Using Computational Fluid Dynamics to Predict the Onset of Cavitation ($x_{Fz}$)

Presenter:

Alan H. Glenn
Academic Education and Degrees:
1987 Doctoral Thesis on Eliminating Screech in Control Valves, Brigham Young University, Provo, Utah
Present Position: Principal Engineer at Flowserve Corporation—Flow Control Division

Associate Author:

Gifford Decker
Academic Education and Degrees:
2006 Masters Degree in Mechanical Engineering, Brigham Young University, Provo, Utah
Present Position: Staff Engineer at Flowserve Corporation—Flow Control Division

Originally presented at the Valve World 2008 Conference, Maastricht, the Netherlands
Using Computational Fluid Dynamics to Predict the Onset of Cavitation ($x_{Fz}$)

Alan H. Glenn, Gifford Z. Decker, FLOWSERVE FCD Valtek Control Products

**Keywords:** cavitation, computational fluid dynamics, CFD, $x_{Fz}$, vapor pressure, shear zone, vortices

1 Abstract

Computational Fluid Dynamics (CFD) has been used extensively to successfully model fluid flow in other fields, such as aerospace and pump design. It has not been used as much to model the very complex flow through valves. In this study, however, a high end CFD tool was used to numerically predict the point of incipient cavitation in several complex valve configurations.

Flow conditions at the point of incipient cavitation are used to determine the characteristic pressure ratio, $x_{Fz}$, a parameter necessary for predicting hydrodynamic noise in control valves using the international hydrodynamic noise prediction standard, IEC60534-8-4. In the past, accurate values for $x_{Fz}$ could only be found from expensive and time consuming tests, which were often not feasible, especially for large valves. Using CFD analysis to accurately predict the flow conditions where cavitation begins could greatly decrease the cost of obtaining accurate values for $x_{Fz}$.

In this study, values for $x_{Fz}$ were found using several separate geometries and conditions where cavitation would be present. The value of $x_{Fz}$ was taken as the value of the ratio $(p_1-p_2)/(p_1-p_v)$, where $p_1$ and $p_2$ are the valve upstream and downstream pressures respectively and $p_v$ is the fluid vapor pressure, at the conditions where the lowest value for static pressure minus dynamic pressure, anywhere in the flow field, equaled the vapor pressure, $p_v$. The CFD analyses predicted values for $x_{Fz}$, using this method, which agreed with the values determined from testing.

This paper briefly explains cavitation and its effect on hydrodynamic noise, the approach used to predict $x_{Fz}$ using CFD methods, and the results obtained using these numerical methods. Finally, it shows the comparison between the CFD and test results, from tests performed on the same valve geometries, to validate the CFD predictions.
2 Introduction

Cavitation in liquid flow through control valves is a serious problem; it can cause severe damage to the valve and make it unusable and it can be the source of unacceptably-high sound pressure levels. Cavitation results from a two step process that can occur in a control valve if the pressure drop is high enough relative to the difference between the upstream pressure and the vapor pressure. First, the pressure in the liquid drops to a value below the vapor pressure and vapor bubbles form in localized regions near or downstream of a restriction in the valve. Then, further downstream, the fluid pressure recovers or increases to a pressure above the vapor pressure and the vapor bubbles suddenly collapse. The violent collapse of the bubbles creates pressure pulses that result in significant noise and, if close to a material surface, can cause damage to the material.

Cavitation is a very complex phenomenon that has defied precise prediction by analytical methods. However, with more advanced computer systems and better fluid analytical models, progress has been made to the point where computational fluid dynamics (CFD) can be used to predict some useful information relative to cavitation. This may reduce the amount of testing needed to find the parameters required to predict the noise and, perhaps, to determine the conditions where cavitation can be damaging. This paper discusses CFD analysis methods recently developed that were used to determine the approximate characteristic pressure ratio, $x_{FZ}$, (the point of incipient cavitation) a key parameter needed to accurately calculate the sound pressure level of a control valve in liquid service. The methods are described and compared with results from testing.

