

Cavitating Flows in Hydraulic Multidimensional CFD Analysis

F. Franzoni, M. Milani and L. Montorsi

DISMI – University of Modena and Reggio Emilia

Copyright © 2008 SAE International

ABSTRACT

The effect of cavitation plays a fundamental role in the hydraulic components design and the capability of predicting its causes and characteristics is fundamental for the optimization of fluid systems.

In this paper, a multidimensional CFD approach is used to analyze the cavitating phenomena typical of hydraulic components using water as operating fluid. An open source fluid-dynamics code is used and the original cavitation model (based on a barotropic equation of state and homogeneous equilibrium assumption) is extended in order to account also for gases dissolved in the liquid medium. The effect of air dissolution into liquid water is modeled by introducing the Henry law for the equilibrium condition, and the time dependence of solubility is calculated on a Bunsen Coefficient basis. Furthermore, a simplified approach to turbulence modeling for compressible flows is coupled to the cavitation model and implemented into the CFD code. The turbulence effects on cavitating regions are addressed for different operating conditions.

In the analysis, both basic geometries and water hydraulic poppet valves are addressed, and the numerical results are compared to experimental measurements available in literature. In particular, the reference test case with abrupt section change geometries, such as the forward facing step, is investigated. The recirculating regions, the vena contracta position, the reattachment point and the pressure and velocity fields are calculated under conditions where cavitation is expected. Furthermore, the diverging and converging flows in hydraulic valves are simulated and the influence of the seat shape on cavitation onset, pressure distribution and discharge coefficient is discussed.

INTRODUCTION

Key feature in the design process of fluid power components and systems is the capability of controlling or avoiding the cavitating phenomena. The problems

caused by cavitation are widely known [1, 2] and they can lead to efficiency loss, vibrations and noise, unexpected change in the characteristics of flow rate and flow forces and even erosion of the components when the phenomenon is particularly aggressive [3].

Therefore, the possibility of predicting cavitation is fundamental in hydraulic components design and both experiments and numerical simulation have to be employed and integrated in order to help the understanding of the basics of cavitation occurrence. Particularly, numerical analysis can provide a great amount of data that can extend the experimental area and deepen the insight of the physical phenomenon [4 - 6]. Moreover, due to the increasingly development of computers and numerical tools, the numerical simulation cost in terms of human resources and computational means are now not only acceptable, but also advantageous.

Nevertheless the accuracy and reliability of the numerical models have to be addressed when approaching a new problem or when they are modified to account for a more detailed description of the physical process. The model sensitivity with respect to the fluid-dynamics characteristics of the case studied (such as the Reynolds number in the critical sections, the geometry complexity and the boundary conditions) must be properly addressed before adopting the numerical tool as a design tool.

This paper focuses on developing and tailoring an open source multidimensional CFD code to the analysis of the internal flow-field in hydraulic components. The attention is devoted to the cavitation process which is modeled by means of an approach based on a barotropic equation of state and homogeneous equilibrium assumption.

In literature many approaches to modeling cavitation can be found with varying complexity and computational cost. Principally they can be subdivided into two main groups on the basis of the explicit phases they are accounting for. The first approach calculates the different phases separately, i.e. liquid, vapor and eventually the released gases, and the interface among

them has to be tracked using different methods [8]. In the second approach, only one explicit phase is calculated and the properties of the liquid, vapor and eventually released gases are lumped into one equivalent fluid. The cavitation is then modeled following different methods. Several models propose a bubble dynamic model based on Rayleigh equation for vapor bubbles, in which the cavitation vapor bubbles have to be assumed present initially in the flow [9, 10]. Other methods are based on the Gibbs free energy in order to determine the bubble nucleation events occurring in a unit volume per unit time [11, 12]. Most common methods for the simulation of cavitating flows are based on a barotropic equation of state to describe the constituent relationship between pressure and density [13 - 16].

In this paper the cavitation model based on a barotropic equation of state and homogeneous equilibrium assumption, described in [16], and implemented into the OpenFOAM code is extended in order to account for the effect of dissolved gas. Moreover, a simplified approach to modeling the turbulence effects for compressible flows is coupled to the cavitation model.

The numerical model is validated by comparing the results with the experimental measurements available in literature for an abrupt section change and for a water hydraulic poppet valve. In both test cases different geometries and operating conditions where cavitation is expected are simulated and the results are discussed in terms of recirculating regions, the vena contracta position, the reattachment point and the pressure and velocity fields. The predictive capabilities of three numerical approaches to the reference cases are addressed. In particular the performance of a numerical analysis without cavitation model, with cavitation model and laminar flow assumption and with cavitation model accounting for turbulence is investigated.

