Dissolved air effects on three-phase hydrodynamic cavitation in large scale Venturi- Experimental/numerical analysis

Emad Hasani Malekshah *, Włodzimierz Wróblewski, Mirosław Majkut
Department of Power Engineering and Turbomachinery, Silesian University of Technology, 44-100 Gliwice, Poland

A R T I C L E   I N F O

Keywords:
Hydrodynamic cavitation
Cavitation
Venturi flow
Dissolved air
Shedding mechanism
Density correction

A B S T R A C T

Hydrodynamic cavitation (HC) in the Venturi nozzle, apart from the harmful influence on the devices, can be used to improve a range of industrial processes, such as biofuel generation, emulsion preparation, and wastewater treatment. The present investigation deals with the influence of dissolved air in Venturi cavitating flow based on numerical and experimental approaches. The experimental campaigns have been done in a closed-loop water tunnel equipped with a Venturi test section. The post-processing techniques such as Fast Fourier Transform (FFT), Power Spectral Density (PSD), temporal/spatial Grey Level distribution and mean value grey level distribution are employed to analyse the experimental observations and measurement. The URANS numerical method is modified based on the Density Corrected-Based Model (DCM) to be more adaptable for flows with high differences in density. The results approve the remarkable effect of dissolved air on the configuration of the cavity, its evolution process, and transient/averaged characteristics. It is observed that the incipient point and ratio of sheet cavity length to cloud cavity length are changed. Furthermore, the flow velocity inside of the sheet and cloud cavities is different; as well as, the higher content of dissolved air leads to slower flow velocity inside the cloud cavity. In addition, the shedding frequency is significantly reduced in case of higher level of air content.

1. Introduction

Cavitation is known as a dynamic phase-change process characterized by an alternation of water and vapour phases [1]. The cavitation phenomenon is started by nucleation and followed by the enlargement of cavity bubbles. Considering the operating parameters like pressure and stream velocity, different types of cavitation including sheet cavity, cloud cavity and supercavitation, may occur. Those phenomena can be observed on blades of water turbines, high-speed propellers, and pumps. Furthermore, due to the high impact of the cavitation on the noise, vibration, erosion, performance alternation and structural damage, it is important to analyse this phenomenon and find out the controlling approaches. Although there are several applications in the turbomachinery, the sludge stabilization which increases removal efficiency of dyes or other emerging contaminants in wastewater, is a promising environmental application of hydrodynamic cavitation [2].

Among the main types of cavitation, partial cavitation, which consists of sheet cavity and cloud cavity, is often detected around the hydraulic components and is known to be highly sensitive for negative effects. Such cavitation has a more complex behaviour than other types since the cavitating flow is characterized by strong unsteadiness, major shedding cloud cavity and fully 3D flow. The cavity configuration and corresponding average/instantaneous characteristics of the partial cavitating flow are highly interesting areas of investigation. The detachment region of the cavity where the type of cavity is altered from sheet to cloud cavity is the first interesting region which is generally followed by the re-entrant jet. Furthermore, another critical part is the region where the sheet cavity is formed due to the low-pressure zone. Afterwards, the sheet cavity is shed downstream and violently collapses when it reaches a high-pressure zone. In the present case, the clouds are generated in the vortex shedding which is filled by the numerous vapour bubbles. The re-entrant jet is one of the principal sources producing the shedding cavity. It mainly consists of liquid and penetrates upstream and hit the sheet cavity border. The existence of a re-entrant jet is already observed and approved by numerical and experimental investigations carried out by Malekshah and Wróblewski [3]. The main structures of the partial cavitation in the flow through a Venturi nozzle are characterized in Fig. 1. The air content is also considered which may exist in the form of dissolved air and dispersed bubbles. It is worth mentioning that different parts of cavities from inception to bubbly clouds are shown...
by different scales. The incipient vapour bubbles and bubbly cloud cavity are microscale structures. However, the sheet cavity and attached–detached cavity are categorized as macro-scale. It is to note that the macroscale zones are the regions of interest.

The breakup and shedding process of partial cavitating flow has been studied around the symmetric and asymmetric objects and nozzles using experimental and numerical approaches [4–6]. As already discussed, the re-entrant jet is known as a principal reason for shedding which was firstly observed by Knapp [7] using a visualization technique and a high-speed camera. In an experimental/theoretical investigation, the attached cavities within a two-dimensional convergent-divergent nozzle have been investigated by Furness and Hutton [8]. They conducted many experiments to detect the behaviour of the partial cavity to the re-entrant jet reaction and approved that the re-entrant jet is mainly responsible for the instabilities at the rear part of the cavity. Kawanami et al. [9] studied the generation of cloud cavitation by implanting an obstacle on the foil surface to ban the re-entrant jet toward the leading edge. They declared that the shedding and breaking up rates have been significantly damped when the re-entrant jet hardly reaches the sheet cavity. Stutz and Reboud [10,11] used a double optical probe to evaluate the sheet cavity structure inside a Venturi nozzle. They quantitatively declared the existence of a re-entrant jet at the adjacent foil surface which flows upstream toward the sheet cavity and causes the periodical breaking off. Huang et al. [4,12] and Ji et al. [13–15] formulated the relationship between the re-entrant jet and the large-scale vortex generated at the rear zone of the sheet cavity. They figured out that the reverse pressure gradient and reverse flow close to the wall are the products of a large-scale vortex. On the other hand, it was declared by Kubota et al. [16] that the cloud cavity is convected downstream with a lower velocity than the bulk flows. However, in the experiment conducted by Pham et al. [17], the velocity of the jet stream was found to be in the same order as the main flow.

