

Article

Numerical Prediction of the NPSH Characteristics in Centrifugal Pumps

Matej Štefanič 

Faculty of Mechanical Engineering, University of Novo Mesto, 8000 Novo Mesto, Slovenia;
matej.stefanic@fs-unm.si

Abstract

This study focuses on the numerical analysis of a centrifugal pump's suction capability, aiming to reliably predict its suction performance characteristics. The main emphasis of the research was placed on the influence of different turbulence models, the quality of the computational mesh, and the comparison between steady-state and unsteady numerical approaches. The results indicate that steady-state simulations provide an unreliable description of cavitation development, especially at lower flow rates where strong local pressure fluctuations are present. The unsteady $k-\omega$ SST model provides the best overall agreement with experimental NPSH₃ characteristics, as confirmed by the lowest mean deviation (within the ISO 9906 tolerance band, corresponding to an overall uncertainty of $\pm 5.5\%$) and by multiple operating points falling entirely within this range. This represents one of the first detailed unsteady CFD verifications of NPSH prediction in centrifugal pumps operating at high rotational speeds (above 2900 rpm), achieving a mean deviation below $\pm 5.5\%$ and demonstrating improved predictive capability compared to conventional steady-state approaches. The analysis also includes an evaluation of the cavitation volume fraction and a depiction of pressure conditions on the impeller as functions of flow rate and inlet pressure. In conclusion, this study highlights the potential of advanced hybrid turbulence models (such as SAS or DES) as a promising direction for future research, which could further improve the prediction of complex cavitation phenomena in centrifugal pumps.

Keywords: centrifugal pump; cavitation; net positive suction head (NPSH); computational fluid dynamics (CFD); turbulence modeling; unsteady simulations; cavitation volume fraction; pump performance characteristics



Academic Editor: Zheng-Tong Xie

Received: 17 September 2025

Revised: 16 October 2025

Accepted: 20 October 2025

Published: 21 October 2025

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

Copyright: © 2025 by the author. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

In a time of increasing energy transitions and evolving markets, evaluating the efficiency of hydraulic machines is of growing importance. With known characteristics of a hydraulic machine, it is possible to more effectively select a suitable machine for integration into a system, improve overall efficiency, and consequently reduce electricity consumption and lower costs. An analysis of energy use in hydraulic machines shows that annual energy consumption accounts for 56%, with poorly designed water pumps making up 22% of this share [1].

This is far from negligible and presents a significant challenge for further research and development of hydraulic machines. The primary goal in development of hydraulic machines is to improve machine efficiency.

The development of centrifugal pumps relies on interdisciplinary methods, one of which is Computational Fluid Dynamics (CFD). CFD simulations provide a detailed insight

into flow conditions and enhance the understanding of the hydraulic behavior of pumps. Among the parameters influencing efficiency is the Net Positive Suction Head (NPSH), a key factor for the hydraulic stability of the pump, as it prevents the occurrence of cavitation.

The NPSH characteristic is divided into two parts, $NPSH_R$ and $NPSH_A$ [2,3]. $NPSH_R$ (Net Positive Suction Head required) represents the minimum suction head, at which the pump can operate without exceeding a specified amount of head drop due to cavitation, while $NPSH_A$ (Net Positive Suction Head available) is the actual suction head provided by the system at the pump inlet. Various studies can be found that use advanced virtual evaluation.

In the study [4], the authors used CFD and experimental methods to analyze the prediction of torque on the pivot of guide vanes in a reversible pump-turbine. Pivot stress was measured using instrumentation and compared with simulation results. The $k-\omega$ SST and SARC turbulence models were applied. The simulations achieved a deviation of $\pm 3.9\%$ in turbine mode, while pump mode, due to dynamic flows, required unsteady simulations, which more accurately captured the conditions at low flow rates. The study highlights the role of CFD as a key tool for the optimization of hydraulic machines by reducing the need for prototypes and enabling the analysis of various operating regimes.

A similar study was conducted on a two-stage centrifugal pump [5]. The authors proposed an efficient method for predicting $NPSH_R$ in two-stage centrifugal pumps, eliminating the high costs associated with traditional approaches for determining $NPSH_3$ by reducing inlet pressure. Using a new boundary condition strategy, the $k-\omega$ SST turbulence model, and the Zwart-Gerber-Belamri cavitation model, they achieved a deviation of up to 7.56% from experimental data. The method reduced computational time by up to 54.55% for flow rates ranging from $0.6 Q_{BEP}$ to $1.2 Q_{BEP}$. This enables faster optimization and time savings in industrial pump design.

In the study [6], a new method was developed for determining cavitation in a centrifugal pump. The authors proposed a method for rapid prediction of NPSH in a two-stage centrifugal pump using transient simulations, reducing the number of steps to three with an algorithm for identifying the critical cavitation point. Validation through experiments confirmed the accuracy of the analyses, while the investigation revealed pressure non-uniformity and recirculation at $0.6 Q_{BEP}$.

In the study [7], a numerical and experimental analysis was conducted on the formation of cavitation bubbles and cavitation implosion. The authors analyzed cavitation instabilities in a high-speed inducer of a centrifugal pump, using a combination of numerical simulations and visual experiments with a high-speed camera. At low cavitation numbers, they observed periodic cavitation surges that caused head fluctuations due to dynamic changes in the size of cavitation structures. The numerical results showed good agreement with the experimental data, confirming the reliability of the applied models. The analysis emphasizes the importance of optimization to reduce cavitation effects and provides insight into the dynamics of instabilities. This is crucial for preventing failures in industrial systems.

In the study [8], the authors investigated numerical simulation of flow and cavitation in a centrifugal pump, using the SAS-SST turbulence model and the Kanfoudi-Zgolli cavitation model. The model was previously validated on the NACA66-MOD hydrofoil. Cavitation-free validation showed good agreement with experimental measurements of pressure fluctuations and bubble dynamics. The analysis revealed periodic pressure oscillations caused by rotor-stator interaction and a drop in efficiency of up to 16% at high flow rates. The SAS-SST model successfully captured the turbulent spectrum and cavitation dynamics. The study supports its use for predicting complex cavitation phenomena in industrial applications.

In the study [9], the authors analyzed fluid–structure interaction (FSI) in centrifugal pumps, where cavitation reduces energy efficiency, increases vibrations, and imposes structural loads on the system. Using a two-way FSI model, they examined flow and deformation under various cavitation conditions. A reduction in NPSH triggered rapid bubble growth, leading to blockage of blade channels. Radial forces increased up to 54.4 N, along with pressure fluctuations in the volute. Structural analysis showed a reduction in blade deformation (up to 0.19 mm) and rotor frequencies (0.95 Hz) due to decreased loading. The results highlight the direct impact of cavitation on hydraulic stability and pump lifespan, and demonstrate how FSI integration enables optimized structural design.

In the study [10], the authors developed an efficient numerical method for evaluating NPSH in centrifugal pumps using CFD. By reducing the computational domain to a single blade passage and applying periodicity, the number of mesh elements was decreased by 75%. Testing four mesh densities showed that the coarsest mesh was sufficient for estimating NPSH₃; however, finer meshes were necessary for more accurate pressure calculations. The simulations were validated on seven blade geometries, with an error of less than 5%, confirming the method's reliability. This approach shortens pump development time, crucial for industry, by enabling a rapid assessment of NPSH₃.

In the article [11], the authors analyzed experimental methods for cavitation detection. The experiment included measurements of static pressure, sound, vibrations, and visualization techniques. The main objective of the study was the development of automated and cost-effective systems. Future directions involve the integration of numerical models with advanced methods.

In the article [12], the authors conducted a numerical and experimental analysis of cavitation in a multistage centrifugal pump. They investigated the influence of NPSH on hydraulic performance and cavitation development. CFD simulations showed that reducing NPSH below NPSH₃ leads to a rapid pressure drop, an increase in vapor content, and a decrease in head of up to 9.7%, compared to the experiment. Cavitation first appears at the suction edge of the blades and spreads as NPSH decreases. Additional stage-by-stage analysis revealed a correlation between head losses and cavitation occurring in individual stages.

In the article [13], the authors investigated cavitation development in a centrifugal pump. Numerical simulations and experiments were conducted on a closed-loop hydraulic test system. They analyzed the internal flow and pressure pulsations at the pump inlet and outlet during cavitation onset. The results confirmed the occurrence and development of cavitation through both experiments and simulations. It was found that the inlet pressure is more sensitive to cavitation-induced changes than the outlet pressure. Under severe cavitation, a pulsation frequency of around 30 Hz was observed. Additionally, the pump pressure dropped by approximately 0.77%, compared to non-cavitating conditions, which may indicate the onset of cavitation. According to their observations, these low-frequency pulsations were linked to the shedding and collapse of cavitation structures, which generated vortical motion and secondary flows within the impeller passages. Such coupling between cavitation dynamics and vortex formation is characteristic of the unsteady behavior in pumps operating under severe cavitation.

In article [14], a study was conducted on the importance of numerical stability and the suitability of numerical approaches in the simulation of centrifugal pumps. It was found that steady-state simulations are less appropriate for analyzing unsteady flow regimes due to pronounced numerical oscillations that arise under such conditions. The analysis showed that unsteady simulations capture complex phenomena, such as vortex formation and flow separation, significantly more accurately, particularly in the off-design operating conditions.

In article [15] a study systematically investigates the flow field characteristics in a cylindrical cavitation nozzle jet using various turbulence models, including RANS, SBES, and LES. The authors highlight that while RANS models capture the large-scale flow structures reasonably well, they fall short in accurately predicting cavitation cloud morphology and transient bubble dynamics. In contrast, SBES and LES provide better resolution of small-scale vortices and cloud shedding behavior. The work emphasizes the sensitivity of cavitation predictions to turbulence modeling and suggests that higher fidelity turbulence models are essential for more accurate simulation of cavitating flows.

In article [16] a study explores the three-dimensional dynamics of hydrodynamic cavitation in a Venturi tube via both 2D and 3D numerical simulations using Detached Eddy Simulation (DES). The authors find that URANS models struggle to reproduce the correct cavity morphology, while DES yields improved agreement with experimental observations on cavity shape and turbulence statistics. In 3D simulations, they reveal significant interactions between cavitation and vortex structures, showing dominance of velocity-gradient driven effects over baroclinic torque in vortex production. Local turbulence metrics such as Reynolds shear stress and turbulent kinetic energy are better captured in DES, underscoring the importance of 3D modeling for cavitation–turbulence interactions.

Based on these findings, our study further examined the numerical instabilities in predicting the NPSH characteristics of centrifugal pumps, with the aim of ensuring more reliable numerical results and providing clearer guidelines for industrial applications of CFD simulations.

The research highlights the critical role of advanced numerical simulations and turbulence models in accurately predicting cavitation phenomena, forming the basis for optimizing the hydrodynamic performance of pumps. In recent years, this field has experienced rapid development, with research outcomes increasingly transferred from academic environments into industrial practice. This trend is particularly relevant in light of the European efforts toward a green transition and the sustainable transformation of various technological sectors.

In this context, the article provides a framework for implementing Computational Fluid Dynamics (CFD) in industrial applications related to cavitation modeling. The analytical approaches are supported by experimental validation, aimed at identifying limitations and challenges associated with numerical analyses, especially in the context of unsteady cavitation phenomena. Recent advances, such as those reported in research [17], demonstrate that cavitation performance can be optimized through active control strategies and multi-physics coupling approaches, further linking numerical prediction with practical industrial applications. Emphasis is placed on the connection between theoretical models, computational simulations, and measurements to ensure the robustness of results and their applicability to the real-world engineering systems.

This approach enables a critical evaluation of simulation errors and enhances the ability to predict cavitation, which is essential for the development of more energy-efficient and sustainable hydraulic devices. It is also important to note the growing relevance of such research in the context of the European green transition, as optimizing the energy efficiency of hydraulic systems directly contributes to reducing the carbon footprint and achieving the EU's sustainable development goals, particularly within the framework of the European Green Deal initiative [18]. It should be noted that most pumps operate at 1450 rpm, whereas certain more purpose-built pumps, such as in our case, operate at higher rotational speeds, above ~2900–3000 rpm. This represents one of the challenges, since at such operating conditions the flow behavior becomes more complex and, consequently, the NPSH values are higher.

