078$, this value is close to what was obtained for a low specific speed mixed flow pump where PIV experimental results yielded a stall ratio $\Omega_{RS}$ of $0.087$ and Sinha et al.\textsuperscript{13} whose work showed speeds within the range of 0.052 to 0.078 for a centrifugal pump. For flow rate $0.1 \Phi_N$ the value is higher; $\Omega_{RS} = 0.086$, than that at $0.2 \Phi_N$.

4 | CONCLUSIONS

The study numerically simulates the complex flow structures at part load flow. Three distinct flow rates at stall operating conditions; $0.3$, $0.2$, and $0.1 \Phi_N$ are compared to the BEP flow rate of $1.0 \Phi_N$. The flow
velocity contours at the impeller outlet, the diffuser channel inlet, and the diffuser channel outlets are examined. Also, the pressure pulsation at \( f_{BPF} \) for an impeller-diffuser gap, diffuser channels, and diffuser-volute gaps are investigated. The following conclusions are arrived at based on the findings of the study:

Pressure fluctuation within the impeller-vaned diffuser gap assumes the typical “starfish” shape with the highest-pressure peaks occurring at points, halfway between any two vane leading edges. Pressure fluctuation trend at \( f_{BPF} \) for part load flow show 0.1 \( \Phi_N \) with the maximum amplitude value at point md7; 126°. All other points at 0.1 \( \Phi_N \) are significantly higher than flow rates 0.2 and 0.3 \( \Phi_N \).

Backflow of fluid into the impeller channel occurs largely at the core flow zone and is more intense toward the suction side of the impeller blade; arguably the source of congenial conditions for the initiation of stalling of the diffuser.

In the diffuser channel at \( f_{BPF} \), there is virtually no pressure recovery for monitoring points at the midpoint of the vaned diffuser channel to the outlet of the vaned diffuser channel.

The vaned diffuser begins to stall at \( t_{fN} = 15 \). Turbulence intensity also reduces at this stage relative to the period (0.4554 s) \( t_{fN} = 6 \), which shows the highest flow rate and is characterized by high flow velocity and turbulent intensity. The study shows the diffuser channel is completely stalled at a period (0.8280 s) \( t_{fN} = 20 \), where the features of turbulent intensity and velocity magnitude are the lowest. Furthermore, the second invariant of the velocity tensor; \( Q \)-criterion, reveals a nonstreamwise vortex at the leading edge of the vaned diffuser channel at a phase evolution of \( t_{fN} = 20 \), indicative of the beginning of stall.

The stall cells in the diffuser rotate at a propagation ratio of \( \Omega_{RS} = 0.078 \) of the rotor speeds at 0.2 \( \Phi_N \) and 0.086 of the rotor speeds at 0.1 \( \Phi_N \), the stall speed is much higher as the flow rate approaches the pump shut off.

### NOMENCLATURE

**Greek**
- \( b_2 \): Impeller outlet width, mm
- \( b_3 \): Volute inlet width, mm
- \( C_p \): Pressure coefficient
- \( D_1 \): Impeller inlet diameter, mm
- \( D_2 \): Impeller outlet diameter, mm
- \( D_3 \): Volute inlet diameter, mm
- \( D_4 \): Volute outlet diameter, mm
- \( f_{BPF} \): Blade passing frequency, Hz
- \( f_N \): Impeller rotating frequency, Hz
- \( H_d \): Pump head, m
- \( n_d \): Impeller rotating speed, r/min
- \( n_s \): Specific speed
- \( p \): Static pressure
- \( Q_d \): Pump capacity, m³/h
- \( Q^* \): Diffuser channel discharge, m³/h
- \( \text{RMS} \): Root mean square, m/s
- \( Z \): Blade number

**Latin**
- \( \eta_N \): Pump efficiency
- \( \Phi_N \): Flow coefficient
- \( \Psi_N \): Head coefficient
- \( \omega_{RS} \): Angular velocity of stall cell
- \( \omega_{rot} \): Angular velocity of impeller rotation
- \( \Omega_{RS} \): Normalized rotating stall speed

### ACKNOWLEDGMENTS

This study is supported and funded by the National Natural Science Foundation of China, No: 51576090. And Six Talent Peaks Project in Jiangsu Province, No: KTHY-060.

### CONFLICTS OF INTEREST

The authors declare no conflicts of interest.

