Numerical Study of Blade Roughness Effect on Cavitation in Centrifugal Pumps

Basel Al Bishtawi¹, Gianfranco Scribano¹ and Manh-Vu Tran²

¹ Department of Mechanical Materials and Manufacturing, Faculty of Engineering, University of Nottingham Malaysia, B34, 43500 Semenyih, Selangor, Malaysia
² School of Engineering, Monash University Malaysia, Jalan Lagoon Selatan, 47500 Bandar Sunway, Selangor, Malaysia

Abstract. Cavitation is classified as an undesirable complex multiphase phenomenon due to its negative effects on the structural and flow behaviours of hydraulic machineries, specifically centrifugal pumps. Typically, the pump’s cavitation performance tends to vary due to a multitude of factors, one of which is the impeller blade roughness. In this study, a numerical analysis is conducted, using ANSYS Fluent 20.1, to evaluate the relationship between the impeller blade roughness and the cavitation performance of an ISO 80-50-250 centrifugal pump operating at different pressures and flow rates. The operating conditions include net positive suction head (NPSH) values set at 20, 7.25, 5.71, 2.58, 1.1 m along with outlet mass flow rates of 40, 50, 60 m³/h while the blade roughness height value is changed to 1, 20, 80 μm. The SST k-ω turbulence model, along with the Zwart-Gerber-Belamri cavitation model is used. The results showed that the pressure head increased by approximately 1.64%, 1.55%, and 1.90% at flow rates of 40, 50, and 60 m³/h when the roughness height was increased from 0 μm to 80 μm. However, the increase in roughness improved cavitation inception simultaneously. Meanwhile, the blade suction-side witnessed a delay in cavitation formation, as it shifted downstream towards the trailing edge from its initial point of formation at the blade passage.

1. Introduction

Centrifugal pumps are classified as one of the most critical methods of fluid transportation, as they maintain a simple, compact design and are easy to operate. Moreover, they play a crucial role in many industries, specifically the energy sector [1]. However, a common, yet critical, hydrodynamic problem faced is cavitation. Cavitation is generally defined as an inevitable hydrodynamic phenomenon that results in fluctuations in the pump’s performance due to vapor bubbles constricting the flow path. Furthermore, these bubbles can lead to long-term mechanical damages, as the collapse of cavitation bubbles erode the blade surfaces [2].

Cavitation is the sudden phase shift from water to vapor in flow regions where the local pressure falls below the vaporization pressure of the fluid [3]. Recent studies have shown numerous developments in their numerical methodologies to model cavitating flow behaviours at ranges of operating conditions, initially with two-dimensional (2D) models and eventually with three-dimensional (3D) models by employing Reynolds-averaged Navier-Stokes (RANS) and Large Eddy Simulations (LES) equations coupled with a multiphase model to accurately simulate transient cavitation inceptions in both the impeller and the volute of the pump [4-6]. Moreover, several researches have delved into finding methods to minimize cavitation and optimize pump performance. Some of which discuss the
implementation of grooves, C-grooves, and micro-grooves, to suppress both cavitation and tip leakage vortex [7,8]. Despite improvements in pressure head and flow rate, the micro-groove impeller presents a greater dynamic depression at the passage between the inlet and the inter-blade channel which lead to a weaker anti-cavitation performance. In addition, many researches have been conducted on the influence of the blade shape manipulation on the cavitation performance of pumps [9,10]. For instance, Zhu et al [9] studied the effect of blade thickness ratio on the cavitation performance through the analysis of the pressure coefficient variation across each shape. Coefficient of pressure experienced a drastic improvement by 11.4% linking to an enhanced anti-cavitation performance. However, a smooth surface impeller assumption was employed disregarding its potential effect on results. The significance of roughness considerations was shown in some studies. For a small centrifugal pump, Stepanov et al [11] observed an improved efficiency by 2% to 4% by simply cleaning the volute. Similarly, Lomakin et al [12] examined an efficiency increase from 78% to 89% by polishing all flow parts. Meanwhile, Zariatin et al [13] conducted an experimental investigation scrutinizing the relationship between the surface roughness and pump-turbine power generation. The study resulted in a relationship in which performance increased up to 13.1%, as the surface roughness was decreased from 12 μm to 0.16 μm.

