Flow Measurement and Instrumentation 54 (2017) 56–67

Contents lists available at ScienceDirect

Flow Measurement and Instrumentation

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

Experimental and numerical investigation of the cavitation-induced choked flow in a herschel venturi-tube

S. Brinkhorst[a,*], E. von Lavante[a], G. Wendt[b]

[a] Chair of Fluidmechanics, University of Duisburg-Essen, 47057 Duisburg, Germany
[b] Head of Department "Liquid Flow", Physikalisch-Technische Bundesanstalt PTB, 38116 Braunschweig, Germany

ARTICLE INFO

Keywords:
Cavitation
Choked flow
Herschel venturi-tube
Cavitating nozzle
Liquid flow metering

ABSTRACT

Due to their simple geometry, cavitating Venturi nozzles (CV) are a long time subject of experimental as well as numerical investigations. However, research mostly focused on certain aspects like the comparison of experimental data with numerical cavitation models or the spray development of diesel injection nozzles, but rarely on the choked flow condition itself, especially with regard to liquid flow measurement.

If the pressure decreases due to the local acceleration of the flow to the respective vapor pressure, a choked flow condition similar to the well-known critical flow Venturi-nozzles (CFVN) develops. For the purpose of gaining further insight into the choked flow condition with respect to liquid flow measurement, high-speed camera investigations of a transparent Herschel Venturi-tube configuration, also known as classical Venturi-tube, were performed. Together with pressure and flow rate measurements, they demonstrated the overall stable flow behavior under choked conditions. With additional numerical investigations, phenomena during the onset of the choked condition were clarified. Furthermore, a simple correlation for the calculation of the actual flow rate during the choked condition, including a temperature correction was proposed.

1. Introduction

As a practical realization of the so called traceability requires, hydraulically cavitating Herschel Venturi-tubes were investigated for their highly stable flow rates under choked flow conditions. In addition to accuracy, the traceability of a given measuring device is of essential importance, in particular with respect to the general guarantee of uniformity and comparability of measurements. In gas flow measurement, the so-called critical flow Venturi-nozzles (CFVN) have been widely established as an efficient transfer standard during the last two decades. Comparable solutions for liquid flow measurement did not exist to this day. The analogies observed in experiments with Venturi-nozzles using gas or liquids as working fluids, already observed in 1930 by Ackeret, led the Physikalisch-Technische Bundesanstalt (PTB) to start detailed investigations of cavitating Venturi-nozzle (CV) flows. The intention was to clarify their applicability in liquid flow measurement comparable with the use of critical nozzles for gas metering.

Though Ackeret has already mentioned the experimentally observed analogies between CVs and CFVN in 1930, his work was not focused on flow rate investigations [1]. Numachi et al. investigated different Venturi-tubes, nozzles and orifices under cavitating conditions [2–5]. However, the differential-pressure flow rate measuring devices were investigated to obviate incorrect measurements due to effects of possible cavitation at certain small pressure ratios. Nevertheless, the phenomenon of an enforced constant mass flow rate through the nozzles, while cavitation in the nozzle throat occurred, was investigated and described. Still, a well-directed exploitation of this effect for flow rate metering applications has not been considered then.

Recently, some experimental work has been published focusing on the use of CVs as liquid flow meters, with special emphasis on the choked flow condition. Ghassemi et al. showed the general possibility of the application of CVs as liquid flow meters [6], while Abdulaziz gained further insights into the cavitation process by using an optical flow visualization method [7]. Rudolf et al. characterized experimentally the cavitation regimes of a Herschel Venturi-tube by evaluating the loss coefficient, thereby revealing a unique behavior during the choked condition, as well as during its onset [8]. Ashrafizadeh et al. performed numerical and experimental investigations of different Venturi-tube geometries designed as flow-control devices [9]. Recent experimental investigations by Schmidt using CVs confirmed the high stability of the flow rate during the choked condition [10].

The investigations mentioned so far were supplemented by the already published numerical investigations of CVs by the authors [11,12]. These investigations were mainly focused on geometry influ-

* Corresponding author.
E-mail address: Sven.Brinkhorst@uni-due.de (S. Brinkhorst).

http://dx.doi.org/10.1016/j.flowmeasinst.2016.12.006
Received 13 July 2016; Received in revised form 15 November 2016; Accepted 12 December 2016
Available online 16 December 2016
0955-5986/

Table 1
Properties defined for the numerical simulations, with μ_l and μ_v representing the liquid and vapor viscosity respectively.

property	value
pressure, inlet (P_{01})	3.764 bar (total pressure)
vapor pressure	2333.21 Pa
α_v, inlet	0
T	298 K
ρ_l	998.34 kg/m^3
ρ_v	0.0173 kg/m^3
μ_l	1.001477 mPa s
μ_v	0.009727 mPa s
number density of seeds	1e12 /m^3
initial seed radius	0.001 mm
turbulent intensity, inlet	1%
C_{prod}, C_{dest}	1

ences on the stability and accuracy of the flow rate, with regard to flow metering applications. They demonstrated the advantages of Herschel-tube configurations compared to a typical ISO 9300 CFVN geometry, with respect to the constancy of the flow rate for different pressure ratios within the cavitation induced choked flow condition Table 1.

The recently performed experimental and numerical investigations presented in this paper revealed a new phenomenon during the onset of the choked flow condition, namely the sudden expansion of the vapor cloud. Furthermore, due to high-speed camera observations with up to 140.000 fps, reasons for the steadiness and high stability of the vapor cloud and thus the choked flow rate will be presented.

