Ultrasonics - Sonochemistry xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Ultrasonics - Sonochemistry

journal homepage: www.elsevier.com/locate/ultson

Experimental and numerical investigation on performance of a swirling jet reactor

Giuseppe Mancuso

Centre for Water Resources Research, School of Civil Engineering, University College Dublin, Belfield, Dublin 4, Ireland

ARTICLE INFO

Keywords:
Modeling
Hydrodynamic cavitation
Swirling jet reactor
ANSYS-FLUENT
CFD simulations

ABSTRACT

In this work, a three-dimensional Computational Fluid Dynamic (CFD) analysis of a swirling jet reactor was implemented to gain a better understanding of fluid dynamics into the reactor. The effect of different geometries of the reactor, by considering different diameters of the injection slots of the reactor, on flow velocity and flow pressure distributions was investigated. Firstly, a one-phase model was implemented by considering only water into the reactor. Then, a two-phase model was defined including dissolved air into the water. The inlet flow pressure was set to 0.25 bar to consider non-cavitating conditions and, then, to get more accurate results on fluid dynamics into the reactor due to the absence of cavitating conditions. Data collected from experimental tests were used to calibrate and validate the model. Results of numerical simulations were in good agreement with experimental data, showing for all the geometries a rotating flow around the central axis of the reactor and at the exit of the double cone. The highest flow velocities and flow pressure drops were observed for the reactor geometry with the smallest injection slots diameters. Finally, noise measurements were performed during another set of experimental tests by considering different inlet flow pressures.

1. Introduction

In the past, many studies were carried out with the main aim of preventing the generation of cavitation in hydraulic machinery such as pumps, turbines or valves [1–3]. Hydrodynamic cavitation (HC) is specific multiphase flow of gases and liquid. If the pressure of mixture is equal to the saturated pressure, the vapour cavitation occurs. In case of pressure lower then atmospheric pressure, the air can be observed due to the release from the liquid. The mixture is then considered as the mixture of liquid, vapour and air [4]. The prevention of HC is an important concern in order to avoid severe damage due to the negative effects of cavitation such as erosions, vibrations and noises [1,2]. On the contrary, in recent years there is an increasing interest in using HC process in various important applications, especially in the field of environmental protection [5–8]. In order to cope with a decrease in available water resources worldwide, an increasing demand of drinkable water by population and the more restrictive environmental legislations on water quality, cavitation has come to be increasingly applied as innovative technique in the field of wastewaters treatment [5,9]. Due to its elevated oxidative capability, linked to its ability of generating highly reactive free radicals and thermal hot spots [10,11], HC was used to treat the increasing presence of bio-refractory, toxic or carcinogenic molecules and pathogens in wastewater streams [6,7].

Recently, a novel swirling jet-induced cavitation reactor, named Ecowirl (patented by Econovation GmbH, Germany and commercialized and optimized by Officine Parisi S.R.L., Italy), was successfully used by our research group in order to degrade a toxic and carcinogenic dye (Rhodamine B, RhB) from waste aqueous solutions [6], and to increase the activated sludge solubilisation and the aerobic sludge biodegradability in wastewater treatment plants [7,9]. Based on previous experimental results, it was observed that in the studied swirling jet-induced cavitation reactor some parameters and operating conditions such as fluid temperature, type of fluid, geometry of the cavitating device, flow rate, flow velocity and flow pressure can deeply influence the intensity of cavitation and the way in which it is generated [6,7]. Although the effectiveness of the swirling jet-induced cavitation reactor was experimentally proved, some difficulties such as not accurate measurements of flow velocity and flow pressure into the reactor due to its complex geometry as well as due to the presence of turbulent flow in cavitating conditions have highlighted the fact that a three-dimensional computational fluid dynamic CFD analysis was necessary.

Many researchers focused on the mathematical modelling of different HC devices such as orifice plates [12], nozzles [13], Venturi [12,14], swirling jet [15] or rotor-stator [16] systems. Numerical investigations have been conducted to predict cavitation and to determine whether computational methods can be used as a reliable tool

E-mail addresses: mancuso.giuseppe@hotmail.it, giuseppe.mancuso@ucd.ie.

https://doi.org/10.1016/j.ultsonch.2018.08.011
Received 29 May 2018; Received in revised form 27 July 2018; Accepted 10 August 2018

to evaluate the performance characteristics of cavitating devices. However, the definition of a mathematical problem can be difficult because it can include very complex geometries of HC devices in addition to turbulent flows and cavitating conditions.

Palau-Salvador et al. [17] performed numerical predictions of cavitation flows based on CFD for simple geometries, such as orifices, nozzles and Venturi systems, using the commercial CFD code FLUENT 6.1. Navickas and Chen [18] studied the internal flow characteristics of cavitating Venturi by means of the FLOW-3D three-dimensional fluid flow program. Their results indicated that numerical methods are effective in obtaining relative magnitudes of significant parameters affecting the performance of the cavitating device. In order to optimize a multi-hole injector nozzle, He et al. [19] modelled the three-dimensional nature of the nozzle flow investigating on the effect of the geometry and dynamics factors on the spray characteristics in turbulent and cavitating conditions. Müller et Kleiser [20] developed a numerical method for the vortex breakdown in a compressible swirling jet non-cavitating flow. Ashrafizadeh and Ghassemi [14] performed an experimental and numerical investigation on the performance of small-sized cavitating Venturi. Badve et al. [21] made a mathematical model describing the shear rate and pressure variation in a complex flow field created in a rotor-stator type HC reactor.

