ABSTRACT
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 can be crucial for the safety of an installation. In this work, two CFD codes, OpenFoam [1] and Ansys-CFX, are used to characterize the flow in a globe valve, with the objective to compare their capabilities in solving cavitating flows in complex 3D geometries. In both codes, an Homogeneous Equilibrium approach is adopted, and phase change is modeled with a similar cavitation model. It is found that both solvers predict correctly the location of vapor cavities, but tend to underestimate their extension. The flow rate is correctly calculated, but in strong cavitating regimes, it is affected by the underprediction of vapor cavities. The force acting on the stem is found to be more sensitive to the computation parameters.

NOMENCLATURE
\( \alpha_i \) Volume fraction of phase \( i \)
\( C_o \) Courant number
\( F_j \) Force component \( j \) (N)
\( K_v \) Flow coefficient
\( \mu_i \) Dynamic viscosity of phase \( i \) (Pa · s)
\( \rho_i \) Density of phase \( i \) (kg · m\(^{-3}\))
\( \dot{m}_{\text{c}} \) Mass transfer rate by condensation (kg/s)
\( \dot{m}_{\text{v}} \) Mass transfer rate by vaporization (kg/s)
\( n_0 \) Nuclei concentration (m\(^{-3}\))
\( p_{\text{in}} \) Static pressure at the inlet (bar)
\( p_{\text{out}} \) Static pressure at the outlet (bar)
\( p_{\text{in},\text{in}} \) Total pressure at the inlet (bar)
\( p_{\text{vap}} \) Vapor pressure (Pa)
\( p_v \) Phase change pressure threshold (Pa)
\( \Delta p \) Pressure drop across the valve (Pa)
\( Q \) Volumetric flow rate (m\(^3\)/h)
\( R \) Bubble Radius (m)
\( S \) Shear strain (/s)
\( Sh_j \) Hydraulic surface, (m\(^2\)) for force component \( j \)
\( \sigma \) Cavitation number

Copyright © 2016 by ASME
INTRODUCTION

In nuclear power plants and petrochemical installations, some specific control valves play a critical role in the functioning of the plants. This is the case of feedwater control valves for the boilers in nuclear reactors, for example. These severe service valves have to be responsive, precise, and perfectly reliable. However, their behaviour can deviate strongly from their normal functioning in case of unexpected circumstances: presence of solid particles in water, or an unwanted phase change. The efficiency and safety of this kind of control valves operating with liquids is highly conditioned by the occurrence of cavitation when they undergo large pressure drops. In the vena contracta that develops in the restriction region, the fluid acceleration is such that the local pressure may decrease below the vapor pressure and generate cavitation. At inception, it is usually not a problem, but when it develops, it can modify the performance of the valve and even limit the flow rate close to choking conditions. In addition, it may generate noise and erosion of the valve components. It is thus important to be able to predict the behaviour of these valves over a wide range of working conditions, including conditions in which there is phase change due to cavitation.

In practice, the occurrence of cavitation is difficult to detect because of the harsh and noisy conditions in which industrial valves usually operate. This is why CFD has become an important tool for the design and characterization of severe service valves. It has been scarcely used up to now for cavitating flows due to the important computational times involved, and the sensitivity of the solvers to empirical parameters. Chen and Stoffel [2] investigate the transient effects of cavitation in a poppet valve during closure, but their computational domain is axisymmetric and limited to the vicinity of the restriction region. Bernard et al. [3] extent the analysis of cavitation to a 3D poppet valve. Beune et al. [4] are the first to publish a validated CFD study with a safety relief valve. They show that taking into account phase change in CFX allows reducing in a spectacular way the overestimation of the mass flow rate in the valve, and the force exerted by the fluid on the stem. Couzinie et al. [5] also use a mixture model in CFX to predict the flow capacity of a safety relief valve in cavitating regimes, and they propose a correction of the liquid vapor pressure to take into account the effect of turbulence. The topology of the flow field is very well validated with experimental data obtained through Particle Image Velocimetry. Regarding globe valves, and control valves in general, little has been done. Davis and Stewart (2002) present a CFD study of the single phase flow in a globe valve, but there is no attempt to model cavitation even though they observe it experimentally. Ferrari and Leutwyler [6] also propose a single phase numerical study of the flow in a globe valve with Fluent. But more importantly, they perform an extensive experimental study, gathering unsteady measurements of flow forces on the stem and flow visualizations on a transparent mock-up.