While some of the previous studies explore the effect of surface roughness on the overall performance of the pump, they tend to neglect the influence on cavitation in the process. Moreover, these studies consider the roughness of all impeller flow parts and disregard the possible isolated effects of individual part’s roughness on pump performance. Therefore, the main objective of this study is to explore the relationship between the impeller blade surface roughness and the cavitation performance of the pump by adopting appropriate cavitation measures, such as the net positive suction head (NPSH).

2. Methodology

2.1. Hydraulic design of the impeller

The geometric parameters specified for the impeller studied are adopted from an ISO 80-50-250 standard centrifugal pump impeller, in which its operational parameters [14] are stated in Table 1 below.

Table 1. Operational parameters of ISO 80-50-250 pump.

| Design Parameter     | Units      | Value |
|----------------------|------------|-------|
| Flow rate, $Q_D$     | m$^3$/h    | 50    |
| Head rise, $H_D$     | m          | 80    |
| Rotational speed, $n$| rpm        | 2900  |
| Number of impeller blades, $Z$ | -   | 5     |

In addition, the geometrical parameters of the impeller generated are reported in Table 2.

Table 2. Geometrical parameters of ISO 80-50-250 pump.

| Parameter            | Units | Value |
|----------------------|-------|-------|
| Impeller inlet diameter, $D_1$ | mm    | 80    |
| Impeller outlet diameter, $D_2$   | mm    | 252   |
| Impeller hub diameter, $D_{ab}$   | mm    | 6.5   |
| Blade inlet angle, $\beta_1$      | degree| 33    |
| Blade outlet angle, $\beta_2$      | degree| 9     |

2.2. Selecting simulation models

In this study, ANSYS Fluent 20.1 is selected to perform three-dimensional numerical calculations using unsteady Reynolds-Averaged Navier-Stokes (U-RANS) equations. Specifically, the Shear Stress
Transport (SST) k-ω turbulence model [15] is chosen. The cavitation model by Zwarts-Gerber-Belamri was utilized [16], as it closely resembles the Rayleigh-Plesset equation. To model the roughness, the modified law-of-the-wall [17] is used in ANSYS Fluent, hence, roughness is accounted for such that:

\[
\frac{u_p u'}{\tau_w \rho} = \frac{1}{k} \ln \left( \frac{E \rho u' y_p}{\mu} \right) - \Delta B
\]

The \( y_p \) term is the distance from the centroid of the wall-adjacent cell to wall \( P \), \( u' \) is the dimensionless velocity, \( u_p \) is the fluid mean velocity at wall-adjacent cell centroid \( P \), \( \rho \) is the fluid density, \( \tau_w \) is the shear stress at the wall, \( \mu \) is the fluid dynamic viscosity, \( k \) is the Von Karman constant, and \( E \) is an empirical constant. The additional term, \( \Delta B \), in equation 1 is used as a correlation to determine the equivalent sand-grain roughness by taking the roughness height \( (K_s) \) and the roughness constant \( (C_s) \) to evaluate the shear stress at the wall and estimate the overall wall roughness effect on flow properties.

2.3. Establishing the calculation domain

The numerical calculations are performed on a fluid domain that resembles the whole impeller, as shown in Figure 1d. Moreover, Figure 1 demonstrates the structured grids in the meshing of the impeller domain.

| (a) | (b) | (c) | (d) |
|-----|-----|-----|-----|

![Structured grids of the calculation domain](image)

Figure 1. Structured grids of the calculation domain: (a) inlet domain (b) outlet domain (c) blade refinement at hub (d) entire calculation domain.

Grid independence test is carried out at the design flow rate specified in Table 1. Eight grids are tested and the effect of mesh number on the pressure head under the NPSH = 20 m cavitation condition at \( Q_D = 50 \text{ m}^3/\text{h} \) is observed. A grid of approximately 980,000 cells is then found sufficient for the calculation.

