15th IFK
Proc. of the 15th International Fluid Power Conference

Modelling and Analysis of the Performance for a Pressure Compensated Flow Control Valve

K. Bhaskar[1,2], N. Johnston[1], A. Plummer[1], S. Westwood[2]

1. University of Bath, 2. Hydraulic System Products Ltd

Abstract.

Pressure compensated flow control valves are an essential requirement in the areas where flow control is necessary to achieve constant speeds regardless of the change in pressure difference or the load on the system. However, maintaining a constant flow rate remains a challenge with increasing pressure drop which directly impacts the performance of the system. The aim of this study is to integrate numerical modelling into the design process for a hydraulic valve in order to predict and optimise its performance. The example of a pressure compensated flow control valve is presented in this paper. The effects of flow forces and flow coefficients on the pressure compensation and flow control are examined. Computational Fluid Dynamics (CFD) models were created and used to obtain the flow forces and flow coefficients for different designs and spool displacements under constant flow rate conditions, and a system model was created in order to examine the performance, incorporating coefficients predicted from the CFD study. Lastly, experimental studies were carried out with the proposed design changes and their impact on the flow control across different pressures were studied.

Keywords. Flow forces, CFD, mathematical model, Orifice, Pressure compensated flow control valve

1. Introduction

Pressure-compensated flow control valves play an important role in the speed control of hydraulic systems. The output flow rate remains nearly constant, regardless of variations in the inlet pressure. The constant rate of flow is ensured by the adjustment of orifice area depending on the fluctuations in the system pressure. However, it is quite challenging to design a simple, stable valve which maintains the flow within a specified tolerance. As the pressure increases, the compensation by reduction of the orifice area introduces flow forces that act on the displacing element such as a spool or a poppet and contribute to deviations in the flow rate. These flow forces arise due to low pressure regions around the exit of the flow and directly influence the performance characteristics of the valve. Accurate prediction of these flow forces requires combining mathematical models, CFD simulations and validation data from experimental tests. Earlier analytical studies on a system level laid the fundamental approach for the modelling of compensated valves. Weber, Johnston and Edge

[1][2] developed steady state CFD models that were used to extract the effective jet angles and discharge coefficients for use in valve dynamic models. The jet angle was identified as the crucial parameter that led to significant changes in the flow force acting on the spool. Faulkner, Johnston and Weber [3] applied this methodology to pressure cartridge valves showing that the geometric parameters such as chamfers and introduction of fillets in place of sharp corners altered the discharge coefficients. The research also emphasised the sensitivity of valve performance to manufacturing tolerances.

Cheng [4] proposed analytical models linking the damping orifice flow characteristics to the transient response time. Parameters such as the spring constant and the pre-compressed length of the spring directly affect the steady flow rate, demonstrating how the behaviour of the flow forces influences compensation accuracy. Borghi et al. [5] conducted CFD-based numerical analysis and experimental studies to determine axial flow forces on compensated spool valves. The dependency of the flow forces on the geometry was modelled and verified through measurements from experimental studies. The Coanda effect was noted to be a significant influence on the actual flow at small openings of the valve. CFD studies and experiments on spool valves by Heraković et al. [6] concluded that the implementation of simple changes to a combination of geometrical parameters of the spool reduced the axial flow forces in the valve and helped eliminate the non-linear digressive flow-force characteristics. The CFD results agreed well with the experimental results. Further studies using CFD and mathematical models implemented in Matlab/Simulink were carried out for a proportional flow control valve by Lisowski et al. [7]. The study explored a triangular shape for the spool opening which reduced the dynamic flow forces acting on the spool. The jet angle was also studied and found to vary with the throttle gap width of the opening area; it was concluded that the functional relationship between the two may be obtained with an approximation.