Up to now, all the numerical studies were conducted with commercial codes. In this paper, we propose to evaluate the capabilities of the open source CFD code OpenFoam to predict the unsteady cavitating flow in a 3D globe valve geometry. For that, we compare the performances of OpenFoam [1] and the commercial CFD package Ansys-CFX, using the experimental data of Ferrari and Leutwyler [6] to validate the results.

NUMERICAL APPROACH

 Governing Equations

To simulate cavitating flows, the two phases, liquid \( l \) and vapor \( v \), have to be taken into account in the governing equations, and the phase transition mechanism due to evaporation-condensation has to be modelled. In this study, the two phases are assumed to be homogeneously mixed and in mechanical equilibrium, following the Homogeneous Equilibrium approach. Hence, only one set of momentum and continuity equations is solved for the mixture. It is assumed that there is no interaction and no slip between vapor bubbles. The VOF (Volume of Fluid) technique is used for tracking the interface between liquid and vapor.

Since liquid and vapor are assumed to be perfectly mixed within each cell of the mesh, the density and viscosity of the mixture are expressed as a function of the liquid and vapor volume fractions, \( \alpha_l \) and \( \alpha_v \), respectively:

\[
\rho = \alpha_l \rho_l + \alpha_v \rho_v \quad (1)
\]

\[
\mu = \alpha_l \mu_l + \alpha_v \mu_v \quad (2)
\]

The subscripts \( l \) and \( v \) stand for the properties of pure liquid and pure vapor respectively. The constraint condition to fulfill is:

\[
\alpha_l + \alpha_v = 1 \quad (3)
\]

To close the system, a transport equation for \( \alpha_l \) is needed. Using Eqn.(1) in the continuity equation and considering the mass transfer between phases due to cavitation, the transport equation for \( \alpha_l \) can be derived as [7]:

\[
\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{U}) = \frac{\dot{m}_c^+ - \dot{m}_c^-}{\rho_l} \quad (4)
\]

where \( \dot{m}_c^+ \) is the mass transfer rate by vaporization and \( \dot{m}_c^- \) the one by condensation.
Cavitation Model

The source term of mass transfer (RHS of Eq. 4) requires an appropriate cavitation model. Different cavitation models can be found in literature, among which the full cavitation model of Singhal et al. [8]. Singhal et al. consider an extra phase for the non-condensable gases present in the liquid. In this work, only two phases (liquid and vapor) are considered and the cavitation model of Sauer and Schnerr [7] is used in OpenFoam. In Ansys-CFX, the model of Zwart [9] is implemented, which is in essence very similar to the one of Sauer and Schnerr. Both models consider an initial amount of micro spherical vapor bubbles with a radius $R$, which constitute nucleation sites for cavitation. They grow and collapse according to the bubble pressure dynamics bubble governed by the first order Rayleigh Plesset equation (5):

$$
\dot{R} = \frac{dR}{dt} = \sqrt{\frac{2}{3} \left( \frac{P(R) - P_{\infty}}{\rho_l} \right)}
$$

where $P(R)$ is the pressure in the liquid at the bubble interface and $P_{\infty}$ the pressure far from the bubble. In the model of Sauer and Schnerr [7], the vapor fraction is calculated based on the volume of the spherical nuclei and their number per cubic meter of liquid, $n_0$, as:

$$
\alpha_v = \alpha_i \cdot n_0 \cdot \frac{4}{3} \pi R^3
$$

$p(R)$ is set to the equilibrium vapor pressure, $p_{eq}$, and $P_{\infty}$ to the ambient cell pressure, $p$. The combination of equations (4), (5) and (6) gives the final expression for the mass transfer source terms:

$$
\dot{m}_v^+ = C_v p_v (1 - \alpha_v) \frac{3 \alpha_v}{R} \sqrt{\frac{2}{3} \left( \frac{\max(0, p_{eq} - p)}{\rho_l} \right)}
$$

$$
\dot{m}_v^- = C_v p_v (1 - \alpha_v) \frac{3 \alpha_v}{R} \sqrt{\frac{2}{3} \left( \frac{\max(0, p - p_v)}{\rho_l} \right)}
$$

where $R$ is derived from Eqn. (6).

As inputs, the model requires therefore the volumetric concentration of nuclei $n_0$, their initial radius, $R$, and the empirical coefficients $C_v$ and $C_i$. The later depend on the state of dearation of the liquid and the mean flow. To quantify the influence of those parameters in the OpenFoam solver, a sensitivity study was led for a sharp orifice [10], concluding that there was no significant difference on results for $n_0 \leq 10^{10} m^{-3}$ and $10^{-7} \leq R \leq 10^{-3} m$. In this work, the value of the parameters are set at $R = 5 \cdot 10^{-6} m$, $n_0 = 10^{14} m^{-3}$, $C_v = 1$ and $C_i = 2$.

In contrast, Zwart [9] defines $n_0$ as the number of nuclei per unit of total volume, which means that $\alpha_i$ vanishes in equation (6) and, therefore, the mass transfer terms in this model have slightly different expressions:

$$
\dot{m}_v^+ = F_i p_v (1 - \alpha_v) \frac{3 \alpha_v}{R} \sqrt{\frac{2}{3} \left( \frac{\max(0, p_{eq} - p)}{\rho_l} \right)}
$$

$$
\dot{m}_v^- = F_i p_v \frac{3 \alpha_v}{R} \sqrt{\frac{2}{3} \left( \frac{\max(0, p - p_v)}{\rho_l} \right)}
$$

where $\alpha_{nuc}$ is the initial volume fraction of nuclei, while $F_i = 50$ and $F_c = 0.01$ are empirical constants. In the expression (9) of $\dot{m}_v^+$, the use of $\alpha_{nuc}$ allows taking into account the fact that the nucleation site density decreases when $\alpha_v$ increases. In Ansys-CFX, the parameters are set as $R = 10^{-6} m$ and $\alpha_{nuc} = 5 \cdot 10^{-4}$, which corresponds to $n_0 = 1.2 \cdot 10^{14} m^{-3}$.

Originally, in the models of Sauer and Schnerr and Zwart, the phase change threshold pressure $(p_v)$ is assumed to be equal to the saturation pressure $(p_{sat})$ in the absence of dissolved gases. In this study, the value of $p_{sat}$ is set at 3540 Pa, which is estimated with the Rankine formula for water at 26°C. Nevertheless, several investigations have shown significant effects of turbulence on cavitation flows, e.g. [11]. Several authors [8] and [12] suggest to take it into account by integrating in time the contributions of $\dot{m}_v^+$ and $\dot{m}_v^-$ assuming a probability density function of the pressure fluctuations due to turbulence. Bouziad [12] proposes a simple approach based on a correction of $p_v$ using the shear strain to modify the bubble pressure. The corrected value of $p_v$ becomes:

$$
p_v = P_{sat} + (\mu + \mu_t) S
$$

where $S$ is the shear strain and $\mu_t$ is the turbulent viscosity. Couzin et al. [5] evaluate the effect of using this approach in a safety relief valve flow simulated with Ansys CFX. They show that turbulence effects contribute to an increase of up to 500 % of the vapor threshold pressure $(p_v)$, which influences directly the location and extension of cavitation. It must be noticed that this kind of approach is highly depend on the quality of turbulence prediction, therefore an appropriate turbulence model should be chosen.

