Validation of CFD predictions using process data obtained from flow through an industrial control valve

J Green¹, R Mishra¹, M Charlton² and R Owen²

¹ University of Huddersfield, Queensgate, Huddersfield, HD1 3DH, UK
² Weir Valves & Controls UK Ltd, Britannia House, Huddersfield Road, Elland, West Yorkshire, HX5 9JR, UK

E-mail. j.r.green@hud.ac.uk

Abstract. This study uses the experimental flow test data to validate CFD simulations for a complex control valve trim. In both the simulation and the experimental flow test the capacity of the trim (Cv) is calculated in order to test the ability of CFD software to provide a design tool for these trims. While CFD tests produced results for the capacity which were consistent across a series of five different simulations, it differed from the experimental flow data by nearly 25%. This indicates that CFD simulations need to be properly calibrated before being used in designing complex valve trims.

1. Introduction

Industrial control valves often handle flows with very high pressure drops (conditions often referred to as severe service). In order to cope with these pressure drops geometrically complex valve trims, with many stages of pressure loss, are designed to prevent undesirable side effects such as cavitation and high noise. There are many different product designs for control valve trims produced by different manufacturers, one such design is the X-Stream trim. This design uses cylindrical obstructions (Figure 1 shows a single disc, multiple discs are stacked upon each other to form a full trim stack) in the flow field to control the pressure drop while minimizing cavitation/noise. Generally these valves are designed using a combination of numerical, analytical and experimental tools to meet process requirements of the valve capacity, which is given by Equation (1) [1].

\[ Cv = C \times Q \times \left( \frac{SG}{\Delta P} \right)^{\frac{1}{2}} \] (1)

In the above equation Cv is the valve capacity (in this case, for liquid flows), Q is the volumetric flow rate, SG is the specific gravity, \( \Delta P \) is the pressure drop. The constant C is a conversion factor depending upon the units of the flow rate and the pressure drop. Traditionally the capacity is represented in imperial units and as such C=1 if the flow rate is in USG/min and the pressure drop is in pounds per square inch.

The CFD software have proven to be an extremely powerful design tool and are increasingly being incorporated into the design stage for many different products [2,3,4,5]. CFD allows for the testing of many different designs without the expensive necessity of manufacturing and flow testing every design. The disadvantages associated with this methodology are that the results obtained from CFD
software need to be demonstrated to be accurate before they can be used because of inherent uncertainty in assumed flow variables. This study is aimed at testing the ability of the CFD software to simulate flow through this complex valve trim and use the flow test data to validate the results obtained. To obtain valve capacity, the flow field within the valve trim needs to be simulated using CFD software to find the pressure drop for a given flow rate for a single trim disc. This data can be used to calculate the $C_v$ for the trim stack and when combined with the $C_v$ values for the valve body and valve seat, the $C_v$ for the complete valve can be estimated.

![Figure 1. X-Stream Trim Disc.](image)

Shown in figure 1 is the trim disc, a stack of which makes up the lower 60% of the valve trim. The remaining 40% of the trim was made up of a multiple hole single cage trim. During the experimental testing, the valve was never opened to more than 60% and as such the drilled cage portion has been ignored in the CFD analysis. The CFD analysis was carried out only on one of the four flow paths on this disc to find the flow capacity of one path, which is then used to find the capacity of one disc by multiplying the single path capacity by four. This methodology is adopted since the simulation of a full disc would require significantly more memory and CPU time to perform without necessarily improving the accuracy of the prediction. The capacity for a single disc can then be used to estimate the trim capacity as a function of valve opening which can be directly compared against the data obtained from the field tests.

2. Computational Fluid Dynamics Methodology
A consistent methodology has been developed for the CFD simulation of X-Stream discs to ensure that the calculated value of capacity is as accurate as possible (to within the uncertainties of the CFD model used). This CFD methodology includes creation of the single disc flow path within the CFD software, which is then discretized using edge mesh parameters to ensure that the density of the final volume mesh produced should be optimum for accurate simulation. Boundary conditions are provided for the inlet, outlet and walls (mass flow inlet and pressure outlet) corresponding to the actual flow conditions. The modeled geometry is then exported to the flow processor for flow simulation. Figures 2(a) and (b) show the mesh used in present investigation. Figure 2(a) highlights the inlet faces whereas 2(b) shows the outlet as the downward facing (towards the viewer) face in the inner solid region (termed outflow region).
Once imported into the flow simulator the flow specific values are applied to the boundary conditions. The k-ε turbulence model is used to simulate the turbulent flow conditions within the trim. The flow equations are solved iteratively within the software until the simulation is deemed to be converged. Convergence is assumed to have been reached based on three indicators, firstly the residuals are both low and constant over a series of iterations, secondly the pressure value at the mass flow inlet ceases changing and finally the contours of pressure and velocity show a reasonable degree of symmetry across the 45° line (see figures 3(a) and (b) for an example of these contours). Once the flow field has been simulated the pressure loss across the trim is calculated, which can then be used to find the capacity of the single flow path using the relationship given in Equation (1). Further simulations are carried out to ensure the accuracy of these predictions. These involve modifications to the mesh (to establish mesh related effects), changing the boundary condition (to establish accuracy of boundary parameter used) and the geometry (to establish effects of any simplifications). The simulations used in the present investigation are as listed below:

1. Mesh: The mesh is adapted within CFD based upon gradients of the velocity profile (where there are locally high changes in velocity, additional cells were added).
2. Boundary Conditions: A second flow rate is chosen (for the mass flow inlet) and convergence is once again achieved as per above (for both the original mesh and an adapted mesh).
3. Geometry: The geometry is changed to incorporate a larger physical outlet which should more realistically represent the conditions found in the valve itself. This is then meshed and imported into CFD software and tested using the methodology above.

The three modifications suggested above should produce very little change in the calculated value of capacity if the original simulation is accurate. A large change would suggest that the original simulation was not sufficiently accurate. During the CFD simulation of flow through the trim a range of flow rates of 0.5 to 3 kg/s/flow path was chosen, which was within the range of installed use for this trim.

3. Results and Analysis
A total of five different simulations have been carried out. The first four use two different inlet mass flow rates each with original and adapted meshes, the fifth result uses a modified geometry where the outlet is extended to better represent the true features within the valve trim and this modification is shown in figure 3.
Figures 3(a) and (b) show the velocity contours (data taken from vertical mid-point of the flow path), with the flow direction as shown on figure 3(a), for the flow through two different valve trim geometries as explained earlier. The flow features through the trim region containing the cylinders is similar in both tests (however flow rate varies which explains the significant differences in local velocity magnitudes). Clearly peak velocities occur at the areas where the cross sectional area is smaller i.e. between the cylinders on each row and in the diagonal gaps between cylinders on neighboring rows. Additionally each row of cylinders has a zone directly ahead and behind it (relative to the inward radial bulk flow direction) which has minimal flow velocity.

Figure 3(a) also shows fluid jets in the outlet zone which stay quite separate from each other rather than quickly merging into a single flow. This is an important consideration from the point of view of hydrodynamic noise generation [6]. The outlet zone in figure 3(b) does not show these jets at all. This is due to the proximity of the wall immediately after the fluid leaves the trim itself which forces the jets to merge and flow to the outlet.

Figures 4(a) and (b) show the pressure contours for both the modified and original design geometries. Overall the nature of the contour plots is similar. The difference in local values can be attributed to the different boundary condition values used (inlet mass flow rate). For the modified geometry this flow rate was increased to cover a broader range of flow rate values. Again flow
features differ only in the outflow region significantly since in the original geometry there is an apparent build up of pressure against the “inner” wall. While this build up is present in the modified geometry, it is much less noticeable. As noted before while there are differences between the two pressure fields locally, it can be seen that these do not significantly impact the calculation for the flow capacity, which can be seen in Table 1. Localized “peak” pressures occur against the leading and trailing points of each cylinder (relative to the radially inwards bulk flow direction), which correspond to local minima in velocity, as expected.

The numerical values from the five simulations carried out are presented in table 1 in the form of flow rates and inlet pressures. In all cases the outlet pressure was set to 0 gauge so the numerical value of the inlet pressure is equal to the pressure drop. The relationship in Equation 1 is then used to calculate the capacity (Cv) for the single path for each of the five simulations. “Flow 1” and “Flow 2” are simulations with two different flow rates carried out using the original geometry (see figure 3 or 4) each using the original mesh and an adapted one while “Mod Geom” uses the modified geometry.

