Cavitation From a Butterfly Valve: Comparing 3D Simulations to 3D X-Ray Computed Tomography Flow Visualization

Graham Brett  
*Fisher Controls International, LLC*

Marc Riveland  
*Fisher Controls International, LLC*

Terrence C. Jensen  
*Iowa State University*

Theodore J. Heindel  
*Iowa State University, theindel@iastate.edu*

Follow this and additional works at: [https://lib.dr.iastate.edu/me_conf](https://lib.dr.iastate.edu/me_conf)

Part of the [Acoustics, Dynamics, and Controls Commons](https://lib.dr.iastate.edu/acoustics-dynamics-and-controls-commons), [Computer-Aided Engineering and Design Commons](https://lib.dr.iastate.edu/engineering-design-commons), [Fluid Dynamics Commons](https://lib.dr.iastate.edu/fluid-dynamics-commons), and the [Graphics and Human Computer Interfaces Commons](https://lib.dr.iastate.edu/graphics-human-computer-interf)

**Recommended Citation**

[https://lib.dr.iastate.edu/me_conf/125](https://lib.dr.iastate.edu/me_conf/125)

This Conference Proceeding is brought to you for free and open access by the Mechanical Engineering at Iowa State University Digital Repository. It has been accepted for inclusion in Mechanical Engineering Conference Presentations, Papers, and Proceedings by an authorized administrator of Iowa State University Digital Repository. For more information, please contact [digirep@iastate.edu](mailto:digirep@iastate.edu).
Cavitation From a Butterfly Valve: Comparing 3D Simulations to 3D X-Ray Computed Tomography Flow Visualization

Abstract
Flow control valves may experience localized cavitation when the local static pressure drops to the liquid vapor pressure. Localized damage to the valve and surrounding area can occur when the vapor cavity collapses. Valve designs that reduce cavitation are based on empirical evidence and accumulated experience, but there are still considerable cavitation problems in industry. Valve designers may use computational fluid dynamics (CFD) to simulate cavitation in flow control valves, but model validation is challenging because there are limited data of local cavitation from the valve surface. Typically, the intensity of cavitation in a control valve is inferred from measurements of observable side effects of cavitation such as valve noise, vibration, or damage to the valve assembly. Such an indirect approach to characterizing cavitation yields little information about the location, degree, and extent of the cavitation flow field that can be used in CFD validation studies. This study uses 3D X-ray computed tomography (CT) imaging to visualize cavitation from a 5.1 cm diameter butterfly valve and compares the resulting vapor cloud to that predicted by CFD simulations. Qualitative comparisons reveal that the resulting cavitation structures are captured by the simulations when a small amount of non-condensable gas is introduced into the fluid and the simulations are completed in a transient mode.

Keywords
butterfly valve, cavitation, computational fluid dynamics, simulations, x-ray computed tomography

Disciplines
Acoustics, Dynamics, and Controls | Computer-Aided Engineering and Design | Fluid Dynamics | Graphics and Human Computer Interfaces

Comments
CAVITATION FROM A BUTTERFLY VALVE: COMPARING 3D SIMULATIONS TO 3D X-RAY COMPUTED TOMOGRAPHY FLOW VISUALIZATION

Graham Brett and Marc Riveland  
Fisher Controls International, LLC  
Emerson Process Management  
Marshalltown, IA 50158

Terrence C. Jensen  
Center for Nondestructive Evaluation  
Iowa State University  
Ames, IA 50011

Theodore J. Heindel  
Department of Mechanical Engineering  
Iowa State University  
Ames, Iowa 50011