Furthermore, the diverging and converging flows in hydraulic valves are simulated and the influence of the seat shape on cavitation onset, pressure distribution and discharge coefficient is discussed.

NUMERICAL APPROACH TO CAVITATION

The CFD analysis is carried out by means of the open source computational fluid dynamics code OpenFOAM. Bounded central differencing is used for the discretization of the momentum, second-order upwind for subgrid kinetic energy, and the mixture fraction. Pressure-velocity coupling is achieved via a PISO (Pressure Implicit Splitting of Operator algorithm) similar procedure. The second-order implicit method is used for time integration scheme.

The cavitation model used in this work is based on a barotropic equation of state approach and it assumes that the two phases (gas and liquid) have equal velocities and temperatures at any point of the flow (i.e. homogeneous equilibrium) [17]. The assumption of

isothermal flow is also made. The selected barotropic equation used in the modeling reads as follows:

$$\frac{d\rho}{dt} = \psi \frac{dp}{dt} \quad (1)$$

where ψ is the mixture compressibility.

The parameter that accounts for the amount of fluid which is in the liquid or gaseous phase is the following:

$$\gamma = \frac{\rho - \rho_{l,sat}}{\rho_{ref} - \rho_{l,sat}} \quad (2)$$

where the reference gas density (ρ_{ref}) is determined as the maximum value between the vapor density of the liquid fluid and the dissolved gas density at the current local pressure:

$$\rho_{ref} = \max(\rho_{v,p}, \rho_{g,p}) \quad (3)$$

The amount of dissolved gases which are locally released is calculated by introducing the Henry law for the equilibrium condition and accounting for the time dependence of solubility on a Bunsen coefficient basis. The released gas amount is firstly calculated from Henry law and the initial volume fraction of dissolved gas. The influence of time on gas absorption and dissolution is then accounted for by means of the Bunsen coefficient, c_B ; thus the time variation of the released gas volume fraction, ε_c , can be calculated as follows [18 – 20]:

$$\frac{d\varepsilon_c}{dt} = \frac{\varepsilon - c_B p - \varepsilon_c}{\tau_c} \quad (3)$$

where ε is the gas – liquid saturated mass ratio, and τ_c is the time constant of the absorption or dissolution of the gas into the liquid medium.

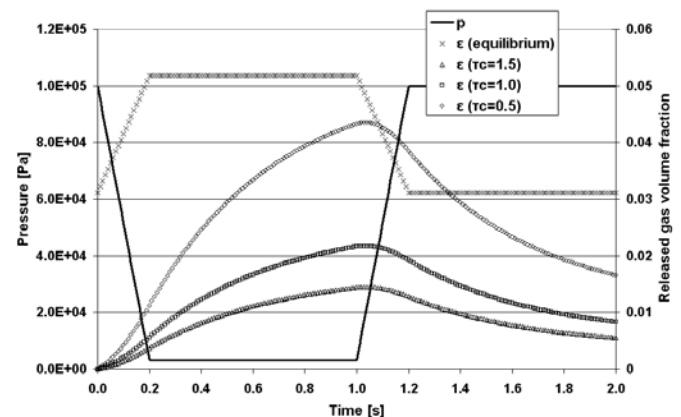


Figure 1: Released gas volume fraction during gas expansion and compression cycle.

The Bunsen coefficient can range approximately from 0.02 for water to 0.09 for mineral oil [19], while it is very difficult to find in literature typical values of the time constant τ_c . Unfortunately, this lack of data is a stringent restriction for the gas release estimation. In fact, the amount of gas released during an expansion process is greatly influenced by the time constant, as well as the amount of gas which is dissolved during compression. Figure 1 shows the volume fraction between the released gas and the liquid during an expansion from 1 bar to the liquid vapor pressure, a settling phase and a subsequent compression back to 1 bar. The time for the expansion and compression phases is set to 0.2 s while the settling phase is equal to 0.8 s. Different time constants are compared, and it is possible to notice that the released gas volume fraction is far from the equilibrium value both for the expansion and the compression processes. Thus, the released gas is not dissolved immediately and it lasts for a time period comparable with the process itself.

The cavitation model originally implemented in OpenFOAM code does not account for the effects of turbulence. Therefore, the code is modified in order to account for the turbulence in the compressible flow equations. The approach to turbulence modeling is simplified since the effects of the liquid – gas/vapor interface are neglected, and this point remains an area for further investigation. In all the simulations carried out for the present work the turbulence is modeled by means of the two zonal version of the k- ω model, known as the shear stress transport model [21].

The grids used in the simulations are constructed paying careful attention to comply with the requirements on the y^+ values for the k- ω SST formulation. Therefore, the y^+ are kept smaller than 3 in all the domain walls.