3 Cavitation noise

The noise produced by liquid flow is relatively low until the pressure drop is high enough that cavitation begins. Fig. 1 below is a typical plot of the sound pressure level, $L_p$, versus the differential pressure ratio, $x_F$. The parameter $x_F$ is a ratio of the pressure drop across the valve divided by the pressure difference $(p_1 - p_v)$ where $p_1$ is the upstream pressure and $p_v$ is the vapor pressure of the liquid at the upstream temperature. When the pressure drop is very low, and $x_F$ is low, there is no cavitation and the sound pressure level is low. As $x_F$ increases, $L_p$ increases gradually until the point where cavitation just begins. The value of $x_F$ at this point is called the characteristic pressure ratio and designated as $x_{FZ}$. In the plot below, the region with no cavitation goes from $x_F = 0$ to $x_F = x_{FZ} = 0.57$. Once cavitation begins, the sound pressure level increases very rapidly as $x_F$ increases. The international hydrodynamic noise prediction standard IEC 60534-8-4, requires an accurate value of $x_{FZ}$ to predict hydrodynamic noise. It is one of the few parameters required by the standard that cannot be easily determined by just knowing the valve geometry and service conditions. In the past, it could only be determined experimentally but this paper...
presents a method of calculating it by CFD analysis that was found to be reasonably accurate for the rotary and linear valves and for the severe service valve trims analyzed.

![Graph of Lp vs. xF for Anti-cavitation Trim AC4]

**Fig. 1:** Sound pressure level, \( L_p \), versus \( x_F \) for anti-cavitation trim tested, designated as “AC4”, showing value of \( x_{Fz} \) (i.e. point of incipient cavitation).

## 4 CFD Analysis Setup and Assumptions

### 4.1 Introduction

The CFD analyses were set up by applying a constant pressure at the inlet of the geometry and a time varying pressure at the outlet. The assumption of symmetry was used to reduce the size of the models whenever possible. **Fig. 2** below shows a geometry (segmented ball valve) that was used in the CFD analyses to illustrate how the boundary conditions were applied.
The flow was modeled as turbulent flow using the Reynolds Averaged Navier-Stokes (RANS) k-ε turbulence model. Any boundary layers were modeled by implementing a wall treatment. Traditional CFD turbulent flow models, like the k-ε model, implement wall treatments in the boundary layer, such as the log law, which assume that the boundary layer effects (turbulent vortices) are represented well enough by relationships from empirical data for averaged values of velocity and pressure. Other models such as Large Eddy Simulation (LES) or Detached Eddy Simulation (DES) attempt to model the actual vortices inside the boundary layer, but require a very fine mesh in the boundary layer. The cost of computation for LES and DES is still not practical for everyday engineering problems. Two models for analyzing the pressure field for comparison to the vapor pressure and the onset of cavitation were developed. It was anticipated that the values for $x_{Fz}$ predicted by these two models would encompass the true $x_{Fz}$ (found by testing). These models are described in more detail below.

### 4.2 Calculated Pressure Model

The calculated pressure model refers to the pressure calculation based on the traditional pressure result obtained from solving the k-ε model. Therefore, the model predicts cavitation when the pressure in a finite volume cell is less than the vapor pressure: cavitation occurs if $p_{\text{cell}} < p_v$ (where $p_{\text{cell}}$ is the volume cell pressure, and $p_v$ is the vapor pressure of the fluid) [1]. This model works well when the cavitation results from large flow effects in the free stream. However, when the cavitation is mostly occurring due to localized low pressures in individual vortices of highly turbulent flow generated in the shear zone (i.e. the boundary layer or separated flow region) the calculated pressure field model is inadequate, due to the inability to accurately and easily predict the physics of such vortices using traditional CFD turbulent flow models.
4.3 Fluctuating Pressure Model

