Energy Conversion and Management 78 (2014) 776–786

Contents lists available at ScienceDirect

Energy Conversion and Management

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

Numerical simulation and experimental validation of the cavitating flow through a ball check valve

José R. Valdés [a,*], José M. Rodríguez [a], Raúl Monge [a], José C. Peña [a], Thomas Pütz [b]

[a] *Instituto Tecnológico de Aragón, María de Luna 8, 50018 Zaragoza, Spain*
[b] *TRW Automotive, Carl Spaeter Strasse, 56070 Koblenz, Germany*

ARTICLE INFO

Article history:
Received 20 September 2013
Accepted 25 November 2013
Available online 24 December 2013

Keywords:
Cavitation
Valves
Fluid flow
Numerical simulation
CFD
Testing
Validation

ABSTRACT

The main objective of this work is to perform and validate a series of CFD simulations of the cavitating flow through a ball check valve. Experimental tests are performed in order to obtain the mass flow rate through the valve under different operating conditions, inducing or preventing the appearance of cavitation by conveniently adjusting the pressure level on the valve outlet port. The measurements are compared with the results of numerical (CFD) simulations of the fluid flow through the valve, with and without the inclusion of a cavitation model. The characteristic flow coefficient of the valve and the hydraulic forces on the ball are analysed.

1. Introduction

The aim of this work is to run and validate numerical CFD simulations of the liquid flow through a ball check valve with and without cavitation. The ultimate aim is to validate the accuracy of the cavitation model used in the CFD simulations that have been run in order to reproduce the incompressible, turbulent flow through a ball check valve used in the Hydraulic Control Unit of an automotive ABS system. Depending on the operating conditions of the valve, cavitation phenomena may or may not appear, yielding significant differences in the mass flow rate through the valve and thus in the flow coefficient of said valve.

Cavitation is a phase transformation that occurs in liquid flows when the local pressure drops below the saturation pressure, with gas cavities that appear, develop and finally collapse when the pressure is high enough. In hydraulic components, cavitation may lead to problems such as vibrations, pressure pulsations, noise and erosion on solid surfaces. Therefore, cavitation and its potential effects must be considered in the design process of hydraulic systems.

Valves are generally used to control the flow rate in hydraulic systems. Ball check valves are designed to open under certain conditions and allow an alternative fluid discharge path. In the design of such valves, it is important to know the characteristics of the flow inside the valve, as well as the flow coefficient for its subsequent application in design and validation phases; for example, in the development of lumped parameter models of complete hydraulic systems for control, performance or stability analysis [1]. In many cases, because the valve must work with outlet pressure levels that are close to the atmospheric pressure, the appearance of cavitation is not avoidable. However, it is necessary to know it and take into account its influence on the mass flow rate, flow forces and flow coefficients. The inception of cavitation yields additional energy losses and density variations that can significantly decrease the mass flow rate, rendering the fluid discharge process less efficient. The flow forces acting on the moving parts are also modified due to the limitation of the minimum static pressure that can be reached. In hydraulic control valves, having different forces for similar pressure differences is a fact that must be accurately known in order to correctly design the control software. In check valves, the net forces acting on the ball must be analysed in order to ensure that the valve opens when necessary.

Existing cavitation models fall into two classes: the Volume Of Fluid method based on the interface tracking and the homogeneous equilibrium flow method [2]. The VOF method deals with bubble dynamics by solving the vapour–liquid interface, but most of the practical cavitating flows are approached using the homogeneous flow theory, due to its reduced complexity and computational cost. In this theory, the fluid is considered as a vapour–liquid mixture without explicit phase interfaces containing a large number of spherical bubbles where the liquid–vapour mass

---

[*] Corresponding author. Tel.: +34 976011160.
E-mail address: jrvaldes@ita.es (J.R. Valdés).

0196-8904/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.enconman.2013.11.038

Nomenclature

A	cross-sectional area
d	contraction small diameter
D	contraction large diameter
K	loss coefficient
n	bubble density
P	static pressure
P_{in}	inlet pressure
P_T	total pressure
Q	volume flow
R_b	bubble radius
V	velocity

Greek symbols

α	vapour volume fraction
Δ	increment
ϕ	valve orifice diameter
ξ	flow coefficient
μ	viscosity
ρ	density

Subscripts

l	liquid phase
v	vapour phase
b	bubble

transfer is governed by a vapour transport equation. This equation contains two source terms that account for the mass transfer and that are modelled based on the Rayleigh–Plesset equation [3], which describes the growth of a single vapour bubble in a liquid.

Many cavitation models based on the homogeneous flow theory have been put forward in recent years, such as Singhal et al. [4], Schnerr and Sauer [5], Kunz et al. [6] or Zwart et al. [7].

