

Numerical Investigation of Unsteady Flow Behavior in a Centrifugal Pump under Off-Design Conditions

M. Elsherbiny¹

m_elsherbiny@sh-eng.menofia.edu.eg

Menofia University

S. M. Selim²

Menofia University

Abraham Engeda

Michigan State University

Ahmed Hegazy³

Menofia University

H. A. Abdalla⁴

Menofia University

Article

Keywords: Centrifugal pump, rotating stall, off-design operation, URANS, SST $k-\omega$, pressure pulsation, CFD, flow instability

Posted Date: September 21st, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-10950731/v1>

License:   This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Additional Declarations: No competing interests reported.

Numerical Investigation of Unsteady Flow Behavior in a Centrifugal Pump under Off-Design Conditions

M. Elsherbiny ^a, S. M. Selim ^a, Abraham Engeda^b, Ahmed Hegazy ^a, H.A. Abdalla ^a

^a Mechanical Engineering Department, Faculty of Engineering, Menofia University, Shebin Elkom, Egypt.

^b Turbomachinery lab, Michigan State University, East Lansing, MI, USA

Corresponding author: M. Elsherbiny (m_elsherbiny@sh-eng.menofia.edu.eg)

ABSTRACT

Because system demand changes, centrifugal pumps may need to operate off their design points. This can cause unsteady flow with complex behavior. This understanding highlights the need to address flow instability phenomena such as rotating stall, which involves flow separation, pressure pulsations, and circumferential non-uniformity, and therefore influences pump performance. This paper studies unsteady flow phenomena in a single-stage centrifugal pump operating at part load through an extensive numerical examination. Transient simulations were performed using the Unsteady Reynolds-Averaged Navier–Stokes (URANS) equations coupled with the SST $k-\omega$ turbulence model. A sliding mesh technique was used to capture rotor-stator interaction and time-dependent flow patterns accurately. The numerical model was validated against Particle Image Velocimetry (PIV) data, and the predicted velocity distributions and vortex structures showed good agreement.

The study is intended to demonstrate the flow properties and their behavior under off-design conditions. The results show that rotating stall is caused by boundary-layer separation on the suction side of the impeller blades. This separation creates low-momentum recirculation zones that evolve into rotating stall cells. These cells intermittently block flow paths, causing large pressure variations and flow redistribution. At partial-load operation, energy dissipation at the impeller inlet dominates. At about 41% of the design flow rate, a transitional step takes place as significant instability begins to arise. The results provide new insight into the relationship between intake distortion, vortex behavior, and outlet recirculation, and improve the design and operational reliability of centrifugal pumps.

Keywords: Centrifugal pump, rotating stall, off-design operation, URANS, SST $k-\omega$, pressure pulsation, CFD, flow instability

1. INTRODUCTION

Centrifugal pumps are extensively used in various industrial applications, including water supply, power generation, petrochemical processing, and irrigation systems, because of their reliability and ability to deliver high flow rates with stable performance [1,2]. Centrifugal pump flow looks simple, but it is very complicated. It contains three-dimensional turbulent structures affected by rotation, pressure gradients, and rotor-stator interactions [3,4].

Rotating stall is one of the most significant instabilities in turbomachinery. It is characterized by localized blockage regions (stall cells) that travel circumferentially in the impeller. These regions have lower static pressure than the surrounding flow and induce vortex generation and secondary flow near the impeller inlet, as Emmons et al. first described [5]. Stall cells cause circumferential flow non-uniformity, pressure pulsations, vibration, and possible structural damage [6–9].

Laser Doppler Velocimetry (LDV) [10] and Particle Image Velocimetry (PIV) experimental investigations [11,12] have given valuable information about internal flow structures. Flow instability is closely correlated with the boundary-layer separation on the suction side of impeller blades under adverse pressure gradients [13].

Meanwhile, numerical simulations have become necessary to analyze unsteady flow behavior. Advanced techniques such as Detached-Eddy Simulation (DES) and hybrid turbulence models are able to resolve vortex dynamics and pressure fluctuations [14–16]. Moreover, new studies have been directed to prediction of the rotating stall evolution under transient conditions [17]. Fluid-structure Interaction has also been studied for pump systems [18, 19]. Researchers have also examined the effect of diffusers [20].

However, the detailed mechanisms of the inception, propagation, and spatial development of rotating stall in centrifugal pumps are still not fully understood. In particular, the coupling of upstream flow distortion, intra-passage vortex dynamics, and outlet recirculation, and their combined effect on transient hydraulic loading, have not been fully characterized within a unified framework.

The novelty of the present work resides in a detailed analysis of the evolution of the rotating stall in a spatially resolved way over a few impeller radii, combining the flow characterization upstream with a time-resolved vortex dynamic. Existing literature mainly concerns the inlet condition or local flow behavior in impeller. Conversely, the present work systematically investigates the interaction between the inlet pre-rotation, the formation of stall cells in the blade passages and the redistribution of the flow at the outlet under different part-load conditions. The study also finds a transitional operating regime related to the onset of severe instability. It gives new insight into the function of reverse flow and vortex interaction in stall propagation.

Hence, the present study focuses on investigating the transient flow behavior and rotating stall mechanisms in a single-stage centrifugal pump operating under off-design conditions. Computational Fluid Dynamics (CFD) simulations are performed using the Unsteady Reynolds-averaged Navier Stokes (URANS) equations coupled with a shear stress transport (SST) turbulence model. The interaction between the rotating impeller and the stationary casing is modeled using a sliding mesh technique, which is necessary to capture accurately time-dependent flow structures. The evolution of velocity fields, pressure pulsations, and vortex structures is emphasized to gain deeper insight into flow instability mechanisms and their impact on pump performance, operational stability, and structural reliability.

Nomenclature

C_p – Pressure coefficient

dV_{ol} – The control volume (m^3)

dt – Time step (sec)

F – Force (N)

f – Frequency (HZ)

G_k – Generation of turbulence kinetic energy

$G\omega$ – Generation of specific dissipation rate

k – Turbulence kinetic energy (m^2/sec^2)

p – Static pressure (Pa)

p_t – Instantaneous static suction pressure (Pa)

Q – Flowrate (m^3/sec)

Q_c – Q-criterion ($1/sec^2$)

S_{ij} – Mean rate-of-strain tensor ($1/sec$)

$S\phi$ – Source term including centrifugal, Coriolis, and body forces

t – Time (sec)

T – Cycle time (sec)

U_2 – Circumferential velocity at impeller outlet (m/sec)

$\vec{V}$ – Flow velocity vector (m/sec)

$\vec{V}_g$ – Moving mesh velocity (m/sec)

W – Relative velocity (m/sec)

Y_k – Dissipation of turbulence kinetic energy

Y_ω – The dissipation of ω

Z – number of blades

Greek Symbols

β – Flow angle (deg)

ρ – Fluid density (kg/m^3)

ω – Specific dissipation rate ($1/sec$)

Ω – Enstrophy (m^2/sec^2)

ϕ – General scalar variable in conservation equation

Γ – Diffusion coefficient

σ_k – Turbulent Prandtl numbers for k

σ_ω – Turbulent Prandtl numbers for ω

subscripts

axial – axial direction

d – designed

r – rotating shaft

X – x direction

Y – Y direction

2. NUMERICAL MODELLING

The internal flow in the centrifugal pump is modeled as a three-dimensional, incompressible, isothermal turbulent flow. The governing equations are the Reynolds Averaged Navier Stokes equations, which conserve mass and momentum [3,4]. The computational domain consists of the inlet pipe, the rotating impeller, and the stationary volute casing. The geometry is discretized using an unstructured mesh, with local refinement near blade surfaces and leading edges to capture strong velocity gradients and boundary-layer development.

2.1. Governing Equations

The unsteady Reynolds-averaged Navier–Stokes (URANS) equations, which embody the conservation of mass and momentum, are the governing equations.

The continuity equation can be written as:

$$\nabla \cdot (\vec{V} - \vec{V}_g) = 0 \quad (1)$$

The general scalar transport equation is given by

$$\rho \frac{d}{dt} \int \phi dV_{ol} + \rho \int \phi (\vec{V} - \vec{V}_g) \cdot \vec{dA} = \int \Gamma \nabla \phi \cdot \vec{dA} + \int S_\phi dV_{ol} \quad (2)$$

In this equation, ϕ denotes a general transported variable, Γ is the diffusion coefficient, and S_ϕ is the source term that encompasses contributions from centrifugal, Coriolis, and body forces.

2.2. Turbulence Modeling

The SST k- ω turbulence model is utilized in the present work because it is highly accurate in predicting the flow separation under adverse pressure gradients and is suitable for turbomachinery applications [14].

The transport equation for turbulence kinetic energy k is:

$$\rho \frac{d}{dt} (k) + \rho \nabla \cdot [k(\vec{V} - \vec{V}_g)] = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k - Y_k \quad (3)$$

The transport equation for specific dissipation rate ω is:

$$\rho \frac{d}{dt} (\omega) + \rho \nabla \cdot [\omega(\vec{V} - \vec{V}_g)] = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \nabla \omega \right] + G_\omega - Y_\omega \quad (4)$$

where the production terms are defined as:

$$G_k = 2\mu_t S_{ij} S_{ij} \quad (5)$$

$$Y_k = \rho \beta^* k \omega \quad (6)$$

The SST formulation use the k- ω formulation in the vicinity of the walls and the k- ϵ formulation in the free stream. This increases the accuracy of the model for the leading-edge separation, the trailing edge wake, the hub-shroud vortices and the curvature-driven turbulence. LES approaches [15,26] enhance the resolution of fine-scale vortical structures, whereas URANS with SST closure present a pragmatic balance between computational cost and predictive accuracy for comprehensive transient multi-revolution simulation.

2.3. Computational Domain

The computational domain consists of an inlet pipe, a rotating impeller, and a casing, as shown in Fig. 1. The geometry is described in detail by Krause et al. [12] and Feng et al. [20]. The pump model is meshed with an unstructured mesh of about 3.2 million elements, with local refinement near the blade surfaces to resolve boundary layers and vortex structures. Grid independence was verified according to the procedure proposed by Feng et al. [20].

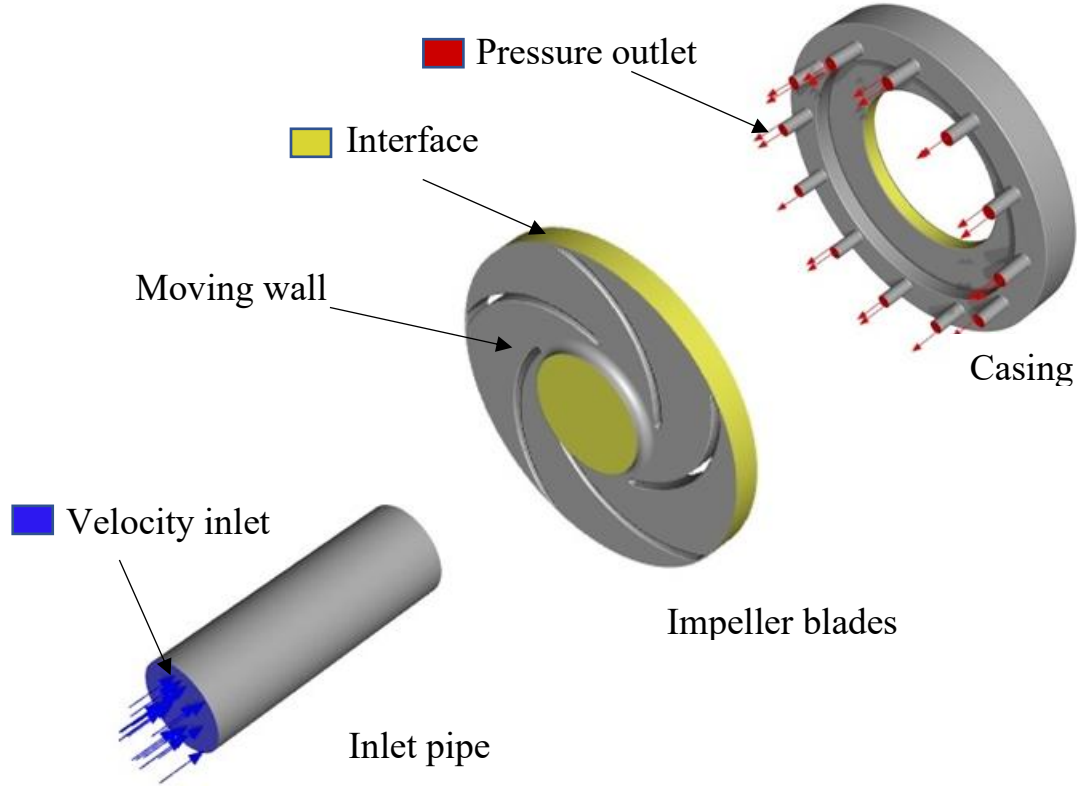


Fig. 1. Computational domain and mesh distribution of the centrifugal pump.

2.4. Numerical Method

The numerical simulations are performed using ANSYS Fluent with a pressure-based transient solver. The finite volume method discretizes the governing equations. Second-order spatial discretization schemes are applied to the convective terms, while second-order implicit time integration captures transient flow behavior accurately. Pressure–velocity coupling is achieved using the coupled algorithm. Residual convergence criteria are set to 10^{-6} to ensure numerical stability and accurate prediction of oscillatory flow phenomena.