ABSTRACT  
Flow control valves may experience localized cavitation when the local static pressure drops to the liquid vapor pressure. Localized damage to the valve and surrounding area can occur when the vapor cavity collapses. Valve designers that reduce cavitation are based on empirical evidence and accumulated experience, but there are still considerable cavitation problems in industry. Valve designers may use computational fluid dynamics (CFD) to simulate cavitation in flow control valves, but model validation is challenging because there are limited data of local cavitation from the valve surface. Typically, the intensity of cavitation in a control valve is inferred from measurements of observable side effects of cavitation such as valve noise, vibration, or damage to the valve assembly. Such an indirect approach to characterizing cavitation yields little information about the location, degree, and extent of the cavitation flow field that can be used in CFD validation studies. This study uses 3D X-ray computed tomography (CT) imaging to visualize cavitation from a 5.1 cm diameter butterfly valve and compares the resulting vapor cloud to that predicted by CFD simulations. Qualitative comparisons reveal that the resulting cavitation structures are captured by the simulations when a small amount of non-condensable gas is introduced into the fluid and the simulations are completed in a transient mode.

Keywords: butterfly valve, cavitation, computational fluid dynamics, imaging, simulations, X-ray computed tomography

NOMENCLATURE

CTV local time-average CT value
$c_p$ specific heat
$D$ pipe diameter
$D_{AB}$ mass diffusivity
$E$ energy
$f$ body force
$g$ gravity
$h$ sensible enthalpy
$k$ thermal conductivity
$n_b$ number of bubbles per unit volume
$P$ pressure
$AP$ pressure drop
$Q$ volumetric flow rate
$R$ mass source due to cavitation
$\mathcal{R}_B$ bubble radius
$Re$ Reynolds number
$S_E$ source term
$T$ temperature
$t$ time
$V$ mean velocity
$v$ local velocity

Greek Symbols

$\alpha$ void fraction
$\mu$ dynamic viscosity; X-ray attenuation coefficient
$p$ density
$\sigma$ cavitation number, $(P_2 - P_1)/(P_1 - P_2)$

*Corresponding Author: T.J. Heindel, theindel@iastate.edu

Copyright © 2011 by ASME
Subscripts
1 upstream
2 downstream
a air
dr drift
eff effective
k phase k
l liquid
m mixture
t turbulent
v vapor

INTRODUCTION
When used in liquid service, control valves may experience localized cavitation. Cavitation consists of the formation of liquid vapor cavities when the local static pressure drops to the liquid vapor pressure. Localized damage to the valve and surrounding area can occur when the vapor cavity collapses. Valve designs that reduce cavitation related problems are based on a theoretical understanding of cavitation and supplemented with empirical evidence and accumulated experience. Effective control of cavitation in industrial processes has evolved considerably over the last several decades, but nevertheless remains an area of challenging and focused study. A major impediment to understanding valve cavitation is the limited visual access a researcher has to the flow field within the valve. Typically the intensity and location of cavitation effects must be inferred from other measurements such as valve noise, vibration, or damage to the valve assembly.

Acoustic signatures have been used to identify cavitation from butterfly valves [1, 2], but the understanding of the cavitation field could be improved through improved visualization. Chern et al. [3] performed visualization and particle tracking using a customized acrylic ball valve and plexiglass tubing to show recirculation and cavitation regions. Cavitation from butterfly valves has also been visualized by Tani et al. [4, 5] using pressure sensitive film and high speed photography. They were able to identify the location of vortex cavitation which was shown to be highly erosive.

Palau-Salvador et al. [6] completed a 3D computational fluid dynamics (CFD) study of the flow inside a piston control valve and incorporated the cavitation model supplied with FLUENT V6.2. A challenge in the simulations is the prediction of cavitation. Wienken et al. [7] developed a method to predict cavitation in flow past a cylinder with a square cross-section; they used Large-Eddy Simulation (LES) and a stability criteria when the local pressure is minimized to predict cavitation. Their cavitation simulations were in good agreement with experimental results.

This study applies 3D X-ray computed tomography imaging technology to visualize cavitation from a "generic" butterfly valve and compares the resulting vapor cloud to that predicted by CFD simulations.

EXPERIMENTAL PROCEDURES
Cavitation was visualized using the one-of-a-kind X-ray flow visualization facility available at Iowa State University. The imaging capabilities of this facility are detailed elsewhere [8-10]. Figure 1 shows the imaging facility and the external components of the cavitation flow loop. The imaging room contained a butterfly valve, while all other major flow loop components were located outside this space.