For the numerical simulations, the commercial code CD-adapco STAR-CCM+10.04.011 (CCM+) was used, solving the three-dimensional, Reynolds-averaged Navier-Stokes (RANS) equations for turbulent, incompressible, unsteady, cavitating two-phase flow.

The experiments were carried out using the experimentation and water meter test rig (EWZP) of the PTB.

2. Numerical cavitation model

The commercial CFD program CCM+, used an interface capturing approach via the Volume-of-Fluid-method (VOF-method) to model the cavitating two-phase flow. The cavitation itself was based on the inertia controlled bubble growth theory obtained by Rayleigh [13]. The capability of this solver has already been extensively validated by different authors [14–17]. Due to the VOF-method treating the vapor and liquid phase as a mixture, only one set of the mass and momentum equations is solved, where the physical properties of the mixture or equivalent fluid were functions of the physical properties of its constituent phases and their volume fractions. In addition to the mass and momentum conservation, an equation had to be solved governing the transport of the volume fraction α between the two phases:

$$\frac{\partial \alpha_v}{\partial t} + \vec{\nabla}\cdot(\alpha_v \vec{u}) = \Gamma_v \tag{1}$$

The subscript v denotes the vapor phase, while the definition of the volume fraction of vapor is: $\alpha_v = V_v/V$, with V_v being the volume of vapor inside the control volume V. The phase change due to cavitation is governed by the source term Γ_v on the right side of Eq. (1), whereby Γ_v is usually split into two terms $\Gamma_v = \dot{m}^+ + \dot{m}^-$. $\dot{m}^+$ denotes the rate of evaporation and $\dot{m}^-$ the rate of condensation. In fact, most cavitation models differ only in the formulation of this source term. In CCM+, the common Sauer cavitation model was used for the calculation of the phase change. The Sauer cavitation model is based on the assumptions of having homogeneously distributed seed bubbles with equal radius R_b. These bubbles remain spherical at all times and the number density of seeds n_0 together with the initial seed radius $R_{b,0}$ was a model parameter to be set. The two phases were treated as incompressible and slip velocity between bubbles and liquid was neglected. Thus, according to the Sauer cavitation model the terms modeling the rate of evaporation and condensation were:

$$\dot{m}^+, \dot{m}^- = C\frac{\rho_v \rho_l}{\rho_m}\frac{3\alpha_v(1-\alpha_v)}{R_b}\sqrt{\frac{2}{3}\frac{p_v - p}{\rho_l}} \tag{2}$$

With $C = C_{prod}$ for evaporation and $C = C_{dest}$ for condensation and the growth/collapse of a bubble being calculated by the simplified Rayleigh-Plesset equation. The vapor pressure p_v is the saturation pressure corresponding to the liquid temperature, p denotes the pressure surrounding the bubble, ρ_l denotes the liquid density and ρ_m denotes the density of the vapor-liquid mixture, calculated by:

$$\rho_m = \alpha_v\rho_v + (1-\alpha_v)\rho_l \tag{3}$$

Due to the use of the simplified Rayleigh-Plesset equation, viscous and surface tension effects on the rate of change of the bubble radius were neglected.

Turbulence was modeled by the realizable k-ϵ two-layer model, including an all-y^+ wall treatment [18]. The chosen turbulence model has already been proven to predict satisfying results for cavitating flow [9,19,20].

Nevertheless, it should be mentioned that no preferred turbulence model, as well as cavitation model, exists in literature. This led to a plethora of cavitation-/turbulence-model combinations. Though the choice of either model usually has a large impact on the numerical results, at least globally matching results between each model can be obtained by carefully adjusting the empirical constants [21].

For time and space discretization, second order accuracy was used. All numerical results presented were obtained with full 3D simulations. For a proper time wise resolution a time step of 10^{-4} s was chosen. The numerical mesh was modeled based on a grid sensitivity analysis presented in [11] and was created using the CCM+ grid generator. With a grid refinement in the cavitating zone the resulting mesh consisted of about 1 million cells.

For all simulations the following boundary conditions (b.c.) have been applied: A stagnation pressure inlet b.c. for defining a constant total pressure at the inlet, while a constant static pressure has been defined at the outlet; at the wall the no-slip b.c. was used.

3. Experimental setup

For the experimental investigations, the PTB provided their experimentation and water meter test rig (EWZP), with a schematic representation shown in Fig. 1. The volumetric flow rate (Q) was measured by a magnetic flow meter (MID). Furthermore, the differential pressure across the nozzle (ΔP) as well as the static pressure (P1) and the temperature at the nozzle inlet were measured. The EWZP has an expanded measurement uncertainty of 0.05%. Details of the meters used can be found in Table 2.

For the measurements, the stagnation pressure in front of the nozzle could be controlled by the rotational speed of the pump in combination with the valve position. The valve was located about 50 pipe diameter behind the nozzle. For visual inspection of the cavitation, as well as the use of high-speed camera recordings, the investigated Herschel Venturi-tube was manufactured out of acrylic glass. The dimensions of the experimentally and numerically investigated Herschel-tube are presented in Fig. 2. For the visualization of vapor

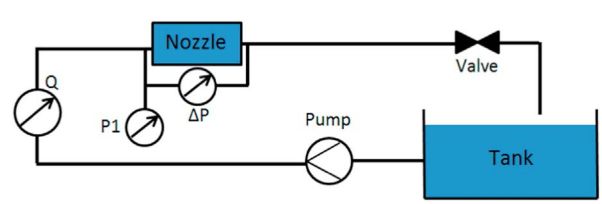

Fig. 1. Schematic representation of the EWZP of the PTB.