To simulate the turbulent cavitating flows, it is often assumed that the mixture of liquid–vapour in the two-phase cavitation model and liquid–vapour-air in the three-phase cavitation model, are homogenous phases. Also, the variation of mixture density is calculated based on either a barotropic equation of state (EOS) [18,19] or a transport equation [20,21] during the cavitating flow. The cavitating flows are usually categorized among high-Reynolds number flows where the turbulence also plays an inevitable role in the prediction of its unsteady characteristics. The URANS approaches are used in many works because they can predict averaged flow characteristics with low computational cost compared to other numerical models. The disadvantage of the URANS approach is the poor capability of resolving details of flow structures. It is mostly devoted to the prediction of averaged flow structures [22–25]. Wrablewski et al. [26] carried out a numerical
investigation of the cavitating flow when the air is taken into consideration. They used URANS numerical approach with RNG $k-\varepsilon$ turbulence model. To deal with the presence of air as the third phase, they adopted two approaches of 2-phase and 3-phase in which the phases are assumed as water/vapour-air and water/vapour/air, respectively. In both approaches, the mixture model was employed. Based on the comparison of numerical results with conducted experiments, it was declared that the 3-phase approach, which considers water, vapour and air as three separated phases, gives better prediction over the cavity structure and unsteady characteristics. Furthermore, the influence of air on the dynamic characteristics and cavity configuration was demonstrated. The larger cavity with more steady behaviour was the outcome of adding dissolved air. In another work, the three-phase cavitating flow was addressed based on numerical/experimental investigation by Wrblewski et al. [27]. The cavitating flow in presence of air was visualized and corresponding unsteady characteristics were measured using a high-speed camera and pressure transducers. The global and local features of the cavitation in the flow around the ClarkY hydrofoil were exported. The numerical simulations were carried out based on the URANS model considering three phases water, vapour and air. Also, two levels of oxygen contents including 2.6 ppm and 5.5 ppm are measured during the test, and the same conditions were adopted for numerical

### Table 1

| Mesh Symbol | Number of elements | Number of nodes on Venturi surface |
|-------------|--------------------|-----------------------------------|
| M1          | 51,000             | 230                               |
| M2          | 57,300             | 260                               |
| M3          | 59,500             | 290                               |
| M4          | 61,500             | 300                               |
| M5          | 63,500             | 310                               |

Fig. 3. Schematic of measuring and visualization systems.

Fig. 4. Computational domain, dimensions, and grid distribution.
The results confirmed the noticeable impact of dissolved air on the enlargement of the cavity and reduction in shedding frequency. To overcome the incapability of the standard turbulence models in perfect prediction of instability of cavitating flow, Coutier-Delgosha et al. [28] proposed a modification over the dynamic viscosity which avoids its overestimation. They proved the positive effect of modifications on the simulation of cavity evolution and corresponding unsteady characteristics. To simulate the cavitation inside a Venturi channel, Chen et al. [23] adopted a modified density corrected model coupled with an energy equation. Also, the heat transfer effect is taken into consideration. Their simulations were in good agreement with the experimental data in the same geometry and operating conditions. Malekshah et al. [29] concentrated on the cavitating flow around the Clark-Y hydrofoil with dissolved air as the third phase. Because the RNG k-epsilon model overestimates viscosity and yields poor predictions, the turbulence model is changed using the density corrected model (DCM) and filter-based density correction model (FBM). The numerical results and the experimental data are also compared. It is determined that when the improved turbulence models are used, the numerical prediction will be more accurate.
The present work aims to analyse the impact of dissolved air on the cavitating Venturi flow using experimental observations and numerical simulations. Experiments are conducted at three cavitation numbers and two dissolved air levels. The URANS simulations are carried out to predict the transient and average features of the cavitation process.