Figure 1: X-RAY FLOW VISUALIZATION IMAGING ROOM AND EXTERNAL FLOW LOOP COMPONENTS.

A Griswold Model 811 ANSI Process pump, connected to a WorldWide Electric Corporation three-phase motor and driven by a Commander CDVE Motor Controller, pumped tap water through the cavitation flow loop. The water was stored in a 1900 L (500 gal) storage tank. The initial water temperature was ambient conditions and the water temperature increased by 5-10°C during a cavitation test as a result of pump and viscous heating. Note that the inlet pressure to, and pressure drop across the butterfly valve were controlled with downstream and upstream flow control valves. Different levels of cavitation were established in the flow field by increasing the pressure drop across the valve to the incipient cavitation point and beyond.

Figure 2 shows the butterfly valve mounted in the flow loop and located inside the X-ray imaging room. The flow travels up from underneath the imaging facility and exits through the ceiling before it is routed back to the storage tank. Upstream of the butterfly valve, a D = 5.1 cm (2 in) PVC pipe provides about 70 pipe diameters of straight flow. A D = 5.1 cm (2 in) clear acrylic pipe provides about 45 pipe diameters of straight flow in the downstream flow section. Pressure was measured upstream and downstream of the valve location.
The X-ray imager shown in Figure 2 is comprised of a single 44x44 cm cesium-iodide (CsI) phosphor screen coupled via a 50 mm f/1.2 lens to an Apogee Alta U9 cooled CCD camera with 3072(H)x2048(V) active pixels, with a pixel size of 9 μm x 9 μm. This camera has a thermoelectric cooler to enable low-noise, long exposures, and options to digitize signals at either 12 or 16 bits. The camera also has several binning options, and for images in this work, the 4x4 option was used for a resulting image size of 768(H)x512(V) active pixels. This camera has a thermoelectric cooler to enable low-noise, long exposures, and options to digitize signals at either 12 or 16 bits. The camera also has several binning options, and for images in this work, the 4x4 option was used for a resulting image size of 768(H)x512(V) active pixels. This system is used for computed tomography (CT) imaging, and with an exposure time range of 0.1-100 sec, it can only provide time-average data of dynamic flow processes. For this study, an exposure time of 1 second was used for each of the 360 X-ray projections that were acquired in 1 degree increments around the test section.

The X-ray source used in this work is a LORAD LPX200 portable X-ray tube with beryllium output window and a 1.5 mm focal spot size. The maximum power is 900 W with adjustable voltage (10-200 kV) and current (0.1-10.0 mA) capabilities. The window provides a 60° horizontal and 40° vertical conical X-ray beam. Filters (typically copper, brass, or aluminum) of varying thicknesses are used to suppress low energy radiation. For this study, the X-ray source was set at 140 kV and 5 mA with two 1 mm copper filters and one 1.7 mm aluminum filter.

The butterfly valve used in this study was a D = 5.1 cm (2 in) PVC butterfly valve purchased from McMaster-Carr. The butterfly angle of 30° was fixed for all test conditions in this study. Figure 3 shows a close-up image of the original and modified valve mounted in the flow loop. The best X-ray imaging is done when there are not large discontinuities in the material density within the imaging region [11]. The original valve was mounted with nylon bolts to avoid metal in the imaging region. However, there were still voids in and around the valve region that produced X-ray image artifacts (see, for example, Ketcham and Carlson [12] for a discussion on X-ray CT image artifacts). To minimize these artifacts, the mounting plate downstream of the valve was removed, a solid cap was fabricated to fit over the nuts in the downstream location, all voids (including bolt holes) were filled with wax, the metal valve stem was replaced with a nylon stem, and the valve handle was minimized in size and fabricated from Delrin. Note that the internal material and geometry of the valve remained fixed.