In contrast to previous works, this study provides a systematic evaluation of how different turbulence models and transient approaches affect the accuracy of NPSH prediction in centrifugal pumps. The research combines numerical and experimental analyses to verify model performance and establish practical guidelines for reliable cavitation assessment in industrial CFD applications. The novelty of this work lies in linking turbulence–cavitation interaction with quantitative error evaluation between steady and unsteady simulations. Furthermore, the study presents one of the first high-speed (above 2900 rpm) unsteady CFD verifications of NPSH prediction in centrifugal pumps, demonstrating the applicability of time-resolved approaches for reliable cavitation assessment under industrial operating conditions.

2. Materials and Methods

Pumps are the key components of hydraulic systems, where three fundamental parameters dominate: power, head, and efficiency. Understanding these quantities is essential for the design and performance analysis of the system. The pump head defines the energy transferred by the pump to the fluid per unit weight. It represents the hydraulic work of the pump and depends on the pressure difference between the inlet and outlet, as well as on the properties of the fluid. According to ISO 9906 it is mathematically expressed by the equation for the pump head H . Key parameters are total pressure p_t , fluid density ρ , z elevation head, V velocity and gravitational acceleration g :

$$H = (z_2 - z_1) + \frac{p_{t,2} - p_{t,1}}{\rho g} + \frac{V_2^2 - V_1^2}{2g} \quad (1)$$

Pump efficiency represents its energy effectiveness and measures the ratio between useful hydraulic power and input mechanical power. The energy efficiency value is calculated using the efficiency equation, where the key parameters are flow rate Q , pump head H , fluid density ρ , and input power P_s :

$$\eta = \frac{\rho g H Q}{P_s}, \quad (2)$$

The input power P_s is the mechanical power supplied to the pump via the drive shaft. It depends on the torque T and the angular velocity ω , and is mathematically described by the equation for input power. Input power is directly related to energy efficiency: higher efficiency requires lower input power for the same hydraulic output:

$$P_s = T\omega \quad (3)$$

In the numerical model, H and T are calculated variables, while all other quantities are parameters. The torque T is obtained by integrating the pressure and shear forces over the rotor surface. The parameters are interdependent: head and flow rate determine the hydraulic power, while efficiency links the hydraulic and input mechanical power. This enables a comprehensive analysis of the pump's hydraulic performance. The relationship between these parameters allows for the optimization of the pump selection for the numerical rating conditions.

2.1. Prediction of the NPSH Characteristics

Cavitation is the phenomenon of vapor bubble formation and implosion in a liquid due to a local drop in absolute pressure down to the vapor pressure p_v at constant temperature. This leads to a phase transition from liquid to vapor, causing shock waves, material erosion, noise, and vibrations [19]. NPSH_A represents the net positive suction head available at

the pump inlet. The value is calculated using the equation for the available suction head according to the standard ISO 9906 [3]. The ISO 9906 standard provides guidelines for the acceptance tests of rotodynamic pumps, including the permissible tolerances for key performance parameters. The value takes into account the hydraulic losses $H_{loss,s}$, h_s water level, the vapor pressure of the fluid p_v , atmospheric pressure p_a , g acceleration due to gravity and density ρ :

$$NPSH_A (m) = \frac{p_a - p_v}{\rho \cdot g} - h_s - H_{loss,s} \quad (4)$$

The key condition for preventing cavitation is: $NPSH_A > NPSH_R$. $NPSH_R$ represents the minimum net positive suction head required by the pump to prevent cavitation. It is defined as the value at which the pump head drops due to cavitation by 3% [19–21] in accordance with the ISO 9906 standard [3]. In this context, $NPSH_A$ denotes the available pump head, $NPSH_i$ defines the lowest pump head without any changes to the machine's characteristics (the critical limit) and $NPSH_R$ the required net positive suction head. In water pumps, there are two typical types of cavitation: suction cavitation and discharge cavitation. In centrifugal pumps operating at reduced flow rates, suction recirculation can induce an asymmetric pressure distribution and the formation of strong vortices at the impeller inlet. These vortices may locally reduce the pressure below the vapor pressure, causing cavitation on the pressure side of the blades, thereby reducing operational reliability and potentially leading to damage, even when the $NPSH_A$ is significantly higher than the $NPSH_R$ [22]. Discharge cavitation occurs at low flow rates, when excessively high discharge pressure causes flow separation from the blades, leading to a local pressure drop below the vapor pressure. The relationship between NPSH and cavitation can be illustrated using cavitation curves $NPSH-H$ for pumps. The aim of the study was to numerically investigate the impact of cavitation on the pump, and in this research, NPSH refers to the value of $NPSH_R$.

2.2. Dimensionless Numbers

The cavitation number represents the ratio between pressure forces and flow inertia, where p_0 is the absolute pressure at the reference point, p_v is the vapor pressure, T_f the fluid temperature, ρ the density, and v_0 the reference velocity:

$$\sigma_v = \frac{p_0 - p_v(T_f)}{\frac{1}{2}\rho v_0^2} \quad (5)$$

A modified version is used for pumps, in which $p_{1,inlet}$ represents the pressure at the pump inlet, p_v vapor pressure, T_f fluid temperature, ρ fluid density in V_p^2 is the peripheral (rotor) velocity of the impeller:

$$\sigma_v = \frac{p_{1,inlet} - p_v(T_f)}{\frac{1}{2}\rho V_p^2} \quad (6)$$

The initial cavitation number σ_{vi} is the value at which cavitation first occurs at any point in the system. For operation under cavitation-free conditions or at the threshold where the first cavitation bubble appears, the following condition must be met:

$$\sigma_v > \sigma_{vi} \quad (7)$$

where the threshold σ_{vi} depends on various factors that influence the flow conditions within the machine and are considered in fluid mechanics, such as flow rate, geometric

characteristics, viscosity, gravity, surface tension, turbulence level, thermal parameters, and wall roughness [23]. A lower value of the initial cavitation number indicates better system adaptation to flow conditions.

2.3. Numerical Analysis

2.3.1. Governing Equation

Governing equations are known as the Navier–Stokes equations. For an incompressible Newtonian fluid with a constant density in a Cartesian coordinate system, they can be written in the following way:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (8)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho} \left(-\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \right) + f_i \quad (9)$$

By taking into account the strain rate tensor and dynamic viscosity, the linear relationship between stress and velocity gradient is expressed as:

$$\tau_{ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (10)$$

By considering the mean and fluctuating velocity components, the RANS (Reynolds-Averaged Navier–Stokes) formulation is expressed as:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho} \left(-\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2} \right) - \frac{\partial \overline{u'_i u'_j}}{\partial x_j} + f_i \quad (11)$$

For the steady-state case, the time-dependent component is neglected, therefore:

$$\frac{\partial u_i}{\partial t} = 0 \quad (12)$$

2.3.2. Numerical Model

The analyses were carried out using the ANSYS CFX software, Version 2025R1 (ANSYS Inc., Canonsburg, PA, USA). The RANS approach was used for modeling turbulent flow. The High Resolution Scheme (HRS) was applied for discretization, which automatically adapts between first and second order schemes, prioritizing accuracy while maintaining numerical stability. Curvature correction was also applied, enhancing the model's ability to predict swirling flows by influencing the production terms of turbulence [24]. Kato-Launder production limiter was used, which restricts the overproduction of turbulence kinetic energy, resulting from a high shear stress rate [25]. For all simulations, double precision was used. The cavitation models are based on the Rayleigh–Plesset equation, describing the growth of a single vapor bubble in a liquid [26,27]. CFD simulations were performed on a workstation with 64-core AMD Epyc Dual CPU hardware (Supermicro, San Jose, CA, USA).

The choice of turbulence model has a significant influence on cavitation prediction accuracy, as turbulence governs vapor formation, bubble transport, and local pressure fluctuations near blade surfaces. Therefore, the selection of models suitable for representing both global and near-wall flow phenomena is essential.

For modeling turbulent flow near walls, we used k – ϵ and k – ω SST model. The advantages of the k – ω SST model lie in its ability to accurately predict flows in regions with high pressure gradients and separation, while maintaining the robustness of the k – ϵ model when using coarse meshes. This means that it enables an automatic switching between a

low-Reynolds model and the use of wall functions. As a result, it is one of the most widely used turbulence models in industry [24]. Although higher-order models such as DES, SBES, or LES offer superior resolution of transient vortex structures, their computational cost remains prohibitive for large-scale industrial pump simulations. The $k-\omega$ SST model therefore represents a practical compromise between accuracy and robustness, capable of capturing near-wall separation and low-flow vortex behavior with reasonable computational requirements. This makes it particularly suitable for industrial CFD analyses of cavitation phenomena. The rotor was modeled using the Frozen Rotor approach, with a full 360° fluid domain including all blades of the impeller and the corresponding stationary components. The frozen-rotor interface was applied over the complete circumferential (360°) contact surface between the rotating and stationary domains to ensure accurate data transfer across the interface. A no-slip boundary condition for mass and momentum was applied to all walls, and the surfaces were modeled as smooth walls. The pump operated at 2900 rpm.

The CFD simulations were carried out by following the sequential procedure described below. In the first phase, the basic energetic characteristics of the centrifugal pump were calculated, namely the pump head, efficiency, and power. This was followed by an analysis of cavitation conditions at the cavitation number of 1. The initial calculation value was based on the flow rate determined in the first step. A total of 100 iterations were performed. The third phase involved analyses at a cavitation vapor coefficient of 50, during which at least 1000 iterations were used to capture stable behavior. The cavitation vapor coefficient is described in more detail in the referenced sources [28,29]. Mean diameter of the bubble is $2\text{ }\mu\text{m}$ in all analyses. In the fourth phase, the unsteady calculation was carried out. The basic energetic characteristics were evaluated over 270 time steps, representing three full impeller revolutions and sufficiently high inlet pressure. In the fifth phase, the development of cavitation was monitored by systematically lowering the inlet pressure. Again, the values were assessed over 270 time steps. Including the calculation from the fourth phase, this totals 540 time steps, corresponding to six impeller revolutions.

Based on our previous experience and the findings of other researchers, in unsteady calculations, we [30–32] set the time step to 0.00023 s , which corresponds to a physical step of 4° . We used 10 inner iterations, which is sufficient for fast unsteady calculations in industrial applications with minimal hardware resources.

2.3.3. Computational Domain and Mesh

The computational domain was divided into four regions, i.e., intake, impeller, diffuser, and volute, to allow for a better control over the mesh generation. Table 1 presents design specification of model pump.

Table 1. Design specification of model pump.

Design Parameters	Value
Rotational speed, N (rpm)	2900
Impeller inlet diameter, D_1 (mm)	100
Impeller outlet diameter, D_2 (mm)	270
Number of blades, z (blades)	6

A combination of hexahedral and tetrahedral elements was used. This approach significantly reduces the number of control volumes and allows mesh adaptation based on the geometric complexity of each region. Hexahedra were applied in geometrically simpler parts of the geometry, while vertical were used in more complex areas. To better capture the boundary layer, an inflation function was used, along with localized surface

meshing. Care was taken to control skewness, aspect ratio, expansion ratio, and orthogonal quality. This ensured compliance with the target mesh quality criteria y^+ , which is essential for an accurate representation of flow conditions near the wall. Figure 1 shows the computational domain.