2. Experimental facilities and procedure

The experiments were conducted using hydraulic installation built and mounted at the laboratory of the Department of Power Engineering and Turbomachinery, The Silesian University of Technology. The schematic of the installation along with the main components is illustrated in Fig. 2. The installation is a closed-loop circuit equipped with a replaceable test section. The operation fluid inside the circuit is water which flows through the 200 mm pipes using an electric pump with a power of 30 kW. The manual valve and the electromagnetic flowmeter are installed after the pump. The water stream flows through the pipe upward around 5 m to reach the section. Before the test section, the straightener is installed to reduce the vorticity of the water stream. In addition, the pipe is connected to the test section using a cross-section inverter. The water stream passes the test section and Venturi nozzle which is the region of interest. Then, the cross-section is changed from rectangular to circular using a shaped diffuser. Afterwards, the pipe heads to the tank which is located on the ground floor. The tank of 1.5 m$^3$ volume is designed to keep the required water for the experiment; as well, as to adjust the pressure level inside the circuit. For this purpose, an internal elastic airbag is mounted inside of the tank which may be inflated using the controlled compressed air system. This enables the test rig is capable to be operated with the same flow rate and different pressure levels in order to set up different operating conditions. To reduce forces and vibration propagation, three elastic compensators, one before the tank, one after the pump and one between the tank, and the pump were inserted.

The cavitation test chamber has a rectangular cross-section and includes a Venturi nozzle. The transparent window, which was made of polycarbonate, was placed at the one sidewall of the test chamber to enable optical access and observations. The length $L$, height $H$ and width $W$ of the chamber are equal to 700 mm, 189 mm and 70 mm, respectively. The ratio of chamber height to width was fixed as $H/W = 2.7$. Hence, the ratio of throat height $H_{th}$ to a height of chamber $H$ is defined as $AR = H_{th}/H = 0.6$. Also, the throat length (i.e. the distance of throat from the inlet) is 196 mm.

![Numerical-predicted time-dependent distributions of vapour and volume fractions and corresponding Continuous Wavelet Transform (CWT) for different cavitation numbers.](image-url)
The experimental tests were conducted based on two specific levels of dissolved oxygen of 4.01 mg/l and 6.66 mg/l. Based on Henry’s law, it corresponds to the air content of 10.25 mg/l and 17.03 mg/l, respectively. The current levels of air only include the dissolved air; as a result, the amount and effect of non-dissolved air bubbles are not taken into account. The multifunction meter CF-401 was employed to measure the oxygen levels before and after the experimental campaign in steady conditions. The average value of the oxygen is reported in this work. The ranges of oxygen levels for the first and second experimental campaigns were 4.32–3.71 mg/l and 7.22–6.15 mg/l, respectively.

It is worth mentioning that two aeration processes were performed to increase the level of air content. For this purpose, the air was injected into the water channel when the facility is running. Then, the injection was stopped; however, the facility was still running for 1–2 min. So, it could be assured that the injected air was perfectly dissolved into water. In the next step, the water sample was taken from the water channel, and the level of dissolved air was measured. Also, the sampling and measuring were repeated after each experimental campaign. The average value of dissolved air is reported in the present work. The amount of dissolved air is calculated during the tests; however, the cavitating flow is characterized using the visualization and frequency measuring over the shedding process. Therefore, other approaches exist to analyze the feature of cavitating flow such as vibration noise measurement [30].

![Simulated cavity evolution in two sequential cycles (σ = 2.02 with low air content)](image-url)
The schematic of the measuring and visualization systems is demonstrated in Fig. 3. The unit consists of a pressure regulator, low/high-frequency pressure sensors, vibration sensors, fast/ABS pressure transducers, data acquisition system, multi-LED lighting, high-speed camera, and computer. Twelve pressure sensors are installed at the surface of the Venturi profile, and two sensors at the inlet and outlet of the chamber. Among the sensors, three are fast-frequency and the rest are low-frequency sensors. The model of low-frequency sensors is APLISENS PC-28 with a full-scale (FS) of 160 kPa and an accuracy of 0.16 %. The pressure waves were transmitted to the measuring cluster using impulse tubes. The pressure fluctuations of $P_{\text{inlet}}$, $P_3$ and $P_8$ were detected using high-frequency miniature pressure sensors of XP5 type with amplifier ARD154. The maximum detectable pressure for XP5 is 500 kPa with an accuracy of 0.25 %. The temperature of the water was measured by the resistance thermometer APLISENS CT-GN1 Pt100 with a full scale of 0–100 °C and accuracy of ±(0.15 K + 0.002°C). The electromagnetic flowmeter UniEMP-05 DN200 was used to measure the flow rate up to 1080 m$^3$/h with an accuracy of ± 0.25 %. To measure the vibration generated by the cavitating flow, the vibroacoustic signal at the outer wall of the chamber was measured using two piezoelectric sensors. The stiff piezoelectric accelerometers KD35 (RTF) were installed at the external wall of the chamber and located before and after the throat. The accelerometers were connected with the 0028 (RTF) type charge amplifier connected with the fast analog-to-digital converter AC 16 bit, 250 kS/s. The system was calibrated before the experiments using the electrodynamic vibration calibrator EET101 (RTF) type. The upper value of the error was less than 5 %. The measurement system was set based on the National Instruments module NI USB 6216. In addition, the NI/PXI-6255 module cooperated with the pressure measuring cluster consisting of sets of pressure sensors and measuring transducers. The executive elements and the data acquisition process were managed by a system programmed in the LabView environment. The visualization unit

Fig. 9. Visualization of cavity evolution ($\sigma = 2.06$ with low air content, $t = 8.3$ ms to $t = 55.4$ ms).
consisted high-speed camera, lighting and monitor. The high-speed video camera Phantom VEO 710 is used to record the cavitating flow. The recording speed was set to 7000 fps with a resolution of 1280 × 800 pixels. The MULTILED L48-XF was utilized for lighting purposes.

