International Journal of Heat and Fluid Flow 64 (2017) 41–54

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

# Numerical investigation of periodic cavitation shedding in a Venturi

Boris Charrière[a], Eric Goncalves[b,*]

[a] Université Grenoble-Alpes, LEGI, UMR 5519 CNRS, 38041 Grenoble, France
[b] ISAE-ENSMA, institut Pprime, UPR 3346 CNRS, 86961 Chasseneuil, France

ARTICLE INFO

*Article history:*
Received 4 May 2016
Revised 31 January 2017
Accepted 31 January 2017

*Keywords:*
Cavitation
Re-entrant jet
Pressure waves
1-fluid simulation
Void ratio equation

ABSTRACT

Unsteady partial cavitation is mainly formed by an attached cavity which presents periodic oscillations. Under certain conditions, instabilities are characterized by the formation of vapour clouds convected downstream the cavity and collapsing in higher pressure region. Two main mechanisms have been identified for the break-off cycles. The development of a liquid re-entrant jet is the most common type of instabilities, but more recently, the role of pressure waves created by the cloud collapses has been highlighted. This paper presents one-fluid compressible simulations of a self-sustained oscillating cavitation pocket developing along a Venturi geometry. The mass transfer between phases is driven by a void ratio transport equation model. The importance of traveling pressure waves in the physical mechanism is put in evidence. Moreover, the importance of considering a non-equilibrium state for the vapour phase is exhibited.

## 1. Introduction

Cavitation is the formation and activity of vapour bubbles or cavities. It may occur through the development or the enlargement of bubbles or cavities present in the liquid. The phenomena is well observed in ship propellers, turbines, pumps and hydrofoils and is often called *hydraulic or hydrodynamic cavitation* (Knapp et al., 1970). The cavitation development may be the origin of several negative effects, including noise, vibrations, performance alterations, erosion and structural damages. This makes cavitation an important issue in design and operation, which should be controlled, or at least well understood.

Among the principal types of cavitation that may develop, partial cavitation pockets are often observed in hydraulic machines and is known to be responsible for severe damage. Such cavitating flows can have a complex behaviour where the cavity is characterized by a strong unsteadiness, transient cavities shedding downstream and a completely 3D flow even in a 2D configuration. Two regions are generally of interest and are driving the cavitation pattern: the cavity detachment region which is related to the cavitation onset, and the cavity closure where vapour structures formed in the low pressure zones are transported downstream and collapse violently when they reach the higher pressure zone. These shedding of vapour clouds may have different origins. In some cases, the clouds are generated by vortex shedding filled with bubbles or produced by periodic disturbances imposed by the mean flow. For pumps and turbines, these disturbances can be caused by rotor-stator interactions. The experimental and numerical studies identify the presence of two main mechanisms: a re-entrant jet and pressure waves generated by the collapse of larger structures. The re-entrant jet is mainly composed of liquid which penetrates the attached cavity from downstream and flows upstream along the solid surface. The motion of a re-entrant jet is central for partial cavitation instabilities but the mechanism which drives the phenomenon remains unclear.

The development of a liquid re-entrant jet was primary described by Knapp et al. (1970). de Lange et al. (1994); de Lange and de Bruin (1998) proposed an experimental study of the phenomenon on NACA profiles. Initially, sheet cavity extends on the suction side. When it reaches a sufficient length, the re-entrant jet set up at the closure of the cavity and is directed toward the leading edge. Then, it divides the cavity in a rear part transformed into bubble cloud and convected by the main stream and a front part reduced to a tiny sheet cavity which grows again and starts a new cycle. Kubota et al. (1989) measured the velocity of the cloud and showed it was convected with a lower velocity than the bulk flow. Finally, the cloud collapses when it reaches the downstream high pressure area. The same cycle was depicted by Kawanami et al. (1997) and they showed that the phenomenon can be aborted by introducing an obstacle along the wall. The jet velocity was measured by Pham et al. (1999) and was found to be of the same order of magnitude as the free-stream. Callenaere et al. (2001) have

---

[*] Corresponding author.
*E-mail addresses:* Eric.Goncalves@ensma.fr, eric.goncalves@ensma.fr (E. Goncalves).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.01.011

**Nomenclature**

*Abbreviation*

| | |
|---|---|
| CFD | Computational Fluid Dynamic |
| CFL | Courant–Friedrichs–Lewy number |
| DFT | Direct Fourier Transform |
| EOS | Equation Of State |
| KL | Smith $k-\ell$ turbulence model |
| RANS | Reynolds-Averaged Navier–Stokes |
| RMS | Root Mean Square |

*Greek symbols*

| | |
|---|---|
| $\alpha$ | Void fraction |
| $\Psi$ | Second turbulence variable |
| $\kappa$ | von Karman constant |
| $\lambda$ | Mixture thermal conductivity |
| $\lambda_t$ | Mixture turbulent thermal conductivity |
| $\mu$ | Mixture dynamic viscosity |
| $\mu_t$ | Mixture dynamic eddy viscosity |
| $\rho$ | Mixture density |
| $\rho_l^{sat}$ | Density of liquid at saturation |
| $\rho_v^{sat}$ | Density of vapour at saturation |
| $\sigma$ | Cavitation parameter |
| $\overline{\overline{\tau}}$ | Mixture total stress tensor |