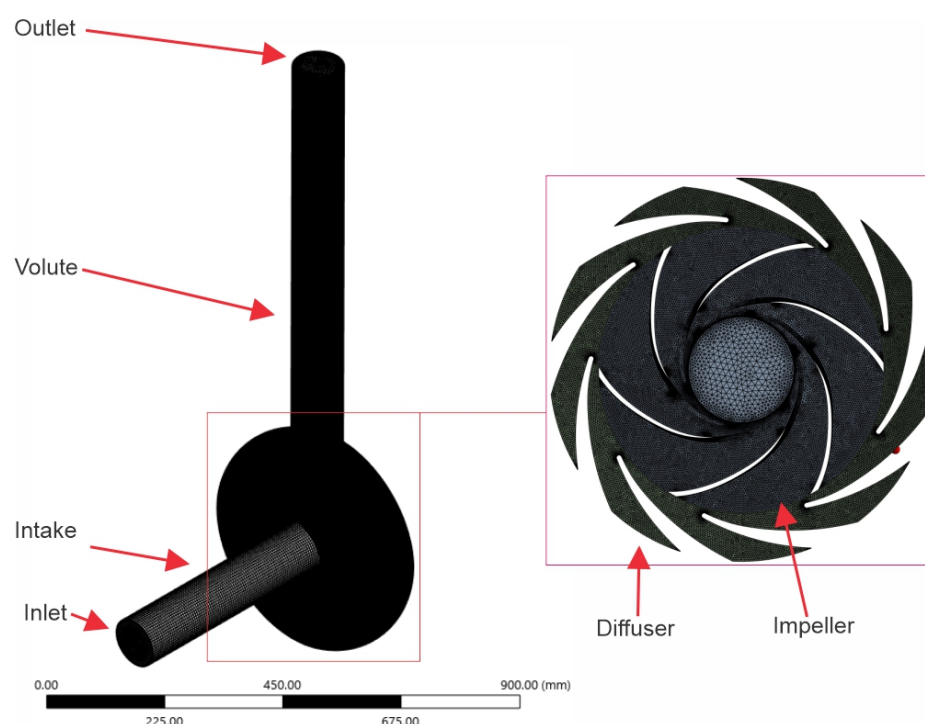


Figure 1. Computational domain.

The difference in the number of finite volumes is determined by the turbulence model. Low-Reynolds number turbulence models require a description of the viscous sublayer and $y^+ < 1$. This group also includes the $k-\omega$ SST model. However, this condition does not apply to models that use wall functions where applicable $30 < y^+ < 300$. We must check the mesh quality using the metrics of aspect ratio, skewness, and orthogonal quality [26,33]. By refining the mesh, the solution was independent of the computational mesh. The final mesh consisted of (Table 2):

- Mesh I ($k-\omega$ SST): approximately 20 million finite volumes,
- Mesh II ($k-\epsilon$): approximately 10 million finite volumes,
- Mesh III ($k-\omega$ SST): approximately 16 million finite volumes, with a denser mesh in the impeller and diffuser regions, and a coarser mesh in the other parts of the domain.

Table 2. Number of mesh volumes.

	Number of Volumes		
	Mesh I	Mesh II	Mesh III
Intake	101,574	101,574	101,574
Impeller	6,998,383	3,206,358	6,998,383
Diffuser	2,674,931	1,214,287	1,706,753
Volute	10,186,937	5,464,427	6,968,725
Total	20,043,825	9,986,646	15,775,435

The suitability of the mesh near the walls is determined by the value of the parameter y^+ , where y is the distance of the first mesh point from the wall, u_τ is the friction velocity, and ν is the kinematic viscosity.

$$y^+ = \frac{yu_\tau}{\nu} \quad (13)$$

The values of the dimensionless number y^+ based on the selected mesh are shown in the table below (Table 3).

Table 3. y^+ of the mesh.

	y^+ Value					
	Mesh I		Mesh II		Mesh III	
	$y^+ \min$	$y^+ \max$	$y^+ \min$	$y^+ \max$	$y^+ \min$	$y^+ \max$
Intake	208	278	207	280	216	268
Impeller	0.002	0.86	1.19	365	0.001	0.67
Diffuser	0.006	2.42	0.74	393	0.1	82
Volute	0.045	229	3	281	0.54	241

Figure 2 shows the distribution of y^+ values on the surface of the impeller. On the left is the result for Mesh I, in the center for Mesh II, and on the right for Mesh III.

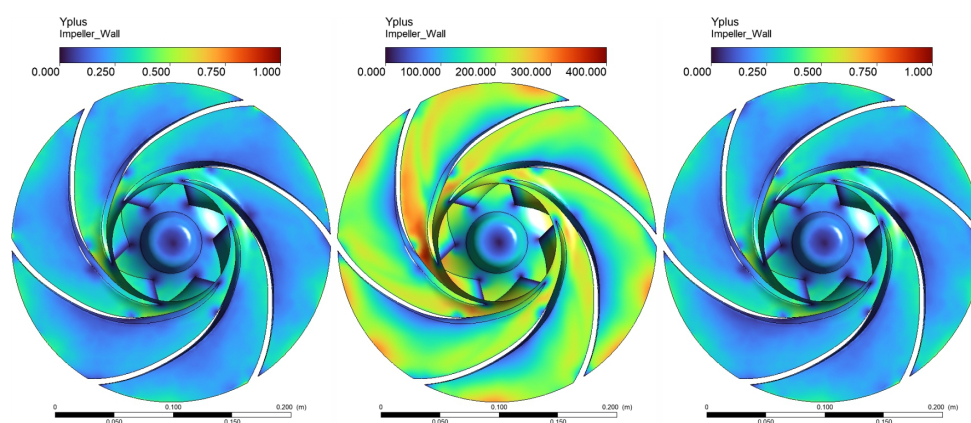


Figure 2. y^+ value on impeller.

A brief grid-independence study was also carried out on three mesh densities (Mesh I–III). The variation in the predicted head (H) between meshes was below 3.5%, confirming mesh independence. The computational grids were generated based on previous experience and validated analyses from earlier studies, ensuring an appropriate balance between accuracy and computational cost. The near-wall resolution was verified through the y^+ analysis already presented in table (Table 3). The mesh was refined and adapted according to the requirements of each turbulence model to maintain the appropriate y^+ range and ensure accurate near-wall flow representation.

2.3.4. Boundary Condition

For boundary conditions, it is essential to properly define the parameters that are critical for accurately representing the numerical model. The operating pressure was set to 0 Pa; pressure was applied at the inlet and flow rate at the outlet. For cavitation modeling, a vapor pressure of 3200 Pa was used. The density of water was set to 997 kg/m³, and the dynamic viscosity to 0.000891 N·s/m².

2.4. Experiment

The measurement setup for validating the pump characteristics was implemented in a closed-loop test rig, partially designed in accordance with the guidelines of the ISO 9906 standard. Such systems are commonly found in industrial settings. The test rig includes the following main components, as shown in the figure (Figure 3).

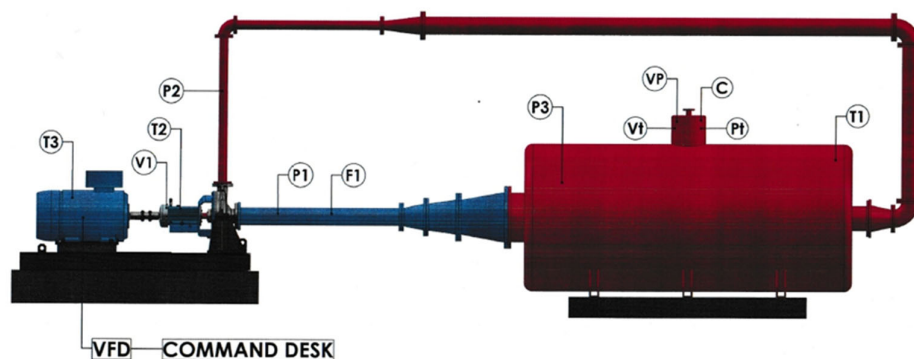


Figure 3. Closed test rig system.

The pump operated at 2900 rpm. To determine the NPSH characteristics, the inlet pressure was systematically reduced until a 3% drop in the pump head was reached. Measurements were performed at different flow rates; however, during NPSH testing, the flow rate was kept constant for each measurement point. Due to test rig limitations, the sensor locations did not strictly follow the ISO 9906 standard. Therefore, relative uncertainties were estimated based on the calibration certificates and statistical analysis of measurement repeatability.

No physical measurement is entirely free from uncertainty, as both systematic and random errors occur in every measurement. The total error consists of the influence of the alignment of the measurement coordinate system, the systematic error of the measuring instrument, and the random error component. Systematic errors, originating from the characteristics of the measuring device, installation, or operating conditions, cannot be reduced by repeated measurements. In contrast, random errors can be minimized through repetition, since the random error of the mean of independent measurements is smaller than that of an individual measurement.

For an accurate execution of the performance acceptance tests, well-defined calibration and verification procedures are required. These include determining the calibration range or spot-check requirements for each instrument, defining whether all instruments must be verified before and after the acceptance tests, and establishing conditions for recalibration during or after testing. For each type of instrument, the calibration range and the number of verification points must be specified. Before and after conducting the acceptance tests, the measuring instruments and devices used must be identified, zero readings must be taken under the defined conditions to detect possible drift, and the data acquisition system must be verified through repeated measurements under controlled operating conditions. Manual sample calculations must confirm the correct acquisition, transmission, and processing of the test data. Where appropriate, the mechanical torque in bearings and shaft seals should be measured to determine whether correction factors are necessary.

The measurement results are also affected by the quality of the water used during testing. This depends on the presence of nuclei, such as air or gas bubbles with a diameter smaller than 50 μm , the content of dissolved gases, and whether the water is completely degassed or saturated with dissolved gas.

In cases where the test rig cannot meet the strict requirements for precise acceptance testing, an alternative is industrial testing in accordance with the ISO 9906 standard.

This standard defines the permissible deviation of the measurement results for various performance parameters. For Grade 3, the permissible tolerances are $\pm 3.5\%$ for flow rate, $\pm 2.0\%$ for rotational speed, $\pm 3.0\%$ for torque, and $\pm 3.5\%$ for total pump head. The total relative uncertainty in NPSH is calculated as:

$$e_{\text{NPSH}} = \sqrt{e_Q^2 + e_n^2 + e_H^2} = \sqrt{0.035^2 + 0.02^2 + 0.035^2} \approx 0.0534 \text{ or } 5.34\% \quad (14)$$

Accordingly, the measured values in this study should be considered with a relative uncertainty of $\pm 3.5\%$ for flow rate, $\pm 3.5\%$ for head, and $\pm 5.34\%$ for NPSH, and these relative uncertainties have been consistently applied in the analysis and in the interpretation of the results. It should be noted that in industrial measurements, the overall measurement error can be somewhat higher due to less controlled testing conditions, compared to laboratory setups.

The measuring sensors used in the experimental setup are listed in Table 4.

Table 4. Measuring sensors. P1–P3 and Pt denote pressure measurement points; F1 indicates the flow rate sensor; T1–T3 are temperature sensors; Vt denotes the vacuum sensor; VFD represents the variable frequency drive measurement of power, rpm, current, and voltage.

	Measuring Quantities	Location and Label
1.	Pressure measurements	P1, P2, P3, Pt
2.	Flow rate measurements	F1
3.	Temperature measurements	T1, T2, T3
4.	Vacuum measurements	Vt
5.	Power, rpm, current, voltage	VFD true Command desk

The following measuring instruments were used during the tests: for the flow rate measurement, an ultrasonic flow meter KROHNE OPTISONIC 3400 (DN200, Duisburg, Germany) was used, with a measurement range of 0–1130 m³/h and a relative accuracy of $\pm 0.3\%$. It was installed at a distance of 10D; pressure was measured using a WIKA S-20 sensor (WIKA, Klingenberg am Main, Germany), with a range of 0–100 bar and an accuracy of 1%; temperature was measured using a resistance temperature detector (RTD) PT100 (WIKA, Klingenberg am Main, Germany), with a range of 0–100 °C and an accuracy class of 1 (± 0.2 K).