3. Numerical approach
3.1. Multiphase numerical model

The homogeneous mixture model is employed to perform the numerical simulation of multiphase flow. Based on the mixture model, the three phases of water, vapour and air are assumed as a single homogeneous fluid with the same velocity field and negligible slip velocity between the continuous and dispersed phases. Based on the above-mentioned assumptions the governing equations read:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]

\[
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = \nabla \mathbf{p} + \nabla \cdot \left[ \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \right] + \rho g
\]

\[
\rho = \rho_l \alpha_l + \rho_v \alpha_v + \rho_{ng} \alpha_{ng}
\]

\[
\mu = \mu_l \alpha_l + \mu_v \alpha_v + \mu_{ng} \alpha_{ng}
\]

In the present simulations, the body force is neglected due to the minor effect of the body force on the cavitation. As a result, the last term on the right-hand side of equation (2) is not taken into account. Since it is intended to take the air into consideration, the third term with subscript ng which denotes the non-condensable gas is added to equation (3). As such, the mixture consists of three phases (i.e., water, vapour, and air), and the mixture model solves the continuity equation for the vapour volume fraction and air volume fraction. In addition, the mass transfer between the liquid and vapour phases is modelled. The equations are as follows:

\[
\frac{\partial \rho_l \alpha_l}{\partial t} + \nabla \cdot (\rho_l \alpha_l \mathbf{u}) = R
\]

\[
\frac{\partial \rho_{ng} \alpha_{ng}}{\partial t} + \nabla \cdot (\rho_{ng} \alpha_{ng} \mathbf{u}) = 0
\]

\[
\alpha_l + \alpha_v + \alpha_{ng} = 1
\]

The mass transfer between the liquid and vapour phases is governed by the source term \( R \) denoting the mass transfer per volume unit in both evaporation and condensation processes. It is worth mentioning that there is no mass transfer between the air phase with other phases. In this regard, the source term in Eq.5 is zero.

The Zwart-Gerber-Belamri (ZGB) cavitation model is employed to calculate the source term in the mass transfer equation (eq.4). In this respect, the source term \( R \) to describe evaporation and condensation is expressed by the following equations [31]:

\[
R_e = F_{\text{vap}} \frac{3 \alpha_{\text{nuc}} (1 - \alpha_c) \rho_v}{R_B} \left( \frac{2 \rho_v - \rho_c - \rho_l}{\rho_l} \right) \quad \text{if } \frac{\rho_v}{\rho_c} > 1
\]

\[
R_c = -F_{\text{cond}} \frac{3 \alpha_c \rho_v}{R_B} \left( \frac{2 \rho_v - \rho_c - \rho_l}{\rho_l} \right) \quad \text{if } \frac{\rho_v}{\rho_c} < 1
\]

where the empirical coefficients \( F_{\text{vap}} = 50 \) and \( F_{\text{cond}} = 0.1 \) are adopted for the water cavitating flow at ambient temperature. Also, the nucleation site volume fraction \( (\alpha_{\text{nuc}}) \) is assigned to \( 5 \times 10^{-4} \), the fixed spherical bubble radius is equal to \( 1 \times 10^{-6} \) m.
3.2. Turbulence model

The RNG $k - \varepsilon$ is employed to model the turbulent cavitating flow and is defined as follows:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho u k) = \nabla \cdot \left( \mu + \mu_t \frac{\rho k^2}{\rho} \right) \nabla k + G_k - \rho e,$$

(8)

Fig. 11. Morphological analysis of incipient point, sheet cavity and cloud cavity ($\sigma = 2.02, 2.06, 2.14$, low and high air contents).

Fig. 12. Velocity analysis of sheet cavity and cloud cavity ($\sigma = 2.06$, low and high air contents).
\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u u) = \nabla \cdot \left( \left( \mu + \mu_t \right) \nabla \epsilon \right) - \frac{c_1^2}{k} \epsilon G_k - c_2 \rho \epsilon^2 \frac{k^2}{\epsilon} 
\]

(9)

where \( \mu_t = \rho C_\mu k^2 / \epsilon \) defines the turbulent viscosity with \( C_\mu = 0.084 \) [32]. Furthermore, \( k, \epsilon \) and \( G_k \) show the turbulent kinetic energy, turbulent energy dissipation rate and production of turbulent energy term, respectively.

