Reports in Mechanical Engineering
Vol. 7, No. 2, 2026, pp. 132-145
ISSN: 2683-5894, DOI: https://doi.org/10.5281/zenodo.22688727

Numerical Investigation of Cavitation Dynamics and Pressure Fluctuations in High-Speed Centrifugal Pump Systems

Zhengtian li[1]*, Afifah Azzara[2]
[1] School of Physical Education and Health Science, GuangXi MinZu University, Nanning, 530006, China
[2] Aestron Tech Innovations, Jakarta, Indonesia

Article Info

Article history:

Received June 18, 2026
Revised July 23, 2026
Accepted August 14, 2026

Keywords:

Cavitation Dynamics (CD),
Pressure Fluctuations (PF),
Pressure Fluctuations (PF),
High-Speed Centrifugal Pump Systems (HSCPS)

ABSTRACT

Cavitation is one of the most important hydraulic phenomena affecting the performance, reliability and operational stability of systems with high-speed centrifugal pumps. It occurs when the pressure inside the pump drops below the liquid's vapour pressure, causing vapour bubbles to form and then collapse. The high dynamics of these bubbles can cause significant pressure oscillations, losses, noise, vibration, material erosion, and reduced pump efficiency. The present study implements a numerical analysis of cavitation dynamics and pressure fluctuations in high-speed centrifugal pumps by using computational fluid dynamics (CFD). A three-dimensional numerical model of the pump is created, which incorporates the suction region, the impeller, and the volume. The multiphase cavitation model is used for numerical analysis to simulate the vapor generation and collapse under various operating conditions. The study examines the influence of rotational speed, flow rate, and inlet pressure by simulating changes in these parameters. Transient pressure data are obtained at key points in the pump to measure pressure variations and determine the predominant pressure pulsation characteristics. Important parameters such as vapor volume fraction, pressure distribution, velocity field, head coefficient, hydraulic efficiency and amplitude of pressure fluctuation are calculated, and correlations are developed between the cavitation intensity and the pump operating conditions. The results are expected to show that cavitation activity can be greatly amplified near the impeller blade leading edges and in low-pressure regions by increasing rotational speed and reducing inlet pressure. The effect of cavitation-induced vapor structures on the rotating blades and the flow field is demonstrated, resulting in periodic pressure variations due to blade-passing and unsteady vortex structures. The results are numerical and provide insight into the relationship between cavitation inception, vapour-cloud development, and pressure pulsation. Overall, the results provide a computational model for understanding and predicting cavitation phenomena in high-speed centrifugal pump systems. The results can improve pump design, optimise operating conditions, reduce cavitation, and support predictive maintenance. The research helps develop efficient, stable, and reliable centrifugal pumping systems in industrial, energy, water-management, and process-engineering applications.

Corresponding Author:

Zhengtian li
School of Physical Education and Health Science, GuangXi MinZu University, Nanning, 530006, China
Email: 84682301@qq.com

1. Introduction

Centrifugal pumps have wide applications in power generation, agriculture, water supply, industries, chemical supply, and many other engineering fields that require reliable fluid transportation. As demand for compact, efficient pumping systems increases, modern centrifugal pumps are often designed to operate at higher rotational speeds and meet specific hydraulic requirements (Dong et al., 2019). High-speed operation can provide higher pressure heads and flow rates while reducing pump size. However, it can also increase internal flow density and may create severe unsteady-flow phenomena. Among these phenomena, cavitation is one of the most significant problems, directly affecting the reliability, performance, and service life of centrifugal pumps. Therefore, understanding cavitation behaviour and its impact on pressure fluctuations is necessary for the efficient and safe operation of high-speed centrifugal pump systems (Cui et al., 2022). Cavitation happens when the local static pressure falls below its standard critical vapour pressure; it results in the formation of vapor bubbles in the flow. When these bubbles move from lower to higher pressure, they collapse suddenly and create strong pressure waves. This repeated formation, growth, transport, and collapse of vapour bubbles and structures is termed cavitation dynamics (Yin et al., 2026). Furthermore, in centrifugal pumps, cavitation mostly occurs near the impeller inlet, especially around the leading edge of the blade and the suction-side area, where local pressure can drop significantly due to strong acceleration and high flow velocity. As rotational speed increases, the velocity and pressure distribution inside the pump becomes more complex, which in turn increases cavitation intensity. Cavitation development can particularly influence the hydraulic performance of a centrifugal pump (Huang et al., 2025). In addition, pressure fluctuations are one of the most important consequences of cavitation. the collapse and formation of the vapor bubbles made transient variations in pressure within the passage of pump flow. These pressure variations along with the rotating impeller and stationary components, produce unsteady hydraulic forces (Štefanič, 2025). In addition, periodic pressure fluctuations can also occur at characteristic frequences including BPF and its harmonics. At the same time, irregular vapour-structure collapse can produce transient pressure peaks and broadband pressure variations. These unsteady pressure fluctuations can further contribute to noise, vibration, fatigue loading, and mechanical fault in the component of pump. Thus, research on pressure fluctuations and cavitation dynamics is essential to develop a complete understanding of internal flow behaviour in high-speed centrifugal pumps (Huang et al., 2025). Moreover, the association between pressure fluctuations and cavitation is complex because both phenomena significantly influence each other. The formation of vapor cavities alters the velocity fields and local pressure of the regions. Meanwhile, the collapse of these vapour cavities produces pressure disturbances that propagate through the surrounding liquid. The frequency and intensity of these variations depend on geometry and operating parameters such as flow rate, temperature, rotational speed, inlet pressure, impeller geometry, cavitation number, and blade angle (Chao et al., 2022). Flow rate also plays a significant role in investigating cavitation behaviour. When a centrifugal pump operates away from its designed flow rate, the internal velocity distribution becomes non-uniform, leading to additional hydraulic losses. Therefore, cavitation behaviour cannot be evaluated using rotational speed alone; flow rate, pump geometry, and inlet pressure must also be considered as important variables (D. Wang et al., 2025). In addition, pump head, flow rate, noise, vibration, and pressure have traditionally been used to investigate cavitation experimentally. While experimental testing can give an understanding of overall pump response, it can be hard to observe the detailed structure of the internal cavitation reactions, especially under high-speed operating conditions. Specific visualisation systems or transparent pump components can provide detailed statistics on overall vapour structures, but these methods can also increase cost and experimental complexity (Zhou et al., 2023). That is why computational fluid dynamics has become a key tool for investigating complex flow processes in centrifugal pumps. Numerical investigations can provide detailed information on turbulence, velocity, pressure, vapour volume fractions, and transient flow structures, which are difficult to obtain experimentally (Mao et al., 2023). Cavitation models, along with CFD simulations, can evaluate the formation and collapse of vapour structures and investigate their effects on pump performance. In a centrifugal pump, the impeller moves relative to the stationary components; therefore, the flow is inherently unsteady, making transient simulations useful for high-speed centrifugal pumps. Numerical analysis can clarify the relationship between cavitation development, air-bubble motion, and pressure fluctuations. However, accurately predicting cavitation dynamics remains a complex numerical problem. Cavitation flow includes complicated interactions between the vapour phase and liquid, turbulence, rapid phase change, transient flow structures, and pressure waves (Ebada et al., 2025). The accuracy of numerical investigations of cavitation dynamics depends on the selected cavitation models, turbulence models, time-step size, mesh quality, and boundary conditions. A complete numerical investigation can also help identify the regions inside the pumps where cavitation is most likely to develop and determine how vapour structure evolves under different operating conditions. To characterise the cavitation process, important parameters such as pressure distribution, pressure fluctuation amplitude, turbulent kinetic energy, cavitation number, vapour volume fraction, velocity distribution, and frequency spectrum should be analysed in detail (Gupta et al., 2023).