Mao et al. [8] analysed a bidirectional throttling slide valve using CFD and experiments. It was found that the asymmetric vortices formed caused unequal steady flow-force curves and a guiding shoulder with an annular cavity was proposed that effectively reduced the asymmetry in the flow forces. Altare, Rundo and Olivetti [9] coupled the 3D CFD studies with a 0D dynamic model to study the effects of flow force compensation on a conical poppet pressure relief valve. The research experimentally explored the use of a deflector geometry for reducing the flow force. The data extracted were supplied as an input to a lumped parameter model of the valve. The study highlights the important role of the relation between the geometry, the flow forces and the deflection of the flow jet. Amirante et al. [10] described a CFD analysis of flow forces in a hydraulic proportional valve with circumferential notches. The CFD predictions were validated with experimental flow rate data. A reduction in the flow forces was observed at large valve openings, but smaller benefits were obtained at small openings. There was a decrease in the discharge coefficient with the valve opening due to changes in the flow direction. The study further shows that alterations to the spool profile could help in further reductions of the flow forces.

Across the studies, flow forces are shown to have a direct effect on the flow performance of the valve, and the jet angle as well as the geometry emerge as the principal contributors. The present study adopts an approach for the estimation of the flow forces and the flow coefficients and combining them with a mathematical model using Matlab/Simulink to optimise the performance of the pressure compensated flow control valve.

2. DESCRIPTION OF THE VALVE MODEL

The pressure compensated flow control valve examined in this study was developed jointly by Hydraulic System Products Ltd and University of Bath. The valve is of cartridge type with a flow control range between 20 lpm to 30 lpm and a working pressure up to 250 bar. The valve consists of a main body with large radial holes as the outlet, a sliding spool with smaller radial holes which would act as the variable metering orifice, a needle and a sliding 'hat' that would form the fixed orifice. There is an adjustable spring to set the flow limit manually. Figure 1 shows the cross-section of the valve.

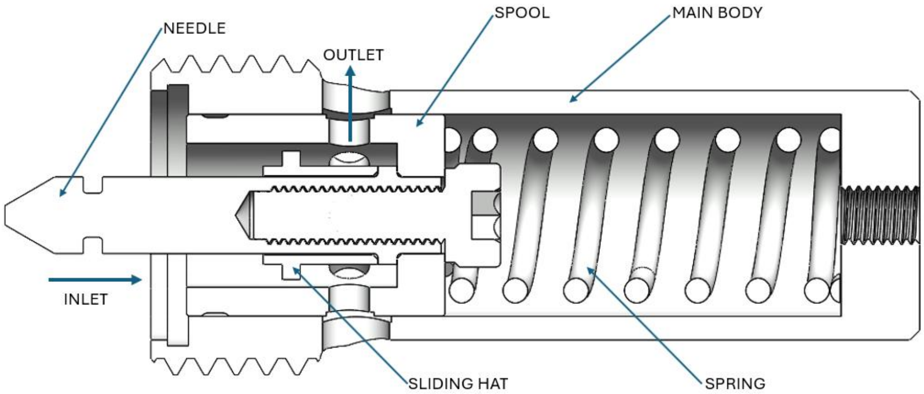


Figure 1: Cross-section of the pressure compensated flow control valve.

3. ANALYSIS OF FLOW USING CFD MODELLING

3.1. CFD model for the geometry and flow paths

A parametric CFD model was developed by tracing the flow path from the CAD model. The model consists of an inlet, radial outlets for the spool and the radial outlets for the main body. The interface between the radial outlets of the spool and the body are connected by a 0.2 mm deep groove. The spool displacements are set from 0 mm from the fully open, normal position as shown in Figure 1, to 3.25 mm which leaves a smaller orifice opening of 0.15 mm, with the displacement of the spool for closing the orifice completely being 3.4 mm. Separate CFD simulations are conducted for each opening of the valve based on the spool displacement under steady flow conditions to test the entire range of the valve opening. A quarter model was used for the CFD studies as the model is axisymmetric, and this saves computational resources and time. Figure 2 shows the quarter model used in the CFD studies.