The standard form of the turbulence model usually overestimates the turbulent viscosity. Also, no treatment over the standard turbulence model is applied to deal with the high jump in density of the mixture. To overcome the damping effect, the standard turbulent viscosity is modified based on the Density Correction based Model (DCM) which was first proposed by Coutier-Delgosha et al. [28]. Using this correction, the turbulent viscosity is simply reduced in the region with a mixture of vapour and liquid. As a result, the damping effect of the standard turbulence model will be compensated. The modified turbulent viscosity is given as follows:

\[
\mu_t = f(\rho) C_\mu k^2 / \epsilon, 
\]

(10)

where,

\[
f(\rho) = \rho_s + \left( \frac{\rho_v - \rho_s}{\rho_v - \rho_l} \right)^n \left( \rho_v - \rho_s \right).
\]

(11)

where the constant \( n \) is set to 10.

4. Computational domain, meshing and grid independence analysis

The computational domain’s length and height, as well as the boundary conditions shown in Fig. 4, match the real physical dimensions of the test chamber. However, due to the symmetrical geometry of the Venturi nozzle, half of the domain is used for numerical simulation to reduce the computational cost. In addition, the calculation domain is extended at the outlet side to avoid backflow. The extended section is of the same length as the Venturi nozzle. The velocity inlet and pressure outlet boundary conditions are set on the left and right walls, respectively. Furthermore, the bottom wall, which is the surface of the Venturi nozzle, is assumed to be a non-slip, stationary surface; as well as the top wall is set as symmetry. Fig. 4 shows an overview of the grid. The domain is divided into three primary blocks. The grid is considered finer near the throat; as well as, adjacent to the wall. Based on the selected mesh distribution according to grid independence analysis, the computational domain has 70 and 260 nodes along the edge normal and along the Venturi surface, respectively. The domain had an overall width of 1.0 mm discretized by 3 layers. The whole mesh consisted of 61 k hexahedra elements and the value of \( y^+ \) on the wall was less than 1.

The grid was generated in ICEM CFD software. It was a 2D structured grid extruded to the overall width by 3 layers of 0.333 mm thickness each. Five different meshes were examined. Their parameters are summarized in Table 1.

The pressure distributions along the Venturi surface obtained for five various grid sizes are compared with the present experimental results for one case (\( \sigma = 2.02 \) with low air content). Based on the comparison (see Fig. 5), it is determined that mesh distribution M4 is the best match for the experimental data and should be used for further investigations.

5. Results and discussion

The cavitating Venturi flow is studied based on experimental and numerical methods with special emphasis on the effect of dissolved air. Two levels of dissolved air and three cavitation numbers are taken into consideration. The transient pressure fluctuations are recorded using pressure transducers on the surface of Venturi, and the cavity evolution is visualized using the high-speed camera. Hence, the cavitation features are predicted using numerical simulations. The FFT, PSD and temporal/spatial grey level distribution are employed for the post-processing.

The averaged pressure distributions at the wall of the Venturi nozzle for various cavitation numbers and air contents are shown in Fig. 6. Considering the fact that the pressure distribution along the flow
channel is crucial in showing the cavitation collapse process, special attention was paid to the pressure variation in various locations at different flow conditions. The averaged pressure stays constant and close to the saturated vapour pressure during cavitation inception and development regions. It experiences a significant increase in the collapse region. The averaged pressure increases slowly along the flow channel in the downstream region where no cavitation exists. At high cavitation numbers, the unfavourable pressure gradient region is higher than at low cavitation numbers, resulting in rapid collapse and severe shock. However, it is observed that the general trend of the averaged pressure distribution is similar in experimental measurements and numerical calculations, it is understood that the collapse process happens further from the throat in numerical simulations. Although the impact of dissolved air on the averaged pressure distribution is negligible in inception and development regions, the pressure magnitude slightly drops when the amount of dissolved air is higher, regardless of the cavitation number.

The distributions of vapour and air volume fractions over the flow time and corresponding Continuous Wavelet Transform (CWT) are presented in Fig. 7. The following time-dependent distribution is predicted using numerical simulation. The Continuous Wavelet Transform (CWT) is employed to calculate the corresponding shedding frequency during various stages of the simulation. The continuous wavelet transform (CWT) is a formal (i.e., non-numerical) tool that provides an overcomplete representation of a signal by letting the translation and scale parameter of the wavelets vary continuously. The time-dependent distributions of volume fractions include two parts demonstrating the cases with low air and high air contents. It is concluded that the averaged vapour volume fraction remains at the same level when the air content enhances; however, its amplitude considerably rises. The latest
observation declares that a larger cloud cavity emerges inside of the Venturi nozzle when the dissolved air increases. The level of dissolved air influences not only the structure of the cavity but also considerable impacts its dynamics. Regardless of the cavitation number, the shedding frequency is reduced when the level of dissolved air enhances. The shedding frequency decreases more in the cases with a higher cavitation number. As a result, the influence of dissolved air on the dynamic of the cavity is more considerable at the higher cavitation number.