2.5. Boundary Conditions

The working fluid is water. A rotating coordinate system is set for the pump impeller and the gap, and a stationary system is set for these components. Interfaces exist between these connecting faces for the two reference systems, where the sliding mesh strategy is applied. The boundary conditions are as follows. The inlet boundary condition is specified at the domain inlet as a uniform velocity corresponding to the prescribed flow rate, and the total pressure is assigned at the domain outlet. An interface matches the flow properties between the inertial and non-inertial domains. The impeller rotates at 600 rpm in the rotating domain in the clockwise direction, while all solid surfaces use no-slip wall conditions. The simulation time spans 20 impeller revolutions in order to ensure the transient solution reaches periodic stability.

2.6. Monitoring Radii Configuration

In order to simultaneously capture the unsteady flow evolution across all impeller passages, multiple monitoring radii (R1 to R4) were established along the mid-span, as detailed in Table 1 and illustrated in Fig. 2. The figure defines the measurement locations (R1–R4), the azimuthal reference ($\theta = 0$), and the circumferential direction, enabling detailed tracking of flow evolution across blade passages. The specific location of each impeller passage is identified by the prefix CH and a subscript number. For the complete flow visualization, we coupled additional sectional planes and iso-surfaces in the impeller domain. This allows the exact tracking of flow acceleration, jet-wake interactions, secondary flow structures and harmonic amplification.

Table 1: Dimensions of the measuring radii

Measuring Radius	R1	R2	R3	R4
Value	82.776 mm	102.05 mm	120.85 mm	137.448 mm

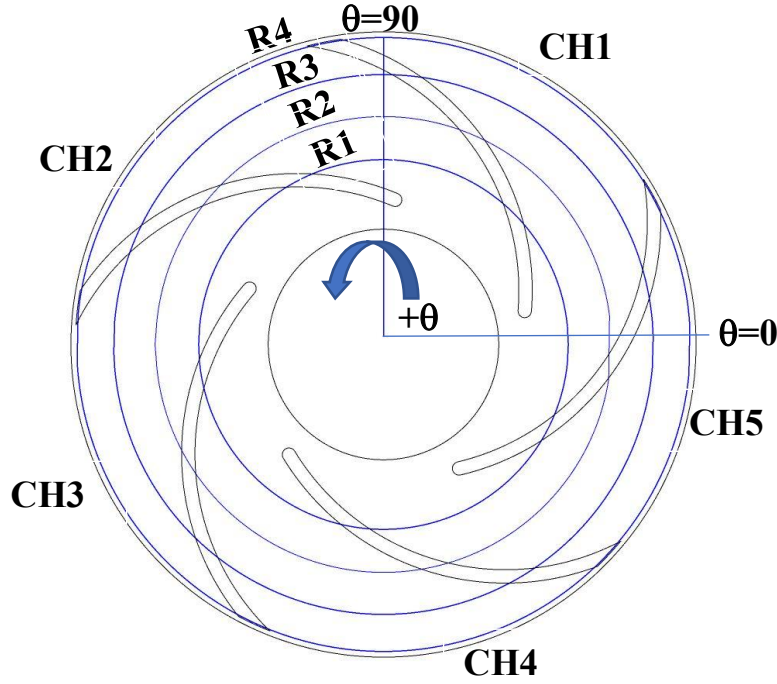


Fig. 2. Configuration of monitoring radii within the impeller.

3. MODEL VALIDATION

A comparison of the numerical results with PIV measurements reported by Krause et al. [12] shows good agreement in the vortex structures and flow separation patterns. Small differences appeared, which may be due to measurement limits and model assumptions.

3.1. Flow at 50% Design Flow Rate

At 50% of the design flow rate (Figs. 3a-b), strong recirculation vorticity, as shown by PIV and simulation results, blocks the impeller passages. PIV measurements show localized flow separation and two vortical structures of different sizes near the suction side of the blade (Fig. 3a). However, due to the numerical simulation assumption, the position of the smaller vorticity size (Fig. 3b) shows a slight difference between the two results. Large vortices appear near the suction side, partially blocking the flow passage. The simulation accurately captures vortex location and flow separation.

3.2. Flow at 30% Design Flow Rate

Two large vortices formed between the impeller blades, as shown in Figs. 3c-d, indicated by both experimental and numerical results. The positions of both vortices are approximately well-determined, and the most attractive feature is the simulation of the flow behavior near the leading edge of the suction side. The simulation shows stronger vortices and flow asymmetry. Minor discrepancies in vortex size are attributed to measurement timing and modeling assumptions.

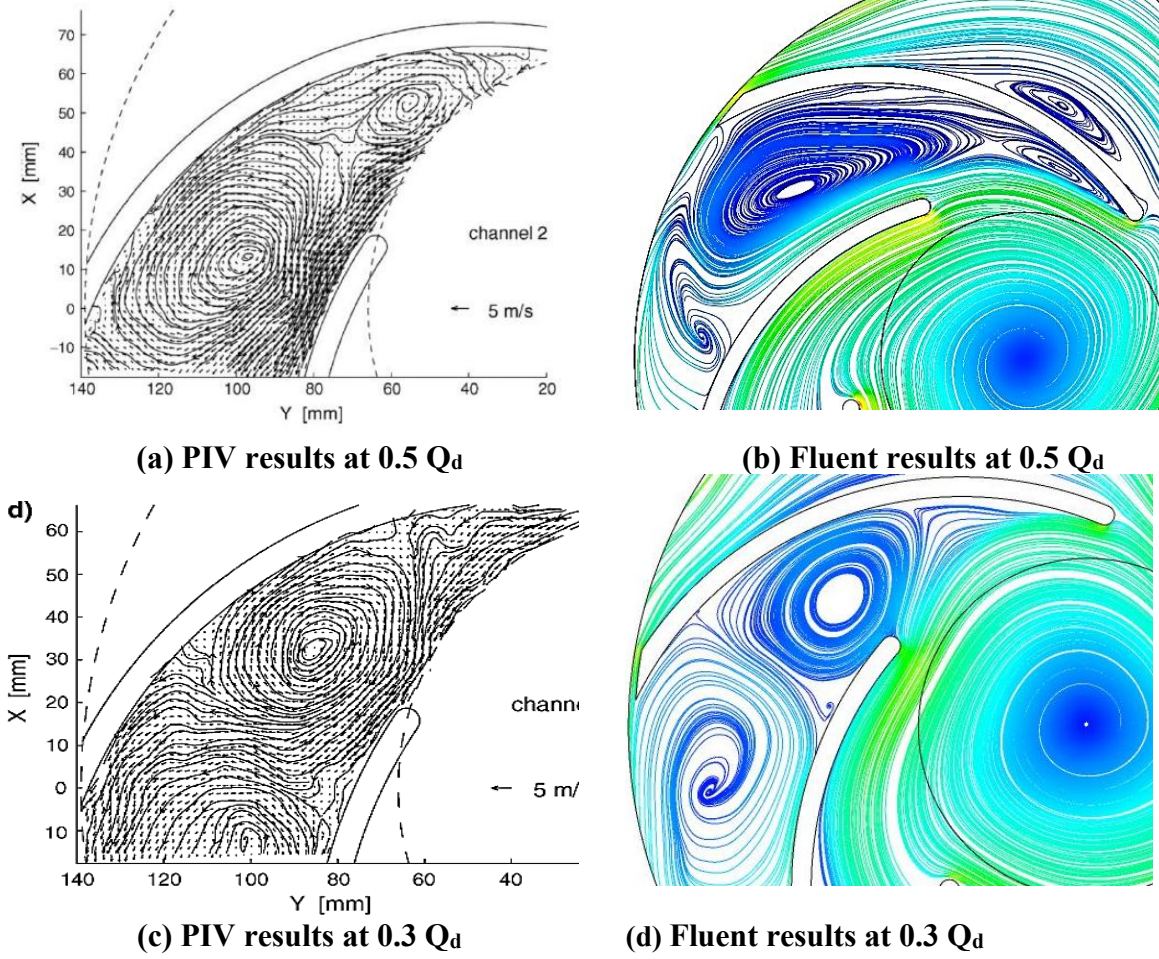


Fig. 3. Comparison between experimental (PIV) and numerical results under off-design conditions.

4. NUMERICAL RESULTS AND DISCUSSION

Transient CFD simulations were used to investigate the complex unsteady flow behavior inside the centrifugal pump operating at off-design conditions. Special attention was paid to the formation, evolution, and propagation of stall cells and the effect on velocity distribution, pressure pulsations, and streamlines inside the pump.

The numerical investigation was conducted at three part-load operating conditions corresponding to 30%, 41%, and 50% of the design flowrate (Q_d). These conditions represent typical operating regimes in which centrifugal pumps may experience flow-stable and non-stable stall (rotating stall phenomena) [6, 13]. The results are discussed in terms of pressure behavior, radial velocity field (V_r), tangential velocity (W_q), and relative velocity magnitude (W), as indicated in Fig. 7.

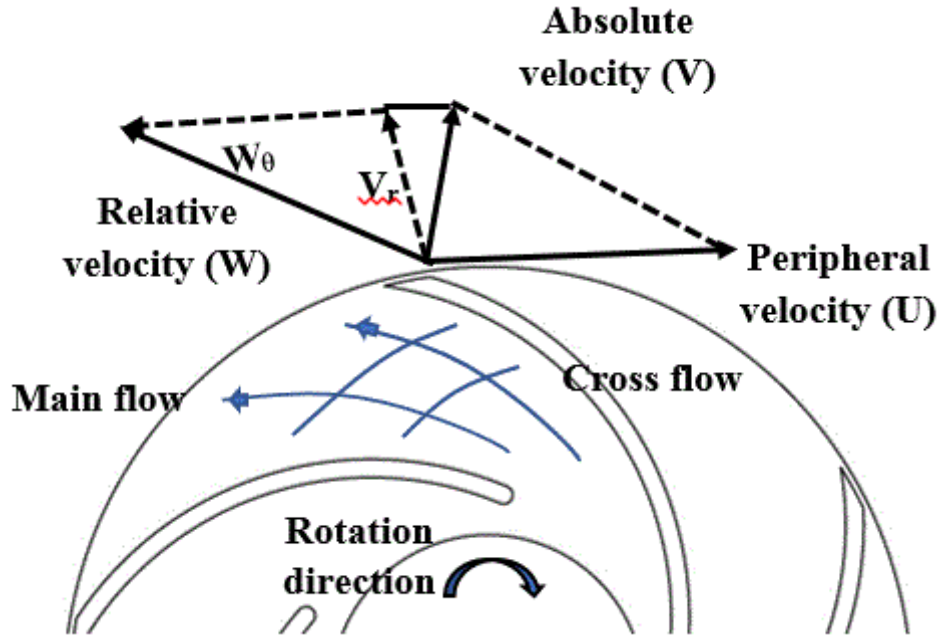


Fig.4, Velocity vector relationship in the impeller

4.1 Flow Behavior inside the Pipe

Three cross-sectional planes (Planes 1–3) are defined upstream of the impeller eye to analyze the development of axial and whirl velocity components at different flow rates (50%, 41%, and 30% Q_d). The planes are set perpendicular to the axial coordinates through the pipe as mentioned in Table 2 and shown in Fig.5

Table 2: Dimensions of the measuring planes

Measuring planes	Plane 1	Plane 2	Plane 3
Value in Z direction	329 mm	38 mm	28 mm

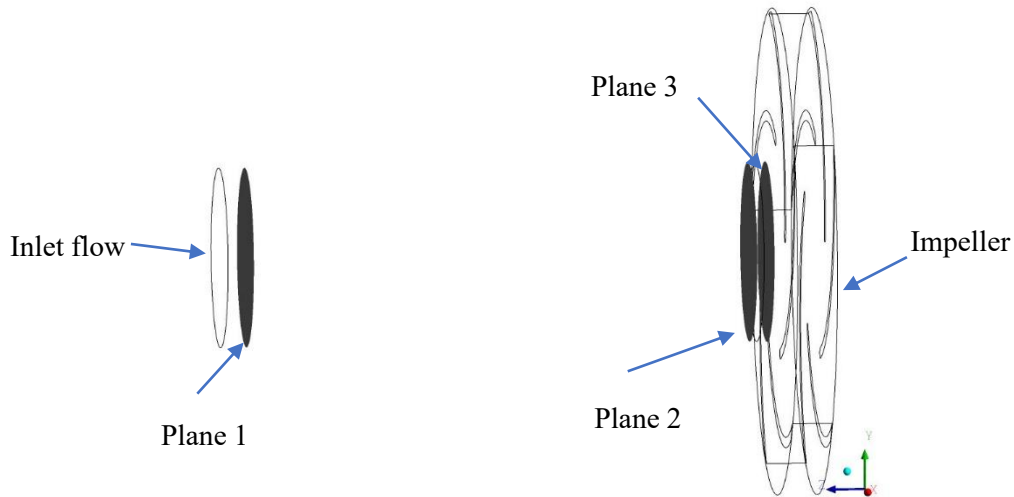


Fig. 5. Location of upstream measurement planes.

At Plane 1, located farthest from the impeller, the flow remains relatively uniform inside the pump section across all compared flow rates. Axial velocity depends on flow rate and increases in the

pipe center, as shown in Fig. 6; however, the whirl velocity component is negligible, indicating the downstream flow does not affect this location.