The second method for predicting the onset of cavitation includes an approximation for the pressure fluctuations that occur due to turbulent mixing in the shear zone or boundary layer. The approximation assumes that the pressure fluctuations will be on the order of the averaged velocity squared, $V^2$, at any point in flow field: 

$$p = \frac{1}{2} \rho V^2,$$

where $p$ is the approximate fluctuating pressure, $\rho$ is the density of the fluid, and $V$ is the velocity magnitude [2]. This fluctuating pressure is then subtracted from the average pressure at the same point in the flow field. The result is a measure of the approximate effect of the individual vortices in the shear zone on the pressure drop. This method is most effective for flows where there is large flow separation wherein the boundary layer is highly turbulent and large compared to the flow area, such as valve trims with small holes and channels designed to reduce cavitation.

To implement the fluctuating pressure model, a user defined function was created in the CFD program that took the pressure in the volume cell and subtracted $\frac{1}{2}$ times the density times the velocity squared of the volume cell:

$$p_{BL} = p_{cell} - \frac{1}{2} \rho_{cell} \cdot V_{cell}^2.$$ 

Where $p_{BL}$ is the predicted minimum pressure in the flow field, $p_{cell}$ is the pressure in the volume cell, $\rho$ is the density of the volume cell, and $V_{cell}$ is the velocity in the volume cell.

5 Results

5.1 Introduction

Several different types of geometries were analyzed including nine anti-cavitation trims, and three standard valve configurations: a segmented ball valve (SBV), a rotary butterfly disk valve (BDV), and a standard globe style valve (SGV). Each of the geometries that were analyzed using CFD, were also physically tested to determine the onset of cavitation for validation of the cavitation prediction methods. In all the tables and figures in the results section the data referred to as CFD1 represents the $x_{F_z}$ predictions using the fluctuating pressure model, and the data referred to as CFD2 represents the $x_{F_z}$ predictions using the calculated pressure model.

5.2 Anti-Cavitation Trims

The results for the anti-cavitation trims are presented below in Table 1 and in Fig. 3.
Table 1: Tabulated results for anti-cavitation trims including test results

<table>
<thead>
<tr>
<th>No</th>
<th>Fluctuating Pressure Model</th>
<th>Test</th>
<th>Calculated Pressure Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC1</td>
<td>0.508</td>
<td>0.478</td>
<td>0.628</td>
</tr>
<tr>
<td>AC2</td>
<td>0.501</td>
<td>0.500</td>
<td>0.601</td>
</tr>
<tr>
<td>AC3</td>
<td>0.544</td>
<td>0.540</td>
<td>0.609</td>
</tr>
<tr>
<td>AC4</td>
<td>0.578</td>
<td>0.570</td>
<td>0.617</td>
</tr>
<tr>
<td>AC5</td>
<td>0.485</td>
<td>0.580</td>
<td>0.586</td>
</tr>
<tr>
<td>AC6</td>
<td>0.355</td>
<td>0.475</td>
<td>0.474</td>
</tr>
<tr>
<td>AC7</td>
<td>0.510</td>
<td>0.610</td>
<td>0.632</td>
</tr>
<tr>
<td>AC8</td>
<td>0.578</td>
<td>0.610</td>
<td>0.711</td>
</tr>
<tr>
<td>AC9</td>
<td>0.554</td>
<td>0.738</td>
<td>0.745</td>
</tr>
</tbody>
</table>

Fig. 3: Bar graph with CFD results compared to test results CFD1 represents fluctuating pressure model, and CFD2 represents calculated pressure model (data from Table 1).
In the above bar graph, the blue columns represent the values of $x_{Fz}$ predicted using the CFD fluctuating pressure model (approximation of the shear zone pressure fluctuations), the red columns represent the values of $x_{Fz}$ from the testing, and the beige columns represent the value of $x_{Fz}$ using the CFD calculated pressure model. From the above plot it can be seen that the results of the cavitation prediction were very good for the anti-cavitation trims. All but one of the trims (AC4) are within 6% of either the fluctuating pressure model or the calculated pressure model, and AC4 is within 9%.