3. Results and Discussion

The main focus of the analysis was on the numerical prediction of the NPSH characteristics of the centrifugal pump, as it directly influences the onset and development of cavitation. CFD makes it possible to predict NPSH beyond the limits of experimental testing [10]. In addition, the basic energetic characteristics, such as power, head, and efficiency, were calculated. Mesh density could affects head and efficiency predictions [6]. The central part of the research addressed the CFD challenges in simulating the cavitation phenomena. Cavitation modeling requires resolving low-pressure regions precisely [34]. The analysis focused on the influence of turbulence model selection, computational mesh density, and the time-dependence of the simulations, all of which are key for accurately capturing locally low-pressure regions within the rotor. Mesh refinement yields larger shifts in NPSH curves than turbulence model choice [35]. The results confirm the complexity of numerically predicting NPSH, especially at lower flow rates, where flow conditions are more unstable and harder to capture using steady-state approaches. Transient simulations capture cavitation at part-load better than steady ones [35–37]. The analysis covers the pump’s operating range between 0.6 Q_{BEP} and 1.2 Q_{BEP} [38], which is broader than the range achievable through experimental measurements. Simulations allow you to analyze

points outside the measured area [5]. The limitations of the measurements were mainly related to the design of the test rig. This represents one of the key advantages of numerical simulations: the ability to analyze pump performance even under extreme or off-design operating conditions. The comparison of different turbulence models showed that the choice of the model has only a minimal impact on the prediction of cavitation characteristics, whereas mesh density has a significantly greater influence. A finer mesh allows for a better resolution of local flow phenomena and more accurate representation of low-pressure regions, where cavitation bubbles are more likely to form. The main challenge in the CFD-based prediction of the NPSH characteristics lies in lower flow rates. Part-load cavitation is highly unsteady [35]. During the measurements, the pump's operating range extended up to $1.1 Q_{BEP}$. Missing values, particularly in regions outside the available experimental data, were estimated by using the linear interpolation and extrapolation. That carries the risk of distorting non-linear head drop characteristics [34]. The overall operating range of the pump considered in this study spans from $0.6 Q_{BEP}$ to $1.2 Q_{BEP}$.

3.1. Energetic Characteristics

In all figures presenting the pump performance and NPSH characteristics (Figures 4–11), head and efficiency are shown in relative form (H_r and η_r), normalized to their values at the best efficiency point (BEP).

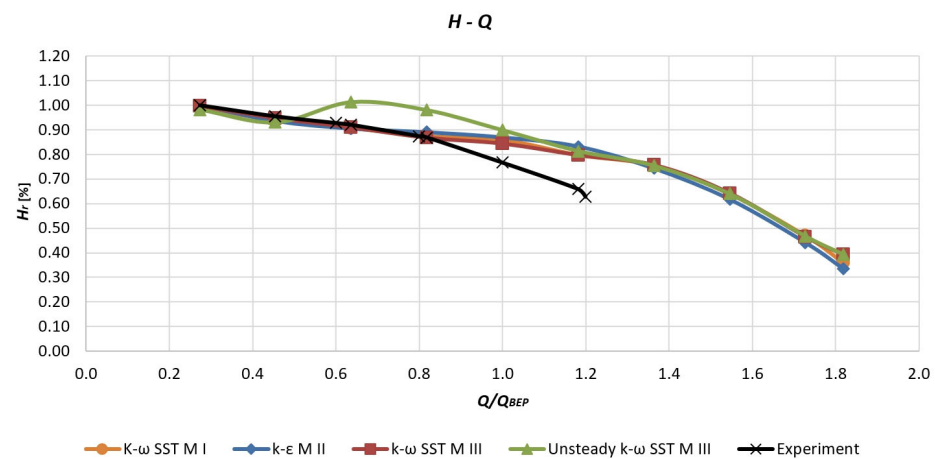


Figure 4. H - Q characteristics.

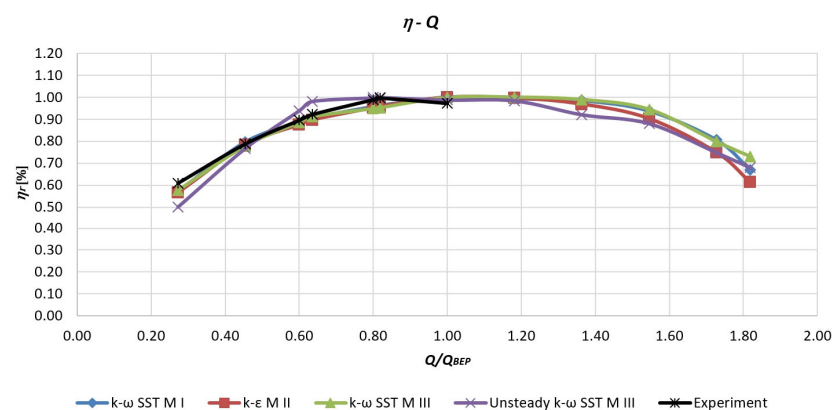


Figure 5. η_r - Q characteristics.

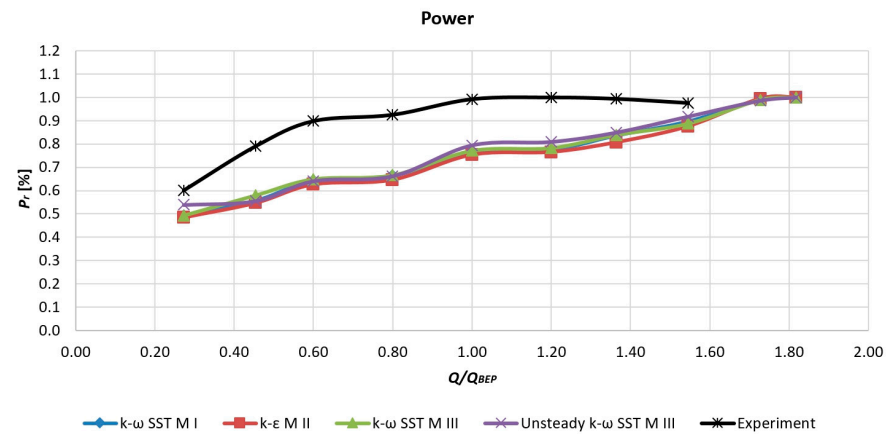


Figure 6. Power characteristics.

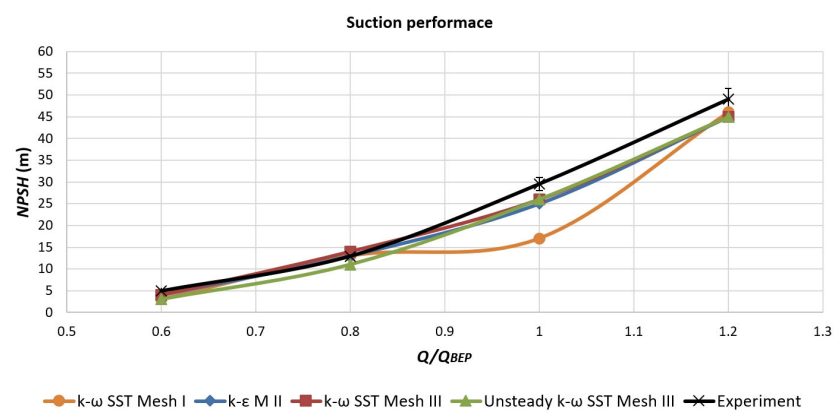


Figure 7. Suction performance.

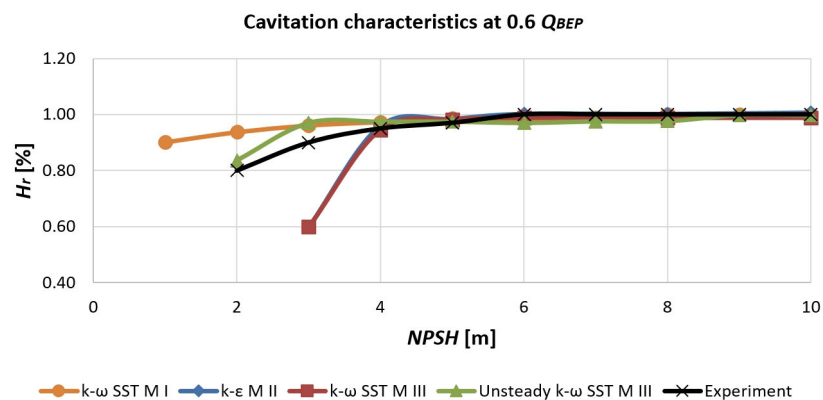


Figure 8. Cavitation characteristics at $0.6 Q_{BEP}$.

Figure 4 shows the comparison between the numerically and experimentally obtained H – Q characteristics of the centrifugal pump. As expected, the pump head decreases with increasing flow rate, following the typical shape of the curve. The simulation results align well with the experimental data, particularly near the best efficiency point (Q_{BEP}). A slight deviation appears around $0.7 Q_{BEP}$, where the unsteady calculation diverges slightly. Numerical results match well in the range from $1 Q_{BEP}$ to $1.8 Q_{BEP}$, while in the experimental data, the pump head is slightly lower between 1 and $1.2 Q_{BEP}$. The deviations in the numerical simulations arise primarily from the simplifications in the numerical model. Nevertheless, the numerical approach successfully captures the general shape of the H – Q curve, confirming its suitability for predicting the basic hydraulic behavior of the pump.

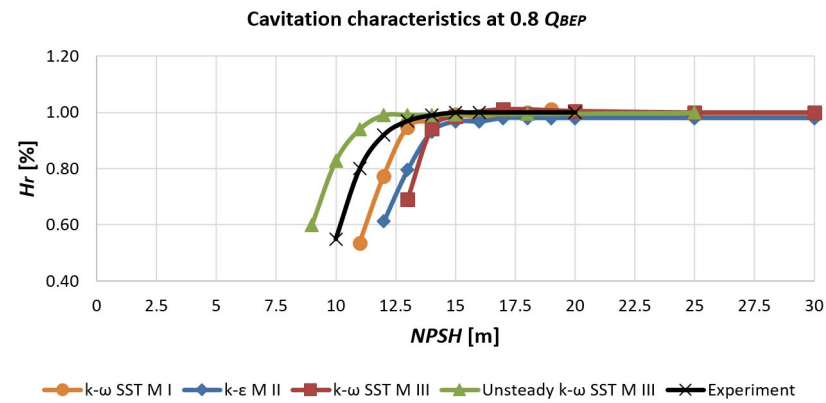


Figure 9. Cavitation characteristics at $0.8 Q_{BEP}$.

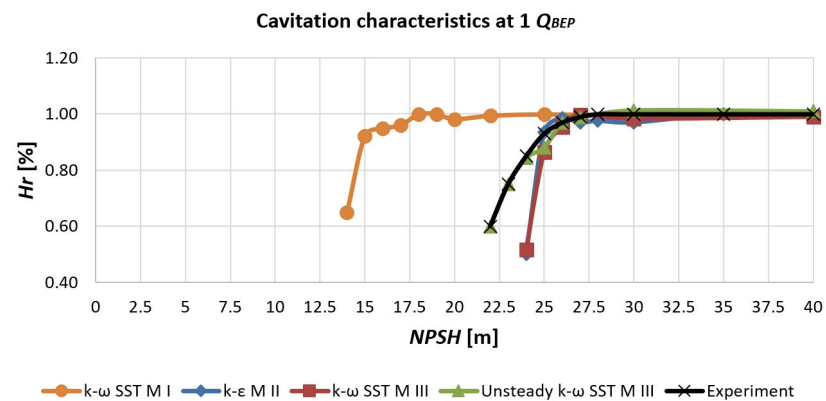


Figure 10. Cavitation characteristics at $1 Q_{BEP}$.

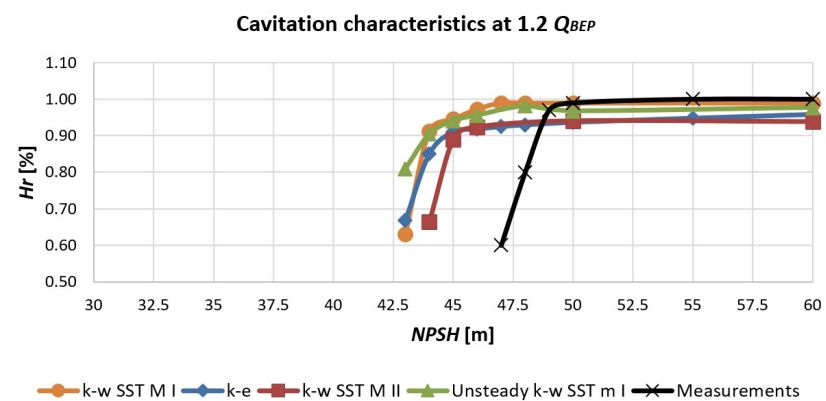


Figure 11. Cavitation characteristics at $1.2 Q_{BEP}$.