The cavity evolution based on the numerical simulation for two cycles (i.e., fifth and sixth cycles) for $\sigma = 2.02$ with low air content is represented in Fig. 8. It should be noted that half of the computational domain is taken into consideration due to the symmetric geometry. Also, the interference effect between the bottom and upper parts of the cavitation inside the Venturi is neglected since those cavity closures are not merged in the considered operating conditions of this study. Two sequential cycles are presented to declare the possible similarity and differences in the cavity structure and evolution. It is observed that structures of sheet and cloud cavities are almost similar however, the separation point happens slightly earlier in the fifth cycle. The cycle starts with the inception of the sheet cavity. The incipient cavity is extended gradually. Then, the extended sheet cavity is torn from the separation point. As a result, two regions of the cavity appear. The torn part of the cavity enlarges and changes to a cloud cavity and the remained sheet cavity is shrunk till disappears. Afterwards, limited numbers of small cavities emerge at the throat of the Venturi nozzle which are shed significantly fast to reach the cloud cavity. The small cavities are merged into a cloud cavity making it larger. The adverse pressure that occurs adjacent to the Venturi surface causes separation of the cavity from the surface. The influence of the re-entrant jet on cavity separation is considerable and will be discussed in the following section. Finally, the separated cloud cavity sheds downstream.

The cavity evolution and the location of the re-entrant jet front between $t = 8.3$ ms to $t = 55.4$ ms are depicted in Fig. 9 based on experimental visualization. The images show the cavity evolution within half of the test section along with the Venturi nozzle for the case with $\sigma = 2.06$ and low air content. It should be noted that the average form of the cavity structure at the bottom and upper parts inside the Venturi is almost similar; however, the instantaneous parameters such as the location of the re-entrant jet and shed cloud cavity may have differences. The time range is selected to declare the behaviour of the re-entrant jet in a cycle. In some of the first images between $t = 12.3$ ms to $t = 15.7$ ms, the generated vortex is depicted by drawing the schematic arrows. It is observed that the shedding vortex is generated repeatedly in the cloud cavity region. The shedding vortex is inflated, detached and excessively shed to downstream. Going further downstream, the shedding vortices collapse when they reach the high-pressure zone outside of the cloud cavity region. Using images between $t = 15.7$ ms to $t = 46.4$ ms, it is focused on the location of the re-entrant jet. The red spots and the white dash line demonstrate the front of the re-entrant jet and its average movement, respectively. It is revealed that the re-entrant jet penetrates toward the throat being close to the wall. The re-entrant jet is not steadily moved forward or backward as it’s pushed forth and back temporarily. Moving forward ($S_1$) and backward ($S_2$) took 16.5 ms and 10.8 ms respectively, which means that the penetration process needs to overcome the main flow.

The temporal-spatial grey level distributions at four different cross-sections ($x/L = 0.2, 0.35, 0.5, 0.65$) for the cases with $\sigma = 2.14$, low and high air contents are represented in Fig. 10. The cross-sections are specifically selected to show the behaviour of different regions like incipient zone, sheet cavity, cloud cavity and bubbly cloud cavity. First, it is observed that the strongest cavity region is generated in sections B and B’, which are located near the throat and inside of the sheet cavity. Moreover, remarkable intensification of the cavity length has happened when the level of air content enhances which is obviously detectable at all cross-sections. In the case with a higher amount of dissolved air, the grey level distributions declare that many scattered bubbles existed in the chamber, especially around the throat where the pressure level is lower than in other regions.

To investigate the impact of cavitation number and level of air content on the structure of the cavity, morphological analysis is provided, as shown in Fig. 11, using the temporal-spatial grey level distribution. The ratio of inception point $x_i$ and length of Venturi nozzle $L_v$ is defined by $x_i/L_v$. At a higher cavitation number, it is concluded that the inception point is closer to the throat edge meaning that the cavitation is generated earlier locally. Similarly, the inception point gets closer to the leading edge by increasing air content. In addition, to compare the length of sheet cavity and cloud cavity with the length of full cavity zone, the ratios of $L_{sheet}/L_{cavity}$ and $L_{cloud}/L_{cavity}$ are defined, respectively. Comparing the length of the sheet cavity with the full cavity zone, it is clear that, in the case with a higher cavitation number, the cavity mainly consists of the sheet cavity regardless of the level of air content. The ratio $L_{cloud}/L_{cavity}$ is inversely related to the cavitation number for both levels of air content. By adding dissolved air, the increment rate of $L_{cloud}/L_{cavity}$ is higher than $L_{sheet}/L_{cavity}$. The latest observation demonstrates the higher effect of dissolved air on the sheet cavity than on the cloud cavity.

To analyse the velocity of flow inside of the sheet cavity and cloud cavity, the velocity analysis is carried out using temporal-spatial grey distribution level for the case with $\sigma = 2.06$, low and high air contents. For this purpose, the average angle of the grey level distribution in combination with a specified point, which shows the borders of the sheet cavity and cloud cavity, can be used, as shown in Fig. 12. It is worth mentioning that yellow, red and white spots point out the start of sheet cavity, end of sheet cavity and end of cloud cavity, respectively. In the case with lower air content, the flow velocity inside the sheet cavity is around $16$ m/s which is greater than the main flow velocity. However, the flow velocity in the cloud cavity is almost $6$ m/s. As such, not only the flow velocity inside the sheet cavity is higher than the main flow velocity, but also it is higher than the velocity of the cloud cavity more than two times. By adding the dissolved air, flow velocities inside the sheet cavity and cloud cavity are almost equal to $19$ m/s and $11$ m/s, respectively, which are higher than flow velocity. It is concluded that adding the dissolved air results in an increase in flow velocity in the cavity zone.