Most of the test data is closer to the fluctuating pressure model prediction (AC1, AC2, AC3, AC4, and AC8); the reason for this can be seen by further investigation of the flow area where the cavitation is predicted to occur. For the trims AC1, AC2, AC3, AC4, and AC8 the cavitation was predicted to happen in an area where large flow separation occurred and the boundary layer was very large and turbulent. **Fig. 4** shows an example of this cavitation region for AC2.

![Vector plot for AC2 showing where cavitation was predicted to occur, in the region of large flow separation.](image)

The above figure shows flow coming from a rectangular channel into a circular hole. As shown, the flow separates from the wall as it turns the corner from the
channel into the hole; which results in a very large region of separated flow. In the above example the separated flow area is greater than half the diameter of the hole, which would suggest that turbulent mixing and vortices in the shear zone will have a large effect on the minimum pressure.

For the other trims the test data was closer to the calculated pressure model predicted value (AC5, AC6, AC7, AC9); for these trims the cavitation was predicted to occur in regions where there was little or no flow separation, but very high velocity gradients resulting in low pressure drops. **Fig. 5** shows an example of this type of region for AC5.

![Vector plot for AC5](image)

**Fig. 5**: Vector plot for AC5 showing where cavitation was predicted to occur, in the region of high velocity gradient.

In the above vector plot, flow is exiting a hole and entering a channel with a much smaller area than the hole. This decrease in flow area results in a large velocity gradient as the flow must speed up to maintain the flow rate. However, unlike the flow in **Fig. 4**, the boundary layer is much smaller (less than one third the channel width) and reattaches quickly suggesting that the shear zone effects in this flow are not as significant as in **Fig. 4**.
5.3 Standard Valves

The results for the standard valves are presented in Table 2 and Fig. 6 below.

Table 2: Results for standard valves including test results

<table>
<thead>
<tr>
<th>Valve</th>
<th>Fluctuating Pressure Model</th>
<th>Test</th>
<th>Calculated Pressure Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segmented Ball Valve (SBV)</td>
<td>0.115</td>
<td>0.208</td>
<td>0.198</td>
</tr>
<tr>
<td>Butterfly Disk Valve (BDV)</td>
<td>0.172</td>
<td>0.133</td>
<td>0.276</td>
</tr>
<tr>
<td>Standard Globe Valve (SGV)</td>
<td>0.221</td>
<td>0.204</td>
<td>0.338</td>
</tr>
</tbody>
</table>

Fig. 6: Bar graph with CFD results compared to test results CFD1 represents fluctuating pressure model, and CFD2 represents calculated pressure model (Table 2).
The above results show a similar trend as seen in the results for the anti-cavitation trim results. In the above plot, the \( x_{Fz} \) values are plotted on the y-axis, the CFD fluctuating pressure model results are represented by the blue columns, the test results are represented by the red columns, and the CFD calculated pressure model results are represented by the beige columns. All the test results are within 5% of either the fluctuating pressure model or the calculated pressure value. Each valve was investigated in more detail to determine the type of flow regime that was prevalent when cavitation occurred.

5.3.1 Segmented Ball Valve (SBV)

The measured \( x_{Fz} \) for the SBV was very close to the CFD calculated pressure value; this would suggest that the cavitation is mostly due to high velocity gradients and or large pressure drops and not due to high turbulent mixing or vortices in the shear zone. **Fig. 7** and **Fig. 8** show the velocity magnitude contour plot through the half-plane of the SBV valve and a close up vector plot of the cavitation region respectively.

![Velocity contour plot for the Segmented Ball Valve (SBV) at 53.8% open.](image)

**Fig. 7**: Velocity contour plot for the Segmented Ball Valve (SBV) at 53.8% open.
Fig. 8: Vector plot of the Segmented Ball Valve (SBV) analysis showing where cavitation occurred.

In the above vector plot, the flow is passing through the port of the segmented ball, the resulting flow is similar to a jet, as can be seen there is very little shear near the point of maximum velocity because there is very little fluid flow in the region of separation. This leads to the conclusion that there is very little shear zone effect on the minimum pressure for this flow.

