Taimoor, Asim, Rakesh, Mishra, Matthew, Charlton and Carlos, Oliveira

Capacity Testing and Local Flow Analysis of a Geometrically Complex Trim Installed within a Commercial Control Valve

Capacity Testing and Local Flow Analysis of a Geometrically Complex Trim Installed within a Commercial Control Valve

Taimoor Asim[1], Rakesh Mishra[1], Matthew Charlton[2], Carlos Oliveira[2]

[1] University of Huddersfield, Huddersfield, UK

{t.asim, r.mishra}@hud.ac.uk

[2] Weir Valves & Controls Ltd., Elland, UK

{matthew.charlton, carlos.oliveira}@weirgroup.com

Abstract. Industrial control valves often handle flows with very high pressure drops (conditions often referred to as severe service). In order to cope with such pressure drops, geometrically complex valve trims with many stages of pressure drops, are designed to prevent undesirable side effects such as cavitation etc. There are many different product designs for control valve trims produced by different manufacturers. One such design uses cylindrical obstructions in the flow field to control the pressure drop. The design of these trims is based on their capacity (Cv) values. With the advent of advanced computational tools, such as Computational Fluid Dynamics based codes, it has become possible to numerically test these trims over a wide range of flow conditions. Hence, this study presents capacity testing and local flow analysis of a complex geometry trim, installed within a commercial control valve for severe service. The results show that the capacity of this particular design of trim decreases as the valve opening position decreases.

1 Introduction

The control valve and trim considered in the present study have been designed specifically as a continuous resistance to minimise the negative aspects of control valves, such as excessive noise and cavitation. These effects commonly occur during severe service environments such as in boiler feed-water, and also in highly erosive conditions such as oil and gas sector. The patented trim design numerically analysed in this work uses a staggered cylindrical arrangement on discs which are stacked on top of each other in order to control the flow.

Nomenclature

Cv	Capacity of the valve $\left(\sqrt{m^7/Kg}\right)$
ΔP	Pressure drop across the valve (kPa)
SG	Specific gravity of the working fluid (-)
VOP	Valve Opening Position (%)
Cv_{Trim}	Capacity of the Trim $\left(\sqrt{m^7/Kg}\right)$

Advanced CFD tools have proven to be extremely powerful design tools and increasingly being incorporated into the design stage for many different products [2, 3, 4, 5]. CFD allows for the testing of various designs without the need to manufacture and flow test each and every single trim design. In the present study, CFD has been used to test a continuous resistance trim with a cylindrical arrangement of stacked discs. The working fluid considered here is water with specific gravity of 1.

2 Numerical Modelling

According to the industrial standard [1], the test setup for control valves should be such that the inlet and the outlet of the valve should be attached with straight pipes of lengths 2D and 6D, where D is the nominal diameter of the pipeline. Figure 1 depicts the trim being tested in the present study. The trim comprises of 11 discs and 5 rows of cylinders.

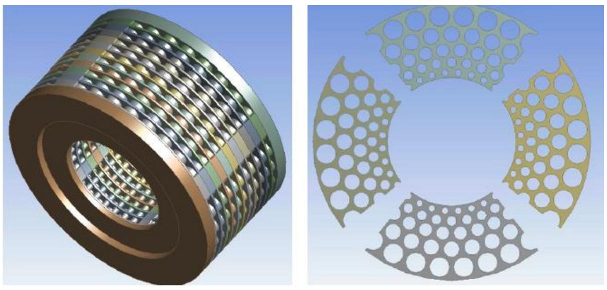

Fig. 1. Numerical model of the trim

Hybrid meshing has been used for the meshing of the flow domain. The inlet and outlet pipes have been meshed using hexahedral mesh elements. The control valve, with the trim installed, has been meshed using tetrahedral mesh elements with a maximum size of 3mm. It has been observed that by using these mesh sizing, reasonably accurate results are being predicted by CFD solver.
Three dimensional Navier-Stokes equations, together with the continuity equation, have been numerically solved in an iterative manner for the steady flow of water

through the valve and the trim. The scope of the present study is limited to the iso-thermal and incompressible flow within the aforementioned control valve, and hence pressure based solver has been used, keeping the density of water constant at 998kg/m^3. As mentioned in [6, 7], extreme pressure and velocity gradients exist in these types of trims. Hence, Shear Stress Transport – kω model has been used to model turbulence within the control valve and the trim. SST-kω model enables accurate predictions of flow properties at the near wall region i.e. within the boundary layer where extreme flow gradients are expected. Furthermore, it behaves as a standard k-ε model in the regions away from the walls.

The boundary conditions specified to the numerical model have been defined as follows:

- Inlet – Pressure Inlet (variable kPa)
- Outlet – Pressure Outlet (0 kPa gauge)
- Trim & Valve – Wall (stationary walls)

3 Results and analysis

The aforementioned control valve has been tested numerically to evaluate the capacity of the trim. The results of this study have been presented in the following sub-sections.

3.1 Validation of the Numerical Model

In order to justify that the aforementioned mesh size settings are enough to predict the capacity of the valve/trim with reasonable accuracy, CFD results for the same have been compared against the experimental/flow-loop test results, so that the CFD results can be validated against the experimental results. Because the flow-loop tests were carried out at much lower pressure differentials, the boundary conditions were specified accordingly. The results are shown in Table 1 clearly show that there is a good agreement between experimentally found capacity of the trim (Cv_{Trim}) and CFD based Cv_{Trim}.