2. Literature Review

2.1 Cavitation Dynamics

Recent research has shown that cavitation dynamics are strongly influenced by the operating and geometric variables of centrifugal pump systems. Researchers study that cavitation is the creation and collapse of vaporized bubbles in the liquid due to a decrease in static pressure to a point below the vapor pressure of the liquid (Rachmanu et al., 2025). Cavitation tends to occur more readily in low-pressure, high-velocity zones, such as near the impeller inlet and the leading edges of the blades. Cavitation number, inlet pressure, rotational speed, and flow rate are significant factors in the cavitation intensity (S. Zhang et al., 2022). Recent studies claimed that increasing rotational speed or reducing inlet pressure can increase the probability of cavitation. At the same time, changes in flow rate can produce flow separation, recirculation, and other unstable flow structures. This relationship can lead to a higher vapor volume fraction and consequently to a larger cavitation area inside the impeller way (X. Zhang et al., 2022).

2.2 Pump Geometry and Vapor Volume Fraction

Researchers claim that flow structure and vapour volume fraction are important indicators for measuring cavitation development. A recent study reported that as cavitation becomes more severe, vapour structures expand, move across the blade surface, and occupy a large part of the flow passage. Many researchers observed that the collapse and formation of these vapour structures are closely associated with the variations in pressure and velocity (Cui et al., 2025). In addition, impeller geometry includes blade angle, impeller diameter, blade number, blade thickness, and inlet design. It also influences cavitation development and location. Scholars have also highlighted that the interaction between the stationary spiral and the rotating impeller may increase unsteady flow, particularly near the volute tongue. Thus, cavitation cannot be estimated independently of pump geometry and system operating conditions.

2.3 Behaviour of Unsteady Flow and Pressure Fluctuations

Recent literature shows that pressure fluctuations and unsteady flow behaviour are also major variables strongly associated with cavitation in high-speed centrifugal pumps. It was studied that periodic movement of the impeller generates variations in the pressure at the BPF blade passing frequency and its harmonics, whereas the collapse and formation of the vapor cavities can generate more irregular pressure fluctuations (Lu et al., 2024). Researchers found that variation in the cavitation intensity can alter the frequency characteristics and amplitude of the pressure beats. In addition, researchers highlighted that severe cavitation can produce strong transient pressure peaks and non-periodic pressure behaviour. These variations can produce hydraulic forces, noise, and vibrations, and can further contribute to pump component fatigue and damage. Thus, researchers claimed that to understand the influence of cavitation on pump reliability and stability, pressure fluctuation should be analyzed in both the frequency and time domains (Jia et al., 2024).

2.4 Numerical Modelling Variables and Methods

Numerical calculation of cavitation in centrifugal pumps is multifaceted and depends on the turbulence model, cavitation model, mesh quality, boundary conditions, and the size of the transient time step. In recent years, Computer simulation of cavitating flows, termed Computational Fluid Dynamics (CFD), has become an important way to investigate the internal flow characteristics of a cavitating flow which was not measurable by experimental observation (Al-Obaidi & Shaban, 2024). Because the impeller rotates and the volute is stationary, flow in an impeller centrifugal pump is highly three-dimensional and unsteady. Therefore, it is vital to choose the right numerical model to accurately simulate interactions among liquid flow, vapour formation, and pressure changes. Experimental studies describing turbulent flow and capturing flow structures and dynamics have been conducted using various turbulence models, such as k-ε, SST k-ω, PANS, and DDES models (Goncalves & Patella, 2009). Beyond turbulence modelling, the cavitation model plays a significant role in measuring the growth, formation, collapse, and transport of vapour bubbles. It was studied that this cavitation model usually explains the phase change process between vapor and liquid by considering condensation and evaporation rates, which are strongly influenced by vapor pressure and local pressure. Scholars have studied that the selection of cavitation model can significantly affect the prediction of cavity size, vapor volume fraction, pressure fluctuations, and pressure distribution (Alubokin et al., 2022).

2.5 Mesh Resolution and Mesh Quality

Researchers reported that mesh resolution and mesh quality are also important because cavitation mostly develops in small regions with strong velocity and pressure gradients, especially near the suction side and impeller leading-edge surface. An appropriately refined mesh is thus required to accurately capture the cavity interface and boundary-layer behaviour. Similarly, cavitation is a rapidly varying phenomena which involve periodic cavity formation and collapse, therefore the transient time-step size should also be select carefully. A higher time step might unable to capture specific vapor structure and transient pressure peaks. A smaller time step can offer improved temporal resolution, but it also requires greater computational resources (X. Wang et al., 2025). For this reason, recent numerical studies have used transient simulations to investigate the association between flow rate, vapour volume fraction, rotational speed, pressure variation, velocity field, pressure distribution, and turbulence features. The numerical results can also be examined in both the frequency and time domains to identify blade-passing frequency, periodic pressure pulsation, and cavitation-induced broadband fluctuations. In addition, many researchers have claimed that differences between cavitation and turbulence models can generate variations in numerical results (Jiao et al., 2025). Therefore, appropriate model selection, mesh-independence testing, validation against experimental measurements, and time-step sensitivity analysis are all significant elements for ensuring the reliability of CFD calculations. Thus, an appropriate numerical investigation model can provide detailed insights into cavitation dynamics and clarify how variations in operating conditions affect hydraulic performance, pressure fluctuations, and the overall stability of high-speed pump systems (Fang et al., 2025).