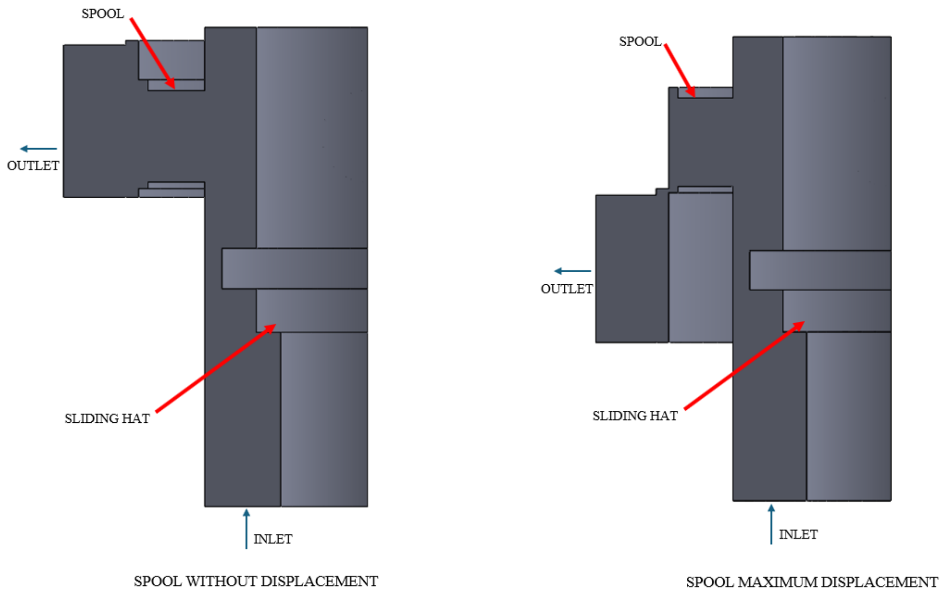


Figure 2: CFD quarter model

3.2. Mesh and mesh settings.

The meshing of the model and the CFD studies were carried out using the ANSYS FLUENT software which also enabled the application of local mesh refinements. The smallest element size was limited to 0.005mm with the largest size being 0.08mm. 5 boundary layers were added to the wall regions with an initial height of 0.005mm for better resolution near the boundary layers. Each model consisted of about 1.4 million poly-hexcore cells with a maximum skewness of 0.56 and a minimum orthogonal quality of 0.5. Figure 3 shows the mesh for the quarter model of the flow path.

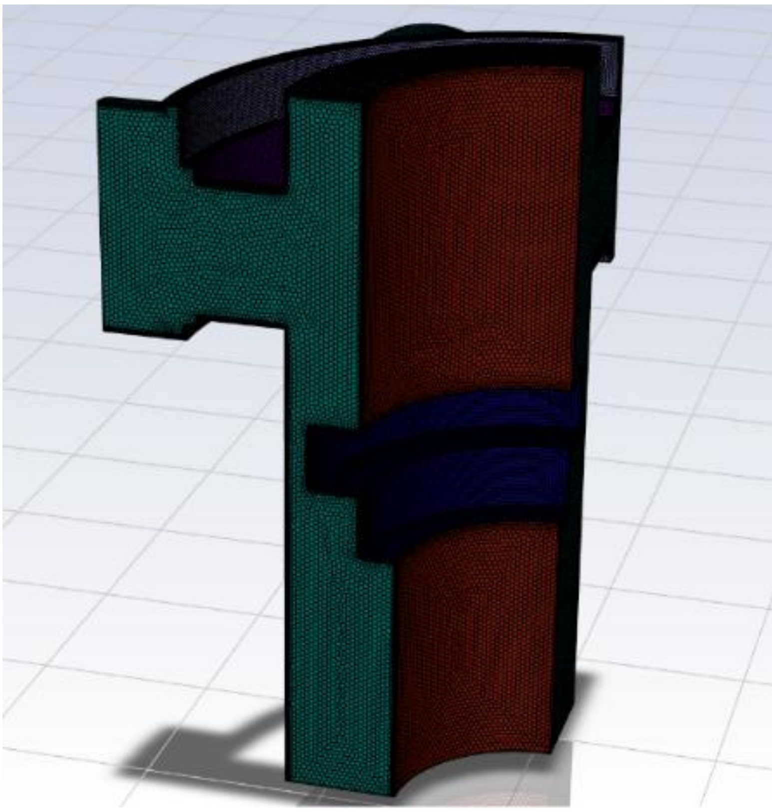


Figure 3: Mesh geometry for the CFD quarter model

3.3. CFD model parameters

A pressure-based model was used for the studies. There are many different turbulence models which can be used for the flow through the valve. The 'Realisable k-ε' model is equipped with enhanced wall treatments and considers the pressure gradient effects. Several CFD studies including [1], [7], and [10] support the use of this model for internal flow through valves, and so this model was chosen.