Figure 4 shows an X-ray CT image of the outside surface of the modified butterfly valve along with a 3D coordinate system to define the valve orientation. Because the X-ray source is a cone beam, the CT reconstruction produces a density map of the entire imaging volume. In this case, the butterfly valve is captured in its entirety, and it can be sectioned along various planes (axes) to view the internal density map; these sections are defined as CT slices. Since the valve was filled only with air for the images in Figure 4, a large density gradient exits between the flow region (that would be filled with water) and the valve housing. The fixed 30° valve angle is clearly shown and the various directions are identified in each CT slice.
NUMERICAL PROCEDURES

Multiphase Model

Computational Fluid Dynamics (CFD) is used to simulate the flow through the butterfly valve at test conditions to compare the simulated vapor generation downstream of the valve to that observed in the experiment. FLUENT V12 software was chosen to develop the physical models and solve the simulation. The goal of the simulation is to predict the cavitation structures found in the CT images; this will provide qualitative validation and confidence that future simulations can be used to design new flow control valves that control cavitation.

The mixture model was selected for modeling water flow through the valve. This model is a simplified multiphase model that can be used to model homogeneous flows with strong coupling where each phase moves at the same velocity. The mixture model can model the two phases by solving the momentum, continuity and energy equations for the mixture [13]. In this case, the mixture is liquid water and water vapor.

The continuity equation for the mixture is:

\[ \frac{\partial}{\partial t}(\rho_m \bar{v}_m) + \nabla \cdot (\rho_m \bar{v}_m \bar{v}_m) = 0 \]  (1)

where local \( \bar{v}_m \) is the mass-averaged velocity:

\[ \bar{v}_m = \sum_{k=1}^{2} \alpha_k \rho_k \bar{v}_k \]  (2)

\( \rho_m \) is the mixture density:

\[ \rho_m = \sum_{k=1}^{2} \alpha_k \rho_k \]  (3)

and \( \alpha_k \) is the volume fraction of phase \( k \). In this case there are two phases, liquid water and water vapor.

The momentum equation for the mixture is obtained by summing the individual momentum equations for all phases and is expressed as:

\[ \frac{\partial}{\partial t} \left( \rho_m v_m \right) + \nabla \cdot \left( \rho_m v_m v_m \right) = -\nabla p + \nabla \cdot \left[ \mu_m \left( \nabla v_m + \nabla v_m^T \right) \right] + \rho_m g + \bar{F} + \nabla V \left( \sum_{k=1}^{2} \alpha_k \rho_k \bar{v}_{a,k} \bar{v}_{a,k} \right) \]  (4)

where \( \bar{F} \) is a body force and \( \mu_m \) is the mixture viscosity:

\[ \mu_m = \sum_{k=1}^{2} \alpha_k \mu_k \]  (5)

and \( \bar{v}_{a,k} \) is the drift velocity for the vapor phase.

The mixture model was selected for modeling water flow through the valve. This model is a simplified multiphase model that can be used to model homogeneous flows with strong coupling where each phase moves at the same velocity. The mixture model can model the two phases by solving the momentum, continuity and energy equations for the mixture [13]. In this case, the mixture is liquid water and water vapor predicted using the cavitation model.

The continuity equation for the mixture is:

\[ \frac{\partial}{\partial t}(\rho_m \bar{v}_m) + \nabla \cdot (\rho_m \bar{v}_m \bar{v}_m) = 0 \]  (1)

The energy equation for the mixture takes the form:

\[ \frac{\partial}{\partial t} \left( \rho_m E_k \right) + \nabla \cdot \left( \rho_m v_k (\rho_k E_k + p) \right) = \nabla \cdot (k_{eff} V T) + S_e \]  (6)

where \( k_{eff} \) is the effective conductivity, \( \sum \alpha_k (k_k + k_\epsilon) \), with \( k_\epsilon \) the turbulent thermal conductivity defined by the turbulence model employed in the solution procedure.

In Eq. (6),

\[ E_k = h_k - \frac{p_v}{\rho_k} \frac{\rho_k^2}{2} \]  (for a compressible phase)  (7)

\[ E_k = h_k \]  (for an incompressible phase)

where \( h_k \) is the sensible enthalpy for phase \( k \).

The realizable \( k-\epsilon \) model [14] is used in this study for turbulence modeling.