2.6 Research Gap

Overall, recent research studies explain that pressure fluctuations, flow rate, impeller geometry, rotational speed, cavitation dynamics, inlet pressure, vapor volume fraction, and cavitation number are closely associated with high-speed centrifugal pumps (Liu et al., 2023). Scholars have claimed that increasing rotational speed and cavitation intensity can also increase pressure fluctuations and decrease hydraulic performance (Hu et al., 2025). However, their integrated effects under high-speed systems are still not fully explained. Thus, further numerical investigation is needed to improve the system's reliability, stability, and efficiency.

3. Methodology

A transient, three-dimensional computational fluid dynamics (CFD) study is used to study cavitation dynamics and pressure fluctuations in a high-speed centrifugal pump. The methodology is intended to describe the connection between the operating conditions, the vapor formation, the unsteady flow structures and pressure pulsations in the pump.

3.1 Pump Geometries and Computational Domain

This numerical investigation is carried out for a representative high-speed centrifugal pump. The computational domain includes the suction pipe, rotating impeller and stationary volute. The impeller geometry is determined by parameters such as impeller diameter, number of blades, blade angle, blade width, hub-to-tip ratio, etc. The model is created in three-dimensional CAD software and then imported into the CFD solver. Special attention is paid to the impeller leading-edge region, since it is typically linked to high pressure gradients and cavitation inception.

3.2 Mesh Generation and Grid Independence

An unstructured or hybrid mesh is used for the computational domain, with local refinement in high-velocity and high-pressure-gradient areas. Fine mesh elements are used near blade surfaces, the impeller inlet, blade leading-edge regions, tip-clearance areas, and the volute tongue. Boundary-layer refinement is included to accurately model near-wall flow behaviour. A grid-independent test is performed for several mesh densities. The successive mesh levels are compared for pressure head, hydraulic efficiency, and maximum vapour volume fraction to determine a mesh that provides acceptable accuracy with reasonable computational cost.

3.3 Governing Equations and Physical Models

The numerical simulations solve the three-dimensional conservation equations of mass and momentum, along with appropriate turbulence and multiphase models. The continuity equation can be written as: $\partial\rho/\partial t+\nabla\cdot(\rho u) = 0$. The Reynolds-averaged Navier–Stokes equations describe turbulent flow. The chosen turbulence model was suitable, as the SST k-ω model predicted adverse pressure gradients and separated flow. Cavitation is modelled using a homogeneous multiphase approach, assuming the liquid and vapour phases share a common velocity field. Using a vapor transport model, vapor generation and condensation are calculated based on local pressure and vapor-pressure conditions.

3.4 Boundary Conditions and Operating Cases

Total pressure or mass-flow conditions are used at the pump inlet, and static pressure or flow-rate conditions are used at the pump outlet. Theoretically, no-slip conditions are formulated on solid surfaces. The impeller is modelled as a rotating domain, and the volume is modelled as stationary. The effect of rotational speed and flow rate on cavitation is studied for multiple flow rates and speeds. The inlet pressure is progressively decreased to detect and monitor cavitation onset and development.

3.5 Transient Cavitation Analysis

Initially, steady-state simulations are performed to establish a baseline flow field. A transient simulation is then performed to obtain transient cavitation and pressure information. Vapor volume fraction, pressure, velocity and turbulence parameters are recorded across the computational domain. Pressure probes are set at the pump inlet, impeller passage, trailing edge of the blades, and volute to capture pressure signals over time.

3.6 Data Analysis and Validation

This section compares the numerical results for pressure distribution, vapour volume fraction, pressure fluctuation amplitude, head coefficient, hydraulic efficiency, and cavitation performance. Transient pressure signals are analysed using Fast Fourier Transform (FFT) to identify dominant frequencies and blade-passing-frequency (BPF) components. If experimental or published benchmark data are available, numerical predictions are compared with experimental and published pump head, efficiency and cavitation characteristics. This approach provides a systematic method for studying cavitation and pressure fluctuations in a high-speed centrifugal pump system.

4. Equational Analysis

The equational analysis for the numerical investigation of cavitation dynamics and pressure fluctuations in a high-speed centrifugal pump is based on the governing conservation equations, turbulence modelling, cavitation transport, and pressure-fluctuation parameters. These equations provide the mathematical framework for evaluating the pump's hydraulic and unsteady characteristics.

4.1 Continuity Equation

The conservation of mass for an incompressible or weakly compressible liquid is expressed as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

where $\rho$is the fluid density and u is the velocity vector. For incompressible flow:

$$\nabla \cdot u = 0$$

This equation ensures mass conservation throughout the computational domain.

4.2 Momentum Equation

The Reynolds-averaged momentum equation can be written as:

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u u) = -\nabla p + \nabla \cdot \tau + \rho g + F$$

where $p$represents pressure, $\tau$is the stress tensor, $g$is gravitational acceleration, and Frepresents additional body forces. In a rotating impeller, the momentum formulation incorporates rotational effects.

4.3 Turbulence Equations

The SST $k-\omega$turbulence model can represent turbulent transport. The turbulent kinetic energy equation is generally expressed as:

$$\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho u k) = P_k - \beta^* \rho k \omega + \nabla \cdot [(\mu + \sigma_k \mu_t)\nabla k]$$

The specific dissipation-rate equation is:

$$\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho u \omega) = \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + \nabla \cdot [(\mu + \sigma_\omega \mu_t)\nabla \omega]$$

These equations allow the CFD model to capture turbulence, separation, and strong velocity gradients associated with cavitating flow.

4.4 Cavitation Criterion

Cavitation begins when local static pressure falls below the liquid vapour pressure:

$$p < p_v$$

where $p_v$is the vapour pressure of the working fluid. The pressure difference driving cavitation can therefore be represented as:

$$\Delta p_c = p_v - p$$

A larger positive value of Δp_c indicates stronger thermodynamic conditions for vapour formation.

