International Journal of Thermal Sciences 114 (2017) 229–240

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Characterization of high-pressure cavitating flow through a thick orifice plate in a pipe of constant cross section

Behrouz Ebrahimi [a, 1], Guoliang He [a, 1], Yingjie Tang [a], Matthew Franchek [a, **], Dong Liu [a, *], Jay Pickett [b], Frank Springett [c], Dan Franklin [d]

[a] Department of Mechanical Engineering, University of Houston, Houston, TX 77204-4006, United States
[b] Corporate Engineering, National Oilwell Varco, United States
[c] R&D/NOVOS, Rig Technology, National Oilwell Varco, United States
[d] Pressure Control Group, Rig Technology, National Oilwell Varco, United States

ARTICLE INFO

Article history:
Received 22 June 2016
Received in revised form
30 November 2016
Accepted 2 January 2017

ABSTRACT

To validate the applicability of basic cavitating flow theory for high-pressure flow systems, this paper presents a systematic study of high-pressure flow through a thick orifice plate in a constant cross-section pipe over a wide range of operating conditions. A combined theoretical, numerical and experimental approach was employed to explore the two-phase flow characteristics of both the non-choked and choked flows with a maximum upstream pressure of 5000 psi and a maximum Reynolds number of 2×10^6. For the flow configuration used in this work, a critical downstream-to-upstream pressure ratio of 0.45 was identified below which cavitation and flow choking will occur. Furthermore, it was found from the numerical models that the conventional one-dimensional analysis is inadequate in predicting the discharge coefficient and the condition for the onset of cavitation. Subsequently, new theoretical corrections suitable for high-pressure conditions were proposed, based on the numerical simulation results, and validated by the experimental measurements.

1. Introduction

Flow systems in the oil and gas industry are subject to extreme flow conditions, such as multicomponent/multiphase fluids and high-temperature/high-pressure environments, which can be further complicated by the transient conditions during emergency. Hence, it is crucial to control and safeguard these flow systems in real time for oil and gas exploration and production. Failure to do so can lead to disastrous consequences for life, economy and environment. In the BP Macondo incident, for example, the blowout preventer (BOP), which was designed to confine the drilling fluid to the well bore, failed to shut off the flow of high-pressure oil-gas mixture and close the well when the gas kick occurred [1,2]. Among various factors, erosion damage due to sand impact and cavitation was considered to be the main cause for the loss of the sealing function in the upper annular ram and the blind shear ram in the BOP failure [3]. A key issue to ensure the proper operation of BOPs and other high-pressure flow devices alike is to understand and foresee the hazardous flow conditions that may damage the equipment and cause malfunctioning. One particular flow condition of such adverse effect is cavitating flow, which has a significant effect on erosion and often acts as the precursor to cavitation damage.

Cavitating flow is encountered routinely in pressurized flow devices [4–6]. These devices are featured by varying cross sections along the flow path. As the flow passes through a restriction, the velocity accelerates and the pressure decreases due to the Bernoulli principle. If the local static pressure becomes lower than the vapor pressure corresponding to the fluid temperature, the liquid phase will be vaporized, thus inducing cavitation [7]. High compressibility in the liquid-vapor mixture, due to the presence of vapor phase, results in a significantly lower speed of sound (e.g., as low as a few tens of meters per second [4,8,9]) compared to that in single-phase flow. As cavitation intensifies due to, for instance, an increase in the upstream pressure or a decrease in the downstream pressure, the fluid velocity may reach the local sonic speed. When this happens,

* Corresponding author.
** Corresponding author.
E-mail addresses: mfranchek@central.uh.edu (M. Franchek), dongliu@uh.edu (D. Liu).
[1] Equal contributions.

http://dx.doi.org/10.1016/j.ijthermalsci.2017.01.001
1290-0729/

the flow rate ceases to increase regardless of further reduction in the downstream pressure and the flow is choked [10–12]. Subsequently, the vapor bubbles may collapse or implode if the fluid pressure recovers to a value higher than the vapor pressure. The bubble implosions are accompanied by high-speed microjets of fluid and shock waves, which, when impinging on the device surface, cause cavitation damage as well as high levels of noise and vibration [13–15]. Therefore, it is crucial to identify and characterize the cavitating flow conditions in order to predict the maximum achievable volumetric flow rate through the flow systems and to safeguard their safe operations.