The "full cavitation model" developed by Singhal [4] accounts for all first order effects (phase change, bubble dynamics, turbulent pressure fluctuations and non-condensable gases dissolved in the fluid). The expressions for the phase-change rates, which depend upon the local flow conditions as well as the fluid properties, are derived from a reduced form of the Rayleigh–Plesset equation for bubble dynamics. The rate expressions employ two empirical constants, calibrated with experimental data covering a very wide range of flow conditions.

The Schnerr–Sauer model [5] follows a similar approach to derive the expression for the net mass transfer from liquid to vapour, but uses a different expression to connect the vapour volume fraction to the number of bubbles per volume of liquid. The Zwart–Gerber–Belamri model [7] uses a mass transfer expression which is similar to that of Singhal's, except that it is related to the vapour phase density, but not to the liquid phase and mixture densities. It also employs two empirically calibrated coefficients for evaporation and condensation and a correction to the vapour volume fraction to account for the incorrect assumption that the cavitation bubbles do not interact with each other, which is only true during the earliest stages of cavitation when the bubble grows from the nucleation site.

Experimental measurements and CFD simulations of the cavitating flow in different hydraulic systems are widely reported in the literature. Chern et al. [8] performed an analysis, based on experimental observations, of the flow through a ball valve and the appearance of cavitation. Jia et al. [9] performed an analysis of the cavitating flow in a conical spray nozzle by means of numerical CFD simulations using Singhal's cavitation model and the Realizable k–ε turbulence model. The mass flow results are compared with those of experimental measurements, showing a fairly good correlation, with some significant differences for high pressure differences, due to the use of a 2D model. Salvador et al. [10] performed a study of the internal flow in diesel injector nozzles modelling the cavitating flow as a homogenous mixture of liquid and vapour and an RNG k–ε model for modelling turbulence. Before carrying out an analysis of the influence of the needle lift by means of CFD, a validation of the mass flow rate and momentum flux is performed at full needle lift conditions, showing a good agreement. Casoli et al. [11] performed a CFD analysis of a homogenizing valve using a two-phase flow, the Singhal model to describe phase changes and the Standard k–ε approach to model turbulent fluctuations. An experimental validation of the mass flow rate as a function of the pressure difference showed a good correlation using the two-phase flow, but not so with a one-phase flow. Mimouni et al. [12] performed an analysis of the numerical simulation of cavitation phenomena with the NEPTUNE CFD code, which is based on the resolution of the mass, momentum and energy balance for both liquid and vapour phases, and the occurrence of cavitation by nucleation at the wall or by pre-existing cavitation nuclei. The model was validated by comparison with experimental measurements of the void fraction in a critical water flow in a nozzle, obtaining a good agreement under the assumption that most of the cavitation nuclei come from the vapour micro bubbles generated at the wall. Computations of cavitation development downstream an orifice showed also good agreement with experimental visualizations.

Li et al. [13] presented a study using a modified k-ω model to predict the unsteady cavitating flows around 2D and 3D hydrofoils and modelling cavitation with the Schnerr–Sauer cavitation model. The results were qualitatively in agreement with experimental observations of formation and transport of cavitating vortices, but under-predicted the lift coefficients. Zhao et al. [2] performed a numerical simulation and validation of the cavitating flow on a 2D NACA0015 hydrofoil under high pressure and temperature. The Singhal cavitation model was adopted combined with an improved RNG k–ε turbulence model.

These analyses and validations indicate that CFD simulations using homogeneous cavitation models are a valid tool to investigate cavitating flows. All three main k–ε approaches for modelling turbulence (standard, RNG and Realizable) seem to provide good approximation to the measured flows. However none of the previously mentioned works apply the Schnerr–Sauer cavitation model for valve flow simulation, neither do other more recent works such as those of Shang [14], who has applied and validated the Schnerr–Sauer model in external flows around blunt bodies of submarine shape; Aung and Li [15] have applied the Singhal model to an electro-hydraulic servo valve; Mohan et al. [16] have coupled the Schnerr–Sauer model to a spray model for simulating a fuel spray; Li et al. [17] have applied the Schnerr–Sauer model to the cavitating flow around a hydrofoil and Zhang and Chen [18] have applied the Zwart–Gerber–Belamri model to investigate the cavitating flow within a slanted axial-flow pump.

The present work aims to validate the Schnerr–Sauer model for the flow through a ball check valve. Furthermore, no papers have been located that provide numerical and experimental results for a hydraulic valve with and without cavitation, under the same pressure jump conditions. This paper will show numerical and experimental results of the mass flow through a ball valve, under