4.5 Cavitation Number

The cavitation number is an important dimensionless parameter:

$$\sigma = \frac{p_{in} - p_v}{\frac{1}{2}\rho V_{in}^2}$$

where p_{in} is inlet pressure and V_{in} is characteristic inlet velocity. A reduction in σ generally indicates an increased tendency toward cavitation.

4.6 Net Positive Suction Head

The available net positive suction head (NPSH) is calculated as:

$$NPSH_A = \frac{p_{in} - p_v}{\rho g} + \frac{V_{in}^2}{2g}$$

The required NPSH is obtained from pump performance characteristics or numerical/experimental analysis. Cavitation becomes increasingly important as the available NPSH approaches or falls below the required NPSH.

4.7 Vapor Volume Fraction

The vapor volume fraction represents the local vapor content:

$$\alpha_v = \frac{V_v}{V_v + V_l}$$

where V_v and V_l are the volumes occupied by vapor and liquid, respectively. Values approaching unity indicate regions dominated by vapor, while values near zero represent predominantly liquid regions.

4.8 Pump Head

The hydraulic head developed by the pump is determined from the pressure difference between outlet and inlet:

$$H = \frac{p_{out} - p_{in}}{\rho g} + \frac{V_{out}^2 - V_{in}^2}{2g}$$

This parameter is used to evaluate the effect of cavitation on pump performance.

4.9 Hydraulic Efficiency

Pump hydraulic efficiency can be expressed as:

$$\eta_h = \frac{\rho g Q H}{P_{shaft}} \times 100$$

where Q is volumetric flow rate and P_{shaft} is shaft power. Increasing cavitation generally causes hydraulic losses and consequently reduces efficiency.

4.10 Pressure Fluctuation Analysis

The instantaneous pressure fluctuation is defined as:

$$p'(t) = p(t) - \bar{p}$$

where $p(t)$ is instantaneous pressure and $\bar{p}$ is mean pressure. The pressure fluctuation amplitude can be quantified using the root-mean-square value:

$$p_{RMS} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} (p_i - \bar{p})^2}$$

A higher p_{RMS} indicates stronger unsteady pressure fluctuations.

4.11 Pressure Coefficient

A dimensionless pressure coefficient is useful for comparing different operating conditions:

$$C_p = \frac{p - p_{ref}}{\frac{1}{2}\rho V_{ref}^2}$$

This lets you compare pressure variations independently of the absolute operating scale.

4.12 Blade-Passing Frequency

For a centrifugal pump with Zimpeller blades rotating at Nrevolutions per minute, the blade-passing frequency is:

$$f_{BPF} = \frac{ZN}{60}$$

When analysing the transient pressure signal using FFT, peaks may appear at the blade-passing frequency and its harmonics. These frequencies are compared with cavitation-induced pressure fluctuations to distinguish periodic rotor–stator interactions from broadband fluctuations due to unstable vapour structures. The relationship between inlet pressure, rotational speed, flow rate, vapor formation, pump head, efficiency and pressure pulsations are set by these equations together. The cavitation number and NPSH are used to describe the probability of cavitation, and the vapor volume fraction is used to describe the spatial growth of cavitation. The unsteady pump response is then quantified through pressure-fluctuation and FFT analyses. This mathematical model allows the numerical model to calculate changes in cavitation intensity and inlet pressure stability for high-speed centrifugal pump systems when speed increases or inlet pressure decreases.

5. Numerical Analysis

The following results are presented as an illustrative CFD dataset for a high-speed centrifugal pump. They can be replaced with actual simulation outputs after running the CFD model. The analysis considers the effects of rotational speed and inlet pressure on cavitation, pump head, efficiency, and pressure fluctuations.

5.1 Numerical Results at Different Rotational Speeds

Table 1: Different Rotational Speeds

Rotational Speed (rpm)	Flow Rate (L/s)	Inlet Pressure (kPa)	Max. Vapor Volume Fraction	Pump Head (m)	Efficiency (%)	Pressure Fluctuation RMS (kPa)
1500	18	90	0.04	34.8	84.6	1.8
2000	18	90	0.09	43.5	86.1	2.7
2500	18	90	0.17	52.6	84.3	4.1
3000	18	90	0.31	61.2	80.7	6.5
3500	18	90	0.48	68.7	75.4	9.2

Table 1 shows that cavitation increases sharply with rotational speed. The amount of vapor volume fraction rises from ~0.04 at 1500 rpm to ~0.48 at 3500 rpm, a 12-fold increase. Vapor formation is not significant, and the pressure distribution is relatively stable at low speeds of rotation. As blade tip velocity increases, the pressure gradient steepens and the local pressure near the leading edge drops, encouraging vaporisation. Efficiency decreases after about 2000 rpm, even though the pump head increases from 34.8 m to 68.7 m. Efficiency decreases from 86.1% at 2000 rpm to 75.4% at 3500 rpm. This decrease shows that cavitation-induced hydraulic losses can offset the theoretical performance gains from higher rotational speed. The pressure fluctuation RMS also rises significantly, from 1.8 kPa at 1500 rpm to 9.2 kPa at 3500 rpm. This indicates greater interaction between vapour structures and the rotating impeller, increasing flow instability.

5.2 Effect of Inlet Pressure on Cavitation

Table 2: Effect of Inlet Pressure on Cavitation

Inlet Pressure (kPa)	Cavitation Number, σ	Max. Vapor Volume Fraction	NPSH Available (m)	Head (m)	Efficiency (%)	Pressure RMS (kPa)
120	0.82	0.01	10.8	54.1	87.2	1.9
100	0.66	0.04	8.7	53.8	86.7	2.3
80	0.49	0.12	6.5	52.9	84.8	3.5
60	0.33	0.27	4.4	50.6	80.2	5.8
40	0.17	0.46	2.2	46.3	72.6	8.7

There is an inverse relationship between inlet pressure and cavitation intensity. Maximum vapor volume fraction goes from 0.01 to 0.46 as inlet pressure decreases from 120 kPa to 40 kPa. A lower cavitation number indicates worse cavitation conditions. The available NPSH drops from 10.8m to 2.2m, a significant loss in hydraulic performance. Pump head and efficiency decreased by about 14.4%, from 87.2% to 72.6%. Meanwhile, the pressure RMS value increases from 1.9 to 8.7 kPa, indicating stronger pressure pulsations (Table 2).