This turbulence model has been validated thoroughly for a wide range of flows and has been found to perform better than the standard k-ε model [11].

The fluid viscosity is set to 32 centistokes with a density of 860 kg/m^3. The inlet velocity is set at 11.7 m/s while the pressure at the outlet is set to 0 Pa gauge pressure. The convergence criteria were set to 10^{-4} for the momentum and the forces acting on the spool surface. It was found to be difficult to obtain valid, converged CFD results for openings less than 0.15 mm.

3.4. CFD results

The CFD results provided pressure and velocity distributions throughout the flow path. The pressures acting on the spool, the forces acting on the spool, the pressure drops across the orifices and the flow rate at the outlet were extracted from the CFD simulations. The pressure drops across each orifice were determined from the CFD pressure distribution plots.

The total force acting on the spool extracted from the CFD is the resultant force which includes the flow force. The flow force (F_f) is calculated by subtracting the pressure force from the resultant force (F_T) on the spool as shown in equation 1. Figure 5 shows a plot of flow force versus spool displacement, where the displacement of the spool increases as the pressure increases.

$$F_f = F_T - PA_s \qquad (1)$$

Where F_f is the total force acting on the spool, F_T is the total resultant force acting on the spool, P is the pressure acting on the spool at the inlet and A_s is the diametrical area of the spool. Whilst this flow force is related to the axial momentum flux entering as well as leaving a control volume within the spool, it is convenient to relate it to the downstream radial orifice. The effective jet angle is calculated using the flow force relation given by the equation:

$$F_f = \rho Q V \cos\theta \qquad (2)$$

where ρ is the density of the fluid, Q is the flow rate at the exit of the valve, θ is the jet angle and V is the velocity of the flow through the orifice which is calculated by the equation:

$$V = \sqrt{\frac{2\Delta P}{\rho}} \qquad (3)$$

Here, ΔP is the pressure drop across the orifice. The flow coefficient C_q is calculated using the orifice equation (Equation 4) with an approximation of the orifice area (Equation 5) which resembles the lateral surface area of a cylinder formed by the internal grooves on the spool and the main body as shown in figure 4.

$$Q = C_q A_{orifice} \sqrt{\frac{2\Delta P}{\rho}} \qquad (4)$$

$$A_{orifice} = \pi D_o x \qquad (5)$$

Where, $A_{orifice}$ is the orifice area, D_o is the outer diameter of the spool and x is the opening displacement of the orifice. This equation for the area is an approximation to the true orifice area to establish a functional relation between the flow and the orifice area. Any error in the approximation is compensated for in the flow coefficient C_q.

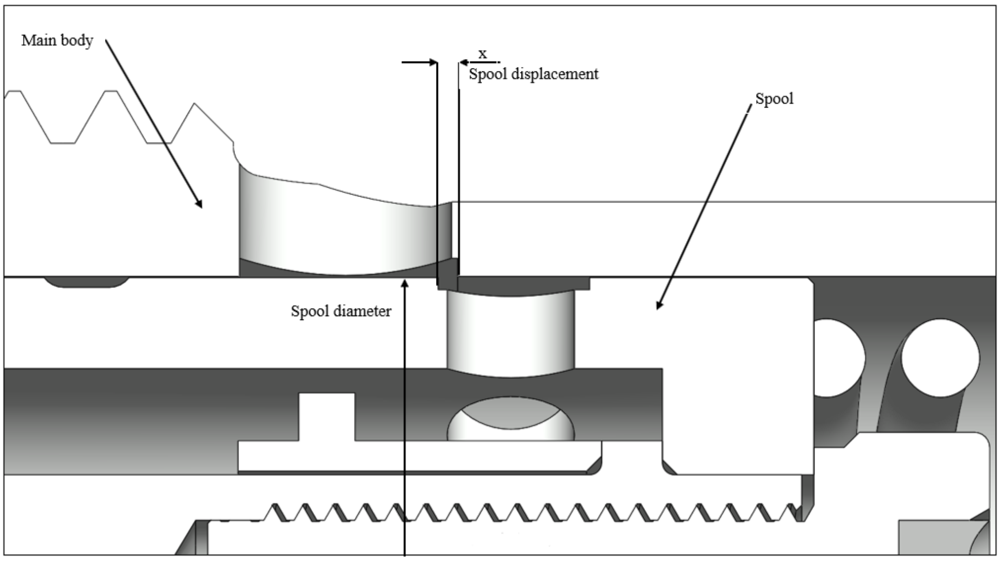


Figure 4: Flow path through the valve for the CFD model.

