CAVITATING FLOW IN A 3D GLOBE VALVE

DANIEL RODRIGUEZ CALVETE[1], ANNE GOSSET[2]
[1]*Grupo Integrado de Ingeniería Universidad de A Coruña, daniel.rcalvete@udc.es*
[2]*Grupo Integrado de Ingeniería Universidad de A Coruña, agosset@udc.es*

Keywords: *Cavitation, interPhaseChangeFoam, Control Valves*

The efficiency of control valves operating with liquids is highly conditioned by the occurrence of cavitation when they undergo large pressure drops. For severe service control valves, the subsequent modification of their performance, due to the presence of vapor in and downstream the restriction, can be crucial for the safety of an installation. In this work, OpenFoam is used to characterize the flow in a globe valve, with the objective to evaluate its capability in solving cavitating flows in complex a 3D geometry. Figure 1 shows the computational domain and mesh of the glove valve under study. The simulations are based on the experimental data of Ferrari et al. [2], who run experiments for a set of cavitating conditions, varying the pressure drop across the valve Δp and the opening valve (lift). In this study, all the simulations are performed for valve openings of 4 and 6 mm. The static pressure at the valve outlet is set at values of 0.4 bar and 0.8 bar respectively. Total pressure is set at the inlet at different values. A structured hex-mesh of $1.65 \cdot 10^6$ cells is designed with Ansys-Icem, with a y^+ varying between 0.2 and 15 in the restriction area.

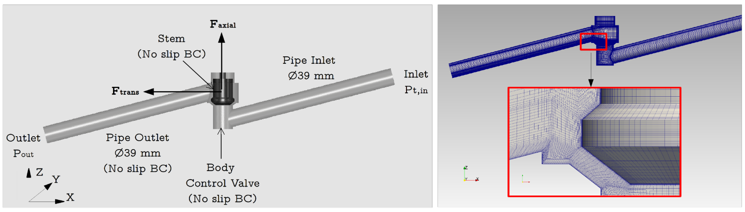

Figure 1: Computational domain and mesh

The interPhaseChangeFoam solver with the SchnerrSauer model for cavitation are used in this study. The nuclei concentration, n and their initial diameter, $dNuc$ are set as $10^{14} m^{-3}$ and $5 \cdot 10^{-5} m$ respectively. The vapor pressure of water is fixed at 3540 Pa, but the effect of turbulence on the phase change threshold is taken into account through a correction of the saturation pressure, p_{vap}, as proposed by Bouziad [1] using the shear strain rate, S:

$$p_v = p_{vap} + (\mu + \mu_t) S \tag{1}$$

We find that turbulence effects contribute to an increase of the threshold pressure of up to 10 times the vapor pressure, advancing phase change and improving the agreement with experiments. Figure 2 shows a comparison of the vapor cavity extension in the glove valve with and without this correction.

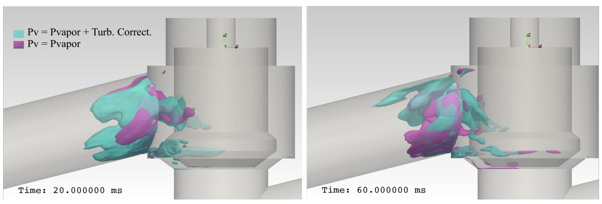

Figure 2: Pressure threshold turbulence correction

Valve flows usually feature important stagnation regions and a high velocity gradient in the restriction, this is why a RANS turbulence model $k - \omega - SST$ is used. Pressure velocity coupling is achieved with the PIMPLE algorithm, with a maximum number of outer correctors equal to 10, a convergence criteria of 10^{-3} for initial residual and two pressure correctors for each iteration. The time step is variable and set by a maximum Co number. To analyse the influence of Co on the CPU time and the quality of results, simulations with $C_{o,max} = 0.9$ and $C_{o,max} = 3$ were performed. Table 1 presents the results in terms of flow rate through the valve, $(Q, m^3/h)$, force acting on the stem, (F_t, N) and the CPU time for 1 s of real flow simulation with 8 core processors. It can be observed that the influence of the time step on the average

Table 1: Influence of the time step

$C_{o,max}$	Δp	Q (m^3/h)	$Force(N)$
0.9	1.46	29.06	102.53
3	1.46	29.25	101.18

results is negligible within this range. Therefore $C_{o,max} = 3$ is considered for all simulations. Euler and limitedLinear (2^o order TVD-Sweby) schemes are used for time and spatial discretization respectively.
Figure 3 illustrates the unsteady behaviour of cavitation by comparing the high-speed experimental observations for $\Delta p = 1.54bar, Q = 23m^3/h$ (lift 4 mm) with the predictions of OpenFoam for $\Delta p = 1.12bar, Q = 23.5m^3/h$ (lift 4 mm). In this close-up, it can be seen how the location of vapor cavities is qualitatively well predicted. However, the extension of vapor through the outlet pipe is underestimated, probably due to an underestimation of the effect of turbulence on phase change.