Finally, the liquid and gaseous phase is tracked by using the volume of fluid (VOF) method where both fluid phases are compressible.

The numerical approach is preliminarily validated by simulating a test case available in literature [22] and comparing the numerical results with the experimental measurements. The selected geometry is a sharp-edged cylindrical orifice and the working fluid is water.

Table 1: Water properties at 293 K used in the simulations

Property	Liquid phase	Vapor phase
Density [kg/m ³]	998	0.17
Viscosity [kg/ms]	1.3E-5	8.8E-4
Compressibility [s ² /m ²]	4.49E-7	6.94E-6
Vapor pressure [Pa]		2,339

The properties of water used in the simulations are reported in table 1 and they are referring to the temperature of 293 K. The dissolved gas fraction is assumed equal to the average value for tap water at ambient pressure, i.e. 0.01%.

In the experiments the effects of varying the diameters of the sharp-edge orifice and the ratio of the inlet diameter (d) and the orifice length (l) have been investigated for different Reynolds numbers of operation. The experimental set-up, as describe in [22], consisted of a tank for the water, a nitrogen gas source for pressurizing the tank and a feed-line fitted with flow control valves for supplying water to the orifice. The modeling of the experimental device focused only on the orifice geometry, and particular care was paid to assure that the flow at the orifice extremes was not influenced by the inlet and outlet of the CFD domain. To do this, two plenums were considered at the orifice inlet and outlet, each one having a diameter and an axial length ten times larger than the orifice diameter. Thanks to the axial symmetry of the geometry and of the boundary conditions, it was possible to account only for a 5° sector of the whole domain, and to simulate the experiments as two dimensional cases. Figure 2 shows an example of the CFD domain used.

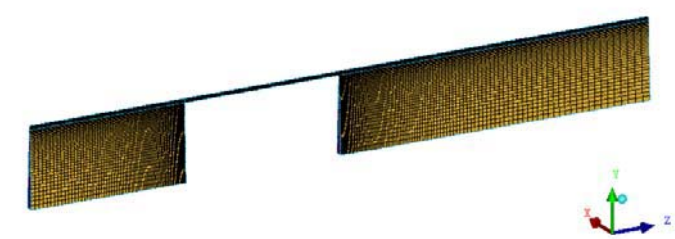


Figure 2: CFD domain for the small sharp-edged cylindrical orifice test case ($d=1$ and $l/d = 20$)

The present analysis focuses only on the geometrical configurations and operating conditions where cavitation was evidenced by the experiments in [22]. Table 2 details the test cases that are accounted for in the simulations.

Table 2: Test cases for the flow through a small sharp-edged cylindrical orifice

Section diameter	l/d	Reynolds number
0.5 mm	20	4.5×10^3 ; 1.1×10^4 ; 1.8×10^4
	50	5.9×10^3 ; 8.5×10^4 ; 1.2×10^4
1.0 mm	20	1.1×10^4 ; 1.8×10^4 ; 3.4×10^4
	50	8.8×10^4 ; 12.6×10^4 ; 3.3×10^4
2.0 mm	20	3.4×10^4 ; 4.9×10^4 ; 7.2×10^4
	50	1.2×10^4 ; 3.4×10^4 ; 5.6×10^4

The numerical results are compared with measurements mainly in terms of the orifice discharge coefficients. Particular care was devoted in calculating the pressure drop and, in order to assess the influence of the abrupt section change only, it was defined as the difference between the inlet static pressure, p_{in} , and the static pressure at the vena contracta position, p_{vc} . The vena contracta position is calculated as the axial coordinate along the cylinder axis where the lowest pressure value can be found [23].

The test cases are simulated by means of three different numerical approaches to cavitating flows analysis, i. e. a numerical approach which accounts for turbulent flows but does not include any cavitation model, a second approach which includes the cavitation model but assumes the flow as laminar and a third numerical approach with accounts both for turbulence and cavitation. Evidently, the computational cost increases from the first approach to the last one, and the accuracy of the results which can be obtained depends on the single problem that has to be simulated. Therefore, the predictive capabilities of the above mentioned numerical approaches are addressed when simulating cavitating flows.