### ORCID

Anthony A. Alubokin  https://orcid.org/0000-0001-7855-7157  
Zhang Ning  https://orcid.org/0000-0003-3158-1197  
Evans K. Quaye  https://orcid.org/0000-0002-5541-9883

### REFERENCES

1. Ullum U, Wright J, Dayi O, et al. Prediction of rotating stall within an impeller of a centrifugal pump based on spectral analysis of pressure and velocity data. J Phys Conf Ser. 2006;52: 36-45. doi:10.1088/1742-6596/52/1/004
2. Lennemann E, Howard J. Unsteady flow phenomena in rotating centrifugal impeller passages. J Eng Gas Turbines Power. 1970;92:65-71. doi:10.1115/1.3445302
3. Krause N, Zähringer K, Pap E. Time-resolved particle imaging velocimetry for the investigation of rotating stall in a radial pump. Exp Fluids. 2005;39:192-201. doi:10.1007/s00348-005-0935-2
4. Gülich JF. Centrifugal Pumps. Springer, Verlag; 2008.
5. Gao B, Guo P, Zhang N, Li Z, Yang M. Unsteady pressure pulsation measurements and analysis of a low specific speed centrifugal pump. J Fluids Eng Trans ASME. 2017;139:1-9. doi:10.1115/1.4036157
6. Dorfler P, Sick M, Coutu A. Flow-Induced Pulsation and Vibration in Hydroelectric Machinery. Springer; 2012.
7. Prunières R, Inoue Y, Nagahara T, et al. Investigation of the flow field and performances of a centrifugal pump at part load. IOP Conf Ser Earth Environ Sci. 2016;49:032015. doi:10.1088/1755-1315/49/3/032015
8. Zhang N, Yang M, Gao B, Li Z, Ni D. Unsteady pressure pulsation and rotating stall characteristics in a centrifugal pump with slope volute. Adv Mech Eng. 2014;(2014):1-11. doi:10.1155/2014/710791
9. Zhou P, Wang F, Mou J. Investigation of rotating stall characteristics in a centrifugal pump impeller at low flow rates. Eng Comput. 2017;34:1849-1873.
10. Brauer O. Part Load Flow in Radial Centrifugal Pumps, PhD Thesis. École Polytechnique Fédérale de Lausanne; 2009.
11. Lutter A, Brenner G. Numerical simulation and evaluation of velocity fluctuations during rotating stall of a centrifugal pump. J Fluids Eng Trans ASME. 2011;133:1-8. doi:10.1115/1.4004636
12. Zhang N, Liu X, Gao B, Xia B. DDES analysis of the unsteady wake flow and its evolution of a centrifugal pump. Renew Energy. 2019;141:570-582. doi:10.1016/j.renene.2019.04.023
13. Gyarmathy G, Inderbitzen A, Staubli T. Visualization of rotating stall in a single-stage centrifugal compressor. La Houille Blanche. 2001:40-45. doi:10.1051/lhb/2001034
14. Maeda H, Jittani Y. Unstable head-flow characteristic generation mechanism of a low specific speed mixed flow pump. J Therm Sci. 2006;15:115-120.
15. Berten S. Hydrodynamics of High Specific Power Pumps for Off-Design Operating Conditions. PhD Thesis. École Polytechnique Fédérale de Lausanne; 2010.
16. Pacot O. Large Scale Computation of the Rotating Stall in a Pump- Turbine using an Overset Finite Element Large Eddy Simulation Numerical Code PAR. 6183. PhD Thesis. École Polytechnique Fédérale de Lausanne; 2014.
17. Posa A, Lippolis A. A LES investigation of off-design performance of a centrifugal pump with variable-geometry diffuser. Int J Heat Fluid Flow. 2018;70:299-314. doi:10.1016/j.ijheatfluidflow.2018.02.011
18. Zhou P, Wang F, Yang Z, Mou J. Investigation of rotating stall for a centrifugal pump impeller using various SGS models. J Hydrol. 2017;29:235-242. doi:10.1016/S1001-6058(16)60733-3
19. Polkot R, Revell A, Craft T, Ashton N. Embedded DDES of 2D Hump Flow. In: Fu S, Haase W, Peng S, Schwamborn D, eds. Progress in Hybrid RANS-LES Modelling. Springer; 2012:169-179.
20. Spalart PR. Strategies for turbulence modelling and simulations. Int J Heat Fluid Flow. 2000;21:252-263. doi:10.1016/S0142-777X(00)00007-2
21. Deck S. Zonal-detached-eddy simulation of the flow around a high lift configuration. AIAA J. 2005;43:2372-2384. doi:10.2514/1.16810
22. Vatsa VN, Lockard DP, Spalart PR. Grid sensitivity of SA-based delayed-detached-eddy-simulation model for blunt-body flows. AIAA J. 2017;55:2842-2847. doi:10.2514/1.J055685
23. Dong Y, Deng X, Wang G. An enhanced version of delayed-detached-eddy simulation based on the v–f model. Flow Turbul Combust. 2019;102:167-188. doi:10.1007/s10494-018-9957-8