Table 1. Validation of CFD results

Valve Opening Position (%)	ΔP (kPa)	Difference between Experimental and CFD based Cv_{Trim} (%)
100	341	5.06
60	370	0.88
10	375	0.33

The different numerical simulations presented in Table 1 were repeated using standard k-ε turbulence model and it was found out that k-ε is, on average, roughly 7% more inaccurate in the prediction of the capacity of the valve/trim as compared to SST-kω turbulence model. It is noteworthy that convergence was judged based on the variations in the mass flow rate within the control valve. The solution has been considered converged once the variations drop below 1%.

3.2 Capacity Testing of the Trim

Figure 2 depicts the variations in the static pressure (in kPa) and the flow velocity (in m/sec) within a single disc of the trim for a differential pressure of 1,000kPa across the valve, when the valve is 10% open. The flow direction is from the outermost row towards the innermost row of the trim. It can be seen that the inlet section has higher pressure as compared to outlet section of the trim. It can be further seen that pressure drops within the trim from one row to another. As water passes between two cylinders, due to area reduction, pressure drops. As water exits a row and enters another one, due to increase in area available for the flow, pressure increases again. Hence, the pressure drops in successive steps within a trim rather than a single continuous drop within the valve alone.

It can be seen in Figure 2(b) that the flow velocity increases between the cylinders of a row, where the area available for the flow to take place is less; hence forming jets. As water exits a row and enters another one, due to area increase, velocity reduces to satisfy continuity.

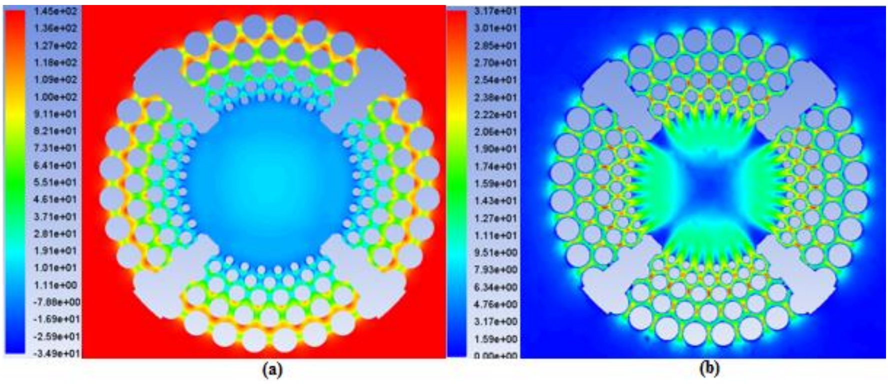

Fig. 2. Variations of flow variables within the trim at 1,000kPa differential pressure and 10% valve opening position (a) Static Pressure (b) Flow Velocity

In order to quantify the capacity of the trim, Figure 3 depicts the capacity test results of the trim for various differential pressures across the control valve.

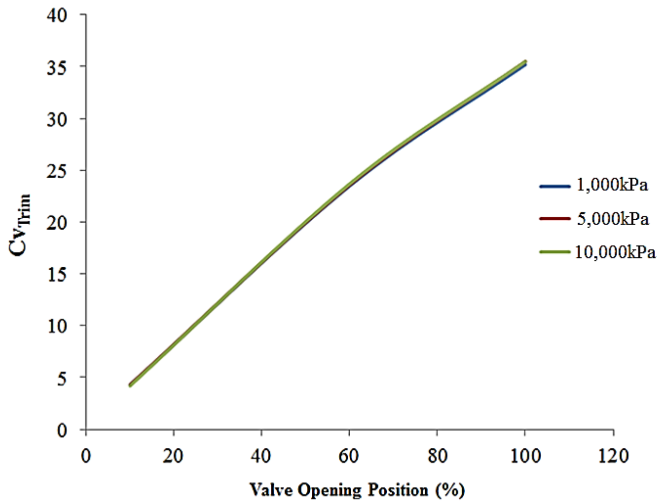

Fig. 3. Variations in Cv_{Trim} of the trim at various valve opening positions and differential pressures

The results presented in Figure 3 points towards two important features. These are:

1. As the valve opening position (VOP) decreases, Cv_{Trim} decreases and vice versa. This is because there is less area available for the flow to take place, and hence the capacity of the valve decreases.

2. Cv_{Trim} remains almost constant for various flow conditions (differential pressures) for the same valve opening position

This suggests that Cv_{Trim} is a function of the valve opening position only, and not the differential pressure across the valve.

In order to express Cv_{Trim} as a function of valve opening position, multiple variable regression analysis has been carried out on the results presented in Figure 3. The following predictor expression has been found out:

$$Cv_{Trim} = \frac{VOP}{\sqrt{7}} \tag{1}$$

In order to check the validity of Eq. (1), Cv_{Trim} values have been calculated using Eq. (1) and compared against the results presented in Figure 3. It has been observed that Eq. (1) gives an average percentage error of 7.4% compared to the results presented in Figure 3, and hence this equation can be used to predict the capacity of the trim with 93.6% accuracy.

4 Conclusions

Detailed Computational Fluid Dynamics based investigations have been carried out on a severe service control valve with a geometrically complex trim installed. Capacity results have been evaluated and compared with the experimental data. It has been shown that the pressure decreases as water exits a row due to area reduction. At the same instance, the flow velocity increases. The capacity results suggest that that Cv_{Trim} values are dependent on the valve opening position only, and independent of the differential pressure across the valve. Furthermore, Cv_{Trim} decreases as the valve opening position decreases due to reduced area for the flow to take place.

A novel predictor expression for the capacity of the trim has been developed that can be used in the design process of such 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. Green J, Mishra R, Charlton M, Owen R 2012 Validation of CFD predictions using process data obtained from flow through an industrial control valve J Physics: Conference Series 164.
7. Green J, Mishra R, Charlton M, Owen R 2012 Local Analysis of Flow Conditions within a Geometrically Complex Control Valve Trim using CFD J Physics: Conference Series 164.