Figure 5 shows the plot of the flow force against spool stroke for different valve openings. Positive flow force indicates the force is acting from left to right in figure 4 helping to close the orifice, while negative flow force indicates the force is acting from right to left helping to open the orifice. It was observed that the flow force steadily increases with a decrease in the outlet orifice area, but for very small openings, the flow force dips to a negative value. This effect is largely due to the change in the jet angle (Figure 6) of the exit flow [1],[2],[7], except that for large openings the inlet momentum is significant.

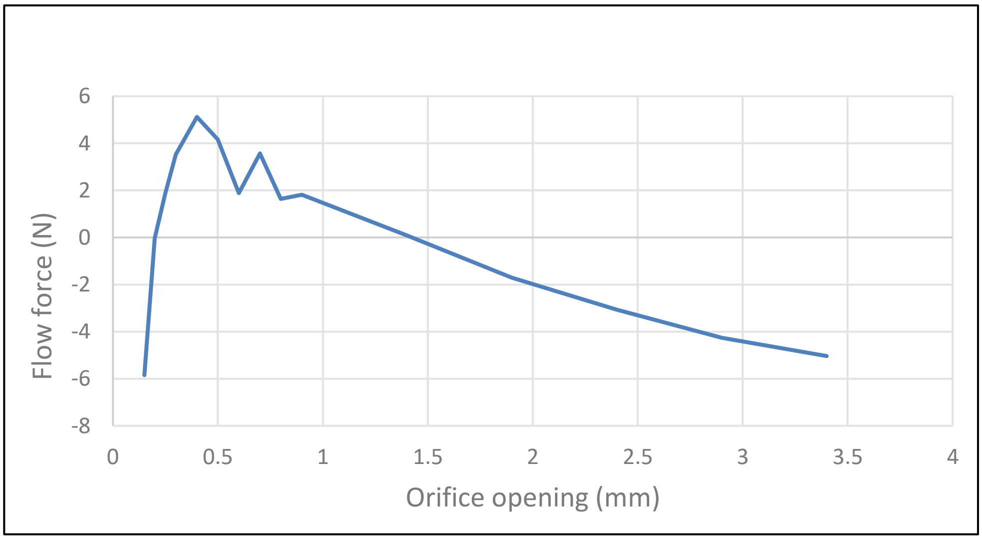


Figure 5: Spool v/s Orifice opening

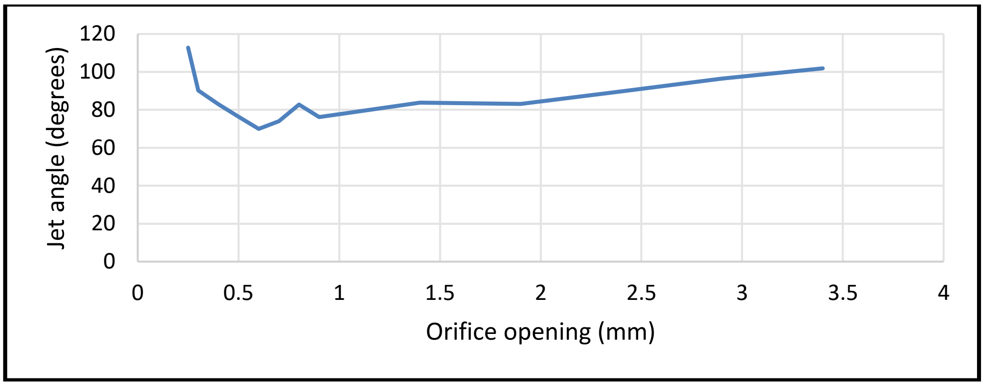
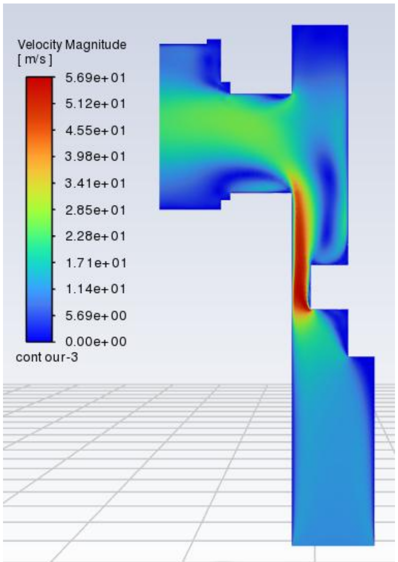
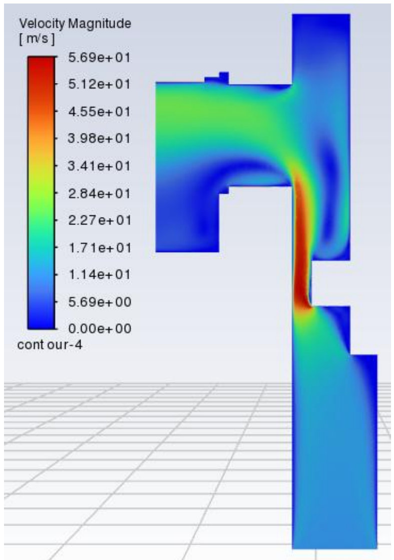


Figure 6: Jet angle v/s Orifice opening

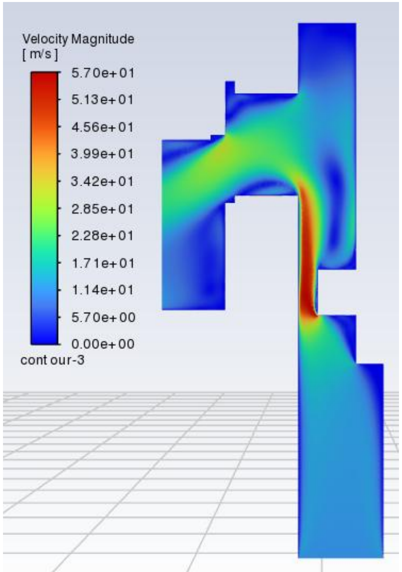
The figures from 7(a) through 7(f) show the velocity distribution of the flow through both the orifices. The angle of the jet is considered as the angle made from the inlet flow line to the line the exit jet follows at the exit of the second orifice.