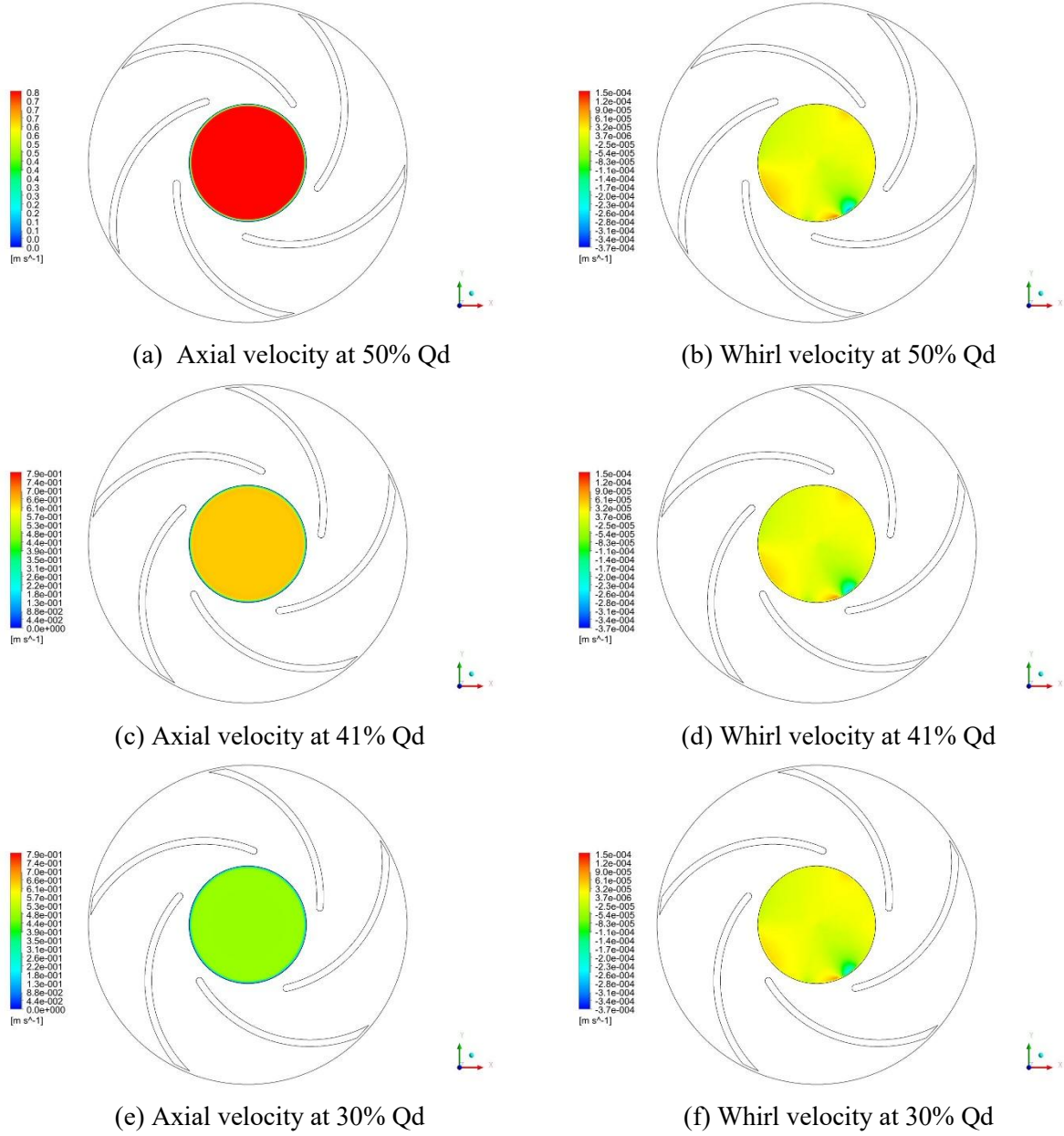


Fig. 6. Velocity components at Plane 1 (near impeller inlet).

Figure 7 illustrates velocity characteristics at Plane 2. The axial velocity increases near the wall and decreases near the center (Figs. 7a, c, and e, as a result of changing the direction from axial toward radial, similar to the flow behavior in the bend direction. Moreover, the flow increases near the suction side of the downstream impeller blade. As the operating condition shifts toward lower flow rates, the stagnant region area inside the pipe increases. Moreover, the absolute whirl, illustrated in Figs. 7b, d, and f, has a positive value because the flow rotates in the same direction as the impeller. However, for stalling cells, the whirl velocity is negative.

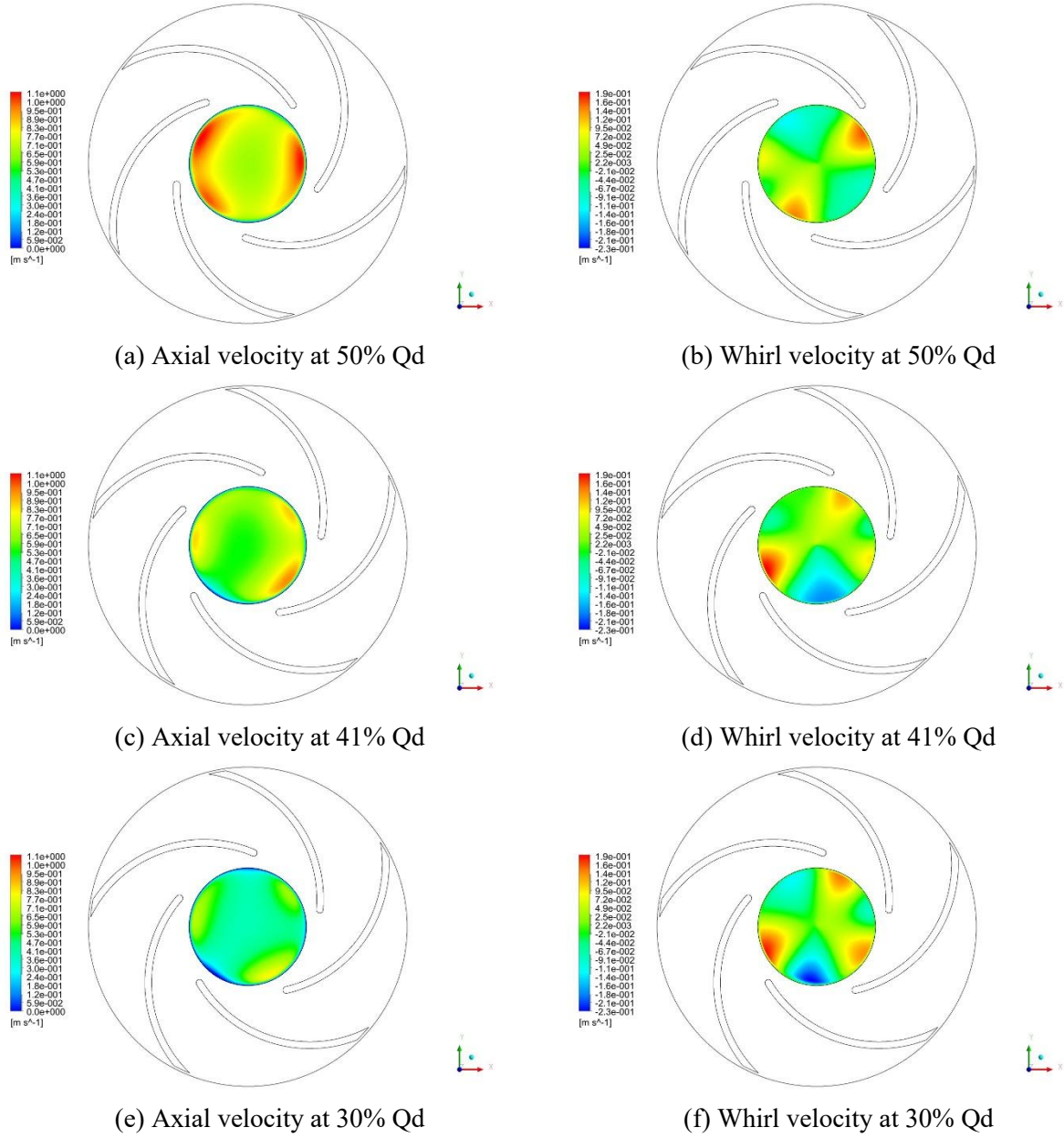


Fig. 7. Velocity components at Plane 2 (intermediate upstream location).

At Plane 3, located closest to the impeller eye and influenced by the rotating boundary wall, the velocity components are shown in Fig. 8. The axial velocity (Figs. 8a, c, and e) indicates a reverse flow on the suction side of the impeller leading edge. Moreover, the stalled passages influence the upstream pipe by indicating a reverse flow inside it. As the handling flow rate decreases, the reverse-flow region increases. Moreover, pronounced pre-rotation with large-scale vortical structures occurred inside the pipe. This is evident from the change in the whirl component from a positive value at the previous plane to a negative value at this plane (Figs. 8b, 8d, and 8f).

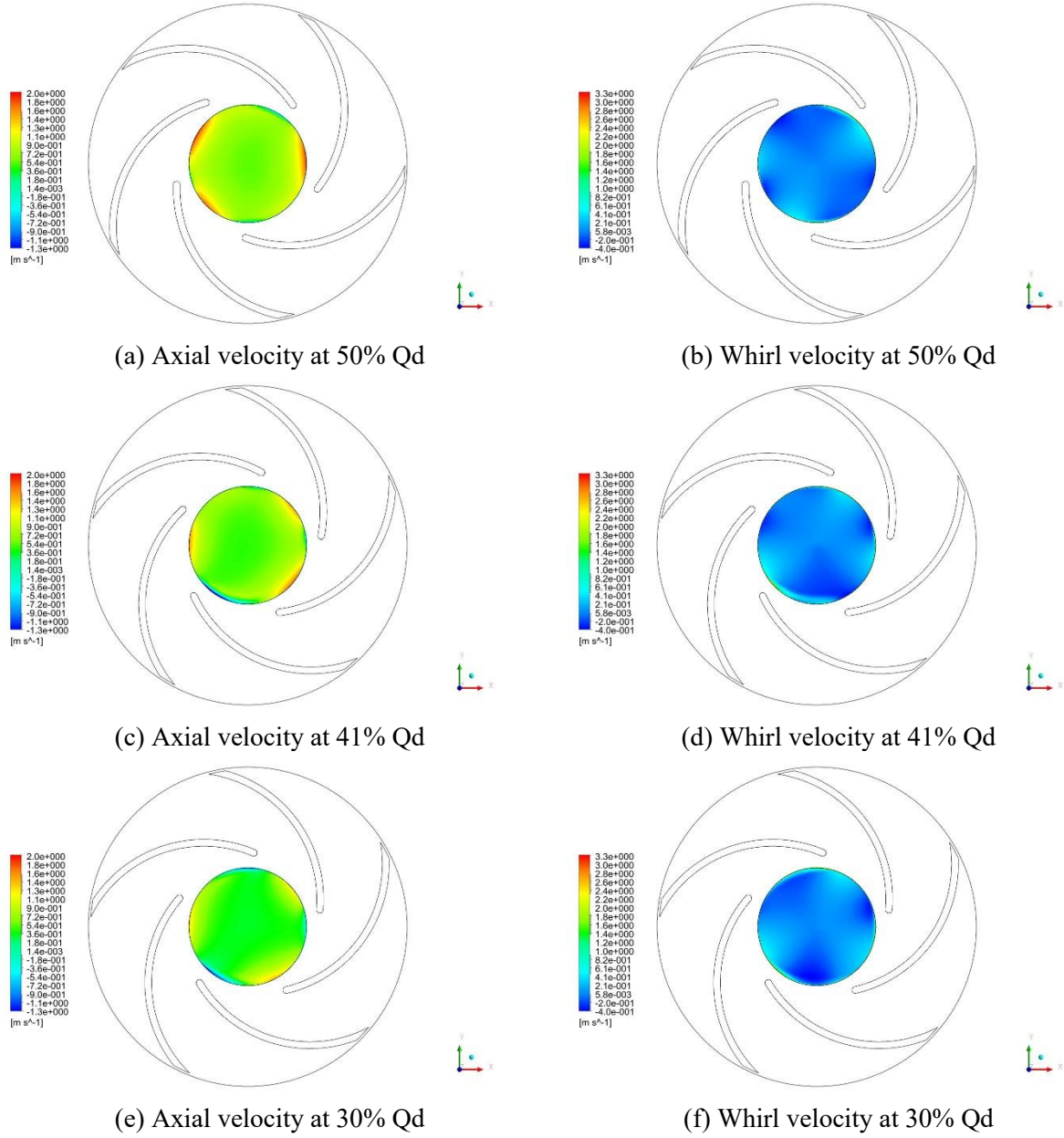


Fig. 8. Velocity components at Plane 3 (farthest upstream).