24. Spalart P, Jou R, Strelets W-H, Allmaras M. Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. In: Chaqunl L, Zhining L, eds. Advances in DNS/LES. Greven Press; 1997:137-147.
25. Spalart P, Deck R, Shur S, et al. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theor Comput Fluid Dyn. 2006;20:181-195. doi:10.1007/s00162-006-0015-0
26. Zhang N, Jiang J, Gao B, Liu X. DDES analysis of unsteady flow evolution and pressure pulsation at off-design condition of a centrifugal pump. Renew Energy. 2020;153:193-204. doi:10.1016/j.renene.2020.02.015
27. Jiyuan T, Guan-Heng Y, Chaqun L. Computational Fluid Dynamics, Second Edition: A Practical Approach. PhD Thesis. École Polytechnique Fédérale de Lausanne; 2019.
28. Ren Y, Zhu Z, Wu D, Li X, Jiang L. Investigation of flow separation in a centrifugal pump impeller based on improved delayed detached eddy simulation method. Adv Mech Eng. 2019;11:1-13. doi:10.1177/1687814019897832
29. He C, Liu Y, Yavuzkurt S. A dynamic delayed detached-eddy simulation model for turbulent flows. Comput Fluids. 2017;146:174-189. doi:10.1016/j.compfluid.2017.01.018
30. Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics the Finite Volume Method. Pearson Education Limited; 2007.
31. Menter, FR. Zonal Two Equation k-omega, Turbulence Models for Aerodynamic Flows. Fluid Dynamics Conference. American Institute of Aeronautics and Astronautics 370L'Enfant Promenade; 1993. pp. 1–21.
32. Bykovsky RK, Jacobsen CB, Pedersen N. Flow in a centrifugal pump impeller at design and off-design conditions-part II: large eddy simulations. J Fluids Eng Trans ASME. 2003:125:73-83. doi:10.1115/1.1524586
33. Spence R, Amaral-Teixeira J. Investigation into pressure pulsations in a centrifugal pump using numerical methods supported by industrial tests. Comput Fluids. 2008;37:690-704. doi:10.1016/j.compfluid.2007.10.001
34. ANSYS: ANSYS Fluent User's Guide. ANSYS Inc.; 2018.
35. EUROPÄISCHE NORM (EN). Rotodynamic pumps — hydraulic performance acceptance tests — Grades 1, 2 and 3 (ISO 9906: 2012). BSI Standards Publication; 2012.
36. Heng Y, Hu B, Jiang Q, Wang Z, Liu X. Stall mode transformation in the wide vaned diffuser of centrifugal compressors. Energies. 2020;13:1-14. doi:10.3390/en13226067
37. Taher H, Mohamed A, Osman B, Mohamed SG. Numerical Investigation of Rotating Stall Characteristics and Proceedings ASME 2014 Power Conference; 2014.
38. Posa A, Lippolis A, Balaras E. Investigation of separation phenomena in a radial pump at reduced flow rate by large-eddy simulation. J Fluids Eng Trans ASME. 2016;138:1-13. doi:10.1115/1.4033843
39. Guelich J, Hughes SF. Review of parameters influencing hydraulic forces on centrifugal impellers. 201, 163–174; 1986.
40. Emmons HW, Kronauer RE, Rockett JA. A survey of stall propagation—experiment and theory. ASME J Basic Eng. 1959;81:409-416.
41. Makay E. Centrifugal Pump Hydraulic Instability, (Project Report). Electric Power Research Institute 3412 Hillview Avenue, Palo Alto, California; 1980.
42. Maxime B. Fluid Dynamics of Cavitation and Cavitation. Springer; 2004.
43. Sinha M, Pinarbasi A, Katz J. The flow structure during onset and developed states of rotating stall within a vaned diffuser of a centrifugal pump. J Fluids Eng Trans ASME. 2001;123:490-499. doi:10.1115/1.1374213
44. Jeong J, Hussain F. On the identification of a vortex. J Fluid Mech. 1995;285:69-94. doi:10.1017/S0022112095000462
45. Zhang Y, Gao B, Alubokin AA, Li G. Effects of the hydrofoil blade on the pressure pulsation and jet-wake flow in a centrifugal pump. *Energy Sci. Eng.* 2021:1-14. doi:10.1002/ese3.865

46. Qin W, Tsukamoto H. Theoretical study of pressure fluctuations downstream of a diffuser pump impeller-part 2: effects of volute, flow rate and radial gap. *J Fluids Eng Trans ASME.* 1997;119:653-658. doi:10.1115/1.2819294

47. Gyarmathy G. *Impeller-Diffuser Momentum Exchange During Rotating Stall.* ASME; 1996.

**How to cite this article:** Alubokin AA, Gao B, Ning Z, Yan L, Jiang J, Quaye EK. Numerical simulation of complex flow structures and pressure fluctuation at rotating stall conditions within a centrifugal pump. *Energy Sci Eng.* 2022;10:2146-2169. doi:10.1002/ese3.1123