Fig. 13 shows the mean value of the grey scale (above) and schematic of the cavity boundary to indicate the impact of cavitation number and the dissolved air content on the structure of the cavity. The mean value is taken over 300 ms of the captured movie covering five periods. Comparing the mean value over different cavitation numbers, it is confirmed that the bigger cavity is generated in the lower cavitation number. In addition, the remarkable influence of the dissolved air on the length of the cavity is visible where the bigger cavity appears in the cases with a higher amount of dissolved air. The boundary of cavities on different cavitation numbers and air contents are sketched schematically using the mean values.

The power spectral density (PSD) based on the pressure distribution recorded by pressure transducers P3 and P8 is presented in Fig. 14. The locations of pressure transducers P3 and P8 are shown on the wall of the Venturi nozzle. It can be seen that P3 is always located inside of the sheet cavity and P8 is also located inside, except case with $\sigma = 2.14$. It should be noted that three fast pressure sensors were used in the present experiments, installed at the inlet and in points P3 and P8. Since the shedding frequency must reflect the frequency of cloud cavity detachment, the pressure distribution which is going to be used to extract the shedding frequency needs to be located outside of the sheet cavity, where there exists a steady cavity region. As such, the pressure distribution inside the sheet cavity may not be desirable for measuring the shedding frequency. As can be seen in Fig. 14, no dominant peaks on the PSD plots can be detected till 1000 Hz for all cases except for P8 in the case with $\sigma = 2.14$. The frequency of 2.3 Hz and 2.6 Hz are measured. Overall, it can be concluded that not only the pressure distribution is not a reliable parameter to measure the shedding frequency since it is
usually influenced by induced pressure shock wave inside of the Venturi nozzle, but also the location of the pressure transducer is so effective in determining the pressure fluctuation.

6. Conclusion

The main purpose of the present experimental/numerical research is to study the effect of dissolved air on the cavitation flow in the Venturi nozzle. For this purpose, experimental tests have been conducted in the closed-loop water tunnel. The water tunnel is equipped with a Venturi test section, as well as the measurement devices such as pressure transducers, vibration transducers and a high-speed camera. The experimental campaigns have been done in two levels of dissolved air and three cavitation numbers. Furthermore, the numerical simulation is carried out and the transient/averaged features of the cavitation process are predicted. Also, the post-processing techniques such as Fast Fourier to study the effect of dissolved air on the cavitating flow in the Venturi nozzle, but also the location of the pressure transducer is so effective in determining the pressure fluctuation.

Data availability

The authors do not have permission to share data.

References:

[1] C.E. Brennen, Cavitation and bubble dynamics, Cambridge University Press, 2014.
[2] G. Mancuso, M. Langone, G. Andreottola, A critical review of the current technologies in wastewater treatment plants by using hydrodynamic cavitation process: principles and applications, J. Environ. Health Science and Eng. 18 (1) (2020) 311–333.
[3] E. Hasani Malekshah, W. Wróblewski, Merging theory-based cavitation model adaptable with non-condensable gas effects in prediction of compressible three-phase cavitation flow, Int. J. Heat Mass Transf. 196 (2022/11/01/2022.), 132379, https://doi.org/10.1016/j.ijheatmasstransfer.2022.132379.
[4] B. Huang, Y.L. Young, G. Wang, Combined experimental and computational investigation of unsteady structure of sheet/cloud cavitation, J. Fluids Eng. 135 (7) (2013) pp.
[5] Z. Wang, B. Huang, G. Wang, M. Zhang, F. Wang, Experimental and numerical investigation of ventilated cavitation flow with special emphasis on gas leakage behavior and re-entrant jet dynamics, Ocean Eng. 108 (2015) 191–201.
[6] T. Liu, B. Huang, G. Wang, M. Zhang, D. Gao, Experimental investigation of the flow pattern for ventilated partial cavitating flows with effect of Froude number and gas entrainment, Ocean Eng. 129 (2017) 345–351.
[7] R.T. Knapp, Recent investigations of the mechanisms of cavitation and cavitation damage, Transactions of the ASME 77 (1955) 1045–1054.
[8] P. Furetta and S. Hutton, "Experimental and theoretical studies of two-dimensional fixed-type cavities," 1975.
[9] Y. Kawanami, H. Kato, H. Yamaguchi, M. Tanimura, and Y. Tagaya, “Mechanism and control of cloud cavitation,” 1997.
[10] B. Stutz, J. Reboud, Experiments on unsteady cavitation, Exp. Fluids 22 (3) (1997) 191–198.
[11] B. Stutz, J.-L. Reboud, Two-phase flow structure of sheet cavitation, Phys. Fluids 9 (12) (1997) 3678–3686.
[12] B. Huang, Y. Zhao, G. Wang, Large eddy simulation of turbulent vortex-cavitation interactions in transient sheet/cloud cavitation flows, Comput. Fluids 92 (2014) 113–124.
[13] B. Ji, X. Luo, R.E. Arndt, Y. Wu, Numerical simulation of three dimensional cavitation shedding dynamics with special emphasis on cavitation–vortex interaction, Ocean Eng. 87 (2014) 64–77.
[14] B. Ji, X. Luo, R.E. Arndt, X. Peng, Y. Wu, Large eddy simulation and theoretical investigations of the transient cavitating vortical flow structure around a NACA66 hydrofoil, Int. J. Multiph. Flow 68 (2015) 121–134.
[15] B. Ji, Y. Long, X.-P. Long, Z.-D. Qian, J.-J. Zhou, “Large eddy simulation of turbulent attached cavitation flow with special emphasis on large scale structures of the hydrofoil wake and turbulence-cavitation interactions,” Journal of Hydrodynamics, Ser. B 29 (1) (2017) 27–39.
[16] A. Kubota, H. Kato, H. Yamaguchi, and M. Maeda, “Unsteady structure measurement of cloud cavitation on a foil section using conditional sampling technique,” 1989.
[17] T. Pham, F. Lazarte, and D. H. Fruman, “Investigation of unsteady sheet cavitation and cloud cavitation mechanisms,” 1999.
[18] B. Charriere, J. Decaix, E. Goncalves, A comparative study of cavitation models in a Venturi flow, European Journal of Mechanics/B 49 (2015) 287–297.
[19] T. Liu, B. Khou, W. Xie, Isentropic one-fluid modelling of unsteady cavitation flow, J. Comput. Phys. 201 (1) (2004) 80–108.
[20] I. Senocak, W. Shyy, Interfacial dynamics-based modelling of turbulent cavitation flows, Part-I: Model development and steady-state computations, Int. J. Numer. Meth. Fluids 44 (9) (2004) 975–995.
[21] A.K. Singhal, M.M. Athavale, H. Li, Y. Jiang, Mathematical basis and validation of the full cavitation model, J. Fluids Eng. 124 (3) (2002) 617–624.
[22] X. Peng, et al., Combined experimental observation and numerical simulation of the cloud cavitation with U-type flow structures on hydrofoils, Int. J. Multiph. Flow 79 (2016) 10–22.
[23] T. Chen, B. Huang, G. Wang, Numerical study of cavitation flows in a wide range of water temperatures with special emphasis on two typical cavitation dynamics, Int. J. Heat Mass Transf. 101 (2016) 886–900.
[24] J. De, X. Luo, Y. Wu, Y. Peng, X. Fu, Partially-Averaged Navier-Stokes method with modified k–e model for cavitation flow around a marine propeller in a non-uniform wake, Int. J. Heat Mass Transf. 55 (23–24) (2012) 6582–6588.
[25] Q. Wu, B. Huang, G. Wang, Lagrangian-based investigation of the transient flow structures around a pitching hydrofoil, Acta Mech. Sin. 32 (1) (2016) 64–74.
[26] W. Wróblewski, K. Bochen, M. Majkut, K. Rusin, E.H. Malekshah, Numerical study of cavitation flow over hydrofoil in the presence of air, Int. J. Numer. Meth. Heat Fluid Flow (2021).
[27] W. Wróblewski, K. Bochen, M. Majkut, E.H. Malekshah, K. Rusin, M. Strozlik, An experimental/numerical assessment over the influence of the dissolved air on the instantaneous characteristics/shedding frequency of cavitation flow, Ocean Eng. 240 (2021/11/15/ 2021.), 109960, https://doi.org/10.1016/j.oceaneng.2021.109960.
[28] O. Coutier-Delsart, R. Fortes-Patella, J.-L. Reboud, Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation, J. Fluids Eng. 125 (1) (2003) 38–45.
[29] E. Hasani Malekshah, W. Wróblewski, K. Bochen, M. Majkut, Evaluation of modified turbulent viscosity on shedding dynamic of three-phase cloud cavitation around hydrofoil – numerical/experimental analysis, Int. J. Numer. Meth. Heat Fluid Flow vol. ahead-of-print, no (2022), https://doi.org/10.1108/HFF-03-2022-0188 ahead-of-print.
[30] G. Mancuso, Experimental and numerical investigation on performance of a swirling jet reactor, Ultrason. Sonochem. 49 (2018/12/01/2018,) 241–248, https://doi.org/10.1016/j.ultsonch.2018.08.011.

[31] A. Kubota, H. Kato, H. Yamaguchi, A new modelling of cavitating flows: a numerical study of unsteady cavitation on a hydrofoil section, J. Fluid Mech. 240 (1992) 59–96.

[32] V. Yakhot, S. Orszag, S. Thangam, T. Gatski, C. Speziale, Development of turbulence models for shear flows by a double expansion technique, Phys. Fluids A 4 (7) (1992) 1510–1520.