Turbulence model

For this 3D pressure driven flow, a URANS model for turbulence is adopted instead of Large Eddy Simulation, disregarded
in this study due to its time cost. There is an important stagnation region below the stem, therefore the $k-\varepsilon$ model in its initial formulation should be avoided as it gives non-physical results when the mean velocity gradient becomes too high. In this case, any two-equation model based on an eddy viscosity limiter can be adopted, especially if the limiter is controlled by the shear strain values such as the $k-\omega$ SST (Shear Stress Transport) or the Realizable $k-\varepsilon$. In this study, the $k-\omega$ SST has been chosen for all the computations, both with Ansys-CFX and OpenFoam. On the other hand, the strong pressure gradients expected in the flow make it necessary to solve accurately the boundary layer up to the viscous sublayer, especially close to the stagnation zone. This means that high resolution mesh at the wall is needed, so that the use of standard wall functions is avoided.

**Computational Domain**

The computational domain is defined from the geometrical model sketched in figure 1 that matches the geometry of a 2” commercial globe valve. It corresponds exactly to the mock-up built by Ferrari and Leutwyler [6] to visualize the flow inside the valve. In cavitating conditions, Couzinet et al. [5] find that the flow is substantially asymmetric in a safety relief valve, this is why the whole domain is simulated here.

![Valve Model and Computational Domain](image)

**FIGURE 1. VALVE MODEL AND COMPUTATIONAL DOMAIN**

The mesh is designed with the meshing software Ansys-ICEM, using a fully structured topology and only hexahedral cells. In order to preserve low $y^+$ at the wall, a special refinement in the restriction is made, as show in figure 3. A focus on the 6 mm lift of the stem is proposed in this study.

A mesh independence study is performed using three different grid sizes, presented in figure 2. Since cavitation develops not only in the restriction region, but also downstream the valve body, in the outlet duct, a special attention is given to the refinement of this region. The results in terms of volumetric flow rate ($Q$), transversal and axial force acting on the stem ($F_{\text{trans}}$ and $F_{\text{axial}}$) are presented in table 1 for the different meshes. Slight differences are found between the coarse and the medium mesh, which become negligible between the medium and the fine mesh. In addition, the extension of vapor cavities is very similar for the two finest grids. Therefore, the medium mesh of 1.65 million cells is chosen as the base mesh.

| TABLE 1. RESULTS OF THE MESH SENSITIVITY STUDY |
|-----------------|---------|-------|---------|----------|--------|
| $\text{Cells x 10}^6$ | $\Delta p$ (bar) | $Q$ ($m^3/h$) | $F_{\text{axial}}$ (N) | $F_{\text{trans}}$ (N) |
| 1.55            | 1.810   | 32.28 | 108.8   | 69.39    |
| 1.65            | 1.810   | 32.53 | 103.6   | 71.33    |
| 1.96            | 1.803   | 32.32 | 103.2   | 71.27    |

For the boundary conditions, a total absolute pressure is fixed at the inlet ($p_{\text{in}}$) and a static pressure is fixed at the outlet ($p_{\text{out}}$) (figure 1).
Numerical Methodology

Two different CFD codes are used in this work, Ansys-CFX and OpenFoam which are both based on the finite volume method. In Ansys-CFX, the solution of the velocity-pressure system of equations is based on a fully coupled approach. This allows solving the flow equations using larger time steps compared to the standard algorithms of pressure-velocity coupling, such as PISO, which is implemented in OpenFoam.

For unsteady flows, one of the main issues with the PISO algorithm is that the rate of convergence becomes very slow when a large time step is used, even if unconditionally stable temporal discretization schemes are used. Consequently, the computational cost of OpenFoam is generally greater than that of Ansys-CFX. It should be underlined that even if the time step can be set to guarantee the stability of the solution, the phase change process due to cavitation is very fast, and the time step must be sufficiently small to capture the relevant phenomena and control the non-linearity generated by the mass transfer term.