*Latin symbols*

| | |
|---|---|
| $C_p$ | Heat capacity at constant pressure |
| $c_{sinus}$ | Minimum speed of sound in the mixture |
| $e$ | Mixture internal energy |
| $E$ | Mixture total energy |
| $F_c$ | Convective flux density |
| $F_v$ | Viscous flux density |
| $k$ | Mixture turbulent kinetic energy |
| $\ell$ | Integral turbulence length scale |
| $P$ | Mixture pressure |
| $Pr$ | Prandtl number |
| $Pr_t$ | Turbulent Prandtl number |
| $Q$ | Mixture total heat flux |
| $Re$ | Reynolds number |
| $S$ | Source term |
| $S_{ij}$ | Stress tensor |
| $T$ | Mixture temperature |
| $u^+$ | Non dimensional wall velocity |
| $u_i$ | Velocity compopent |
| $U_\tau$ | Friction velocity |
| $\vec{V}$ | Velocity vector |
| $w$ | Vector of conservative variables |
| $y^+$ | Non dimensional wall distance |

*Subscript*

| | |
|---|---|
| *inlet* | Inlet value |
| *k* | k-phase |
| *ref* | Reference value |
| *w* | Wall value |

*Superscript*

| | |
|---|---|
| *t* | Turbulent part |
| *v* | Viscous part |

found, using LDV measurements, that the re-entrant jet instability occurs when two conditions are satisfied: first, the adverse pressure gradient must be large enough, which imposes a maximum cavity length, second, the cavity must be not too thin, which imposes a minimum cavity length.

The collapse of a shed bubble cloud is known to create pressure waves, often orders of magnitude greater than the pressure in the mean flow. Reisman et al. (1998) suggested the presence of shock waves as cloud collapse mechanism, which causes rapid change in the void fraction distribution. Arndt et al. (2000) showed for a hydrofoil geometry that two competitive mechanisms can be responsible for cloud cavitation. From experimental measurements coupled with numerical simulations, the study distinguished each mechanism using the quantity $\sigma/2\alpha$, where $\sigma$ is the number of cavitation and $\alpha$ the angle of attack. For large values a re-entrant jet was found to dominate whereas for smaller values bubbly flow shock waves dominated. The transition was found to occur for $\sigma/2\alpha \simeq 4$. Leroux et al. (2004); (2005) has suggested that these pressure pulses may contribute to the motion of the re-entrant jet and the cloud cavitation instability. High-speed visualizations of Saito and Sato (2007) and Sato et al. (2013) about periodic cloud shedding in a convergent-divergent nozzle suggested that the re-entrant motion was triggered by the collapse of shedding clouds and the propagation of bubble collapse. The propagation speed toward upstream was estimated to be on the order of 200–300 m/s. Stanley et al. (2011); (2014) described a complex mechanism causing the instability for periodic cavitation shedding in a cylindrical orifice. Using high-speed visualizations, they identified a combination of a travelling pressure wave generated by the bubble cloud collapse and a translational motion of the re-entrant jet, each with distinctly different velocities. The most recent studies have been proposed on a 8° Venturi nozzle by Ganesh (2015). Based on Laser Doppler Velocimetry (LDV) and X-ray densitometry measurements, their experimental works identified two mechanisms: a re-circulating flow and condensation shock waves, which is the most dominant for a periodic dynamic.

In this paper, we investigate a self-sustained oscillating cavitation pocket developing on a 8° Venturi nozzle by using compressible RANS simulations. For this geometry, the re-entrant jet phenomenon was observed and described, both experimentally (Stutz and Reboud, 1997; Aeschlimann et al., 2013) and numerically (Coutier-Delgosha et al., 2003; Shin et al., 2003; Senocak and Shyy, 2004; Chen et al., 2006; Goncalves, 2011; LeMartelot et al., 2013). The present study focuses on the traveling waves phenomenon and its role in the periodic cloud shedding. Numerical simulations are performed using an in-house finite-volume code solving a four-equation system composed by three conservation laws for mixture quantities and a supplementary equation for the volume fraction of vapour. The mass transfer between phases appearing explicitly in the formulation is closed by assuming its proportionality with the mixture velocity divergence (Goncalves, 2013; Goncalves and Charriere, 2014). Validation and analysis are done with experimental measurements (time-averaged void ratio and velocity profiles and shedding frequency). The importance of the pressure waves propagation in the physical mechanism is exhibited. Moreover, the effect of a non-constant vapour density is studied, that is to say the vapour phase can reach a metastable state.

This paper is organized as follows. We present first the governing equations, including the physical models, followed by an overview of the numerical methods adopted. The Venturi configuration and the associated numerical parameters are then presented. Unsteady simulations are described and compared with experimental data. A deeper analysis of the pressure waves phenomenon is proposed by using spatio-temporal correlations. The last part deals with conclusions and future investigations.

## 2. Equations and models

The numerical simulations are carried out using an in-house finite-volume code solving the one-fluid compressible Reynolds averaged Navier–Stokes (RANS) system.