4.2 Fluid Properties inside the Impeller

4.2.1. Flow behavior at R1 (inlet region)

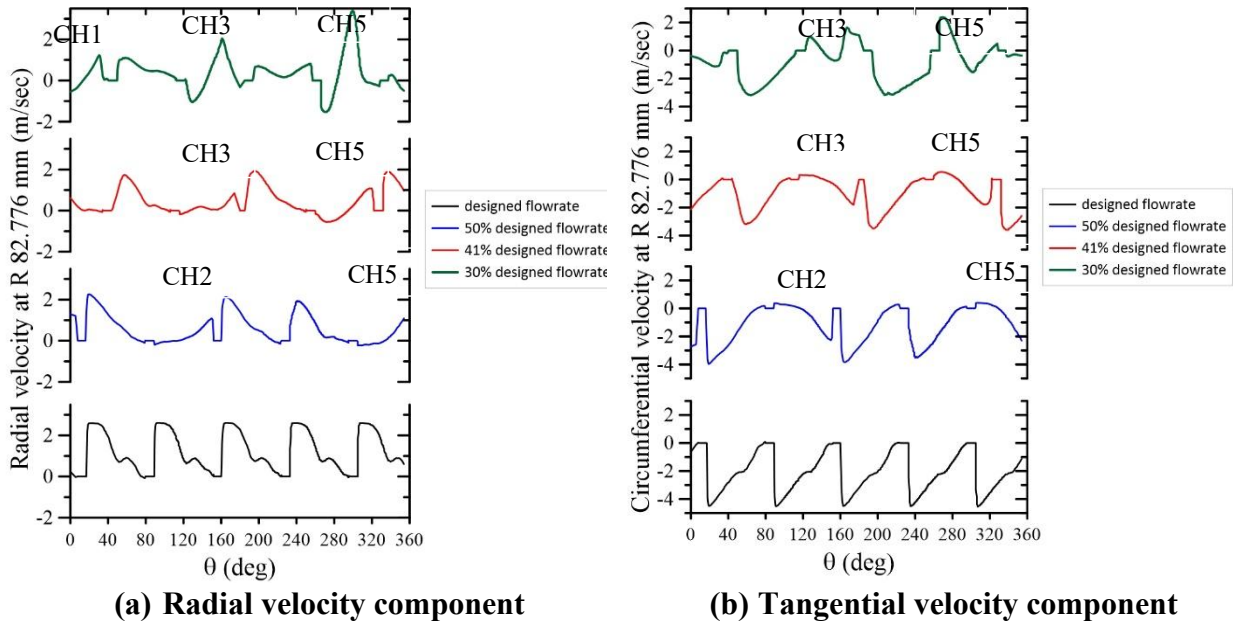
Figure 9 shows distributions of (a) radial velocity, (b) tangential velocity, (c) relative velocity, and (d) pressure coefficient at radius (R1), showing the onset of stall cells, flow separation, and non-uniformity of pressure distribution under part load conditions. The radial velocity component (V_r), which is directly related to the mass-flow distribution in the blade passages (Fig. 9a), is the most important indicator of the impeller through-flow capacity. The radial velocity profile is quite uniform under the design operating conditions and decelerates gradually from the suction side to the pressure side. Conversely, under part-load conditions, the emergence of stall cells, rotating about the axis perpendicular to the main flow (Z-direction), induces a significant deterioration in the radial velocity field, notably within passages CH2 and CH5 at 50% Qd (Fig. 9a). Although stalling is localized in two passages, the radial velocities in the remaining active

passages still fall below design-condition values. This indicates that the flow does not spontaneously redistribute to replicate design-equivalent states in the non-stalled passages perfectly. Furthermore, anomalous fluid behavior near the pressure side of the stalled passages suggests that the primary vortex core originates in the vicinity of the suction side.

As the operating flow rate deviates further from the design point, the number of stalled passages increases, accompanied by a marked amplification in vorticity intensity. This is evidenced by steep radial velocity gradients, such as those observed in CH5 at 0.41 Qd. Notably, at 0.3 Qd (Fig. 9a), the radial velocity field reveals the formation of two differently sized vortices in passages CH3 and CH5. In addition, a U-shaped flow profile emerges at CH4, with a stagnant core in the middle of the passage and active flow restricted to the boundary regions near the blade surfaces. Subsequently, the adjacent passage (CH5) exhibits a flow structure similar to CH3 but characterized by much higher fluctuations in radial velocity, indicating a significantly more intense vortex.

The tangential (circumferential) velocity component (W_θ) provides additional information on the intra-passage vorticity dynamics (Fig. 9b). Negative W_θ corresponds to counterclockwise flow rotation in the same direction as the rotation of the impeller. Under design conditions the flow is almost free of perpendicular (Z-axis) vortices and tangential velocity decreases toward the pressure side due to Coriolis forces. However, vorticity becomes highly pronounced under part-load conditions. At 50%Qd, the stalled cells exhibit minor positive tangential velocity values near the suction surface in CH2 and CH5. Further flow rate reductions sharply increase eddy intensity, reflected in steep gradients in W_θ . The resulting wave-like profile of W_θ (Fig. 9b) confirms that this swirling vorticity is physically constrained between the bounding blades of the respective passages.

From the observations of non-stalled cells at all operating regimes, we find the relative velocity profile (Fig. 9c) to be inversely related to the pressure distribution (Fig. 9d). In the stalling cells the magnitude of the relative velocity exhibits a characteristic double-peak profile. These peaks are proportional to the strength of the swirling of the trapped vortex. Regions with near-zero relative velocity indicate localized flow stagnation, which is clearly observed in CH2 and CH5 at 50% Qd, and in CH3 at 41% Qd. Consequently, the presence of these stall cells profoundly alters the intra-passage pressure field (Fig. 9d). As vortex intensity escalates, the localized pressure drops drastically, falling even below the baseline suction-side pressure—a phenomenon prominently captured in CH3, CH5, and CH1 at 30%Qd, as well as in CH5 at 41% Qd.



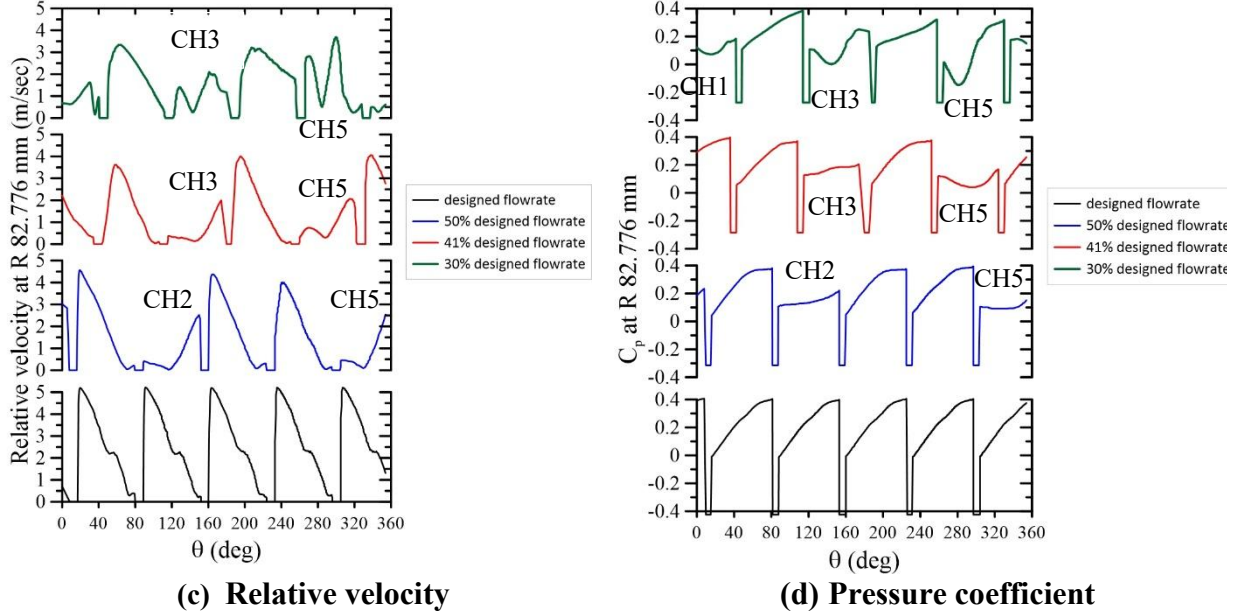


Fig. 9. Flow characteristics at radius R1 (impeller inlet region).

4.2.2. Flow behavior at R2 (Mid-Radius Region)

Fig. 10 presents the variation of stall cells, vortex strength, and pressure non-uniformity along the flow from inlet to outlet. Off-design operation improves vortex interaction, redistributes the flow, and reduces energy-transfer efficiency, as shown by the distributions of the velocity components and the pressure coefficient. At monitoring radius R2, the flow area increases due to the constant blade height, resulting in a radial velocity reduction in the intra-passage at the design condition (Qd) (Fig. 10a). However, the radial velocity at the suction side is approximately 5 times its value at the pressure side. At partial loads (50% and 40% Qd), the radial velocity peak in non-stalling cells goes up to about 30% of the design value. Stalling regions at 50% and 41% Qd are still recognizable by wave-like oscillations in radial velocity (Fig. 10a) and by magnitude in relative velocity near the pressure side (Fig. 10c), as mentioned before.

For the upstream vortices with low swirl (e.g., CH2 and CH5 at 50% Qd), the radial velocity gradient between the pressure and suction sides increases, i.e., the vorticity core approaches. The gradient is increased at lower flow rates (41% Qd for CH3/CH5 and 30% Qd for CH1/CH3). However, the affected area is localized and not intra-passage near the suction side, leading to secondary vorticity generation. Therefore, new vortices emerge near the pressure side of CH3 and CH5 at 41% Qd and near CH1, CH3, and CH4 at 30% Qd. In contrast, the radial velocity fluctuations of CH5 at 30% Qd are diminished, implying the removal of the dominant upstream vortex.

The relative velocity around the pressure side is lower than around the suction side at the design conditions at R2, indicating lower blade loading than at R1. Therefore, optimizing the blade geometry at this radius is expected to result in a smoother energy transfer. Fig. 10b shows that at flow rates below 50% Qd, the circumferential velocity at most stalling passages is positive, meaning the vorticity center is not along radius R2. This is attributed to either a large vorticity volume or the joining of two different vortices, which removes the fluid from the suction side and reduces the energy conversion.

Vortices are further evidenced by humps in the relative velocity magnitude (Fig. 10c) and low-pressure zones in the pressure coefficient plots (Fig. 10d), as indicated in the stalled cells at 41%

and 50% Qd. The physical size of these vortices correlates directly with the area of pressure reduction on the curves. At 30% Qd, vortices affect the remaining impeller passages, as indicated by the middle peak in relative velocity, resulting in minimal pressure variation between the various impeller passages.

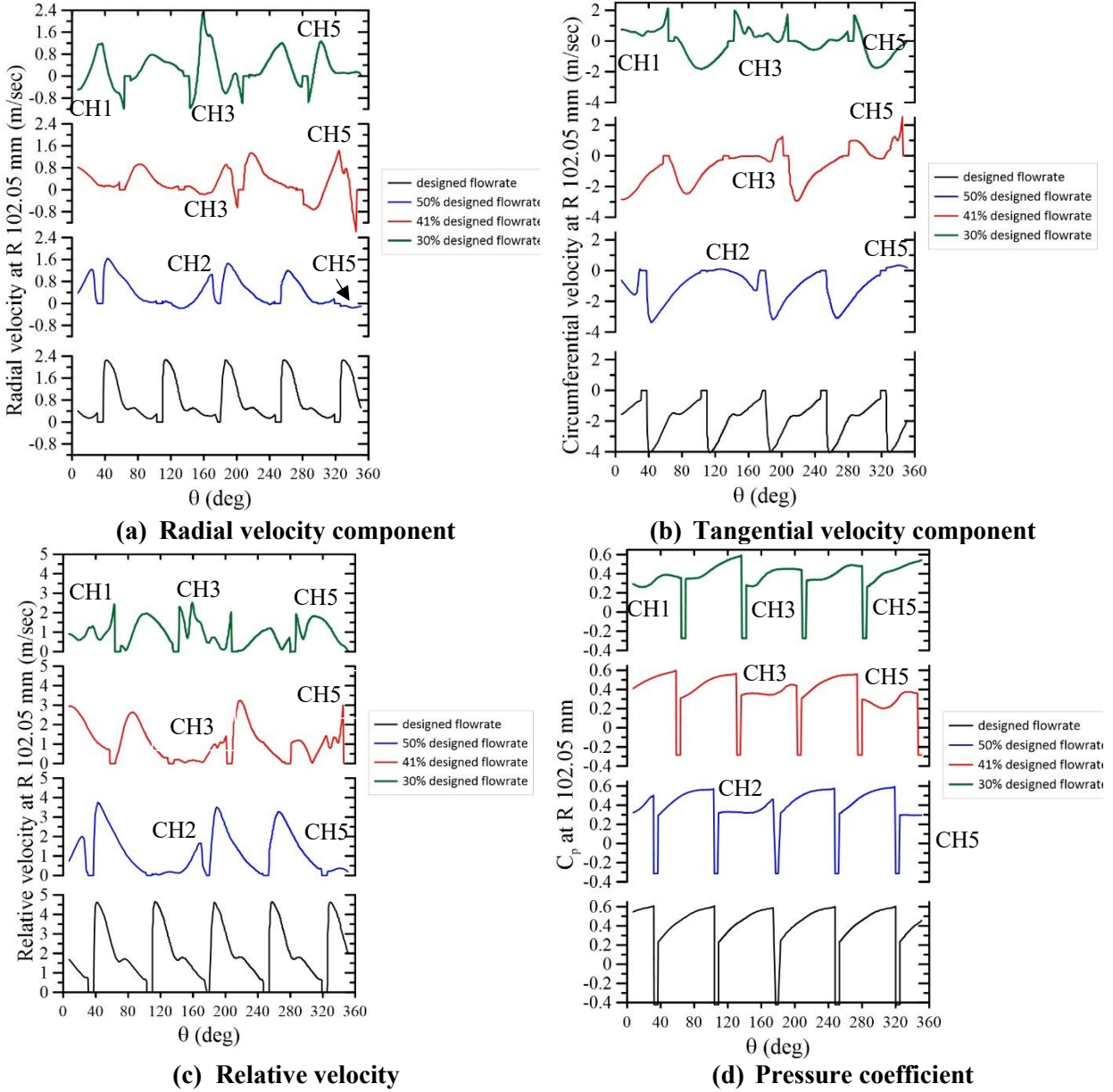


Fig. 10. Flow characteristics at radius R2 (mid-impeller region).