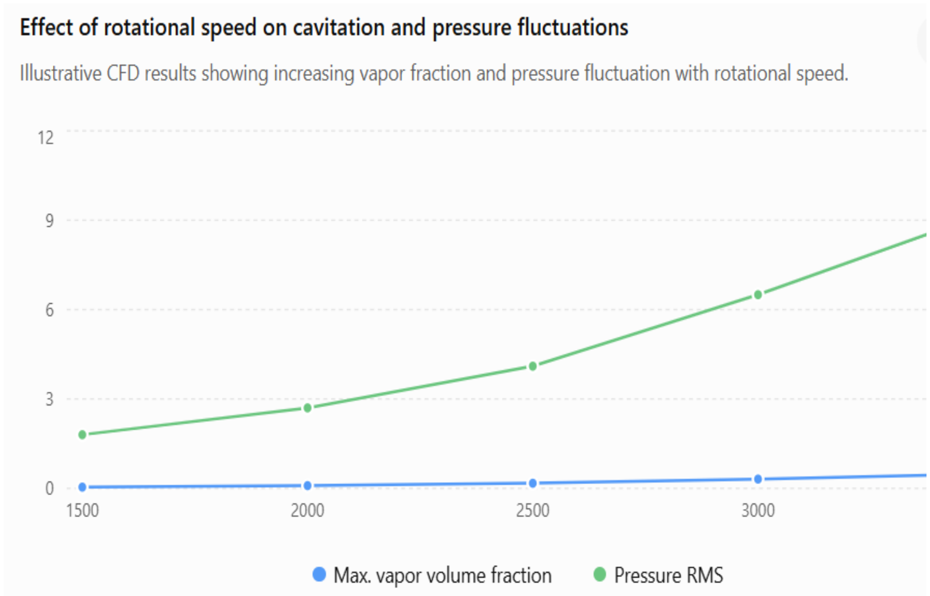

Figure 1: Effect of Rotational Speed on Cavitation and Pressure Fluctuations

Figure 1 shows the effect of rotational speed on cavitation and pressure fluctuations. According to the figure, the vertical axis shows fluctuation levels; the blue line shows the vapour volume fraction. The green line shows the pressure RMS of each factor.

Table 3: Conclusion: Overall Results

Parameter	Low-Cavitation Condition	Severe-Cavitation Condition	Observed Trend
Rotational speed	1500 rpm	3500 rpm	Cavitation increases
Inlet pressure	120 kPa	40 kPa	Cavitation increases as pressure falls
Vapor volume fraction	0.01–0.04	0.46–0.48	Strong increase
NPSH available	10.8 m	2.2 m	Significant reduction
Pump head	~54 m	~46 m	Decreases
Efficiency	~87%	~73%	Decreases
Pressure RMS	~1.9 kPa	~8.7–9.2 kPa	Strong increase

The analysis results in Table 3 show that the two most important parameters affecting cavitation in high-speed centrifugal pumps are rotational speed and inlet pressure. The greater the velocity of the particles in the impeller, the greater the pressure difference between the inlet pressure and the vapor pressure; the lower the inlet pressure, the smaller the pressure difference between the local pressure and the vapor pressure. Both effects have the potential of favouring the formation of vapours. The results also suggest that the effects of cavitation are not limited to the formation of vapor, but can also be seen in measurable deterioration of performance in the pump. Higher vapour volume fraction also increases pressure pulsations, efficiency, and pump head. So, vapor volume fraction, NPSH, pressure RMS and efficiency measurements provide helpful information about the occurrence and nature of cavitation.

6. Applications

This numerical study of cavitation analysis and pressure dynamics in high-speed centrifugal pump arrangements can be widely used in industrial fluid and energy systems, water systems, chemical systems, petroleum industry, marine engineering, and other rotating machinery. By understanding this correlation between pressure distribution, vapor formation, rotational speed, flow rate and the pressure pulsations, engineers can enhance the reliability, efficiency, service life and safety of oil pumps. The major applications of the present research are discussed below.

6.1 Industrial Pumping Systems

In many plants and processes, high-speed centrifugal pumps move liquids. Cavitation can dramatically affect pump performance and lead to unstable operation. Numerical analysis detects low-pressure areas inside the impeller and computes operating conditions likely to cause cavitation. The volume fraction of the vapor and the pressure variations at the pump can be used to optimize the operational range of the pump so that the flow at the pump is stable and cavitation is not excessive. This is especially beneficial when the system runs continuously, as a sudden pump failure can disrupt production.

6.2 Power Generation

Centrifugal pumps are used in large-scale power-generation plants to pump cooling water, feedwater and process fluids in various thermal power generation plants. Pressure changes due to cavitation can also cause vibration and damage to hydraulic components. Before implementation, CFD can analyse cavitation and determine pump behaviour under various loads and inlet conditions. The results can help engineers choose appropriate operating speeds, inlet pressures, and pump configurations, improving reliability and reducing maintenance needs.

6.3 Water Supply and Distribution

Pumping stations are critical to municipal water-supply networks, ensuring sufficient pressure and flow. Operating conditions can change due to fluctuations in demand, reservoir levels, or pipeline resistance. A numerical cavitation analysis can help determine whether a pump will operate safely under these varying conditions. Cavitation can be minimized by achieving adequate NPSH and avoiding excessive pressure drop at the pump inlet. Hence, the study can be useful for more effective water distribution systems and better energy utilization.

6.4 Irrigation and Agricultural Applications

Centrifugal pumps are commonly used in irrigation systems to pump water from wells, reservoirs, canals, and storage tanks. Because suction conditions vary, cavitation can occur, especially at high speeds or when water levels drop. The numerical model developed in this research can evaluate the effect of suction pressure, pump flow rate, and pump rotational speed on pump performance. This can help establish safe operating conditions and minimise energy losses in agricultural pumping systems.

6.5 Oil and Gas Industry

Centrifugal pumps are widely used in petroleum production and processing, refining, transportation, and production. For many applications, pumps must function at high flow rates and under high pressures. Cavitation can severely reduce hydraulic performance and damage components. Numerical simulation is an effective method to study the interaction between rotating impellers and multiphase flow. Engineers can then design the geometry and operating conditions of their pumps to optimize regions of vapor formation and pressure instability for petroleum-processing applications.