2.4. Setting of boundary conditions

A total pressure inlet and a mass flow rate outlet are set as the boundary conditions. In addition, a no-slip boundary condition was set for the blades, hub, and shroud. All three sub-domains are set as rotating reference frames moving at 2900 rpm. Furthermore, the hub, shroud, and blade boundaries are referenced to the rotating reference frame specified. As for the cavitation settings, a vaporization pressure of 3574 Pa, evaporation coefficient of 50, and a condensation coefficient of 0.01 are set for the Zwart-Gerber-Belamri cavitation model. A fully coupled solver is chosen. QUICK is used for spatial discretization. The pressure staggering option (PRESTO!) scheme is applied for pressure. A First Order Implicit scheme was implemented for the transient formulation. Furthermore, a time step of 1.729x10^{-4} s is used to ensure stability of the numerical calculation. The surface roughness is varied, by changing \( K_s \), based on He et al [18]. The variation in operating conditions include NPSH values set at [20, 7.25, 5.71, 2.58, 1.1 m] along with outlet mass flow rates of [40, 50, 60 m^3/h] while the blade roughness height value is changed to [1, 20, 80 \mu m]. The simulation set-up is validated against experimental results of the
same pump by Wu et al [14]. The validation shown in Figure 2 shows a satisfactory agreement with the experimental data.

![Figure 2](image-url)

**Figure 2.** Pressure Head-NPSH curves. Open symbols experimental data by Wu et al [19]. Experimental results are shown as: red open right-pointing triangle \((Q = 1.2Q_D)\), black open square \((Q = 1.0Q_D)\), blue open circle \((Q = 0.8Q_D)\). Simulation results are shown as: red dotted line \((Q = 1.2Q_D)\), black solid line \((Q = 1.0Q_D)\), and dashed line \((Q = 0.8Q_D)\).

3. Simulation results and analysis

3.1. Effect of blade surface roughness on pressure head

Figure 3 illustrates an NPSH Test in which the pump’s cavitation performance is examined at different operating conditions while the blade retains different degrees of roughness. The pressure head demonstrates varying behaviours depending on the operating flow rate. In Figure 3a, the pressure head presents noticeable improvement, as \(K_s\) is increased; the head experiences a 1 m increase from 83.5 m to 84.5 m, as \(K_s\) is increased from 0 µm to 1 µm. However, the positive change in the pressure head begins to slightly diminish, as \(K_s\) is further increased. For instance, this is shown by the 0.3 m increase due to the \(K_s\) change from 1 µm to 20 µm. This trend deviates away from the findings of several studies that suggest a negative correlation between cavitation performance and surface roughness on the impeller by observing a decrease in pressure head with the increase in roughness height [12,18]. Instead, the deduced trend hints at plausible improvement in the pump’s performance against cavitation inception. Generally, the majority of pumps are assumed to be cavitating despite operating at their best efficiency point (BEP), since it occurs at high speeds and not easily perceived [19]. This usually occurs when the pump operates at NPSH values less than the NPSH incipient (NPSHi) value. Hence, the trend indicates that the traces of cavitation that initially form are either prevented from forming, delayed, or no longer block the impeller passage.

![Figure 3](image-url)

**Figure 3.** Pressure Head-NPSH curves for different \(K_s\) values at (a) \(Q = 0.8Q_D\) (b) \(Q = 1.0Q_D\) (c) \(Q = 1.2Q_D\). Blue open square (smooth surface), black open circle \((K_s = 1 \mu m)\), red open left-pointing triangle \((K_s = 20 \mu m)\), green open right-pointing triangle \((K_s = 80 \mu m)\).
3.2. Development of cavitation within the impeller

The presence of a fluctuating trend in the pressure head in Figure 3c may indicate an underlying relationship between the degree of roughness’ influence on the cavitation pattern and the operating conditions at which the pump operates at. Therefore, to obtain a general profile of this relationship, the simulation results are evaluated about the super cavitation states at NPSH = 1.1 m and at overload operating conditions at $Q = 1.2Q_0$.

Figure 4. Instantaneous vapor volume fraction distribution along the hub and shroud due to changing blade surface roughness height value at NPSH = 1.1 m, $Q = 1.2Q_0$ at flow time = 0.35 s.