4.2.3. Flow behavior at R3 (Near Outlet)

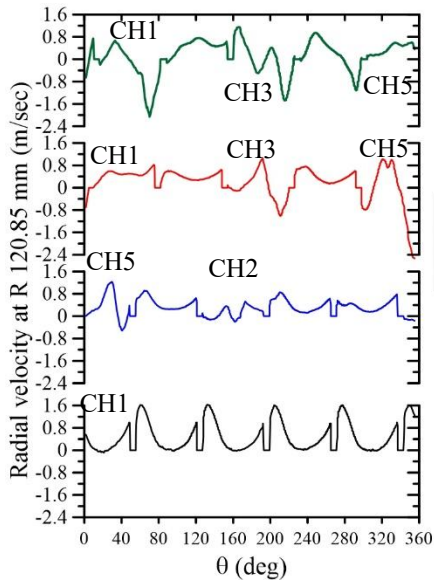
At the radius near the impeller outlet (R3), the design condition (Qd) is characterized by a reduction in radial velocity near the suction side (Fig. 11a). Effective energy transfer and centrifugal forces are evidenced by a "U-shaped" profile with an increase in radial velocity near the pressure side, reaching approximately half the magnitude of the suction side velocity.

At 50% Qd, radial velocity is nearly uniform across non-stalling passages. However, Channel 5 (CH5) develops new vorticity near the pressure side, indicating localized flow separation. A critical feature at this flow rate is the stagnation observed across half of Channel 2 (CH2). This suggests backflow originating from the outer diameter, which obstructs the main flow and forces the fluid to redistribute toward the hub or shroud. As the handling flowrate decreased, swirling vorticity on the pressure side of the blade at the upstream location (R2) increased (CH3 and CH5 at 41% Qd, and CH1 and CH3 at 30% Qd). The high swirling intensity pronounced the high blockage vorticity in the upstream radius. On the other side, the vortices near the suction side are reduced (CH5, at 30% Qd).

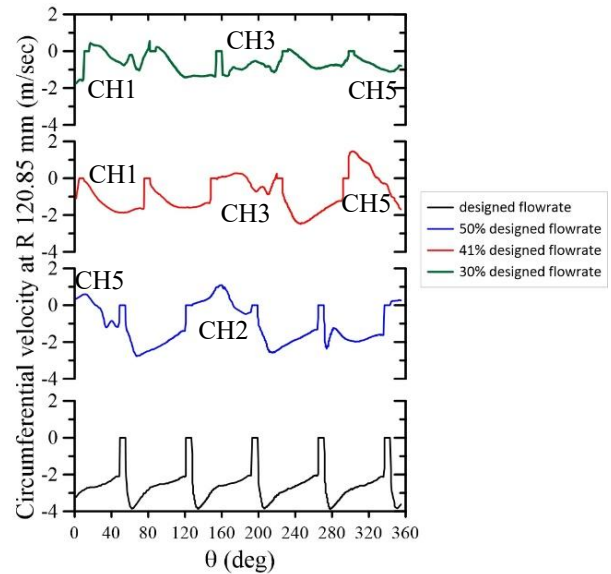
The circumferential velocity at the design condition increases toward the pressure side compared with the previous position, which enhances the Coriolis force (Fig. 11b). In contrast, the wave-like profiles observed at 50% Qd, especially in CH2, indicate the existence of two vortices. This behavior is also reflected in CH3 at 41% Qd, whereas a much stronger vortex is detected at the outlet of CH5.

At 30% Qd relative velocity decreases to about one quarter of its design value (Fig. 11c). The pressure-side magnitude increases at Qd compared to the previous monitoring location, whereas the suction-side relative velocity decreases. For stalled passages, however, the relative velocity is higher near the pressure side, which is the inverse of the design condition. Thus, the high-energy regions shift to the pressure side and split into sub-regions. This is the formal onset of the jet-wake phenomenon.

Generally, the pressure coefficient (Cp) increases as the flow moves toward the outlet (Fig. 11d). The pressure distribution at Qd is uniform for all passages, with stalling passages showing lower Cp trends than non-stalling passages, but similar peak-to-peak values. The pressure difference is more significant at 30% Qd. The pressure imbalance between neighboring passages induces cross-flow at the outlet, producing exit recirculation. Finally, near the outlet at R3, backflow and pressure imbalance between passages become evident, contributing to instability.



(a) Radial velocity component



(b) Tangential velocity component

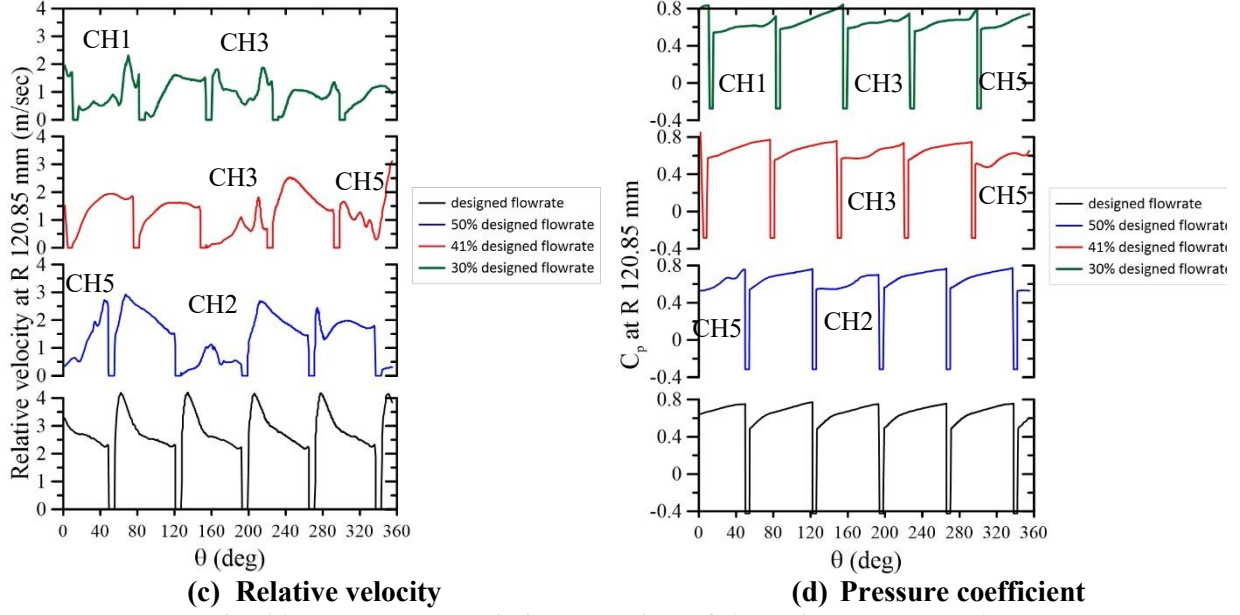


Fig. 11. Flow characteristics at radius R3 (near impeller outlet).

4.2.4. Flow behavior at R4 (Outlet)

At the impeller outlet (R4) under design conditions, the radial velocity profile (Fig. 12a) shows a significant increase, reaching magnitudes comparable to those observed at the suction side of the upstream radius R3. Conversely, most stalling cells at off-design conditions have negative radial velocity, indicating exit recirculation. Moreover, radial velocity fluctuates strongly between stalling and non-stalling passages, especially at 41% Qd and 30% Qd. Most importantly, recirculation occurs near the pressure side of the blade at the outlet. Moreover, the magnitude of radial velocity entering the impeller depends on the swirling that occurs on the upstream radius (R3).

The relative velocity distribution at the outlet shows a clear inversion of the conditions at the inlet (R1). In particular, the relative velocity near the suction side at R4 is smaller than that at the pressure side (Fig. 12c). The change in magnitude of velocity through the passage is mainly due to the dominant effect of centrifugal forces and the accumulation of high energy fluid on the pressure side as the flow moves toward the periphery of the impeller.

Also, the tangential velocity component (Fig. 12 b) and the pressure coefficient (Fig. 12 d) confirm the non-uniformity of the outlet flow, particularly under off-design conditions. The oscillations of the pressure coefficient in the affected channels during the stalling condition (50% Qd) indicate the loss of the steady pressure recovery which agrees well with the localized recirculation zones and the onset of the rotating stall patterns identified in previous sections.