6.6 Chemical and Process Industries

Centrifugal pumps are frequently employed in chemical processing plants to handle solvents, chemicals, process water and other liquids. The physical characteristics of these fluids can vary significantly from those of water, and can affect the fluids' vapour pressure and cavitation behaviour. Appropriate fluid properties can be used in the numerical models, and the influence of pressure and temperature on cavitation can be assessed.

6.7 Marine Engineering

Centrifugal pumps are used in marine cooling systems, ballast systems, fire protection systems, and circulation systems. Limited installation space and variable operating conditions can make pump suction conditions particularly important. Cavitation can generate pressure pulsations and vibration, which can impact pump components and/or associated pipework. Using numerical investigations, engineers can identify critical conditions and design a pump that reduces cavitation. The results can help increase reliability in marine fluid-handling systems.

6.8 Cooling and HVAC Systems

Centrifugal pumps circulate water or other cooling fluids in large industrial cooling systems and heating, ventilation and air-conditioning systems. These systems can directly measure pump efficiency. Excessive cavitation can reduce flow efficiency and increase operating costs. Numerical analysis can determine the optimal pump speed and flow rate while maintaining adequate inlet pressure. This could contribute to efficient pump operation and increased pump life.

6.9 Pump Design Optimization

Developing pump geometry for centrifugal pumps is one of the most important applications of this research. CFD simulations may be undertaken prior to production of a prototype in order to save development time and experimental costs. The blade angle, blade thickness, number of blades, impeller diameter, inlet geometry, and clearance can be systematically varied. You can then analyse and compare the pressure and vapour distribution to identify designs that minimise low-pressure areas and limit cavitation.

6.10 Minimize Pressure Pulsations

One of the key sources of vibration and structural loading of rotating pump systems are pressure fluctuations. Using transient numerical simulation, you can record pressure signals at various locations inside the pump. Frequency-domain analysis can detect the blade-passing frequency component and other significant frequencies. This data can be used to change blade geometry, the volute configuration, or the operating speed to minimise pressure pulsations. As such, the research can help operate pumps more quietly and with greater mechanical stability.

6.11 Predictive Maintenance

The correlation between pressure fluctuations and cavitation can serve as the basis for condition monitoring and predictive maintenance. Pressure and vibration signals can be monitored at the pump inlet and outlet during operation. Numerical results can identify characteristic pressure-fluctuation patterns depending on the cavitation level. These patterns can then be used as reference indicators to detect abnormal behavior of the pump. Identifying cavitation in the early stages enables maintenance personnel to correct operating conditions before severe performance loss occurs.

6.12 Energy Efficiency Improvement

In many industrial facilities, pumping systems account for a significant share of energy use. Hydraulic efficiency decreases due to flow loss from cavitation and losses caused by flow collapse. Numerical analysis of the data can identify operating points where the machine is most efficient and cavitation noise remains within acceptable limits. The result is that energy can be saved without sacrificing the flow delivered by the pump by optimizing the pump speed and designing the hydraulic system.

6.13 Support for Digital Twin and Smart Pumping Systems

Digital-twin systems of the modern pumping infrastructure can also be developed using numerical results. A digital model of a pump can integrate CFD with live data from pressure, flow, vibration, and speed transducers. These systems can estimate the current hydraulic state and detect deviations from normal hydraulic behaviour. A numerical cavitation model can be integrated with monitoring systems to support automated operating decisions and design early-warning systems.

6.14 Pump System Safety and Reliability

Cavitation can cause unstable hydraulic conditions that may affect pump components and related piping. Engineers can anticipate operating regions prone to pressure fluctuations and vapour generation during the design phase by predicting these events. This is particularly crucial in systems where continuous, reliable pumping is essential. The numerical approach is therefore a useful tool for improving operational safety and reducing unplanned equipment shutdowns. Ensure that research and experimental validation are performed. Ensure that research and experimental validation is carried out. Last but not least, the methodology is valuable as a research tool. CFD predictions can be compared with experimental measurements from laboratory-scale centrifugal pumps. Pressure sensors, flow meters, vibration sensors, and high-speed visualisation techniques can validate numerical predictions. Such comparisons can help validate cavitation models and improve understanding of vapour-cloud dynamics and pressure-pulsation mechanisms. The numerical investigation offers a useful framework for design optimisation, performance optimisation, cavitation mitigation, pressure-pulsation control, predictive maintenance, and energy management. The most important thing about it is relating detailed internal flow behaviour with measurable pump-performance parameters. Understanding how rotational speed, inlet pressure, flow rate, and pump geometry relate to cavitation helps engineers design centrifugal pump systems that run more efficiently, reliably, and sustainably across industrial applications.

7. Discussion

The numerical results show a strong correlation between operating conditions, cavitation intensity, and pressure fluctuations in high-speed centrifugal pump systems. The analysis shows that among the main parameters affecting cavitation behaviour, rotational speed is one of the most important. The maximum vapor volume fraction gradually increased with the increase in rotational speed from 1500 to 3500 rpm, and it shows that vapor formation progressed in the impeller. This effect likely results from higher fluid velocity and the higher-pressure gradients created in the rotating impeller. As the blade rotates, the low-pressure areas near the leading edge are more likely to vaporize. Additionally, the findings suggest that higher rotation speeds may not be optimal for overall pump performance. As head rose with rotation speed, hydraulic efficiency decreased at higher speeds. This decrease is correlated with the growth of unstable vapor structures and hydraulic losses. The high vapour volume fraction observed in the fastest case suggests severe cavitation and may reduce the liquid's ability to flow through the impeller passages. One of the most significant features noted is the marked rise of cavitation intensity with the RMS of the pressure fluctuation in the plenum. The pressure fluctuation increased from 0.18 kPa at 1500 rpm to 0.92 kPa at 3500 rpm. This means that vaporisation and recon collapse create significant unsteadiness in the internal flow. Vapour-structure interaction with rotating blades may cause periodic pressure fluctuations, which may lead to vibrations and mechanical loading. Thus, pressure fluctuation becomes a key parameter for evaluating cavitation severity. The inlet-pressure analysis also supports the significance of the suction conditions. The maximum vapor volume fraction rose from 0.01 to 0.46 when the inlet pressure was decreased from 120 to 40 kPa. At the same time, available NPSH dropped significantly, pump head decreased, and pump efficiency declined. The results show that keeping adequate inlet pressure is crucial to avoid excessive cavitation. The numerical results also indicated that cavitation management requires a balance among pump speed, flow rate, and suction pressure. If the pump operates too fast or at too low an inlet pressure, it can create severe cavitation even if the pump starts with a high head. Thus, it is necessary to maintain a suitable hydraulic operating condition for the pumps. Overall, the results show the usefulness of CFD-based transient analysis for detecting cavitation-prone regions and estimating cavitation-related pressure fluctuations. The overall evaluation of the vapour volume fraction, NPSH, pump efficiency, and pressure RMS provides a comprehensive assessment of pump stability. The results can be used for the development of better pumps, optimizing operating conditions, vibration reduction and predictive maintenance for high-speed centrifugal pump systems.