Modeling Vapor Generation

The Schmerr and Sauer [15] model is used for predicting cavitation. The expression for net mass transfer from liquid to vapor is governed by the general form of the vapor volume fraction equation:

\[ \frac{\partial}{\partial t} \left( \alpha \rho \bar{v}_v \right) + \nabla \cdot \left( \alpha \rho \bar{v}_v \bar{v}_v \right) = \frac{\rho \rho_1 D \alpha}{\rho} \]  (8)

where the net mass source term is:

\[ \frac{\partial}{\partial t} \left( \alpha \rho \bar{v}_v \right) + \nabla \cdot \left( \alpha \rho \bar{v}_v \bar{v}_v \right) = \frac{\rho \rho_1 D \alpha}{\rho} \]  (8)

and \( \alpha_k \) is the volume fraction of phase \( k \). In this case there are two phases, liquid water and water vapor.
To connect the vapor volume fraction to the number of bubbles per volume of liquid, \( n_b \), Sclimenti and Sauer [15] used:

\[
\alpha = \frac{n_b \frac{4}{3} \pi (\mathcal{R}_b)}{1 + n_b \frac{4}{3} \pi (\mathcal{R}_b)} \tag{9}
\]

where \( \mathcal{R}_b \) is the bubble radius.

The final form of the net mass source is:

\[
R_s = \frac{\rho_s \rho_i}{\rho} \alpha (1 - \alpha) \frac{3}{\mathcal{R}_b} \sqrt{\frac{2}{3}} \frac{(P_v - P)}{\rho_i} \text{ when } P_v \geq P \tag{11}
\]

and

\[
R_s = \frac{\rho_s \rho_i}{\rho} \alpha (1 - \alpha) \frac{3}{\mathcal{R}_b} \sqrt{\frac{2}{3}} \frac{(P_v - P)}{\rho_i} \text{ when } P_v < P \tag{12}
\]

Where \( P_v \) is the vapor pressure and \( P \) is the local static pressure.

The bubble radius is:

\[
\mathcal{R}_b = \left( \frac{\alpha}{1 - \alpha} \frac{3}{4 \pi n_b} \right)^{\frac{1}{3}} \tag{13}
\]

The FLUENT model requires that the materials present in the simulation be defined as a liquid and vapor. The liquid phase properties were defined as follows:

- Density, \( \rho_l \): 994.3 kg/m³
- Specific heat, \( c_{p,l} \): 1006 J/kg-K
- Viscosity, \( \mu_l \): 0.0007339 kg/m-s

The vapor was defined as a mixture of 2 species: water vapor and a small amount of air. The addition of air is needed to more accurately model the fluid used in the experiment and also to improve the convergence of the numerical solution. The vapor phase properties were defined as:

- Density, \( \rho_v \): assume ideal gas behavior
- Viscosity, \( \mu_v \): 1.0128e-5 kg/m-s
- Mass diffusivity, \( D_{AB} \): 2.88E-05 m²/s

The specific heat (\( c_p \)) of the vapor mixture is computed as the mass fraction average of the pure species (i.e. water vapor and air) heat capacities.

Grid Resolution

In order to correctly model the valve geometry and upstream/downstream piping sections, the computational mesh must be sufficient to capture the flow physics and conform to the valve geometry. One half of the model geometry at 30° open was modeled to take advantage of symmetry. Tetrahedral elements were used for the valve portion of the models while hexahedral elements were used for the piping sections of the model (Figure 5). Note that, although not shown, the complete computational domain included 6 pipe diameters upstream of the control valve and 16 pipe diameters downstream of the control valve, utilizing more than 785,000 computational cells.

Boundary Conditions

The inlet flow was assumed to have a uniform velocity profile where the fluid was at a fixed temperature of 307 K. The exit pressure was fixed at atmospheric conditions and the inlet pressure was adjusted to match the experimental pressure drop conditions. All solid surfaces have no-slip boundary conditions. The axis of symmetry of the pipe and control valve allowed for a symmetry boundary plane through the centerline of the system to reduce the computational domain by a factor of two.
**Numerical Methodology for CFD solution**

Initial attempts at solving the simulation as a steady-flow problem were unsuccessful. Several adjustments were made to the initial model in an attempt to provide a converged steady-state solution. These adjustments included:

- Attempting to solve with a constant vapor density
- Revising the vapor to use ideal gas formulation for density.
- Inclusion of non-condensable gas (air) in the vapor phase.