Figure 5 shows efficiency as a function of flow rate. The numerical results closely follow the experimental data in the region around Q_{BEP} , where the flow is hydraulically stable and the flow field is least disturbed. Deviations are somewhat more pronounced at lower flow rates ($0.25 Q_{BEP}$ to $0.8 Q_{BEP}$) and higher flow rates ($1.4 Q_{BEP}$ to $1.8 Q_{BEP}$), which may be due to the greater sensitivity of efficiency to the accuracy of energy capture within the pump. Efficiency errors grow due to unsteady losses and imperfect loss modeling [39–41].

Figure 6 shows a comparison of pump power as a function of flow rate. The CFD simulations exhibit a consistent trend of increasing power with increasing flow rate, which aligns with the expected behavior of the centrifugal pump. Compared to experimental results, the simulated power values are lower. CFD often underpredicts pump power [42,43]. These lower values are the result of several factors. The CFD model does not account

for mechanical losses, such as bearing and seal friction, and assumes idealized boundary conditions, which reduce the modeled flow losses. Similar findings can be observed in the literature [44–46]. Additionally, numerical dissipation occurs. It should be noted that the experimental pump power represents the mechanical (shaft) power, whereas the CFD-computed power corresponds to the rotor interaction power. In contrast, experimental power measurements typically reflect the entire system, including electromechanical and system-level losses, which are not considered in the CFD analysis. As a result, less energy is required in the simulation, leading to lower predicted power values. Nevertheless, the deviations remain acceptable from the perspective of assessing pump performance.

3.2. Cavitation Characteristics

Figure 7 shows suction performance or NPSH-Q characteristics obtained through the CFD simulations and experiments. The results demonstrate good agreement overall. A notable deviation occurs with the $k-\omega$ SST Mesh I case in the range from 0.9 to 1.1 Q_{BEP} , whereas at higher and lower flow rates, such deviations are not present. The CFD results indicate that the choice of the turbulence model has less impact on the NPSH prediction than mesh density and the selection of simulation time regime. The overall shape of the numerical curves follows the experimental trends, confirming the adequacy of the chosen approach for predicting the suction performance of the pump. When measuring the NPSH characteristics and generating NPSH curves, step sizes in full meters were used, in accordance with standard industrial practice, ensuring measurement repeatability. It is important that the available NPSH ($NPSH_A$) is at least 0.5 m higher than the required NPSH ($NPSH_R$) to prevent cavitation [3,47].

Figure 8 shows a comparison of cavitation curves between experimental and numerical results at a flow rate of 0.6 Q_{BEP} . Larger discrepancies are observed, particularly when using steady-state simulations, where more pronounced deviations occur. The unsteady simulation shows better agreement with experimental results, confirming the importance of including the time component for accurate prediction of cavitation phenomena at low flow rates. The differences between steady-state and unsteady results are most evident in the more pronounced drop in the pump head under steady-state conditions ($k-\omega$ SST Mesh III and $k-\epsilon$ Mesh II). NPSH values at a flow rate of 0.6 Q_{BEP} range between 3 and 4 m. The relative deviation compared to experimental data is less than 10%.

Figure 9 shows a comparison of cavitation curves between experimental and numerical results at a flow rate of 0.8 Q_{BEP} . Some deviations are noticeable, particularly in steady-state simulations and with the coarser mesh ($k-\omega$ SST Mesh III and $k-\epsilon$ Mesh II). NPSH values at a flow rate of 0.8 Q_{BEP} range between 11 and 14 m. The relative deviation compared to experimental data is less than 10%.

Figure 10 shows a comparison of cavitation curves between experimental and numerical results at a flow rate of 1 Q_{BEP} . A larger deviation is observed in the case of $k-\omega$ SST Mesh I, in which the NPSH value is as much as 9 m lower. Despite several repeated runs, the result did not change, and the issue was therefore attributed to the use of steady-state simulations, which fail to capture the actual flow field in the model (vortices, flow separation, etc.). In other cases, no significant deviations were observed. The most accurate agreement with the experimental data was achieved using unsteady simulation, where only a minor deviation was observed at the 25 m value. Slightly larger deviations are also observed in the steady-state simulations ($k-\omega$ SST Mesh III and $k-\epsilon$ Mesh II). NPSH values at a flow rate of 1 Q_{BEP} range between 17 and 26 m. The error compared to measurements is less than 5%, except in the case of scenario 1 ($k-\omega$ SST Mesh I).

Figure 11 shows a comparison of cavitation curves between experimental and numerical results at a flow rate of 1.2 Q_{BEP} . It should be noted that the experimental value in this

case was determined through the previously mentioned extrapolation. At this flow rate, the CFD results show minimal deviations among themselves, with values ranging between 45 m and 46 m, while the experimental value is 49 m.

When considering the ISO 9906 measurement uncertainty of $\pm 5.5\%$, the transient simulation (Mesh III—Unsteady) and the finest steady-state mesh (Mesh III) show the best overall agreement with the experimental results. In these cases, two to three operating points fall entirely within the ISO tolerance band, and the remaining deviations are within the ISO 9906 tolerance limits, i.e., within $\pm 5.5\%$ uncertainty. The deviations summarized in Table (Table 5) were evaluated with respect to these ISO 9906 tolerance limits and represent the NPSH₃ deviation after including the defined measurement uncertainties.

Table 5. Percentage deviations of NPSH₃ between CFD and experiment for different meshes and flow rates.

Q_{BEP}	Percentage Deviation [%]			
	Mesh I ($k-\omega$ SST)	Mesh II ($k-\epsilon$)	Mesh III ($k-\omega$ SST)	Mesh III—Unsteady ($k-\omega$ SST)
0.6	−14.5	−14.5	−14.5	0
0.8	2.5	17.5	0	−9.5
1	−29.5	0	0	0
1.2	−0.5	−2.5	−2.5	−2.5

Larger discrepancies are observed for the coarser steady-state meshes, particularly for Mesh 1 at $1.0 Q_{BEP}$ and Mesh II at $0.8 Q_{BEP}$. Nevertheless, for the finer meshes (Mesh III and Mesh III—Unsteady), the deviations remain well within the tolerance limits. For the unsteady $k-\omega$ SST case (Mesh III—Unsteady), the mean absolute deviation of the NPSH₃ values across all flow rates is 3.0%, with a maximum deviation of 9.5%. This confirms the superior predictive accuracy and consistency of the unsteady approach compared to the steady-state models. When considering all meshes and operating points, the overall mean absolute deviation amounts to approximately 6.9%, indicating a generally good agreement between numerical predictions and experimental measurements across the entire dataset.

The observed deviations between numerical and experimental results can be attributed to several sources of error. The main sources include simplifications in the numerical model, idealized boundary conditions, and the neglect of mechanical losses such as bearing and seal friction. Additional uncertainty arises from the mesh discretization and turbulence model selection, which influence the local pressure distribution near the blade surfaces and therefore the predicted cavitation onset. A combined influence of these factors explains the remaining differences between the simulated and measured NPSH characteristics, especially at low flow rates where recirculation and unsteady vortical structures dominate the flow field.

Figure 12 shows the pressure distribution on the impeller at the 3% head drop ($H_3\%$). It can be observed that the low-pressure regions correspond to the locations of the cavitation bubbles. Once again, the pressure field is captured more accurately using unsteady simulations.

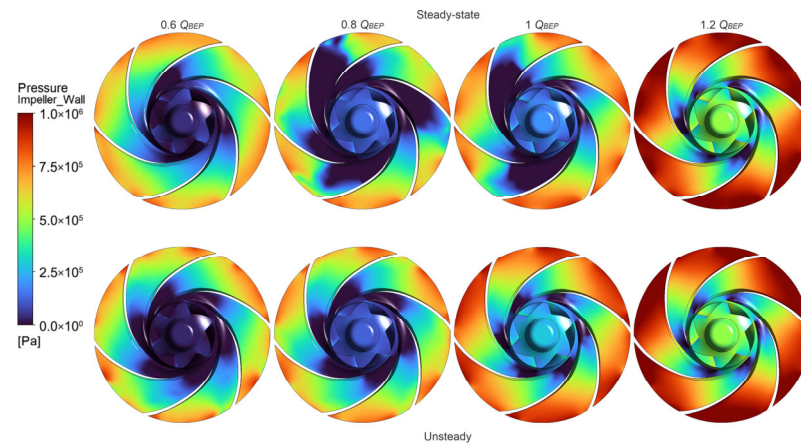


Figure 12. Pressure on impeller at 3% head drop $H_3\%$.

Figure 13 shows steady-state CFD results of cavitation volume fraction at a value of 0.5, covering the flow range between $0.6 Q_{BEP}$ and $1.2 Q_{BEP}$ and head drops of $H_0\%$, $H_1\%$, and $H_3\%$. It is observed that the cavitation bubbles are unevenly distributed, with noticeable asymmetry and instability [14,38,48] appearing between the rotor blade gaps. The cavitation volume fraction threshold of $VF = 0.5$ was selected to represent the interface between vapor- and liquid-dominated regions, following common practice in CFD post-processing of cavitating flows [28,29,32].

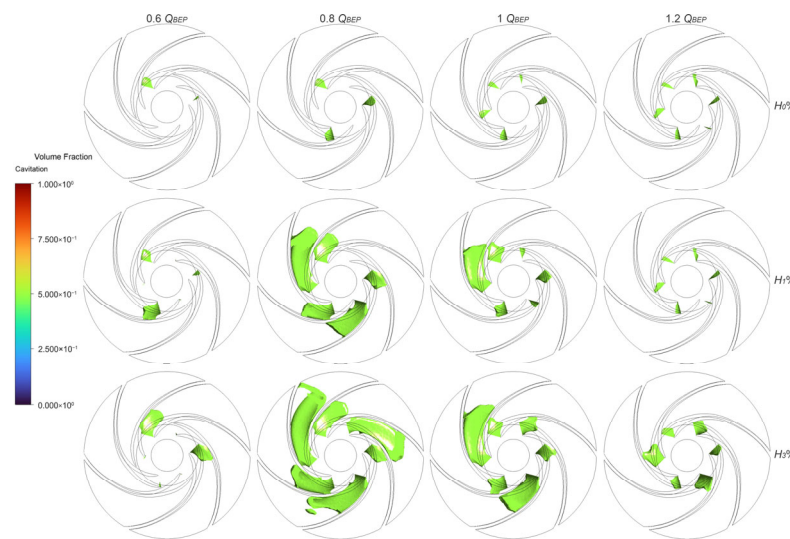


Figure 13. Volume fraction 0.5 (steady-state).

At the $H_0\%$ head drop, the difference in the cavitation bubble shape is minimal. However, at the $H_3\%$ head drop, more pronounced variations in bubble shape begin to emerge particularly between the cases $0.8 Q_{BEP}$ and $1 Q_{BEP}$, and $0.6 Q_{BEP}$ and $1.2 Q_{BEP}$. It is evident that there is less asymmetry in the shape of the cavitation bubbles at higher flow rates. Similar patterns are observed at the $H_3\%$ head drop as in the previous case, with reduced asymmetry in cavitation bubble shapes at higher flow conditions.

Figure 14 shows transient CFD results of cavitation volume fraction at a value of 0.5, across the flow range from $0.6 Q_{BEP}$ to $1.2 Q_{BEP}$ and head drops of $H_0\%$, $H_1\%$, and $H_3\%$. The cavitation bubbles appear evenly distributed, with no signs of asymmetry or instability [14,38,48] between the rotor blade gaps.