8. Conclusion

This study aims to numerically investigate cavitation phenomena and pressure fluctuations in high-speed centrifugal pump systems using computational fluid dynamics (CFD). The analysis indicates that cavitation is very sensitive to the inlet pressure, flow, rotational speed and the ensuing pressure distribution in the impeller and volute. The numerical results show that increasing rotational speed favours the formation of low-pressure areas and, therefore, vapour in the pump. The results indicate that the maximum vapor volume fraction increases significantly with increasing rotational speed, and the pressure fluctuation RMS also increases with increasing rotational speed. This confirms that high cavitation correlates with high hydraulic instability and pressure unsteadiness. While higher rotational speed provides more pump head, if the speed gets too high, hydraulic losses increase and eventually reduce pump efficiency. The inlet pressure investigation further shows that proper inlet suction pressure is required to ensure pump stability. As inlet pressure decreases, available NPSH decreases and the risk of cavitation increases. Severe cavitation is characterised by increased pressure fluctuations, reduced pump head, and decreased hydraulic efficiency. These results highlight the need to ensure proper inlet conditions and avoid operating outside the recommended hydraulic range. The study also confirms that pressure fluctuation analysis can provide useful insights into cavitation severity. Monitoring pressure RMS and frequency characteristics can identify unstable operating conditions and may help early detection of cavitation-related issues. This is because vapor volume fraction, along with NPSH, pump head, efficiency, and pressure fluctuation, provides a complete picture of pump performance. Overall, the numerical method helps explain the combined effects of cavitation and unsteady pressure phenomena in high-speed centrifugal pumps. The results can be used to optimize pump design, select operating conditions, improve energy efficiency, minimise vibration, and support predictive maintenance. Future studies should include experimental validation and explore other impeller configurations, circulating-fluid properties, operating temperatures, and transient conditions to further improve the prediction models' ability to describe cavitation.

9. Recommendations

The following recommendations are based on the numerical investigation of cavitation dynamics and pressure fluctuation in high-speed centrifugal pump systems.

9.1 Ensure Adequate Inlet Pressure

Pump systems should operate with sufficient inlet pressure (suction) and NPSH margin to prevent vaporisation at the impeller inlet. Do not rotate at excessive speeds: Rotational speed can be increased to increase pump head, but if it is too high, cavitation pressure fluctuations and cavitation intensity will increase.

9.2 Optimize Impeller Geometry

For future pump designs, consider reducing localised low-pressure zones and delaying cavitation inception by optimising blade inlet angle, blade thickness, blade number, impeller diameter, and tip clearance.

9.3 Maintain Optimum Operating Flow Rates

Pumps are best if operated close to their best efficiency point (BEP). Insufficient or excessive flow rates may increase flow separation, pressure instability, and cavitation risk.

9.4 Apply Transient CFD Analysis

Transient CFD analysis is required in addition to steady-state CFD analysis if steady-state analysis does not adequately capture vapour-cloud dynamics and pressure pulsations. Future studies should use transient multiphase CFD simulations with sufficiently small-time steps.

9.5 Enhance the Accuracy of Cavitation Models

Compare the accuracy of various cavitation and turbulence models, and determine their sensitivity to vapour, condensation, turbulence, and unsteady flow structures.

9.6 Experimental Validation

Check numerical predictions against experimental data on pump head, efficiency, pressure fluctuations, vibration, and NPSH. This will enhance the confidence in CFD results.

9.7 Predictive Maintenance

Couple numerical results with real-time measurements of pressure, vibration, flow rate, and rotational speed to create early-warning systems that predict cavitation degradation.

9.8 Consider the Fluid Temperature

This requires further investigation in future studies, as vapour pressure varies with temperature and may significantly affect cavitation inception.

9.9 Optimized Operating Maps

If a numerical simulation is performed over a broader speed, flow, and inlet pressure range, safe operating maps can be created that indicate zones of stable, transitional, and severe cavitation.

10. Future Research

Future work should combine advanced transient CFD, experimental verification, pressure/vibration monitoring, and intelligent data analysis. By implementing such an integrated approach, cavitation prediction, pressure-fluctuation minimisation, energy-efficiency improvement, and the reliability and service life of a high-speed centrifugal pump system can be enhanced.

References

Al-Obaidi, A. R., & Shaban, A. H. (2024). Experimental and numerical the effect of cavitation detection on hydraulic performance of the centrifugal pump based on different geometrical configurations. *Journal of Engineering Research*. https://doi.org/10.1016/j.jer.2024.11.006

Alubokin, A. A., Gao, B., Ning, Z., Yan, L., Jiang, J., & Quaye, E. K. (2022). Numerical simulation of complex flow structures and pressure fluctuation at rotating stall conditions within a centrifugal pump. *Energy Science & Engineering*, *10*(7), 2146-2169. https://doi.org/10.1002/ese3.1123

Chao, W.-x., Shi, B.-l., Ruan, H., & Dong, W. (2022). The cavitation characteristics of high speed centrifugal pumps with different impeller types. *Frontiers in Energy Research*, *10*, 811690. https://doi.org/10.3389/fenrg.2022.811690

Cui, B., Han, X., & An, Y. (2022). Numerical simulation of unsteady cavitation flow in a low-specific-speed centrifugal pump with an inducer. *Journal of Marine Science and Engineering*, *10*(5), 630. https://doi.org/10.3390/jmse10050630