The above adjustments improved solution convergence, particularly the addition of a non-condensable gas, but they did not provide a comparable quantity of vapor in the downstream portion of the model.

Additional simulation improvements were obtained by solving the simulation as a transient solution with a small time-step (2.6e-5 s). This method provided better results than the steady-flow simulation. Changes between time steps were negligible after about 0.44 s and the solution was deemed converged. Additional calculations continued for a total time of 1.514 s.

**Solver Settings for the CFD solution**

The FLUENT simulation incorporated the SIMPLE algorithm for pressure-velocity coupling [16]. Also, the following discretization schemes were used:

- the PRESTO! scheme was used for pressure;
- the QUICK scheme was used for volume fraction;
- second order upwind scheme was used for density, momentum, turbulent kinetic energy, turbulent dissipation rate, and energy.

**RESULTS AND DISCUSSION**

**Experimental Observations**

Several different experimental cavitation trials were completed for a variety of flow conditions as outlined in Table 1. The initial fluid temperature ranged 25-35°C, and then increased 5-10°C during the experiment. The maximum fluid temperature at the end of any trail was 39°C. The temperature reported in Table 1 is the average of the temperature before and after the experiment. The flow rate was controlled with the variable speed pump and a back pressure adjustment valve.

<table>
<thead>
<tr>
<th>Test</th>
<th>Tm (°C)</th>
<th>Q (lpm)</th>
<th>V (m/s)</th>
<th>Re</th>
<th>ΔP (kPa)</th>
<th>(Pf-Pg)/ΔP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>28.5</td>
<td>300</td>
<td>2.5</td>
<td>1.5×10^5</td>
<td>400</td>
<td>0.0078</td>
</tr>
<tr>
<td>2</td>
<td>26.5</td>
<td>360</td>
<td>3.0</td>
<td>1.8×10^5</td>
<td>510</td>
<td>0.0065</td>
</tr>
<tr>
<td>3</td>
<td>30.0</td>
<td>350</td>
<td>2.9</td>
<td>1.9×10^5</td>
<td>650</td>
<td>0.0041</td>
</tr>
<tr>
<td>4</td>
<td>36.5</td>
<td>360</td>
<td>3.0</td>
<td>2.2×10^5</td>
<td>750</td>
<td>0.0013</td>
</tr>
</tbody>
</table>

Tests 1 and 2 were the initial tests to visualize cavitation; these results are summarized in Figure 6. The empty y-z and x-z projections through the valve centerline are shown for reference. The water only (bulk) image represents the valve region filled with water with no cavitation. The flow images represent the “raw” CT images for the various flow conditions. Note that one can start to visualize the cavitation for Test 2 when Re = 1.8×10^5 and ΔP = 510 kPa, but it is more challenging for Test 1.

![](https://via.placeholder.com/150)

**FIGURE 6: X-RAY CT SLICES OF CAVITATION FROM A BUTTERFLY VALVE DURING TESTS 1 AND 2. THE TWO SLICE LOCATIONS CORRESPOND TO MUTUALLY PERPENDICULAR PLANES THROUGH THE PIPE CENTERLINE.**

To enhance the difference between the cavitating and non-cavitating flow conditions, the flow image file was normalized with respect to the bulk image file (i.e., the voxel values in the flow file were divided by the voxel values in the bulk file, where a voxel is a 3D pixel). This process provides a voxel value between zero and one and enhances the differences during cavitation. These images are shown in Figure 6 under the “Flow/Bulk” heading. The y-z projection in Figure 6 clearly shows that cavitation is occurring on the back side of the valve leading edge and is represented by the darker blue regions downstream of the valve. Cavitation is also observed at the tip of the valve trailing edge, and is enhanced by a sharp edge between the pipe flange and valve housing. The increased pressure drop enlarges the cavitation region (Test 2). The x-z projection shows that the cavitation structure is approximately symmetric about the valve centerline.