Flow choking and cavitation have been investigated extensively in pressure-driven flow through various devices, including nozzles, poppets and valves. Since the typical flow configuration is represented by an orifice plate, a myriad of experimental and theoretical studies have focused on cavitating flow through orifices of different cross-sectional shapes [16–23]. Kojasoy et al. [16] established the theoretical framework for modeling the pressure drop in both single-phase and two-phase flow through thick- and thin-orifice plates with sudden expansion and sudden contraction. A one-dimensional (1D), single-phase model was first derived and then extended to two-phase flows by using a slip flow model originally proposed for turbulent two-phase pipe flows. The theoretical predictions were validated with experimental data using R-113 as the working fluid. The maximum upstream pressure in the experiments was 87 psi and the Reynolds number (defined in terms of the average orifice velocity and the orifice diameter) was in the range of 800–15,000. Abdulaziz [17] analyzed the cavitating process for water flow through a small type circular Venturi with a convergent part and a divergent part, and proposed a 1D model to predict the critical downstream pressure/upstream pressure ratio for the incipience of cavitation. The model prediction of the vapor void fraction was compared to the results obtained from the light reflection imaging technique. The working fluid was water and the maximum upstream pressure of the Venturi was 73 psi. Ramamurthi and Nandakumar [18] characterized the discharge coefficient of water flowing through small sharp-edged cylindrical orifices. It was found that the discharge coefficient scales with the Reynolds number and the aspect ratio in the attached non-cavitating flows; however, it only depends on the orifice diameter for separated flows and cavitating flows. Ashrafizadeh and Ghassemi [19] investigated the effect of the upstream and downstream pressures and the geometrical parameters on the mass flow rate through cavitating Venturis with similar geometries to that in Ref. [17]. Flow experiments were conducted using water to obtain the critical pressure ratio at the cavitation condition with the maximum upstream pressure of 290 psi. A computational fluid dynamics (CFD) model was also developed to simulate the flow through the Venturi. Nilpueng and Wongwises [20] presented the experimental data on the choked flow of HFC-134a through short-tube orifices. The upstream pressure was varied between 145 and 174 psi. The results indicated that the commencement of choked flow is dependent on the downstream pressure, inlet subcooling and the length-to-diameter ratio of the tube. Long et al. [21] studied the critical cavitating flow in a liquid jet pump under choking conditions. They measured the axial pressure distribution along the wall of the jet pump, which suggests liquid-vapor two-phase flow occurs when the downstream pressure is lower than a critical value. It was also observed the velocity of the two-phase mixture reaches the local sound velocity, leading to unchanged flow rate regardless of further decrease in the outlet pressure. Tomov et al. [22] experimentally investigated cavitating flow of water in a horizontal Venturi nozzle with a maximum inlet pressure of 58 psi. A high-speed camera was used to observe the cavitation regimes, namely, cloud cavitation, quasi-supercavitation and supercavitation. Subsequently, these regimes were aerated by injecting air bubbles. It was found that the phase diagram (a measure of the liquid void fraction) exhibits statistically symmetrical structures with characteristic lengths and frequencies in pure cavitation. However, the symmetry was broken in the aerated cavitation flow. Gaston et al. [23] formulated a theoretical model to describe the motion, growth and decay of cavitating bubbles in a Venturi flow. The liquid flow was solved by the boundary element method, and the 1D Rayleigh-Plesset equation was solved to acquire the velocity of the growing bubble at the interface between the liquid and vapor phase.

In addition to the aforementioned studies conducted at low pressures, there is a large body of literature on high-pressure cavitating flows; however, focused primarily on cavitation-induced erosion in cavitating jets or nozzles [24–28]. Soyama [25] investigated experimentally the effect of nozzle geometry on cavitation in shot peening applications using a high-speed water jet. The experiments were performed at upstream pressures of 2175 psi and 4350 psi and downstream pressures of 14.5 psi and 60.9 psi, respectively. The geometric factors, such as the standoff distance (the distance from the nozzle throat to the surface of the specimen), the throat diameter and the size of the nozzle outlet section, were optimized to increase the aggressive intensity of the cavitating jet. It was also found that a larger upstream/downstream pressure differential significantly enhances cavitation and increases the material erosion intensity. In a follow-up study, Soyama [26] characterized the surface cavitating erosion rates for different upstream/downstream pressure differential, nozzle geometry and standoff distance, and developed a database of the cavitation erosion resistance for various types of specimen materials. Grinspan et al. [27] devised a surface modification process using an oil jet with an upstream pressure up to 9000 psi to introduce the compressive residual stress and increase the hardness of the metal specimen. Cavitation bubbles were observed in the experiments that impinge on the specimen surface to cause severe plastic deformation. Giannadakis et al. [28] numerically simulated the cavitation process inside a diesel injector nozzle with a 7500 psi upstream pressure. The bubble growth, breakup, coalescence and turbulent dispersion were considered in the cavitation flow models. The liquid and vapor volume fractions and the liquid phase velocity were evaluated for different cavitation regimes. The numerical predictions were validated by the experimental data measured by high-speed optical imaging, computed tomography (CT) and laser Doppler velocimetry (LDV).

The literature survey indicates that although the basic theories of cavitating flow have been developed, the available experimental results are restricted to either low-pressure conditions or high-pressure flows through specific cavitating jets or nozzles. To explore the validity of the cavitating flow theories and to provide critical information to relevant engineering applications, this paper presents a comprehensive study of high-pressure cavitating flow through a thick orifice plate over a wide range of operating conditions. The flow characteristics are presented as a relationship of the volumetric flow rate with respect to the upstream/downstream pressure differential. The maximum upstream pressure achieved is 5000 psi. The rest of the paper is organized as follows. First, existing theoretical models are introduced for both single-phase and cavitating flows through an orifice plate. Then, the CFD model for solving the cavitating flow is discussed. Subsequently, the test setup and the experimental methods are presented. Finally, results from the theory, CFD and experimentation are compared, and the key deficiencies of the 1D assumption that is being widely used in the cavitating flow literature are discussed.