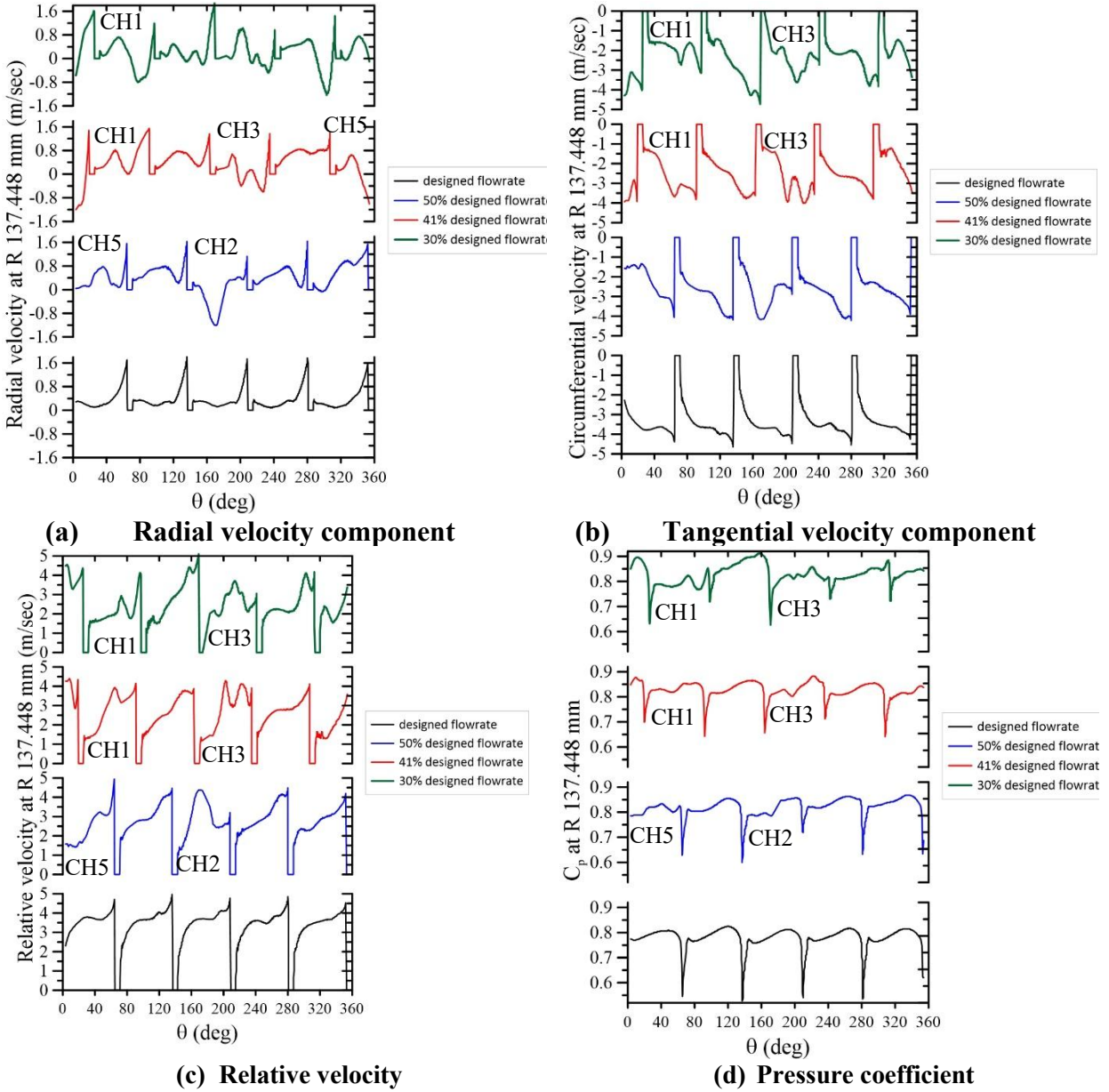


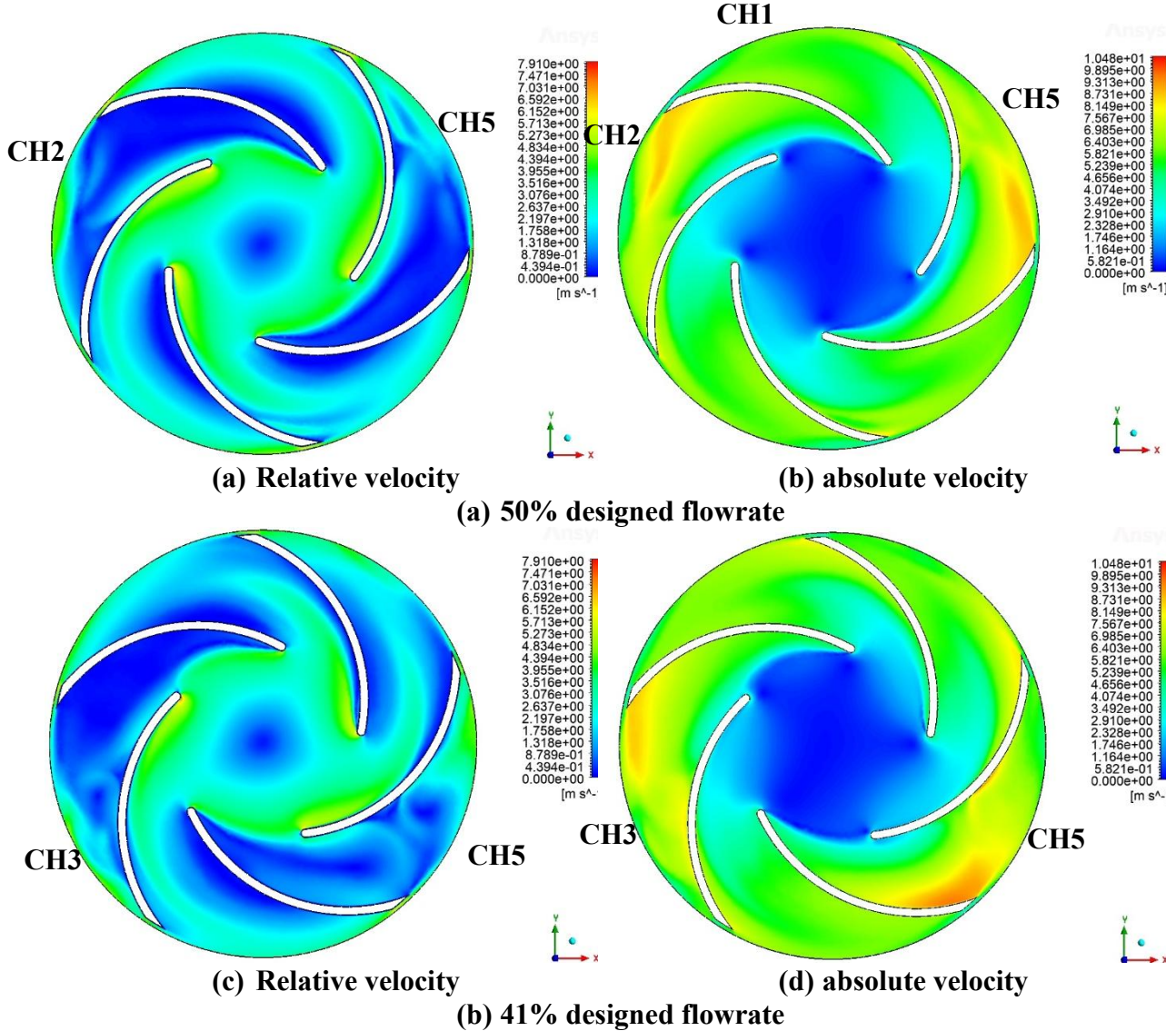
Fig. 12. Flow characteristics at radius R4 (impeller outlet).

In summary, flow behavior depends strongly on the inlet angle at the blade leading edge. Stalled cells disrupt the flow, reducing the effective area and increasing the radial velocity. The positive circumferential velocity indicates that inertia dominates the flow relative to the Coriolis forces. The flow separates from the pressure side because of downstream pressure conditions. This causes low energy conversion. This separation tends to shed existing vortices, but it also creates new vortices in adjacent passages, which eventually sustain rotating stall. The handling flowrate reduction starts with the early vortex. The pressure in the stall passages is not high enough to discharge at the impeller outlet, and exit recirculation becomes pronounced. Therefore, at the outlet of the impeller, non-uniform discharge and strong pressure fluctuations are observed, indicating hydraulic instability

4.3. Rotating Stall Mechanism

4.3.1 Kinematic characteristics of the impeller flow

The deterioration of velocity profiles at off-design conditions is evidenced by the relative velocity contours in Fig. 13. Contours of relative and absolute velocities at (a,b) 50%, (c,d) 41%, and (e,f) 30% of the design flow rate illustrate flow stagnation, blockage, and increasing vortex intensity. At 50% Qd the flow near the suction side of CH2 and CH5 is stagnant relative to the blades, rotating effectively at the impeller speed (Fig. 13a). Thus, these regions have high absolute velocities (Fig. 13b) as the absolute velocity vector is opposite to the stagnant relative velocity. The vortex intensity in CH5 increases when the flow rate is reduced to 30% and 41% Qd. The total area of high absolute-velocity peaks is highest at the lowest flow rate (30% Qd), suggesting widespread flow blockage.



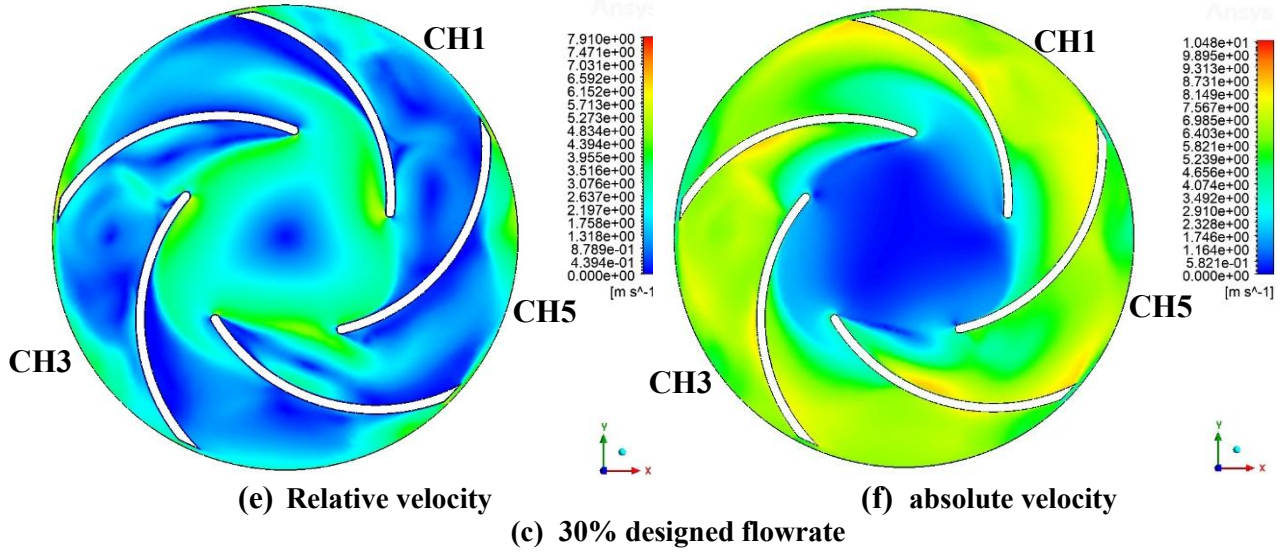
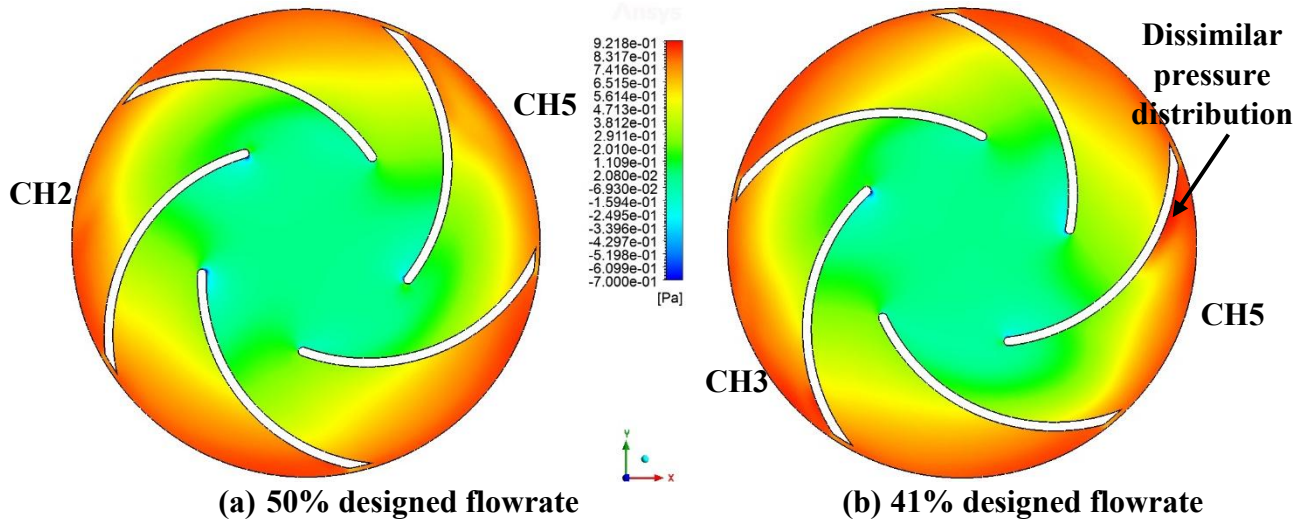


Fig. 13. Kinematic flow structures inside the impeller at different operating conditions.

4.3.2 Pressure recovery and exit recirculation

In order to compare pressures at off-design conditions (different blade loading), the pressure coefficient contour, calculated from $C_p = \frac{p - p_i}{0.5 \rho U_2^2}$, is presented in Fig. 14. Contours at different flow rates show the expansion of low-pressure regions in stalled passages and increasing pressure non-uniformity as flow rate decreases. Pressure coefficient (C_p) contours show poor energy conversion within the stalled cells. The stalling channels (CH2/CH5 at 50% Qd; CH3/CH5 at 41% Qd; and CH1/CH3/CH5 at 30% Qd) are dominated by low-pressure regions (in green). These channels have much smaller high-pressure (red) zones than non-stalling passages. At lower flow rates (41% and 30% Qd), the pressure distribution along the outlet radius is not as uniform. This unbalance leads to flow separation from the blade surfaces and the downstream pressure is higher than the source pressure, leading to exit recirculation.



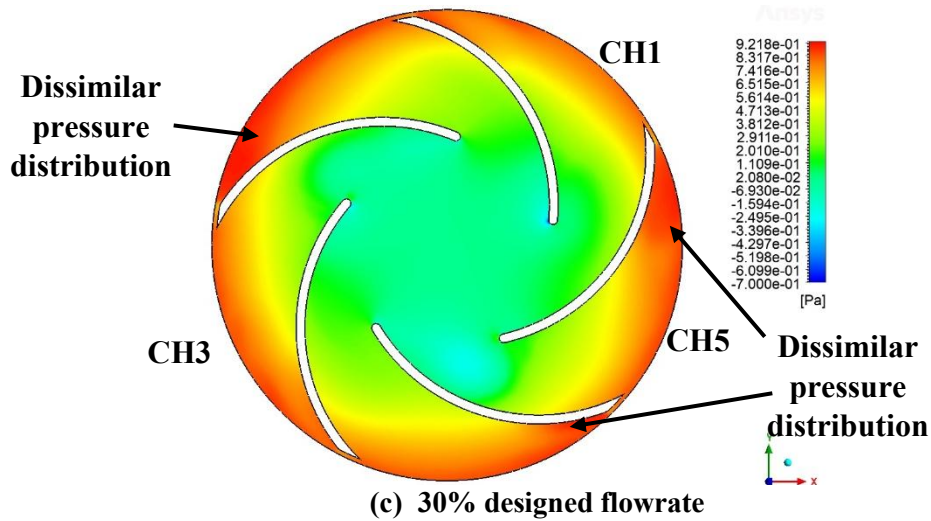
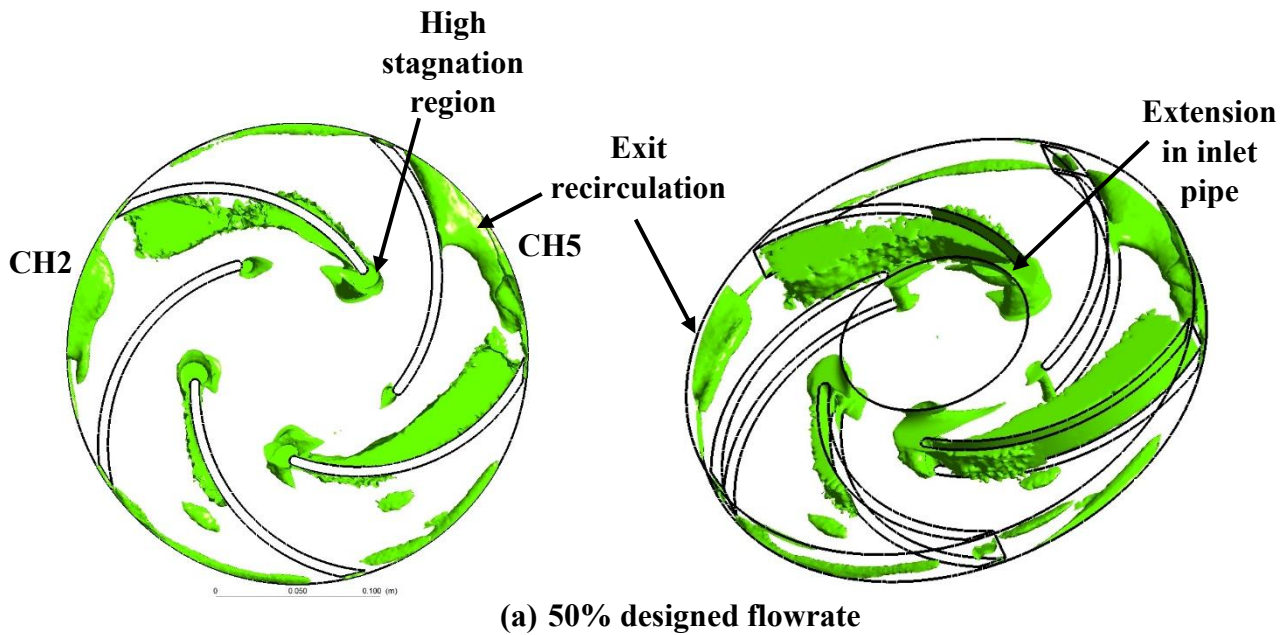


Fig 14. Pressure coefficient distribution within the impeller.