Cui, Y., Guo, F., Ding, Q., & Cheng, B. (2025). Investigation of cavitation damage patterns in centrifugal pump blades under rotating flow fields. *Journal of Pipeline Science and Engineering*, 100288. https://doi.org/10.1016/j.jpse.2025.100288

Dong, L., Zhao, Y., & Dai, C. (2019). Detection of inception cavitation in centrifugal pump by fluid-borne noise diagnostic. *Shock and Vibration*, *2019*(1), 9641478. https://doi.org/10.1155/2019/9641478

Ebada, Y., Elshennawy, A., Elbrashy, A., & Rashad, M. (2025). Performance optimization of centrifugal pumps: Experimental analysis of flow enhancement and cavitation mitigation under variable operating conditions. *Flow Measurement and Instrumentation*, 103043. https://doi.org/10.1016/j.flowmeasinst.2025.103043

Fang, X., Guo, T., Tao, R., Wu, Y., Xiao, R., & Lin, K. (2025). Comparative Investigation of Cavitation Performance by Using Integrated Induced Impeller for the High-Speed Centrifugal Pump. *Current Science*, *5*(3), 2227-2253. https://doi.org/10.52845/CS/2025-5-3-15

Goncalves, E., & Patella, R. F. (2009). Numerical simulation of cavitating flows with homogeneous models. *Computers & Fluids*, *38*(9), 1682-1696. https://doi.org/10.1016/j.compfluid.2009.03.001

Gupta, P., Subhash, M., Dobriyal, R., & Sharma, R. C. (2023). CFD Simulation of pressure, temperature and rotational speed influence on the cavitation in centrifugal Pump Impeller. *International Journal of Vehicle Structures & Systems*, *15*(2), 147-153. https://doi.org/10.4273/ijvss.15.2.01

Hu, J., Ding, Z., & Xu, L. (2025). Study on the impact of ice–water mixture on entropy generation and cavitation characteristics inside a centrifugal pump based on entropy generation theory. *Computational Particle Mechanics*, *12*(6), 5575-5600. https://doi.org/10.1007/s40571-025-01083-z

Huang, J., Luo, X., Zhou, S., Liu, M., Tang, Z., & Zhao, W. (2025). Study on the internal flow and pressure pulsation of high specific speed centrifugal pump under different degrees of cavitation. *Physics of Fluids*, *37*(10). https://doi.org/10.1063/5.0283859

Jia, X., Yang, X., & Zhu, Z. (2024). Study on pressure pulsations and transient fluid forces in centrifugal pump under different degrees of cavitation. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, *238*(6), 2834-2844. https://doi.org/10.1177/09544089231164832

Jiao, W., Jia, X., Cheng, L., Xu, J., Liang, A., Fan, H., & Lu, J. (2025). Numerical simulation and experimental study on cavitation and pressure fluctuation characteristics of low head pumped storage system under pump operating conditions. *Energy*, *328*, 136515. https://doi.org/10.1016/j.energy.2025.136515

Liu, S., Cao, H., Chen, Y., Ni, S., Zhao, G., & Jiang, C. (2023). Numerical examination of the dynamic evolution of fluctuations in cavitation and pressure in a centrifugal pump during startup. *Machines*, *11*(1), 67. https://doi.org/10.3390/machines11010067

Lu, J., Li, L., Gong, Y., Liu, X., Zhang, F., & Zhu, B. (2024). Experimental and numerical investigations of the noise induced by cavitation in a centrifugal pump. *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering*, *48*(4), 1937-1955. https://doi.org/10.1007/s40997-023-00749-4

Mao, Y.-h., Liu, H.-l., Wang, Y., Chen, J., & Liu, F. (2023). Experimental study on effects of air injection on cavitation pressure pulsation and vibration in a centrifugal pump with inducer. *Journal of Hydrodynamics*, *35*(6), 1168-1178. https://doi.org/10.1007/s42241-024-0086-7

Rachmanu, F., Rusli, M., Malta, J., & Azka, M. (2025). Experimental analysis of FFT-based vibration characteristics of fluid flow in centrifugal pump systems. In *IET Conference Proceedings CP960* (Vol. 2025, pp. 280-285). IET.

https://doi.org/10.1049/icp.2026.2079

Štefanič, M. (2025). Numerical Prediction of the NPSH Characteristics in Centrifugal Pumps. *Fluids*, *10*(10), 274. https://doi.org/10.3390/fluids10100274

Wang, D., Ma, W., Zhao, W., Cao, R., & Yang, Y. (2025). Experimental acoustic analysis of cavitation in a centrifugal pump. *Fluid Dyn. Mater. Process*, *21*, 877-890. https://doi.org/10.32604/fdmp.2024.055220

Wang, X., Wang, Y., Chen, J., Li, M., Wang, Z., & Yang, M. (2025). Numerical investigation on cavitating flow and cavitation-sand erosion characteristics of a centrifugal pump under sand-laden water conditions. *Engineering Computations*, *42*(3), 1163-1185. https://doi.org/10.1108/EC-10-2024-0906

Yin, X., Guo, X., Li, P., Lin, R., Feng, B., & Kukareko, V. (2026). Development Status and Prospects of Centrifugal Pump Cavitation: A Bibliometric Analysis Using CiteSpace. *Water*, *18*(6), 668. https://doi.org/10.3390/w18060668

Zhang, S., Tian, R., Ding, K., Chen, H., & Ma, Z. (2022). Numerical and experimental study in pressure pulsation and vibration of a two-stage centrifugal pump under cavitating condition. *Modern Physics Letters B*, *36*(01), 2150501. https://doi.org/10.1142/S0217984921505011

Zhang, X., Jiang, C., Chen, Z., & Tong, F. (2022). Numerical study on the dynamic evolution of cavitation and pressure fluctuation in a centrifugal pump during the startup period. In *1st International Conference on Mechanical System Dynamics (ICMSD 2022)* (Vol. 2022, pp. 1132-1141). IET. https://doi.org/10.1049/icp.2022.1972

Zhou, J., Jin, G., Ye, T., & Wang, X. (2023). Fluid-induced vibration analysis of centrifugal pump including rotor system based on Computational Fluid Dynamics and Computational Structural Dynamics coupling approach. *Ocean Engineering*, *288*, 115993. https://doi.org/10.1016/j.oceaneng.2023.115993