OpenFoam proposes an algorithm that combines both SIMPLE and PISO for unsteady simulations, called PIMPLE. At each time step, PIMPLE basically allows a specific number of SIMPLE iterations within a PISO loop, to converge the solution to a certain tolerance. The SIMPLE mode used between time steps allows underrelaxing the solution, and thus helps reaching a stable solution while using larger time steps. In order to keep the computational time within reasonable limits, the PIMPLE solver is used in conjunction with the cavitation model in OpenFoam.

For time discretization, a second order backward scheme is used in Ansys-CFX. A preliminary study had to be performed with OpenFoam regarding time schemes. The second order backward and Crank Nicholson schemes both resulted in numerical instabilities and divergence of the solution, even with \( C_{o,max} < 1 \). Therefore, a first order implicit scheme was adopted. An adaptive time step based on a maximum Courant (\( C_{o,max} \)) number is set in the OpenFoam solver. A value of \( C_{o,max} = 3 \) is chosen, which implies a time step of about \( 3 \cdot 10^{-8}s \). To analyse the influence of the time step on the solution, a simulation with \( C_{o,max} = 0.9 \) was performed. Table 2 shows the values of the macroscopic parameters of interest (\( Q, \Delta p = p_{in} - p_{out} \) and the forces) in both cases, concluding that there is no significant difference.

With OpenFoam, the simulation running in a cluster with 8 core-processors takes about 86 days of CPU time for 1s of real flow at \( C_{o,max} = 3 \). On the other hand, the simulations in CFX use a time step of \( 10^{-4}s \) and running in a cluster with 6 core-processors, they take about 3 days of CPU time for 1s real flow.

For the spatial discretization, second order bounded schemes are used in both codes.

RESULTS

Operating Conditions

The numerical study is based on the experimental conditions described in [6]. The authors run experiments for a set of cavitating conditions varying the pressure drop across the valve \( \Delta p \) and the valve opening (lift). All the simulations are performed for a valve opening of 6 mm and some tests were also conducted at 4mm. Table 3 reports the conditions corresponding to the different computations.

The outlet pressure \( p_{out} \) is set at values of 0.4 and 0.8 bar, in order to induce earlier cavitation. Cases Dp07 and Dp08, with \( p_{out} = 0.8 \) bar, correspond to non cavitating conditions, in agreement with the experimental observations. In contrast, cases Dp15-18-23, with \( p_{out} = 0.4 \) bar, correspond to fully developed cavitating conditions, as shown in figure 4.

Influence of Turbulence on \( p_e \)

As mentioned above, the correction of \( p_e \) (11) accounts for the influence of turbulence on cavitation. In order to evaluate