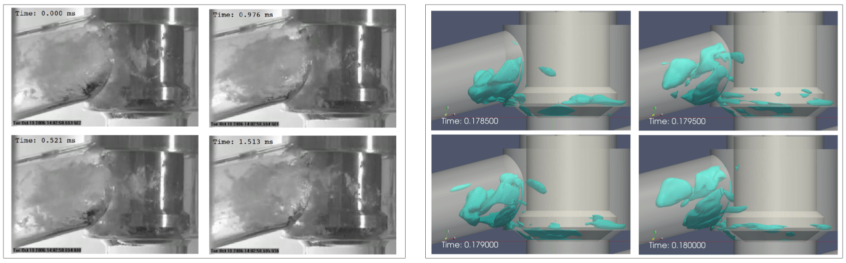

Figure 3: Experimental(Left) and numerical(Right) cavitation sequences, Lift 4mm

For turbulent single phase flows in valves, the flow rate $(Q, m^3/h)$ is a linear function of the square root of pressure drop (Δp). A deviation is observed when vapor cavities become important in the restriction. In the evolution of the volumetric flow rate in figure 4, it can be seen that OpenFoam tends to underestimate the extension of cavitation because the deviation observed in experiments is hardly recovered. The error in the prediction of the flow rate oscillates between 2% and 6%. The evolution of the total force (N) on the stem is illustrated in figure 4, with an error between 5% and 17%. In conclusion, the main valve characteristic are well predicted with the interPhaseChangeFoam solver, although it tends to underestimate slightly the extension of cavitation. This is also what was found with the commercial code Ansys-CFX in a comparative study with OF [3], even to a larger extent. Since it is based on a coupled solver however, CFX involves computational times from one to two orders of magnitude lower than OF.

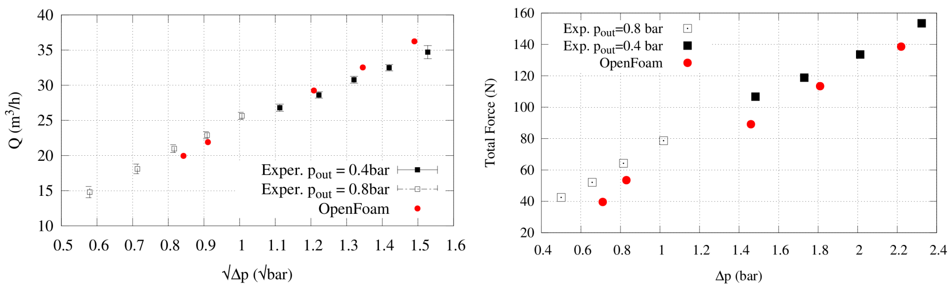

Figure 4: Flow rate vs $\sqrt{\Delta P}$ (left) and force on the stem vs ΔP (right), Lift 6 mm

Acknowledgments

The authors wish to thank Jérôme Ferrari from EDF R& D for providing us with additional experimental data. They also acknowledge the financial support of Xunta de Galicia through grant # EM2013-009.

References

[1] Y. A. Bouziad, “Physical modelling of leading edge cavitation: computational methodologies and application to hydraulic machinery,” Ph.D. dissertation, Ecole Polytechnique Fédérale de Lausanne, 2006.

[2] J. Ferrari and Z. Leutwyler, “Fluid flow force measurement under various cavitation state on a globe valve model,” in *Proc. 2008 ASME Pressure Vessels and Piping Conference. Chicago (IL, USA)*. ASME, 2008, pp. 157–165.

[3] D. Rodriguez Calvete, A. Gosset, and et al., “Numerical simulation of cavitating flow in a globe valve: Confrontation of openfoam and cfx,” in *Proc. of ASME Pressure Vessels and Piping Conference, Vancouver(Under Review)*, 2016.