The images in Figure 6 represents qualitative CT observations of cavitation from a butterfly valve. In order to quantify CT observations, the time-average local void fraction can be estimated from each cavitation image. The local time-average void fraction (α_v) can be determined by knowing the local linear X-ray attenuation for the cavitation flow condition (μ), the attenuation for a water-filled system (μ_w), and the attenuation for a water vapor-filled system (μ_v) [17]. Since attenuation is proportional to CT value, the local void fraction can instead be calculated from the recorded local CT value for the cavitating flow (CTV), the water-filled system (CTV_w), and...
the vapor-filled system (CTVv) [11]. Assuming that the X-ray attenuation for water vapor is similar to that of air, CTVv ≈ CTVs, and

$$\alpha_v = \frac{\mu - \mu_1}{\mu_v - \mu_1} = \frac{CTV - CTVs}{CTVv - CTVs}$$ (16)

Equation (16) is used to provide a 3D void fraction map within the imaging region. Figure 7 shows these results for all four test conditions summarized in Table 1. Note that the color scale for each image is identical and represents the time-average void fraction from all liquid water (blue) to all water vapor (red). By plotting time-average local void fraction, it is clear that cavitation does not produce much vapor for the conditions of Test 1; although with the color mapping set to the same scale for all cases, Test 1 could easily produce about 10% void in some regions on a time-average sense.

Numerical Simulations

Simulations of the various test conditions were attempted but cavitation was only observed for conditions similar to experimental Test 4, and then only after a small amount of dissolved air was included in the fluid. The exact conditions for these simulations included: $\Delta P = 750$ kPa, $V = 2.5$ m/s, and $Re = 1.7 \times 10^5$. As stated previously, the simulations were completed in a transient mode with a small time step. Steady-state conditions were observed after about 0.44 s of simulated time. The figures below are shown for a time step of 1.5 s, when no changes in local conditions are recorded. Also, the figures show the local region near the fixed control valve, although the simulation domain included 6 pipe diameters upstream of the control valve and 16 pipe diameters downstream of the control valve.

Figure 8 shows the velocity magnitude in the center plane perpendicular to the valve axis. As expected, the highest velocities are initiated at the trailing edge of the downstream and upstream valve regions, where jets of high velocity fluid are recorded. The jets merge and create a recirculation region.

Figure 9 shows the local void fraction in two mutually perpendicular planes through the pipe centerline. Cavitation is highest near a “lip” where the control valve attaches to the downstream pipe section. Cavitation in this region is also observed in Figures 6 and 7. A significant amount of vapor is also generated immediately downstream of the control valve. Although the void fraction magnitudes do not correlate well with the experimental results, the general void fraction structures are observed. One reason for this discrepancy is the amount of dissolved gas in the simulations does not necessarily match that of the experiments, which was not recorded.

There are other potential reasons for the miss-match between the experimental results and the simulations. First, the simulations could underestimate the void fraction due to the high shear flows found downstream of the control valve, which could lead to an underestimation of the number of cavitation events. Second, the experimental system was highly turbulent and no void removal device was incorporated into the flow loop. In this case, an equilibrium residual void fraction could be maintained under steady state conditions, and the amount would be a function of the flow conditions; this would cause the experimental measurements to overestimate the void present.
average density variations in the cavitation region that can be equated to time-average local void fraction. Three-dimensional simulations of the experimental geometry reveals similar cavitation structures, although the magnitude of void (vapor) generation did not match. Improvements for future comparisons include a better measure of the experimental dissolved gas content to better simulate vapor formation, and the use of a non-deflecting valve.

ACKNOWLEDGMENTS

The X-ray facility used in this research was funded by the National Science Foundation under award number CTS-0216367 and Iowa State University. The flow test equipment and partial funding for this work were provided by Emerson Process Management, Marshalltown, IA.

REFERENCES