### TABLE 3. EXPERIMENTAL AND NUMERICAL CONDITIONS

<table>
<thead>
<tr>
<th>Cases</th>
<th>$\Delta p$ (bar)</th>
<th>$p_{in}$ (bar)</th>
<th>$p_{out}$ (bar)</th>
<th>$Q$ (m$^3$/h)</th>
<th>$\sigma$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp</td>
<td>0.67</td>
<td>1.62</td>
<td>1.47</td>
<td>0.8</td>
<td>20.98</td>
</tr>
<tr>
<td>Dp07</td>
<td>0.71</td>
<td>1.62</td>
<td>1.51</td>
<td>0.8</td>
<td>19.95</td>
</tr>
<tr>
<td>CFX</td>
<td>0.72</td>
<td>1.62</td>
<td>1.52</td>
<td>0.8</td>
<td>18.86</td>
</tr>
<tr>
<td>Exp</td>
<td>0.83</td>
<td>1.74</td>
<td>1.63</td>
<td>0.8</td>
<td>22.90</td>
</tr>
<tr>
<td>Dp08</td>
<td>0.83</td>
<td>1.74</td>
<td>1.63</td>
<td>0.8</td>
<td>21.90</td>
</tr>
<tr>
<td>CFX</td>
<td>0.83</td>
<td>1.74</td>
<td>1.63</td>
<td>0.8</td>
<td>20.16</td>
</tr>
<tr>
<td>Exp</td>
<td>1.49</td>
<td>2.10</td>
<td>1.89</td>
<td>0.4</td>
<td>28.61</td>
</tr>
<tr>
<td>Dp15</td>
<td>1.46</td>
<td>2.10</td>
<td>1.86</td>
<td>0.4</td>
<td>29.25</td>
</tr>
<tr>
<td>CFX</td>
<td>1.55</td>
<td>2.10</td>
<td>1.95</td>
<td>0.4</td>
<td>27.70</td>
</tr>
<tr>
<td>Exp</td>
<td>1.74</td>
<td>2.50</td>
<td>2.13</td>
<td>0.4</td>
<td>30.78</td>
</tr>
<tr>
<td>Dp18</td>
<td>1.81</td>
<td>2.50</td>
<td>2.21</td>
<td>0.4</td>
<td>32.53</td>
</tr>
<tr>
<td>CFX</td>
<td>1.84</td>
<td>2.50</td>
<td>2.24</td>
<td>0.4</td>
<td>29.70</td>
</tr>
<tr>
<td>Exp</td>
<td>2.33</td>
<td>3.00</td>
<td>2.72</td>
<td>0.4</td>
<td>34.71</td>
</tr>
<tr>
<td>Dp23</td>
<td>2.22</td>
<td>3.00</td>
<td>2.62</td>
<td>0.4</td>
<td>36.24</td>
</tr>
<tr>
<td>CFX</td>
<td>2.29</td>
<td>3.00</td>
<td>2.69</td>
<td>0.4</td>
<td>33.05</td>
</tr>
</tbody>
</table>

its effect, two simulations with and without turbulence correction are performed with the OpenFoam solver for the case Dp15. Figure 5 shows the time evolutions of the vapor volume fraction $\alpha_{v,avg}$ averaged over the whole domain, and of the flow rate. It can be clearly seen how the fluctuations $\alpha_{v,avg}$ grow larger when the correction is applied. Moreover in figure 6, the iso-surfaces $\alpha_v = 0.5$ in both simulations are superimposed at three different time instants, which confirm how the distribution of vapor is influenced by the correction of $p_v$.

**Flow Topology**

Figure 4 shows the extension of vapor cavities at 3 different pressure drops obtained with OpenFoam. To relate the pressure drop with the intensity of cavitation, the cavitation number is defined as:

$$\sigma = \frac{p_{in} - p_{out}}{p_{in} - p_{vap}}$$  \hspace{1cm} (12)

It can be seen how an increase of $\sigma$ from 0.80 ($\Delta p = 0.71$bar) to 0.86 ($\Delta p = 2.22$bar) leads to an increase of the vapor extension towards the valve outlet. In figure 7, the predictions of OpenFoam and Ansys-CFX are compared to the experimental observations at $\Delta p = 1.8$bar. In the numerical results, the cavities are materialized with the isosurfaces $\alpha_c = 0.5$. Qualitatively, the location of vapor is well predicted, but its extension is substantially underestimated by both codes. In addition, Ansys-CFX gives a cavitating regime that is less intense than with OpenFoam. This discrepancy might be due to the differences in the implementation of the cavitation models in both solvers. In OpenFoam, vapor is assumed to have a constant density, while in Ansys-CFX, vapor is treated as a perfect gas and the liquid to vapor density ratio has to be limited to ensure stability of the calculations. It has been shown that this limitation has a direct influence on the mass transfer rates [13]. Deactivating the limitation, we have ourselves observed an increase of vapor cavities.