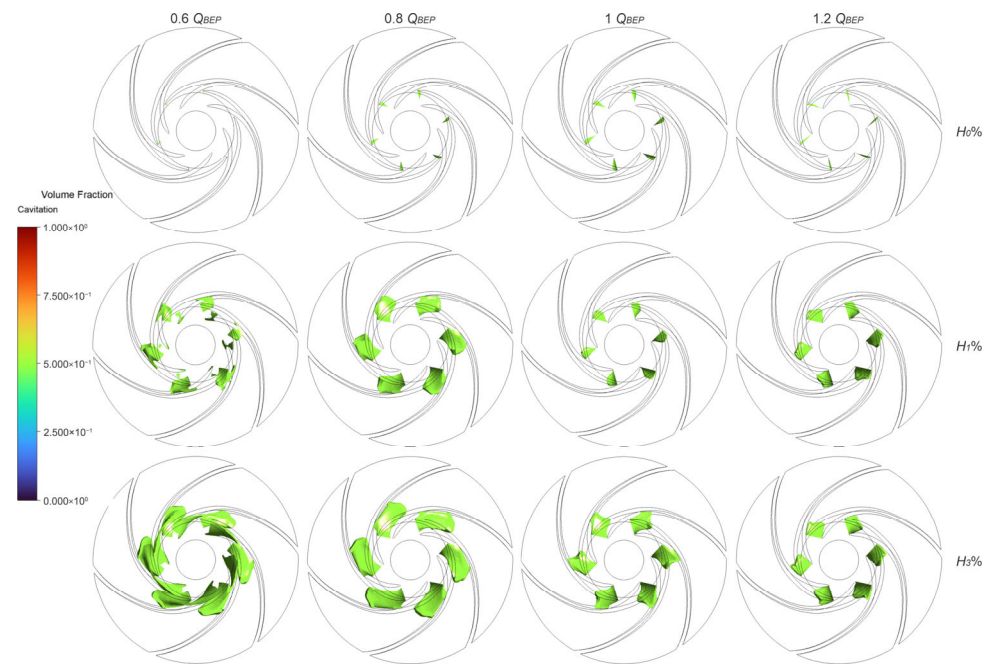


Figure 14. Volume fraction 0.5 (unsteady).

At the $H_0\%$ head drop, the differences in the bubble shape are minimal. For all flow rates and head drops, a consistent increase in cavitation volume is observed as the head decreases. Notably, low flow conditions lead to a significant increase in cavitation, as operating far from the Q_{BEP} point generates larger low-pressure regions on the suction sides of the blades, resulting in more extensive cavitation bubbles.

Figure 15 shows the density distribution on the XY plane (shroud outlet front wall) at a 3% head drop ($H_3\%$). It can be observed that the steady-state simulation exhibits significant asymmetry in the cavitation bubbles, with certain regions (steady-state $0.8 Q_{BEP}$) showing near-complete blockage of the gap between blades from the suction vane edge to the discharge vane edge of the adjacent blade. Once again, the unsteady simulation provides the most accurate representation of the actual flow conditions.

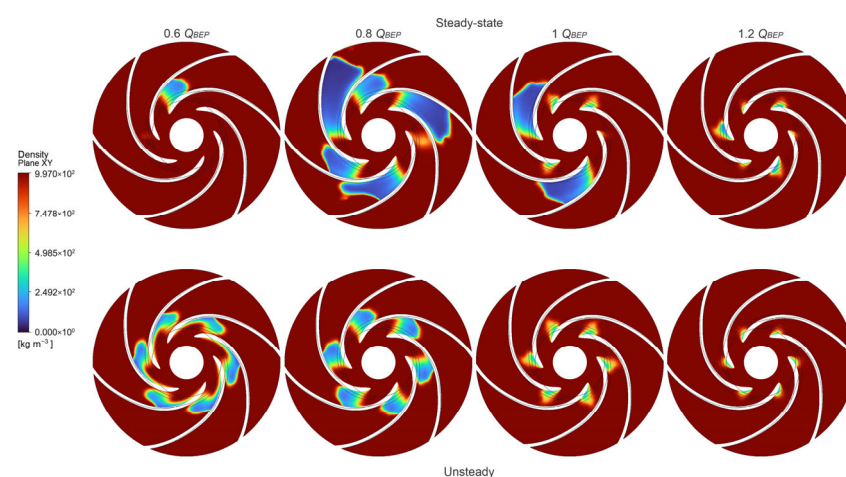


Figure 15. Density on impeller at 3% head drop $H_3\%$.

Next, we examined the volume of the cavitation bubbles. The total volume of the rotor was calculated to be 1036 cm³. This was followed by the calculation of the cavitation bubbles volumes using ANSYS CFD-Post. Table 6 presents the computed volumes of the cavitation bubbles at a 3% head drop ($H_3\%$). The data covers the range from $0.6 Q_{BEP}$ to

1.2 Q_{BEP} , using both steady-state and transient approaches. The results show a significant increase in the cavitation volume with decreasing flow rate, confirming the system's higher sensitivity to cavitation in low-flow regions. The unsteady simulation consistently predicts the cavitation volumes more accurately than the steady-state simulation, highlighting the advantages of the time-dependent approach in capturing the dynamics of local pressure drop and the expansion of the cavitation bubbles. The largest difference between the two approaches is observed at a flow rate of 0.6 Q_{BEP} , where the cavitation volume in the unsteady simulation exceeds the steady-state value by more than six times. This can be attributed to the fact that steady-state models tend to underestimate cavitation development, especially in regimes with significant pressure fluctuations. The trend of decreasing cavitation with increasing flow rate aligns with the expected physical behavior, as the pressure on the suction side of the blades increases, suppressing cavitation formation.

Table 6. Volume of cavitation bubble (VF = 0.5).

	Steady-State cm ³	Unsteady cm ³	Steady-State %	Unsteady %
0.6 Q_{BEP}	3.17	20.68	0.3	2
0.8 Q_{BEP}	39.97	16.77	3.9	1.6
1 Q_{BEP}	19.61	6.46	1.9	0.6
1.2 Q_{BEP}	4.8	3.48	0.5	0.3

Figure 16 presents the cavitation bubble volume at 1 Q_{BEP} . The vapor volume fraction was computed in ANSYS CFD-Post using the built-in Volume Fraction variable for the multiphase field, and subsequently integrated over the computational domain to obtain the total cavitation volume. The steady-state simulation predicts a cavitation bubble volume that is 216% larger compared to the unsteady solution.

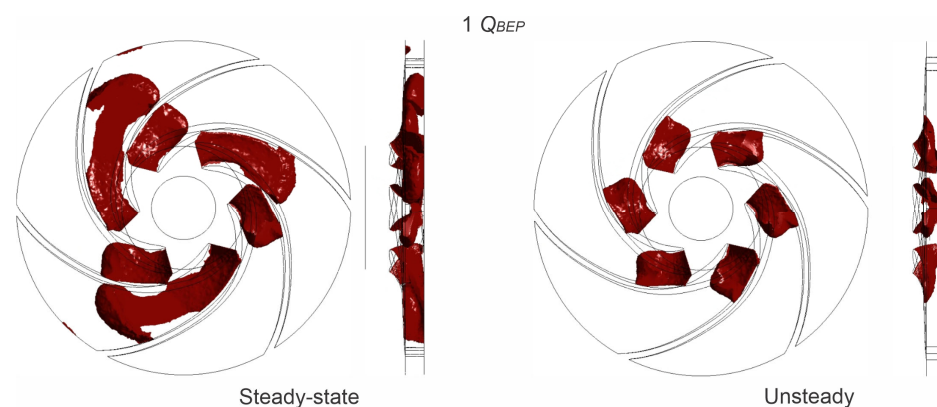


Figure 16. Volume of the cavitation bubble 3% head drop and 1 Q_{BEP} .

Figure 17 shows the convergence of efficiency and pump head in steady-state (left) and unsteady (right) simulations for flow rates ranging from 0.6 Q_{BEP} to 1.2 Q_{BEP} . The pressure reduction was performed stepwise (P1–P13), although in some cases, fewer than 13 pressure steps were applied, and the absolute pressure values and step sizes differed between analyses. Accordingly, the figure illustrates how the pump head and efficiency evolve with decreasing inlet pressure. In steady-state simulations, the pressure was systematically reduced after at least 1000 iterations, which was sufficient for the stabilization of the results. However, significant fluctuations in efficiency and head values were observed, which are attributed to numerical instabilities inherent in the steady-state approach. These instabilities are particularly pronounced in flow regimes with strong vortex activity, as the steady-state solver forces the flow into a condition that is not physically realistic.

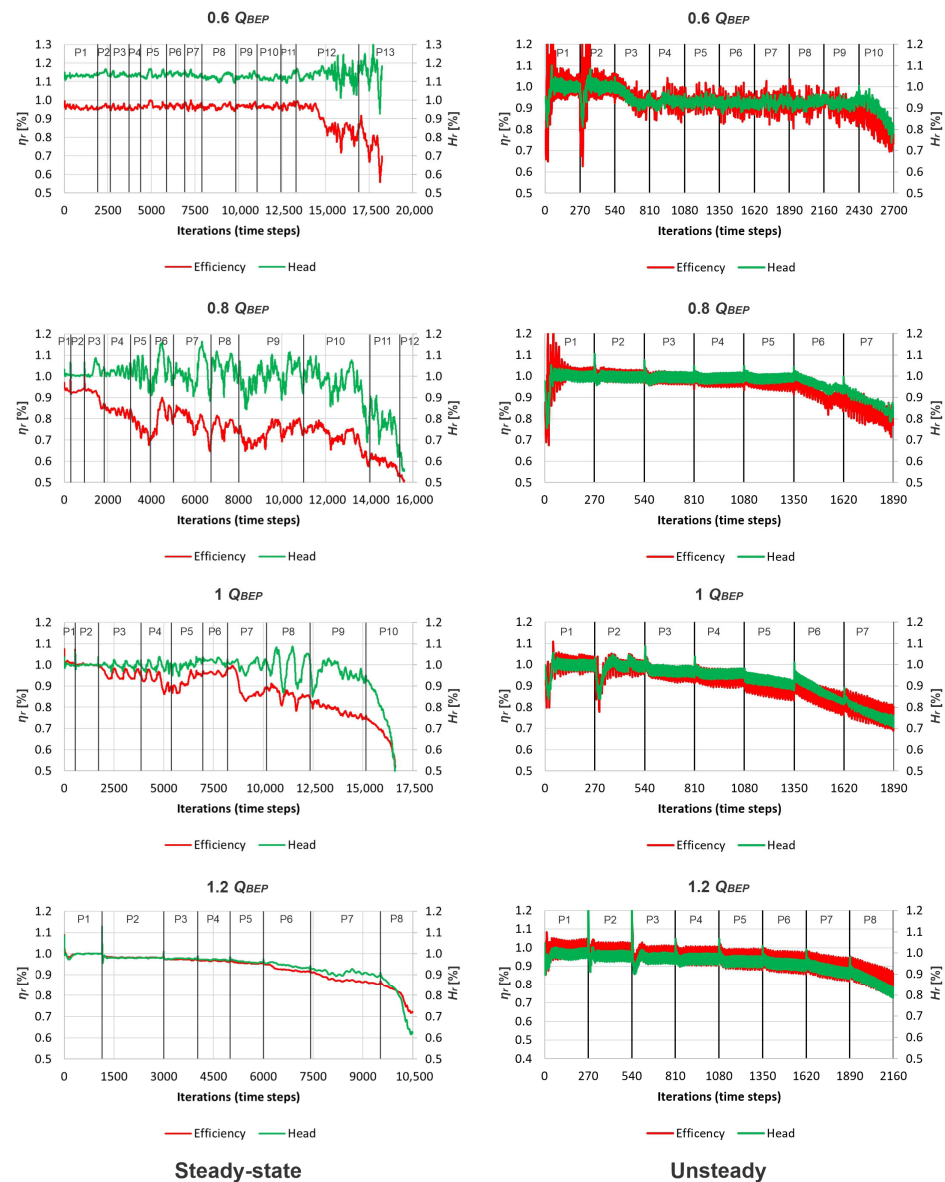


Figure 17. Numerical prediction of pump energy efficiency and total head as a function of pressure drop.