This work is the first step to fully understand the fluid dynamics into the swirling jet reactor. In this study, cavitation conditions were not take into account in order to get more accurate results on fluid dynamics into the reactor. Numerical simulations were implemented by means of a CFD software (ANSYS 16.2) with one- and two-phase models, with the main purpose of gain a better understanding of the fluid dynamics when cavitation does not occur. Then, noise measurements were performed during experiments both with and without cavitation. Fluid dynamics model results of this first step as well as noise measurements might be used in the future to include cavitation in the mathematical model.

2. Materials and methods

In order to widen the knowledge of the HC process into the studied swirling jet reactor and validate the mathematical model with experimental data, a series of tests were performed as reported in following sub-sections.

2.1. Experimental setup

Fig. 1 shows a schematic representation of the experimental setup used in the present work. It was a closed hydraulic loop and it consisted of (1) a feed tank filled with tap water, (2) a mono screw pump (3.0 kW, Netzsch Pumps & Systems GmbH Germany), (3) an inverter (Bonfiglioli Vectron - Active) to adjust the number of pump revolutions; (4) a flow-meter (G2 Stainless Steel Industrial Flowmeters, Great Plains Industries, Inc.); (5) a cavitating device (Ecowirl reactor); (6) a frequency-meter (PCE-VT 2700); three pressure transducers (Afriso) to measure the inlet (P1), the outlet (P2) and the vacuum (P3) pressure, respectively.

Fig. 2 shows a schematic representation of geometry of the studied swirling jet reactor, in which HC is generated by using a multi-dimensional vortices generator, consisting of a frustum-conical pre-swirling chamber (2) preceded by another chamber (1) where are located six injection slots (Fig. 2, cross section A-A) through which the flow enters generating a vacuum-core vortex (4) and a double cone chamber (3) where a triangular plate (Fig. 2, cross section B-B) is placed and in which the flow impacts generating cavitation [6,7] for inlet pressures higher than 2.0 bar.

2.2. Experimental tests

A feed tank of volume of 1000 L was filled with tap water at environment temperature (20 ± 2 °C). Water was pumped through the swirling jet reactor by using the mono screw pump. By adjusting the frequency, the inverter was used to control the pump flow rate (on the range of 0.21–6.57 $m^3 h^{-1}$) and the flow inlet pressure (0.25–4.0 bar). The methods included flow rate, flow pressure and vibration noise measurements. The flow-meter allowed measuring the flow rate of water before being introduced to the swirling jet reactor. Upstream, downstream and vacuum-core vortex pressures were measured by using pressure transducers. Cavitation noise measurements were made by using the frequency-meter. Then, water was discharged again back into the feed tank.

Based on results of previous experimental tests [6,7] and on negative pressure measurements in the swirling jet reactor vacuum zone (Fig. 1, (P3), Fig. 2, (4) and Table 1), the presence of HC at inlet pressures higher than 2.0 bar was assumed. Therefore, flow rate was kept as low as possible (inlet pressure of 0.25 bar) in order to collect data for the numerical simulation in non-cavitating conditions. On the contrary, inlet pressures higher than 2.0 bar were considered to perform noise measurements for cavitating flows.

2.3. Reactor geometrical configurations

In this study, two different geometric configurations of the swirling jet reactor were taken into account by varying the diameters of the six injection slots (Fig. 2, (1) and section A-A). In the studied swirling jet reactor, injection slots were characterized by different upstream and downstream diameters: configurations EW1 (10 mm upstream - 8 mm downstream) and EW2 (5 mm upstream - 3 mm downstream) were used, respectively (Table 1).

Firstly, a comparison between the two reactor configurations, EW1 and EW2, was carried out considering a non-cavitating flow at the same inlet flow pressure of 0.25 bar. In this phase, data collected from measurements were used to calibrate and validate the mathematical model. Reducing the injection slots diameters while keeping constant the inlet flow pressure, from configuration EW1 to configuration EW2, a decrease in both flow rate and outlet flow pressure was measured. Then, higher inlet flow pressures (ranged from 2.0 to 4.0 bar) were used in order to generate HC in the flow. Cavitation noise measurements were performed to quantify the intensity of HC at this stage. Detailed information of operating conditions and parameters is reported in Table 1.

2.4. Cavitation noise measurements

Cavitation noise measurements were performed by means of a frequency-meter (PCE-VT 2700) placed on the external surface at the entrance of the HC reactor double cone, Fig. 1. In this region, it is mainly expected to take place HC, resulting in subsequent noise generation. As the sound probe also received signals from the motor of the pump, it was necessary to reject the signal from motor's site. Tests were carried out using the two different swirling jet reactor configurations, EW1 and EW2, in cavitating flow conditions at inlet pressure of 2.0, 3.0 and 4.0 bar, respectively. Fast Fourier Transform (FFT) analysis was performed on the collected data in order to identify main noise frequencies [4,22].

2.5. Numerical simulations

A numerical approach was employed in order to investigate the fluid dynamics in the swirling jet reactor in non-cavitating flows considering the effect of the reactor geometry on the flow velocity and flow pressure distributions.

Based on experimental investigations, CFD simulations were performed by means of ANSYS FLUENT 16.2 software. First, only water phase was considered in the model. Then, since the flow through the swirling jet reactor is a multiphase fluid, a two-phase model was employed. The primary phase was defined as liquid water (Tables 1 and 2),