4.3.3 Iso-surface analysis and bending effects

Iso-surfaces at -0.01 m/s show stagnant flow regions, vortex cores, and upstream extent of recirculation zones in off-design conditions. Figure 15 shows stagnant zones that are roughly half the volume of the identified vortices. The stagnation points at the leading edges of non-stalling passages remain localized and centrally located. However, the stagnation zones are different over the blade height and even extend upstream into the inlet pipe due to the “bending effect” of the flow entering the impeller eye. Prominent on the side of the shroud. The exit recirculation is mainly observed near the pressure side of stalling passages and slightly increases near the hub side with further flow condition deviation from the design point.



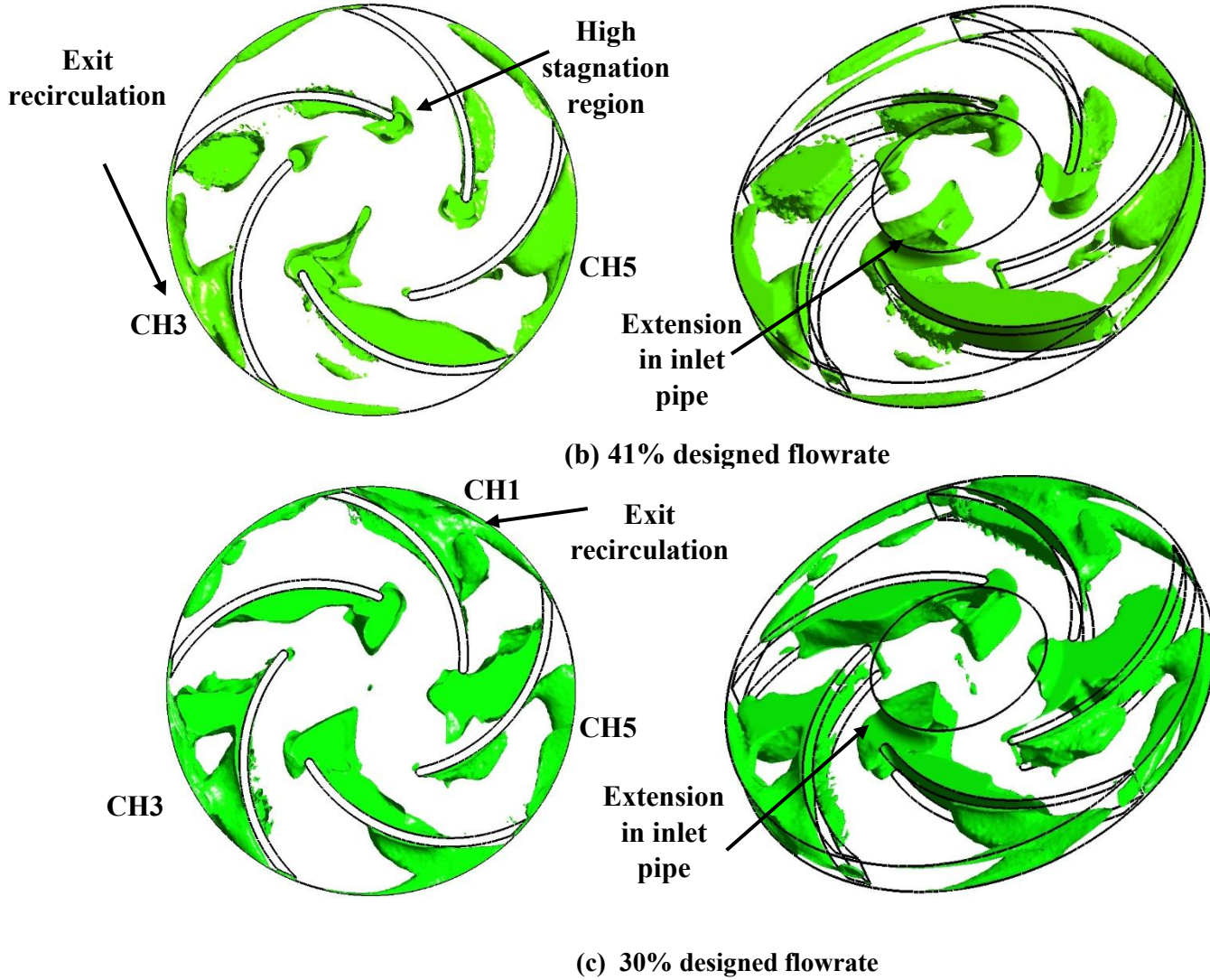
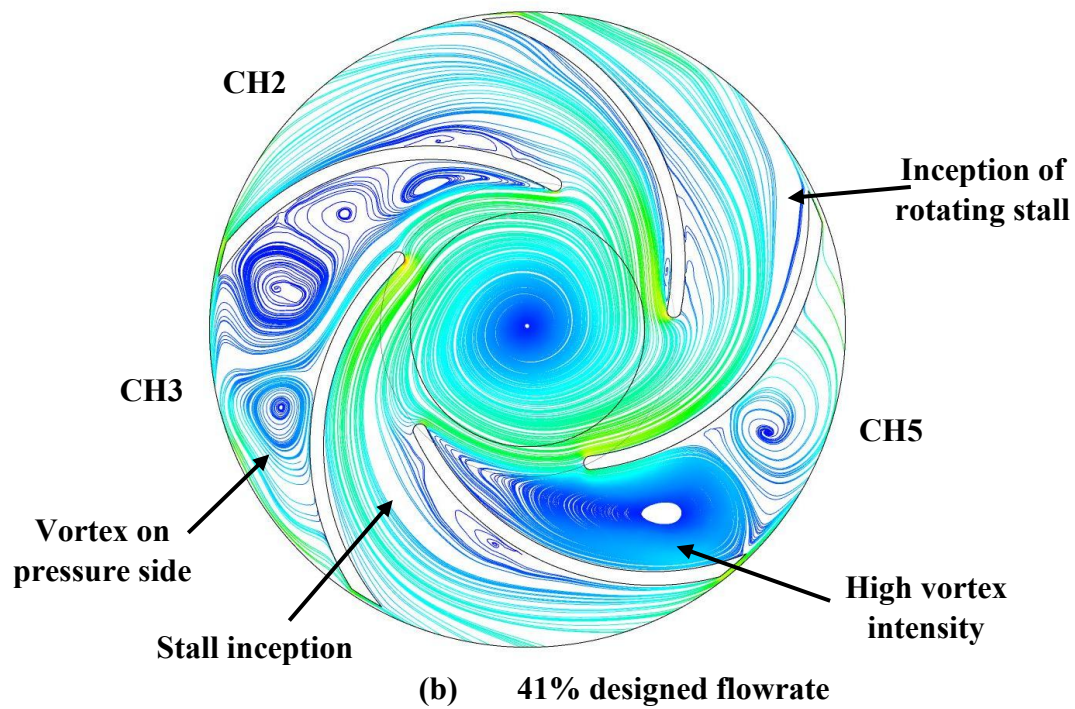
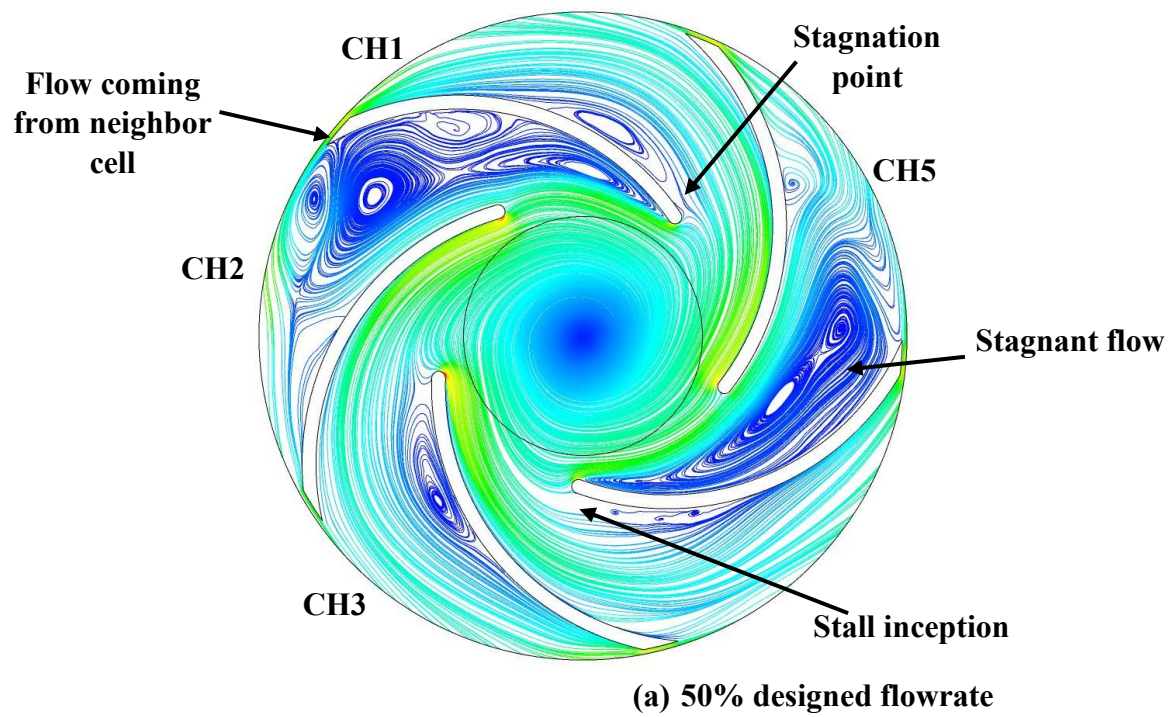


Fig. 15. Iso-surfaces of radial velocity highlighting stagnation and recirculation zones.

4.3.4 Streamline patterns and stall propagation

Figure 16 demonstrates Stall inception, vortex formation and interaction between adjacent passages for (a) 50%, (b) 41% and (c) 30% of the design flow rate. The Rothalpy equation indicates that at high velocity, non-stalling cells tend to have low suction-side pressure. At 50% Q_d , the flow angle drops below the blade angle, causing stagnation on the pressure side (stall inception) and separation on the other side of the blade. Separation at the leading edge of the pressure side (CH1) is caused by Coriolis forces (Fig. 16a) and dominates. However, the separation on the suction side of CH2 causes stall propagation by localized negative pressure.

As the flow rate reduces to 41% Q_d , the incidence angle mismatch worsens, increasing stagnation at the leading edge of CH2 and CH5 where high-intensity stalling vortices completely block the passage (Fig. 16b). Interestingly, a vortex forms on the pressure side of the trailing edge to compensate for the pressure reduction induced by the primary vorticity. At 30% Q_d , the vortex intensity increases while its physical size diminishes, following the conservation of angular momentum.