Due to these fluctuations, it is not possible to reliably determine the 3% head drop point ($H_{3\%}$), especially at lower flow rates between 0.6 Q_{BEP} and 0.8 Q_{BEP} . The unstable behavior at 0.6 Q_{BEP} is strongly influenced by inlet recirculation, a phenomenon well documented in centrifugal pumps. Recirculation at the impeller eye generates vertical backflow structures that destabilize the flow field and directly affect the NPSH prediction. If such effects are not considered, the resulting NPSH curves may be of poor quality or even misleading. In these cases, the head drop is either not clearly visible or occurs abruptly, making interpretation more difficult. In the flow range from 1 Q_{BEP} to 1.2 Q_{BEP} , a more noticeable head drop occurs at specific pressure values, but it remains difficult to pinpoint exactly when it happens, as fluctuation amplitudes often exceed 30% and are not symmetrical. In practice, this makes it difficult to determine an appropriate average or the correct moment to stop the simulation. Frequently, a recurring oscillation pattern emerges after a certain number of iterations, where the head no longer changes regardless of further iterations.

These oscillations are also associated with previously observed patterns in the volume fraction field. These trends are consistent with the vapor volume fraction distributions shown in Figures 13 and 14, where strong oscillations correspond to unstable cavitation clouds in steady-state simulations, whereas unsteady results yield more stable structures. Strong oscillations lead to asymmetric and unstable cavitation distributions. In contrast, unsteady simulations eliminate these issues. Although fluctuations are still present as expected when the time component is included, it is possible to determine accurate average values for both efficiency and head. The amplitude of fluctuations in unsteady simulations remains within 10%, which greatly simplifies the averaging process. This highlights the fundamental difference between steady-state and unsteady simulations, where only the latter provide physically meaningful results for the NPSH determination.

Unsteady simulations were run for at least three revolutions (270 time steps). When necessary, an additional three revolutions were carried out in the event of a strong drop in pump head or energy efficiency. An additional three revolutions were added when a drop in one of the monitored variables was detected. For an accurate NPSH determination, a total of six revolutions was sufficient in all cases. At $0.6 Q_{BEP}$, slightly increased fluctuations in efficiency were observed, though still considerably smaller than in the steady-state case. Although inlet recirculation can also be detected in unsteady simulations at $0.6 Q_{BEP}$, its influence is much weaker and does not prevent a reliable determination of average head and efficiency values. For other flow rates (0.8 – $1.2 Q_{BEP}$), efficiency fluctuations were small and stable, as expected in such transient analyses. At all flow rates, the pump head showed stable and predictable behavior with minimal variation.

For the evaluation of the 3% head drop ($H_3\%$), the following pressure levels were considered: steady-state $0.6 Q_{BEP}$ corresponds to pressure P10, $0.8 Q_{BEP}$ corresponds to pressure P13, $1.0 Q_{BEP}$ corresponds to pressure P8, $1.2 Q_{BEP}$ corresponds to pressure P6. For unsteady simulations, $0.6 Q_{BEP}$ corresponds to pressure P9, $0.8 Q_{BEP}$ corresponds to pressure P6, $1.0 Q_{BEP}$ corresponds to pressure P4, $1.2 Q_{BEP}$ corresponds to pressure P5.

An important contribution of this study is the warning that the head and efficiency curves must be interpreted with caution when cavitation is present. To reduce the risk of misinterpretation, the monitoring of the cavitation structures through volume fraction fields (as illustrated in Figures 13 and 14) is recommended alongside the numerical convergence of global performance parameters. This combined approach ensures a more reliable identification of the NPSH and improves the robustness of the analysis.

Each pressure level (P1–P13) corresponds to a discrete inlet pressure applied during the cavitation test sequence, starting from the highest inlet pressure (non-cavitating condition) and progressively decreasing it until the 3% head-drop point was reached. The number and magnitude of pressure steps varied with the flow rate:

- At $0.6 Q_{BEP}$, the inlet pressure was reduced in uniform steps of about 10 kPa;
- At $0.8 Q_{BEP}$, the first two reductions were 50 kPa followed by 10 kPa steps;
- At $1.0 Q_{BEP}$ and $1.2 Q_{BEP}$, the first two reductions were 50 kPa, then 25 kPa, and finally 10 kPa as cavitation approached.

With the gradual reduction in the inlet pressure, the available suction head of the system is simulated. At the beginning of the NPSH curve determination, larger pressure decrements are applied to accelerate the process, while near the critical cavitation point, smaller increments are required to capture the onset of head drop with sufficient resolution (“fine tuning”). The adjustment of pressure steps therefore depends on the operating regime. Because of these differences in convergence behavior and cavitation sensitivity, a universal or fully automated procedure for pressure reduction cannot be defined—each operating condition requires an individually adapted approach.

These stepwise pressure decreases correspond to the symbols P1–P13 shown in Figure 17 and illustrate how the pump head and efficiency evolve with the gradual reduction in the available suction head in both steady-state and transient simulations.

4. Conclusions

This study confirmed that the CFD simulations provide a valuable insight into the cavitation characteristics of the pump and enable an accurate determination of the NPSH characteristics within the range from $0.6 Q_{BEP}$ to $1.2 Q_{BEP}$. An additional advantage is the ability to perform multiple simulations in a shorter time, compared to experiments covering operating ranges that are not always experimentally accessible. This study also confirmed that a reliable numerical prediction of the cavitation characteristics in a centrifugal pump requires the use of a sufficiently fine mesh, an appropriate turbulence model, and, most importantly, a time-dependent (unsteady) approach. However, such an approach also demands the use of powerful computing systems with high processing capacity. Steady-state computations exhibited large fluctuations that often exceeded 30%, which made a robust identification of the 3% head drop difficult, whereas unsteady computations produced stable mean values of the head and energy efficiency with fluctuation amplitudes within about 10%.

In estimating the $NPSH_3$ value, the most accurate results were obtained using the unsteady simulation combined with Case 3 ($k-\omega$ SST model Mesh III), where deviations from experimental data were minimal. In this case, the flow conditions within the pump were well resolved; the pressure fields were accurately captured, as evident from the volume fraction distribution; and the NPSH characteristics values were precisely predicted.

The results further showed that the steady-state approach systematically underestimates the cavitation volume, especially in low-flow regimes where pronounced low-pressure zones dominate on the suction side of the blades. This observation is consistent with the unstable trends in the head and energy efficiency and with the asymmetric cavitation structures in the steady-state results, while the unsteady results match the vapor volume fraction fields shown in Figures 13 and 14.

Simulations confirmed that the resolution in the space and time of the flow phenomena significantly affects the reliability of cavitation predictions. Unsteady models provide a clearer view of the formation and development of the cavitation bubbles and their spatial distribution across the impeller. Moreover, it was shown that improved meshing plays a key role in accurately capturing local pressure minima, which is crucial for reliably determining the onset of cavitation. For engineering interpretation, the head and energy efficiency curves should be assessed together with the cavitation indicators, for example, the vapor volume fraction monitored during the pressure stepping procedure, to reduce the risk of misidentifying the 3% head drop and the associated $NPSH_3$. In summary, the choice of the numerical approach depends heavily on the required level of accuracy and available computational resources. For basic assessments, steady-state models remain usable; however, when operating near the cavitation limit or outside optimal conditions, the use of an unsteady approach is recommended, as it offers greater result stability and more reliable capture of real, time-dependent flow phenomena. When inlet recirculation is present at $0.6 Q_{BEP}$, steady-state computations are particularly unreliable, so an unsteady approach is essential for a trustworthy NPSH assessment.

Although the presented results confirm the reliability of the unsteady CFD approach for predicting NPSH characteristics, several limitations must be acknowledged. The numerical model assumes idealized flow conditions and simplified cavitation behavior, which may not fully capture transient vapor–liquid interactions under severe cavitation. In addition, the cavitation model is based on homogeneous two-phase flow and a constant bubble

diameter, which can affect the accuracy of local pressure predictions. Experimental measurements were performed on an industrial test rig rather than in a precision laboratory setup, leading to additional uncertainties related to sensor calibration and flow non-uniformity. Future work should focus on employing adaptive mesh refinement and hybrid turbulence models (SAS and DES) to better resolve transient vortical structures and on establishing an automated link between the vapor volume fraction and the 3% head drop criterion to enhance cavitation prediction robustness. Extending the validation to various pump geometries and operating speeds would further improve the general applicability of the proposed methodology.

In future experimental work, advanced diagnostic techniques such as synchrotron X-ray-based multiphase flow imaging could be employed to provide high-resolution validation data for transient cavitation dynamics, as demonstrated in recent studies on multiphase streamflow and densitometry using synchrotron radiation [49].

For future research, it would be worthwhile to explore the application of hybrid turbulence models, such as SAS and DES, which combine the advantages of RANS and LES approaches and enable a more accurate modeling of turbulent structures under demanding flow conditions. In addition, future work should investigate the quantitative link between the computed cavitation bubble volume and NPSH₃, defined by the 3% head drop criterion, with the aim of developing an automatic indicator that combines global performance parameters with local cavitation metrics.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data presented in this study are available on request from the author due to by agreement, the property of the company where most of the measurements were carried out. The data are part of ongoing development for further applied use.

Acknowledgments: I would like to thank my supervisor Andrej Lipej for all the professional support, guidance, and valuable advice during the research.

Conflicts of Interest: The author declares no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

CFD	Computational Fluid Dynamics	
e_H	Measurement uncertainty of head	
e_n	Measurement uncertainty of rotation speed	
e_Q	Measurement uncertainty of flow rate	
ε	Dissipation of turbulence kinetic energy	m^2/s^3
f_i	Vector of the mass density of the external forces	M/kg
FSI	Fluid–structure interaction	-
g	Acceleration due to gravity	m/s^2
H	Pump head	m
$H_{loss,s}$	Hydraulic losses	m
H_r	relative head (ratio H/H_{BEP})	%
HRS	High Resolution Scheme	-
h_s	Water level	m
k	Turbulence kinetic energy	m^2/s^2
μ	Kinematic viscosity	m^2/s
n	Rotational speed	rpm

NPSH	Net Positive Suction Head	m
NPSH ₃	Net Positive Suction Head, the 3% head drop criterion	m
NPSH _A	Net Positive Suction Head available	m
NPSH _R	Net Positive Suction Head required	m
NPSH _i	Net Positive Suction Head, the critical limit	m
P_s	Input power	W
p_0	Pressure at the reference point	Pa
$p_{1,inlet}$	Pressure at inlet	Pa
$p_{t,in}$	Total pressure at inlet	Pa
$p_{t,out}$	Total pressure at outlet	Pa
p_a	Atmospheric pressure	Pa
p_v	Vapor pressure	Pa
Q_{BEP}	Best efficiency point	-
Q	Flow rate	m ³ /s
RANS	Reynolds-averaged Navier–Stokes equations	-
σ_v	cavitation number	-
σ_{vi}	Initial cavitation number	-
SST	Shear Stress Transport	-
T	Torque	N/m
t	Time	s
T_f	Fluid temperature	°C
τ	Viscous stress tensor	N/m ²
u_{ij}	Velocity	m/s
u_τ	Friction velocity	m/s
v	Velocity	m/s
v_0	Reference velocity	m/s
V	Velocity	m/s
x_{ij}	Position vector	m
y^+	Non dimensional number	-
ω	Dissipation of turbulent kinetic energy	m ² /s ³
η	Dynamic viscosity	N·s/m ²
η	Efficiency	-
η_r	Relative efficiency (ratio η/η_{BEP})	%
z (blades)	Number of blades	-
z	Elevation head	m