Finally, figure 8 illustrates the unsteady behaviour of cavitation by comparing the experimental observations with the predictions of OpenFoam, both at $\Delta p = 1.54$bar (lift 4 mm). Two isosurfaces, at $\alpha_c = 0.5$ and $\alpha_c = 0.1$ are superimposed to illustrate the extent of vapor-liquid mixing. Experimentally, the sequence of images is captured by a high speed camera at a frequency of 13KHz, while the vapor fraction fields from the OpenFoam simulation are sampled at 2KHz. The sequences are synchronized in time to compare qualitatively the behaviour of va...
FIGURE 6. VAPOR PHASE ISO-SURFACES ($\alpha_v = 0.5$) WITH AND WITHOUT $p_v$ CORRECTION IN OPENFOAM

por cavities. In this close-up, it can be seen clearly how the vapor cavities grow in the restriction region around the stem, and extends downstream the valve body in both cases. The location of vapor cavities is well predicted and a certain synchronization of the bubble growth and collapse can be seen as well.

Flow curve
For single phase turbulent flows through control valves, the flow rate $Q$ in $m^3/h$ is a linear function of the square root of the pressure drop $\Delta p$ in bar across the valve [14]. This relation is expressed as:

$$Q = K_v \sqrt{\Delta p}$$  \hspace{1cm} (13)
where $K_v$ is the flow coefficient, and the slope of the valve characteristic curve. In fact, $K_v$ represents the volumetric flow rate of water circulating in a valve under a 1 bar pressure drop, at a given valve aperture. In cavitating conditions the characteristic flow curve deviates from this linear behaviour, until it reaches choked flow condition. The later occurs when $Q$ no longer varies with $\Delta p$ due to a vapor blockage at the valve outlet.

The volumetric flow rates obtained numerically are compared to the experimental data [6] in Figure 9. The predictions of OpenFoam and Ansys-CFX present a reasonable agreement with experiments, with an error between 2 and 7% with OpenFoam, and an underestimation of 3 to 11% for Ansys-CFX. However, it can noticed that at the highest pressure drops, the flow rates predicted numerically remain on a linear curve, while the experimental values deviates slightly from their linear evolution. This indicates that both codes tend to underestimate the influence of vapor cavities on the head loss across the valve, as observed before. Table 4 summarizes the values found for the flow coefficient ($K_v$). The prediction of OpenFoam is sensibly closer to the experimental data.

**Local analysis**

To complete the comparison between both codes, figure 10 shows the time averaged velocity profiles in the x direction at 4 different positions along the outlet duct. The velocity is adimensionalised with the mean velocity $U_{ref}$ based on the flow rate. It can be observed that the maximum velocity position and profile shape in the near wall region is similar in both cases. This means that the two codes estimate similarly the vena contracta in the restriction and the resulting jet. Further from the wall, the difference are more substantial. They may be due to the influence of the convective schemes used in both codes [15], and to the differences in the mass transfer rates.

**Forces on the stem**

The experimental study [6] focuses on the components of the force acting on the stem. Figures 12 and 11 compare the axial and transversal force components found experimentally and numerically. It can be seen that both codes strongly underestimate the transversal forces, while overestimating the axial forces. This behaviour where the axial force is dominant corresponds normally to small valve apertures [6]. It is at larger apertures that the force ratio is inversed, possibly due to the acceleration of the

**FIGURE 9.** CHARACTERISTIC FLOW CURVE FOR A VALVE OPENING OF 6 MM

**FIGURE 10.** X-VELOCITY PROFILES AT 4 POSITIONS ALONG THE OUTLET DUCT ($\Delta p = 1.8$ bar) WITH CFX AND OPENFOAM

**TABLE 4.** FLOW COEFFICIENT $K_v$

<table>
<thead>
<tr>
<th></th>
<th>$K_v$</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.</td>
<td>24.46</td>
<td>-</td>
</tr>
<tr>
<td>OF</td>
<td>24.08</td>
<td>1.55</td>
</tr>
<tr>
<td>CFX</td>
<td>22.07</td>
<td>9.77</td>
</tr>
</tbody>
</table>

---

Copyright © 2016 by ASME
fluid under the disk which generates lower pressures. According to experimental results [6], transversal forces are already dominant at a valve opening of 6 mm. Surprisingly, and as shown in figure 13, the total force acting on the stem is well predicted, with a deviation of less than 5% with respect to the experimental values. Axial force measurements are also known to be affected by the friction of the stem in its seating, which may bias the measurement of the hydrodynamic force. Additional simulations have been run at different valve apertures, to understand better the evolution of the force ratio. Eventually, the simulation at $\Delta p = 1.54 \text{bar}$ and 4mm lift (Figure 8) gives an accurate agreement on axial force estimation (less than 1 %). This reinforces the idea of experimental measurement errors at 6 mm lift.