A sensitivity analysis with respect to the mesh resolution is firstly carried out. The different meshes are constructed with a cell dimension in the orifice geometry ranging from 0.025 mm, 0.01 mm and 0.005 mm, hereafter named as Mesh #1, Mesh #2 and Mesh #3. Figure 3 depicts the influence of the grid resolution on the discharge coefficients for each numerical approach. The selected boundary conditions are referring to the test case with $d = 0.5$ and $l/d = 20$ and Reynolds number equal to 1.8×10^4 . The numerical analysis including the turbulence effects but no cavitation model demonstrates to be less sensitive to the cell dimension than the simulations accounting for cavitation. In the case of numerical approach having the cavitation model but no turbulent flow the discharge coefficient becomes more accurate as the mesh density increases, since the

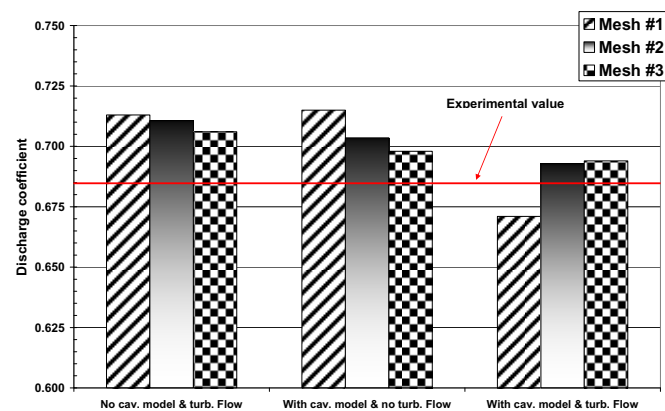
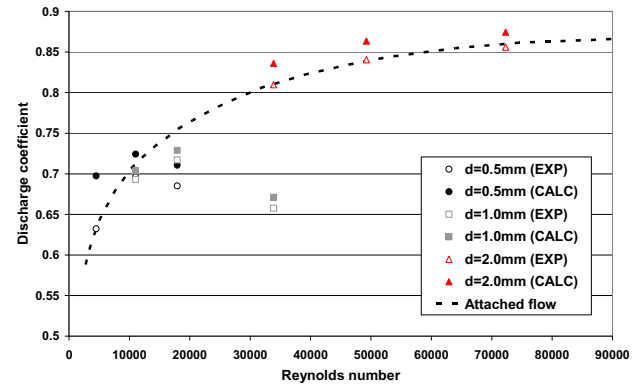


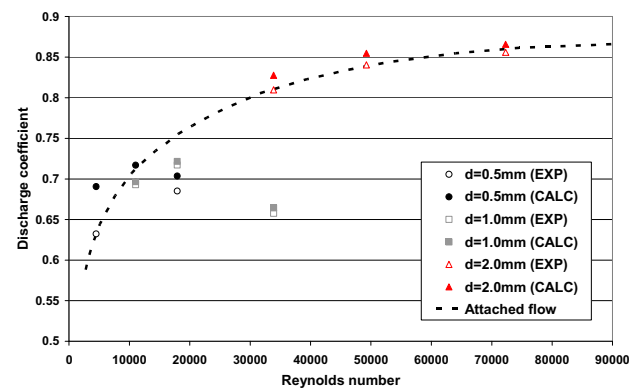
Figure 3: Predicted discharge coefficient for different mesh resolutions and numerical approaches ($d=0.5$ mm, $l/d=20$, $Re=1.8 \times 10^4$)

possibility of capturing the large scale effects of the turbulent flow increases. The most complete numerical approach demonstrate to be less sensitive to the grid resolution and provides almost the same results by using Mesh #2 and Mesh #3. Therefore, all the remaining test cases are simulated by using Mesh #2.

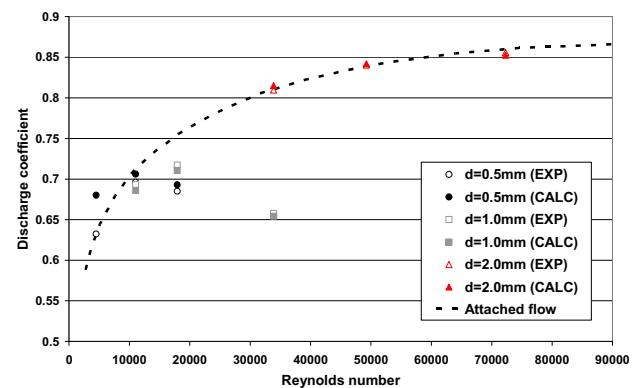
Figures 4 and 5 show the comparison between the experiments and calculations for the selected test case in terms of discharge coefficients. In the Figures the trend of the discharge coefficients in case of attached flow is also plotted (dotted line). This line represents the limit between the hydraulic flip cases and the reattached flow conditions.



a)



b)



c)

Figure 4: Comparison between measured and calculated discharge coefficients for simulations a) without cavitation model and turbulent flow, b) with cavitation model and laminar assumption and c) with cavitation model and turbulent flow ($l/d=20$)