References

1. Napierała, M.A. Study on Improving Economy Efficiency of Pumping Stations Based on Tariff Changes. *Energies* **2022**, *15*, 799. [\[CrossRef\]](#)
2. Ješe, U. *Numerical Study of Pump-Turbine Instabilities: Pumping Mode off-Design Conditions*; Université Grenoble Alpes: Grenoble, France, 2015.
3. ISO 9906; Rotodynamic Pumps-Hydraulic Performance Acceptance Tests-Grades 1, 2 and 3, 2nd ed. International Organization for Standardization: Geneva, Switzerland, 2012.
4. Ocepek, M.; Lipej, A. Numerical and experimental analysis of guide vane pivot torque prediction in the reversible pump-turbine. *Proc. Inst. Mech. Eng. Part A J. Power Energy* **2024**, *239*, 284–294. [\[CrossRef\]](#)
5. Pei, J.; Osman, M.K.; Wang, W.; Appiah, D.; Yin, T.; Deng, Q. A Practical Method for Speeding up the Cavitation Prediction in an Industrial Double-Suction Centrifugal Pump. *Energies* **2019**, *12*, 2088. [\[CrossRef\]](#)
6. Pei, J.; Osman, M.K.; Wang, W.; Yuan, J.; Yin, T.; Appiah, D. Unsteady flow characteristics and cavitation prediction in the double-suction centrifugal pump using a novel approach. *Proc. Inst. Mech. Eng. Part A* **2019**, *234*, 283–299. [\[CrossRef\]](#)
7. Cui, B.; Chen, J. Visual experiment and numerical simulation of cavitation instability in a high-speed inducer. *Proc. Inst. Mech. Eng. Part A J. Power Energy* **2020**, *234*, 470–480. [\[CrossRef\]](#)
8. Ennouri, M.; Kanfoudi, H.; Taher, A.B.H.; Zgolli, R. Numerical flow simulation and cavitation prediction in a centrifugal pump using an SST-SAS turbulence model. *J. Appl. Fluid Mech.* **2019**, *12*, 25–39. [\[CrossRef\]](#)

9. Wu, D.; Ren, Y.; Mou, J.; Gu, Y.; Jiang, L. Unsteady Flow and Structural Behaviors of Centrifugal Pump under Cavitation Conditions. *Chin. J. Mech. Eng.* **2019**, *32*, 17. [\[CrossRef\]](#)
10. Lorusso, M.; Capurso, T.; Torresi, M.; Fortunato, B.; Fornarelli, F.; Camporeale, S.M.; Monteriso, R. Efficient CFD evaluation of the NPSH for centrifugal pumps. *Energy Procedia* **2017**, *126*, 778–785. [\[CrossRef\]](#)
11. Mousmoulis, G.; Anagnostopoulos, J.; Papantonis, D. A review of experimental detection methods of cavitation in centrifugal pumps and inducers. *Int. J. Fluid Mach. Syst.* **2019**, *12*, 71–88. [\[CrossRef\]](#)
12. Rakibuzzaman, M.; Kim, K.; Suh, S.H. Numerical and experimental investigation of cavitation flows in a multistage centrifugal pump. *J. Mech. Sci. Technol.* **2018**, *32*, 1071–1078. [\[CrossRef\]](#)
13. Lu, J.; Yuan, S.; Luo, Y.; Yuan, J.; Zhou, B.; Sun, H. Numerical and experimental investigation on the development of cavitation in a centrifugal pump. *Proc. Inst. Mech. Eng. Part E* **2014**, *230*, 171–182. [\[CrossRef\]](#)
14. Hatano, S.; Kang, D.; Kagawa, S.; Nohmi, M.; Yokota, K. Study of cavitation instabilities in double-suction centrifugal pump. *Int. J. Fluid Mach. Syst.* **2014**, *7*, 94–100. [\[CrossRef\]](#)
15. Xu, Y.; Tian, J.; Wang, Z.; Zhang, J.; Li, S.; Yan, Y.; Ge, M. A comprehensive study on the flow field of cylindrical cavitation nozzle jet under different turbulence models. *Ocean Eng.* **2025**, *315*, 119596. [\[CrossRef\]](#)
16. Apte, D.; Ge, M.; Zhang, G.; Coutier-Delgosha, O. Numerical investigation of three-dimensional effects of hydrodynamic cavitation in a Venturi tube. *Ultrason. Sonochem.* **2024**, *111*, 107122. [\[CrossRef\]](#)
17. Ge, M.; Sun, C.; Zhang, G.; Coutier-Delgosha, O.; Fan, D. Combined suppression effects on hydrodynamic cavitation performance in Venturi-type reactor for process intensification. *Ultrason. Sonochem.* **2022**, *86*, 106035. [\[CrossRef\]](#)
18. Fetting, C. *The European Green Deal*; ESDN Report; ESDN Office: Vienna, Austria, 2020; Volume 53.
19. Vuković, Ž.; Halkijević, I. Kavitacija u centrifugalnim crpkama. *Gradjevinar* **2011**, *63*, 11–17.
20. Poullikkas, A.; Turton, R.K. Two Phase Flow and Cavitation in Centrifugal Pump—A Theoretical and Experimental Investigation. Master's Thesis, Loughborough University, Loughborough, UK, 1992.
21. Hydraulic Institute Standards. *Rotodynamic Pumps Guideline for NPSH Margin*; Hydraulic Institute: Parsippany, NJ, USA, 2012; ISBN 978-1-935762-10-2.
22. Lipej, A.; Mitruševski, D. Numerical prediction of inlet recirculation in pumps. *Int. J. Fluid Mach. Syst.* **2016**, *9*, 277–286. [\[CrossRef\]](#)
23. Michel, J.M. Introduction to Cavitation and Supercavitation. 2001. Available online: <https://apps.dtic.mil/sti/html/tr/ADP012072/> (accessed on 16 September 2025).
24. Menter, F.R.; Kuntz, M.; Langtry, R. Ten Years of Industrial Experience with the SST Turbulence Model. *Turbul. Heat Mass Transf.* **2003**, *4*, 625–632.
25. Kato, M.; Launder, B. The Modelling of Turbulent Flow Around Stationary and Vibrating Square Cylinders. 1993. Available online: <https://cir.nii.ac.jp/crid/1571135649542053888> (accessed on 16 September 2025).
26. Ansys Inc. *ANSYS CFX-Solver Theory Guide*; ANSYS Inc.: Canonsburg, PA, USA, 2020.
27. Ansys Inc. *ANSYS FLUENT Theory Guide*; ANSYS Inc.: Canonsburg, PA, USA, 2020; Volume 20.
28. Zwart, P.J.; Gerber, A.G.; Belamri, T. A two-phase flow model for predicting cavitation dynamics. In Proceedings of the 5th International Conference on Multiphase Flow, Yokohama, Japan, 30 May–3 June 2004; Volume 152, p. 152.
29. Mejri, I.; Bakir, F.; Rey, R.; Belamri, T. Comparison of Computational Results Obtained From a Homogeneous Cavitation Model With Experimental Investigations of Three Inducers. *J. Fluids Eng.* **2006**, *128*, 1308–1323. [\[CrossRef\]](#)
30. Shi, J.; Zhao, Z.; Song, W.; Jin, Y.; Lu, J. Numerical simulation analysis of flow characteristics in the cavity of the rotor-stator system. *Eng. Appl. Comput. Fluid Mech.* **2022**, *16*, 501–513. [\[CrossRef\]](#)
31. Sedlar, M.; Sputa, O.; Komárek, M. CFD Analysis of Cavitation Phenomena in Mixed-Flow Pump. *Int. J. Fluid Mach. Syst.* **2012**, *5*, 18–29. [\[CrossRef\]](#)
32. Cui, B.; Han, X.; An, Y. Numerical Simulation of Unsteady Cavitation Flow in a Low-Specific-Speed Centrifugal Pump with an Inducer. *J. Mar. Sci. Eng.* **2022**, *10*, 630. [\[CrossRef\]](#)
33. Štefanič, M.; Lipej, A. Numerical Investigation and Experimental Validation of Different Air Flow Conditioners using Up-and Down-Stream Pipeline. *J. Appl. Fluid Mech.* **2023**, *16*, 983–991. [\[CrossRef\]](#)
34. Orlandi, F.; Montorsi, L.; Milani, M. Cavitation analysis through CFD in industrial pumps: A review. *Int. J. Thermofluids* **2023**, *20*, 100506. [\[CrossRef\]](#)
35. Shen, Z.; Wang, C.; Zhang, J.; Qiu, S.; Lin, R. Study on Part-Load Cavitation in High-Specific-Speed Centrifugal Pump. *Water* **2024**, *16*, 2180. [\[CrossRef\]](#)
36. Lipej, A. Challenges in the Numerical Analysis of Centrifugal Pumps. *Eng. Technol. Appl. Sci. Res.* **2022**, *12*, 8217–8222. [\[CrossRef\]](#)
37. Lipej, A. Analysis of numerical instability of centrifugal pumps operational characteristics. *J. Mech. Energy Eng.* **2021**, *4*, 357–364. [\[CrossRef\]](#)
38. Zhao, G.S.; Zhao, W.G. Investigation of cavitation instabilities in a centrifugal pump based on one-element theory. *IOP Conf. Ser. Earth Environ. Sci.* **2018**, *163*, 012042. [\[CrossRef\]](#)

39. González, J.; Parrondo, J.; Santolaria, C.; Blanco, E. Steady and Unsteady Radial Forces for a Centrifugal Pump With Impeller to Tongue Gap Variation. *J. Fluids Eng.* **2005**, *128*, 454–462. [[CrossRef](#)]
40. Casimir, N.; Xiangyuan, Z.; Ludwig, G.; Skoda, R. Assessment of statistical eddy-viscosity turbulence models for unsteady flow at part and overload operation of centrifugal pumps. In Proceedings of the 13th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, Lausanne, Switzerland, 8–12 April 2019. [[CrossRef](#)]
41. Wang, C.-N.; Yang, F.-C.; Nguyen, V.T.; Vo, N.T.M. CFD Analysis and Optimum Design for a Centrifugal Pump Using an Effectively Artificial Intelligent Algorithm. *Micromachines* **2022**, *13*, 1208. [[CrossRef](#)]
42. Shao, C.; Zhou, J.; Cheng, W. Experimental and numerical study of external performance and internal flow of a molten salt pump that transports fluids with different viscosities. *Int. J. Heat Mass Transf.* **2015**, *89*, 627–640. [[CrossRef](#)]
43. Derakhshan, S.; Nourbakhsh, A. Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation. *Exp. Therm. Fluid Sci.* **2008**, *32*, 1620–1627. [[CrossRef](#)]
44. Wang, C.; Shi, W.; Wang, X.; Jiang, X.; Yang, Y.; Li, W.; Zhou, L. Optimal design of multistage centrifugal pump based on the combined energy loss model and computational fluid dynamics. *Appl. Energy* **2017**, *187*, 10–26. [[CrossRef](#)]
45. Li, H.; Chen, Y.; Yang, Y.; Wang, S.; Bai, L.; Zhou, L. CFD Simulation of Centrifugal Pump with Different Impeller Blade Trailing Edges. *J. Mar. Sci. Eng.* **2023**, *11*, 402. [[CrossRef](#)]
46. Lei, L.; Wang, T.; Qiu, B.; Yu, H.; Liu, Y.; Dong, Y. The influence of ring clearance on the performance of a double-suction centrifugal pump. *Phys. Fluids* **2024**, *36*, 25133. [[CrossRef](#)]
47. Reeves, T.D.; Bower, J.; Nevěřil, J.; Janigro, A.; Schofield, S. Guide to the Selection of Rotodynamic Pumps. Available online: https://www.europump.net/files/Publications/Guides/Guide_to_the_Selection_of_Rotodynamic_Pumps_Final.pdf (accessed on 16 September 2025).
48. De Bernardi, J. Aspects Expérimentaux et Théoriques des Instabilités de Cavitation Dans les Turbopompes de Moteurs de Fusée. Ph.D. Thesis, Polytechnique de Grenoble, Grenoble, France, 1996.
49. Ge, M.; Sun, C.; Zhang, X.; Coutier-Delgosha, O.; Zhang, G. Synchrotron X-ray based particle image velocimetry to measure multiphase streamflow and densitometry. *Radiat. Phys. Chem.* **2022**, *200*, 110395. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.