Table 1. Capacities calculated from CFD simulation results.

| Mesh | Flow 1 | Flow 1 | Flow 2 | Flow 2 | Mod Geom |
|------|--------|--------|--------|--------|----------|
|      | Original | Adapt | Original | Adapt | Original  |
| Mass Flow Rate kg/s | 0.5 | 0.5 | 1.5 | 1.5 | 3 |
| Inlet Pressure Pa | 3.3E+05 | 3.23E+05 | 2.82E+06 | 2.78E+06 | 1.04E+07 |
| Cv (1 path) | 1.147 | 1.159 | 1.177 | 1.186 | 1.227 |

The results presented in table 1 show a strong agreement between the five simulations for the value of capacity. The three changes described previously have only had a small impact upon the flow capacity value. It can be seen that changing the mesh increases the flow capacity by 1% and 0.75% for the lower and higher flow rates respectively. Furthermore, changing the flow rate increases the flow capacity by 2.54% for the original mesh and 2.28% for the adapted mesh. Also changing the geometry increases the flow capacity by 6.52% (it should be noted that this geometry was tested at a third, higher flow rate than the original test).

Based on the above analysis an average value of 1.2 is used for the flow capacity of a single flow path for further analysis. This results in a Cv of 4.8 for a complete disc. For validation purposes the data obtained from an in-use valve was used. This data was provided from pressure sensors upstream and downstream from the installed valve working on a boiler feedwater line (which contains a liquid with specific gravity of 0.653 at the operating conditions for the valve). The data used in this analysis have been shown in table 2. It should be noted that the flow rates/valve openings correspond to a flow rate of 2.3 and 1.9 kg/s/flow path which is well within the range of this parameter which was tested using CFD which ranged from 0.5 to 3 kg/s/flow path. This demonstrates clearly that the values used during the CFD test match up well to those experienced by the installed trim discs.
Table 2. Provided Results for Flow Test with Analysis.

|                      | Test 1    | Test 2    |
|----------------------|-----------|-----------|
| Inlet Pressure PSI   | 2183.8015 | 2171.303  |
| Outlet Pressure PSI  | 1465.428  | 1494.486  |
| Pressure Drop PSI    | 718.3735  | 676.8165  |
| Flow Rate USG/min    | 1609.2083 | 1646.807  |
| Valve Cv             | 48.5169738| 51.15215  |
| Body Cv              | 602       | 602       |
| Seat Cv              | 220       | 220       |
| Open Trim Cv        | 49.9122962| 52.7954   |
| Open %              | 23.3      | 28.2      |
| Trims                | 7.33858288| 8.88189   |
| Cv 1 disc           | 6.80135366| 5.944162  |
| Cv 1 Path           | 1.70033842| 1.486041  |

Once flow rate, inlet and outlet pressures were known the actual analysis could be carried out to first determine the valve Cv. Which can then be used with the known seat and body Cv to find the trim Cv (for this particular opening). This value can then be reduced to calculate the single disc and single path Cv values which can be compared directly with the results obtained from flow simulations. The average of the flow capacity for the process conditions was obtained as a Cv of 6.4 for a single disc (with a standard deviation, for the two tests, of 0.6). It is clear that the results obtained from CFD simulations and those calculated from experimental flow data differ by a significant amount (the Cv value provided by CFD is calculated as 25% less than that calculated from provided flow data). At this stage it can be concluded CFD analysis needs to be more rigorously analyzed as well as local flow field needs to be measured to understand reasons for this deviation.

4. Conclusions
The aim of this study was to validate CFD simulation results of a valve trim by using experimental flow test data. The results from these two methods were used to find the flow capacity of a single flow path on the valve trim and compare these values. Based on five CFD simulations an average value for capacity of the single path was found to be 1.2. This compares reasonably with two available sets of data from the experimental flow test which were used to find an average value of the flow capacity for a single flow path. This demonstrates that the experimental flow data provided a value for flow capacity which was 25% higher than that estimated from the flow simulations, this indicates that CFD simulations must be used with care when designing the complex valve trims.

References
[1] IEC60534-2-1 Industrial-process control valves part 2-1: flow capacity – sizing equations for fluid flow under installed conditions Edition 2.0 2011-03
[2] Marquis A 2000 CFD in fluid machinery design Proceedings of the Institution of Mechanical Engineers ISSN 0957-6509, 06/2000 Journal of power and energy 214 Issue 6 699
[3] Dvorak P 2003 CFD speeds fan design Machine Design 75 Issue 19 60
[4] Kacik M, Dvorak P 2002 How CFD streamlines fluid designs Machine Design 74 Issue 20 64
[5] Wu J, Shimmei K, Tani, K Sato J 2007 CFD-based design optimization for hydro turbines J Fluids Eng 129 Issue 2 159.
[6] BS EN 60534-8-4 Industrial-process control valves part 8-4 noise considerations – prediction of noise generated by hydrodynamic flow 2005