5.3.2 Butterfly Disk Valve (BDV)

The measured $x_{Fz}$ for the BDV unlike the Segmented Ball Valve (SBV) was closer to the CFD fluctuating pressure model value, suggesting that the cavitation is mostly due to turbulent mixing in the shear zone. Fig. 9 and Fig. 10 show the velocity magnitude contour plot through the half-plane of the BDV valve and a close up vector plot of the cavitation region respectively.
In contrast to the Segmented Ball Valve (SBV), the BDV vector plot (see Fig. 10) shows that there appears to be an area where there is greater probability for turbulent mixing, as seen in the area where the flow is separating near the leading edge. The velocities inside this mixing region are on the same order as the free stream velocities suggesting that the shear zone effect will be significant.

5.3.3 Standard Globe Valve (SGV)
The analysis of the SGV appeared to be like the Butterfly Disk Valve (BDV), with the test $x_{FZ}$ being closer to the CFD fluctuating pressure value. Fig. 11 and Fig. 12 show the velocity magnitude contour plot through the half-plane of the SGV and a close up vector plot of the cavitation region respectively.

**Fig. 11:** Velocity contour plot for the Standard Globe Valve (SGV) at 99.6% open.

**Fig. 12:** Vector plot of the Standard Globe Valve (SGV) analysis showing where cavitation occurred.
The above vector plot shows that the cavitation in the SGV initiated in the region where the flow had separated. In the separated region, the velocity again appears to be significant and acting in a direction opposite to the free stream, indicating that the shear zone effects are more likely to be causing cavitation (see Fig. 12).

5.4 Refined Application of the Fluctuating Pressure Model to Boundary Layer Flow Only

The above application of the two proposed models is sufficient for obtaining a bracketed value of the xFz for the valves and trims analyzed. However, further investigations of the application of the fluctuating pressure model showed that for the valve and trims for which cavitation occurred in the regions where there was very little flow separation or apparent boundary layer effects (AC5, 6, 7, and 9 and the segmented ball valve) the fluctuating pressure initially dropped below the vapor pressure outside the boundary layer in the free stream. It was thus proposed that the fluctuating pressure model be applied only in the boundary layer (including regions of flow separation shear mixing zones etc.) Therefore, an additional analysis was done on AC9 (following the same procedures as all other analyses) that ignored the calculated CFD fluctuating pressure model values in the free stream, while monitoring the results of the fluctuating pressure model in the boundary layer. Fig. 13 below shows the results of the additional analysis compared to the original results.
Fig. 13: Bar graph with CFD results compared to test results where AC9 represents the original analysis and AC9_2 represents the analysis done using only boundary layer fluctuating pressure results. CFD1 represents CFD fluctuating pressure model, and CFD2 represents the CFD calculated pressure model.

The above results suggest that applying the CFD fluctuating pressure model in the entire region can lead to a much smaller $x_{Fz}$ (see Table 1: Tabulated results for anti-cavitation trims including test results, Fig. 3, Table 2: Results for standard valves including test results, and Fig. 6) than what is really happening. However, if the method is applied only in the region of the boundary layer or separated flow as seen in Fig. 13 the results are much closer to the test values.

6 Conclusions

The CFD predictions for $x_{Fz}$ compared to test values have shown that it is possible to predict the onset of cavitation in control valves whether it is due to large scale flow effects, or turbulent mixing in the shear zone. Further more, an investigation of the flow region where cavitation is likely lead to the conclusion that the CFD fluctuating pressure model should be applied throughout the entire flow region, because this can lead to much lower values for the predicted $x_{Fz}$ than the test values. However, if the results of the fluctuating pressure model are monitored only in the boundary layer the result appears to be much closer to the test value. This was shown for the one case of AC9 and it is recommended that
the same type of analysis be done for AC5, 6, 7, and the Segmented Ball Valve to further validate this conclusion.

7 References