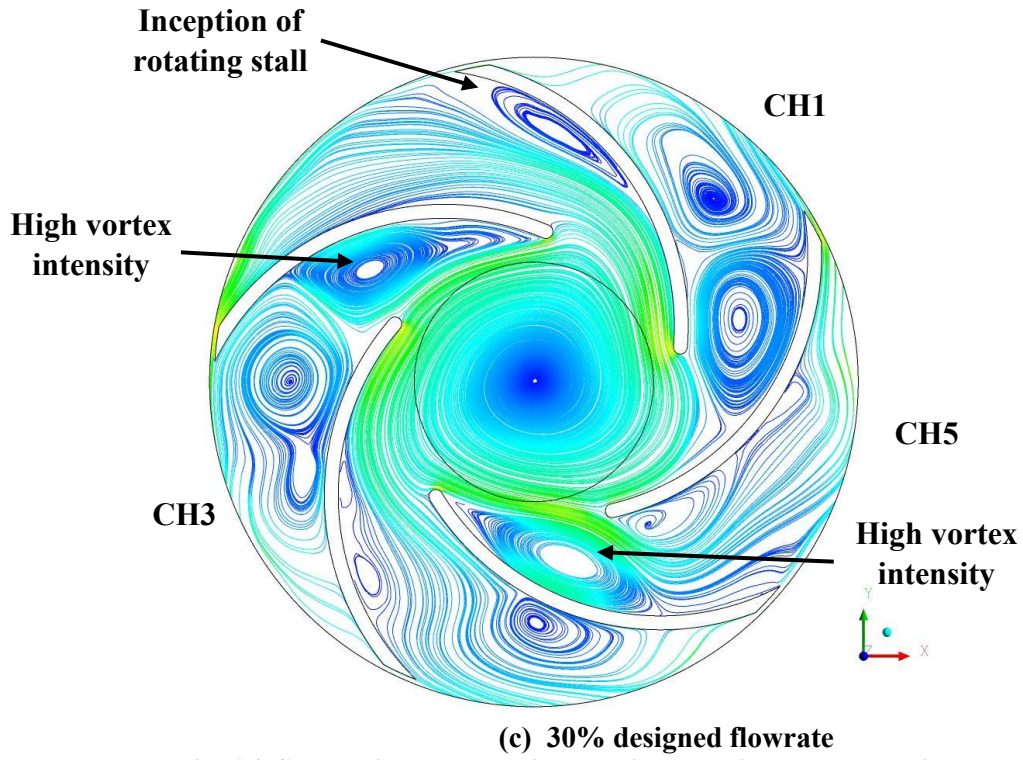
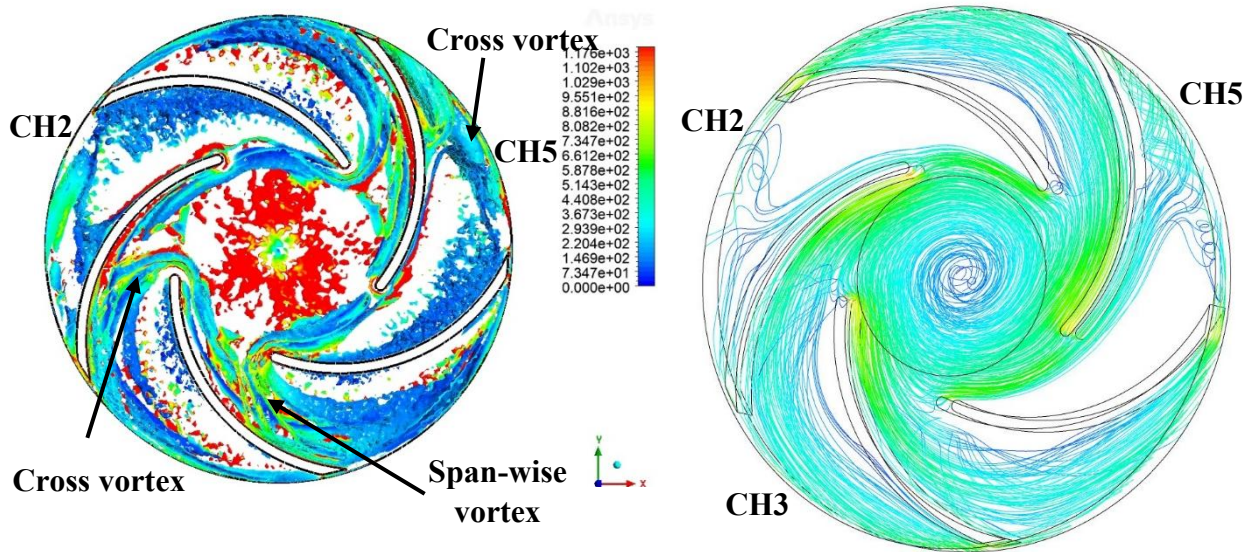


Fig. 16. Streamline patterns illustrating rotating stall evolution.

4.3.5 Vorticity and secondary flows

The spanwise and cross vortices can be illustrated from the Q criterion and 3D streamlines at various flow rates, as shown in Fig. 17. Vorticity distribution and three-dimensional streamlines showing the formation of cross-flow and spanwise vortices under part-load conditions. Installed cells (CH2, CH5 at 50% Qd) the streamlines are forced tangential to the surface of the blade at approximately mid-chord (Figs. 17b, d). Non-stalled channels are free of spanwise vortices, but contain secondary flows near the shroud-side leading edge due to non-uniform pressure distribution along the blade height (the “bend effect”). Velocity fluctuations and high vorticity are most concentrated near the shaft and on the suction side. Swirling intensity, and thus the total vorticity region, is highest at the minimum flow rate of 30% Qd (Fig. 17e).



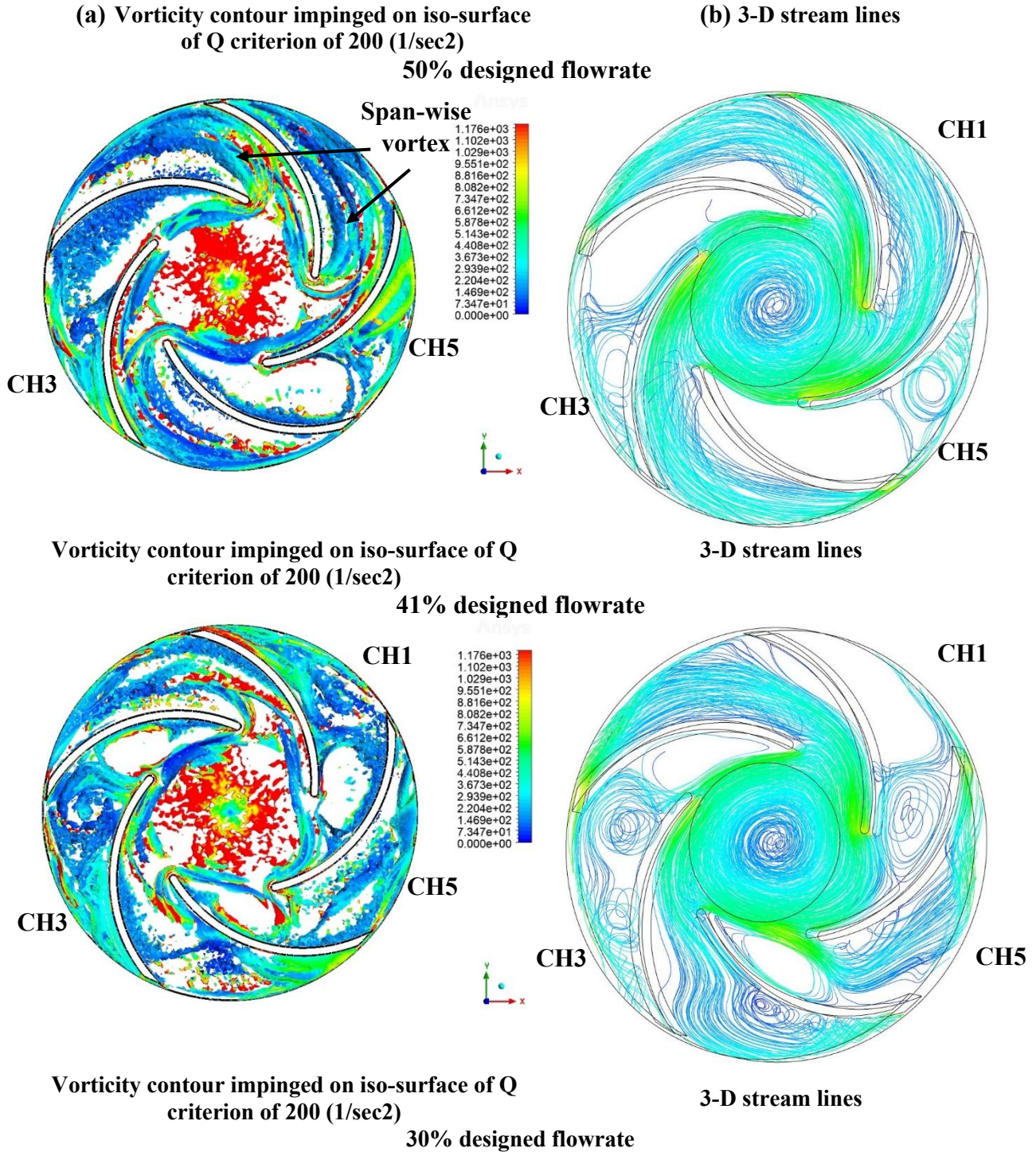


Fig. 17. Three-dimensional vortex structures and flow topology inside the impeller.

In conclusion, the results indicate that rotating stall develops through the following sequence:

1. Boundary-layer separation on the suction side
2. Formation of recirculation zones
3. Growth into stall cells
4. Interaction between adjacent passages
5. Development of outlet recirculation

A transitional regime is identified around 41% of the design flow rate, consistent with findings reported in [6,7].

5. CONCLUSIONS

This study investigated unsteady flow behavior in a centrifugal pump under off-design conditions using transient CFD simulations. The main conclusions are:

1. Flow instability and vortex intensity increase significantly as the operating point deviates from the design condition.
2. The stalling cells are characterized by negative whirl velocity at the inlet pipe.
3. The high relative velocity near the pressure side is characterized by the stalled cell, or the humps on it, which reflect the number of vortices inside the blade passage.
4. Rotating stall originates from suction-side boundary-layer separation.
5. Reverse flow at the impeller inlet is a major contributor to energy loss.
6. More enhancement at the middle of the impeller blade geometry will lead to a highly smooth energy transition
7. Stall cells intermittently block flow passages and redistribute flow.
8. Pressure fluctuations and asymmetric loading increase under part-load conditions.
9. A transitional regime exists around 41% of the design flow rate.
10. Stable operation requires maintaining conditions close to the design point.

Disclosure of interest

The authors declare that they have no personal relationships that could have appeared to influence the work reported in this paper.

Funding

The authors declare no funding.

Data Availability Statement (DAS)

All data generated or analyzed during this study are included in this published article and will be made available upon request from the corresponding author.

6. REFERENCES

- [1] Gülich, Johann Friedrich. Centrifugal Pumps. Springer, Berlin, 2010.
- [2] Lobanoff, Val S., and Robert R. Ross. Centrifugal Pumps: Design and Application, 2nd ed., Gulf Professional Publishing, 2013.
- [3] Japikse, David. Centrifugal Pump Design and Performance. Concepts ETI Inc., Vermont, 1994.
- [4] Brennen, Christopher E. Hydrodynamics of Pumps. Oxford University Press, 1994.
- [5] Emmons, Howard W., Charles E. Pearson, and Howard P. Grant. "Compressor Surge and Stall Propagation." Transactions of the ASME, Vol. 77, 1955, pp. 455–467.
- [6] Tan, Lei, Bo Zhu, Shuliang Cao, and Yun Wang. "Experimental Investigation of Pressure Fluctuations in Centrifugal Pumps under Off-Design Conditions." Journal of Fluids Engineering, 135 (2013): 011102.
- [7] Qu, Lin, Shouqi Yuan, and Yiping Tang. "Unsteady Flow Characteristics of Rotating Stall in a Centrifugal Pump at Partial Flow Conditions." Applied Thermal Engineering, 108 (2016): 108–117.

- [8] Ye, Chao, Dong An, Wei Huang, Yong Heng, and Yong Zheng. "Investigation on Stall Characteristics of Centrifugal Pumps with Guide Vanes." *Water*, 15 (2023): Article 21.
- [9] Fraser, William H., and Robert Milne. "Effects of Rotating Stall on Centrifugal Pump Performance." *Proceedings of the Institution of Mechanical Engineers*, 197 (1983): 123–130.
- [10] Miner, Steven M., Robert J. Beaudoin, and Ronald D. Flack. "Laser Velocimeter Measurements in a Centrifugal Flow Pump." 1989, pp. 205–212.
- [11] Sinha, Manish, and Joseph Katz. "Quantitative Visualization of the Flow in a Centrifugal Pump with Diffuser Vanes." *Journal of Fluids Engineering*, 122 (2000): 97–107.
- [12] Krause, Norbert, Klaus Zähringer, and Eberhard Pap. "Time-Resolved Particle Image Velocimetry Measurements in a Centrifugal Pump." *Experiments in Fluids*, 39 (2005): 192–201.
- [13] Liu, Hong, Ming Tan, and Cheng Wang. "Investigation of Internal Flow Instability in Centrifugal Pumps under Low-Flow Conditions." *Energy Conversion and Management*, 136 (2017): 326–338.
- [14] Zhao, Wei, Ming Wang, Bin Liang, Lei Zhao, and Qiang Liu. "Transient Pressure Fluctuation Characteristics in Centrifugal Pumps Using Delayed Detached-Eddy Simulation." *Intelligent Marine Technology and Systems*, 2024.
- [15] Zhang, Ning, Dong Li, Bo Gao, Dong Ni, and Zhaoxu Li. "Unsteady Pressure Pulsations in Pumps: A Review." *Energies*, 16 (2023): Article 150.
- [16] Liu, Yong, Jian Yang, Ming Fang, Xin Li, Yong Wu, and Yong Lu. "Computational Research on Rotating Stall Mechanisms in Pump-Turbine Systems." *Water*, 17 (2025): Article 2538.
- [17] Zhou, Xiaofeng, Xiaolong Chen, Dong Yu, Yifan Li, and Xia Chen. "Predicting Alternating Stall in Centrifugal Pump Impellers under Transient Conditions." *Processes*, 12 (2024): Article 2635.
- [18] Liu, Hong, Yu Zhang, Xia Chen, Jian Li, and Shun Wang. "Pressure Fluctuation and Rotating Stall Characteristics in Pump-Turbines under Pump Mode." *Journal of Energy Storage*, 94 (2024): Article 112385.
- [19] Li, Zhi, Qiang Zhao, Y. Chen, X. Wang, and L. Zhang. "Mechanical Characteristics of Centrifugal Pump Rotors Based on Fluid–Structure Interaction." *Water*, 18 (2026): Article 229.
- [20] Feng, Jian, Zhen Ge, Hong Yang, Guohua Zhu, Chao Li, and Xiaobing Luo. "Rotating Stall Characteristics in the Vaned Diffuser of a Centrifugal Pump." *Ocean Engineering*, 229 (2021): Article 108955.