![Figure 11: Transversal Force on the Stem](image1.png)

![Figure 12: Axial Force on the Stem](image2.png)

The force on the stem can be characterized through the hydraulic surface, defined as:

$$Sh_j = \frac{F_j}{\Delta p} \quad (m^2)$$

(14)

where $F_j$ stand for the $j$ component of the force, $a$ for the axial forces and $t$ for transversal forces. The mean value of $Sh$ and its standard deviation are reported in table 5.

In order to understand better the induced forces, figure 14 represents the time average pressure contours on the disc obtained with OpenFoam. The axial force is strongly related to the pressure difference between the upper and the lower surfaces of the stem, whereas the transversal force depends on the pressure distribution on the sides of the disc. It can be observed that the contours on the lower surface of the disc are conditioned by the stagnation region which dominates the axial force acting in the positive z-axis direction. The pressure on left side of the stem decreases drastically to values close to the vapor pressure ($p_{vap}$), which explains the transversal forces in negative x-direction. Observing the streamlines in figure 14, a strong acceleration is seen in the restriction, followed by a recirculation region at the entrance of the outlet duct. The influence of the turbulence model is being also investigated: while its influence has been shown to be negligible on the flow rate, it seems to be more important for the forces.

![Figure 13: Total Force on the Stem](image3.png)

**TABLE 5. HYDRAULIC SURFACE, $Sh$**

<table>
<thead>
<tr>
<th></th>
<th>$Sh_a (m^2)$</th>
<th>$\sigma_{Sh_a}$</th>
<th>Error %</th>
<th>$Sh_t (m^2)$</th>
<th>$\sigma_{Sh_t}$</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exp.</td>
<td>10.84</td>
<td>1.01</td>
<td>-</td>
<td>73.41</td>
<td>6.97</td>
<td>-</td>
</tr>
<tr>
<td>OF</td>
<td>47.03</td>
<td>4.20</td>
<td>334</td>
<td>39.18</td>
<td>0.984</td>
<td>45</td>
</tr>
<tr>
<td>CFX</td>
<td>46.59</td>
<td>-</td>
<td>330</td>
<td>33.42</td>
<td>-</td>
<td>54</td>
</tr>
</tbody>
</table>

where $F_j$ stand for the $j$ component of the force, $a$ for the axial forces and $t$ for transversal forces. The mean value of $Sh$ and its standard deviation are reported in table 5.
CONCLUSIONS

In this work, two different CFD codes (OpenFoam and Ansys-CFX) are assessed for the simulation of a 3D cavitating flow in a globe valve that was characterized experimentally. The cavitating model is similar in both cases, with a slight difference in the treatment of the vapor phase. The two codes succeed in predicting reasonably well the flow rate through the valve, but they tend to underestimate the mass transfer by cavitation, which explains that the quality of the predictions decreases slightly in conditions of fully developed cavitation. Qualitatively, OpenFoam and Ansys-CFX calculate correctly the location of cavitation, and the highly unsteady behavior of cavities is well reproduced. Regarding the force exerted by the fluid on the stem, both codes give an axial force larger than the transversal one, in contrast with experiments, possibly due to measurement errors.

In order to understand these issues, more computations are being performed at different valve apertures, and with different turbulence models. Up to now, the simulations at 4mm lift are in good agreement with experiments in terms of axial force prediction. The influence of the treatment of the vapor phase is also under investigation, to try to improve the prediction of vapor extension.

ACKNOWLEDGEMENTS

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

REFERENCES