From Figure 4, it appears that cavitation inception is significantly dominant at the shroud than at the hub. At the shroud, cavitation occurs on both the pressure side and suction side; however, vapor formation predominantly expands from the leading edge downstream of the pressure side towards the impeller outlet. Moreover, cavitation formation at the suction side appears to form later downstream with a lower intensity. The switch in cavitation zone to the pressure side of the impeller is typically due to the overload operating conditions causing mismatch between the flow and the impeller [19]. Similarly, the impeller hub experiences a gradual increase in the vapor volume with increasing blade surface roughness. This may dictate a pressure head drop; however, the increase in roughness also highlights other cavitation behaviours that correspond to a pressure head improvement. For instance, vapor volumes at the pressure side of the blade near the shroud appear to detach from the blade surface near the leading edge and continues to flow downstream without reattaching to any surface. Moreover, vapor volumes formed at the suction side at the hub follow the same behaviour in which increased amount of vapor is released downstream into the impeller passage, as $K_s$ is increased from 1 μm to 80 μm. Furthermore, a significant pattern shown at the suction side of both the hub and shroud is that the cavitation formation is gradually delayed and receding towards the trailing edge, as the $K_s$ value
increases. Hence, this justifies the increasing pressure head trend demonstrated in Figures 3a, 3b, and 3c.

4. Conclusion
Based on the numerical studies conducted on the effect of blade wall roughness on the cavitation inception, the following is concluded:

- The incremental increase of the blade roughness height value, from 0 μm to 80 μm, resulted in an average increase of 1.64%, 1.55%, and 1.90% at flow rates of $0.8Q_D$, $1.0Q_D$, and $1.2Q_D$ respectively.
- The roughness increase delayed the formation of cavitation at the impeller suction side at both hub and shroud.

References
[1] Shah S R, Jain S V, Patel R N and Lakhera V J 2013 CFD for centrifugal pumps: a review of the state-of-the-art Procedia Eng. 51 715
[2] Brennen C 1995 Cavitation and Bubble Dynamics (California: California Institute of Technology)
[3] Tao R, Xiao R, Wang F and Liu W 2018 Cavitation behavior study in the pump mode of a reversible pumpturbine. Renew. Energy 125 655
[4] Yang J, Zhou LJ, Wang ZW and Zhi FL 2013 The effect of cavitation on the hydrofoil dynamic characteristics. IOP Conf. Ser. Mater. Sci. Eng. 52 062018
[5] Uroš Ješe, Regiane Fortes Patella and Matevž Dular 2015 Numerical study of pump-turbine instabilities under pumping mode off-design conditions ASME-JSME-KSME Jr. Fluids Eng. Conf. AJKFluids2015-33501ff
[6] Al-Obaidi AR 2019 Effects of different turbulence models on three-dimensional unsteady cavitating flows in the centrifugal pump and performance prediction. Int. J. Nonlinear Sci. Numer. Simul. 20(3-4) 487
[7] Liu Y and Tan L. 2020 Influence of C groove on suppressing vortex and cavitation for a NACA0009 hydrofoil with tip clearance in tidal energy Renewable Energy 148 907
[8] Skrzypacz J and Bieganowski M 2018 The influence of micro grooves on the parameters of the centrifugal pump impeller Int. J. Mech. Sci. 144 827
[9] Zhu D, Xiao R, Tao R and Wang F 2018 Designing incidence-angle-targeted anti-cavitation foil profiles using a combination optimization strategy Energies 11(11) 3099
[10] Tao R, Xiao R and Wang Z 2018 Influence of blade leading-edge shape on cavitation in a centrifugal pump impeller Energies 11(10) 2588
[11] Stephanoff AJ 1957 Centrifugal and Axial Flow Pumps: Theory, Design, and Application (New York: Wiley)
[12] Lomakin A 1978 Centrifugal and Axial Pump (Beijing: Machine Press)
[13] Zariatin DL, Rahmalina D, Prasetyo E, Suwandi A and Sumardi M 2019 The effect of surface roughness of the impeller to the performance of pump as turbine pico power plant. J. Mech. Eng. Sci. 13(1) 4693
[14] Wu D, Ren Y, Mou J, Gu Y and Jiang L 2019 Unsteady flow and structural behaviors of centrifugal pump under cavitation conditions. Chi. J. Mech. Eng. 32(1)
[15] Menter FR 1994 Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J. 32(8) 1598
[16] Zwart PJ, Gerber AG and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics ICMF 2004 152
[17] Brennen C 2006 An Internet Book on Fluid Dynamics - Law of the Wall (California: California Institute of Technology)
[18] He X, Jiao W, Wang C and Cao W 2019 Influence of surface roughness on the pump performance based on computational fluid dynamics IEEE Access 7 105331
[19] Binama Maxime, Muhirwa A and Bisengimana E Cavitation effects in centrifugal pumps-a review Bianama Maxime. Int. J. Eng. R. Appl. 6 2448