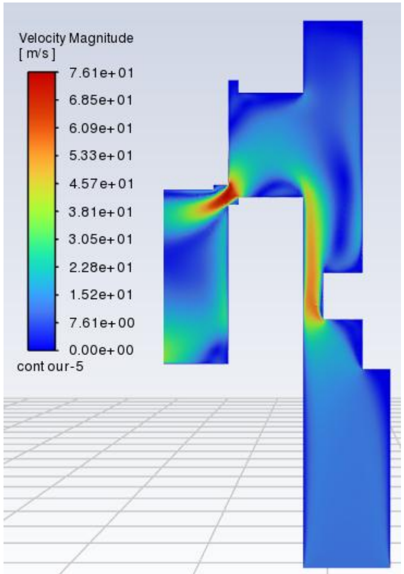
(a): Orifice fully open



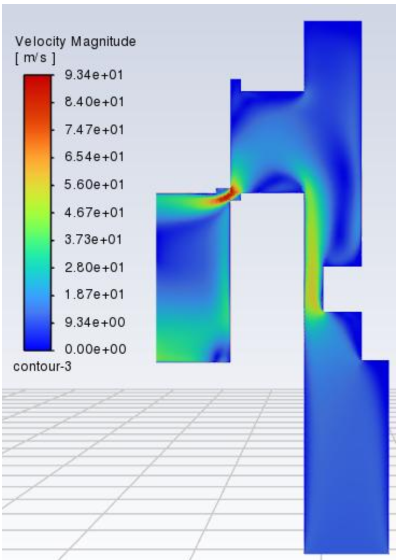
(b):Orifice 2.4mm open



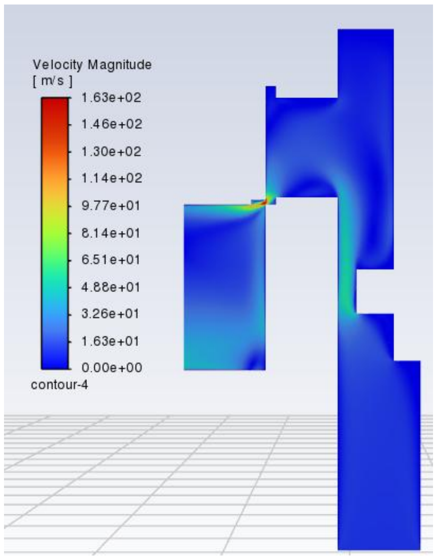
(c): Orifice 1.4mm open



(d): Orifice 0.4mm open



(e): Orifice 0.15mm open



(f): Orifice 0.1mm open

Figure 7: CFD flow patterns through a radial plane, for different openings.

It is observed from the velocity distribution figures that the jet angle starts off greater than 90° and reduces as the area of opening for the variable orifice reduces. The reduction in the jet angle increases the value of cosθ in equation (2). This would indicate smaller quantity of flow forces for large opening of the valve. However, for openings less than 0.4 mm, the jet angle can be seen to increase towards the radial direction, and this correlates with the increase in angle in figure 6. It should be noted that the orifice geometry and jet patterns are complex and three-dimensional because of the radial holes in the spool and housing (figure 3), so the 2D plots in figures 7(a) to 7(f) do not tell the full story.

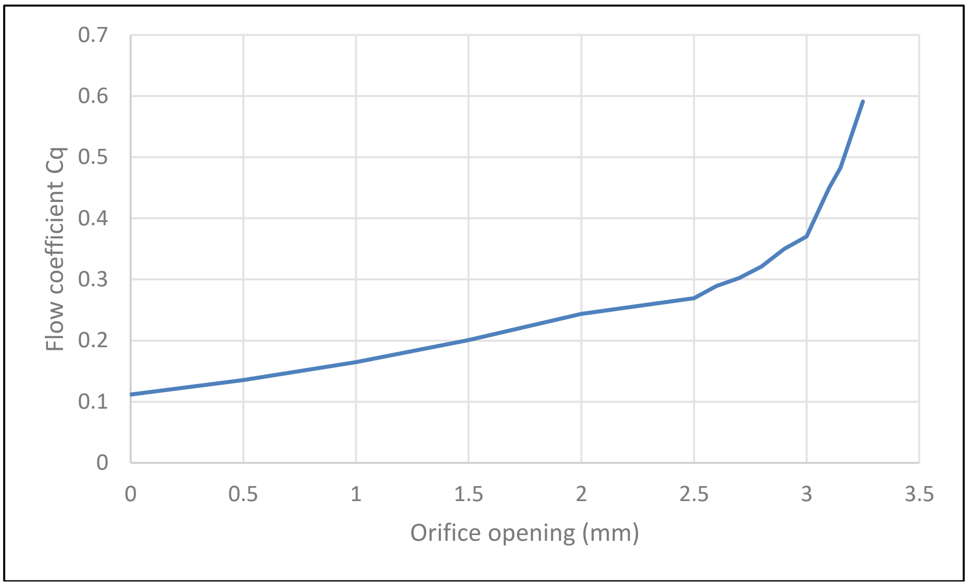


Figure 8: Flow coefficient vs orifice opening

Figure 8 shows the flow coefficients calculated from equation (4) plotted against the opening of the orifice which is a result of the displacement of the spool. The flow coefficient is calculated for flows across the variable orifice. The value of the flow coefficient C_q is observed to increase steadily as the area of opening for the decreases. This change observed is mainly due to the approximation in the calculation of the orifice area. The values of the jet angle and the flow coefficients were input as a table into the mathematical model.

4. Mathematical Model

A mathematical model was developed and implemented in Matlab® Simulink. The model consists of a pressure ramp at the inlet as the input, pressure drop across the input (sliding hat) orifice as well as the variable orifice, a spring-mass system to derive the displacement of the spool, a leakage model and the flow at the outlet of the second orifice. Figure 9 shows the block diagram for the mathematical model.

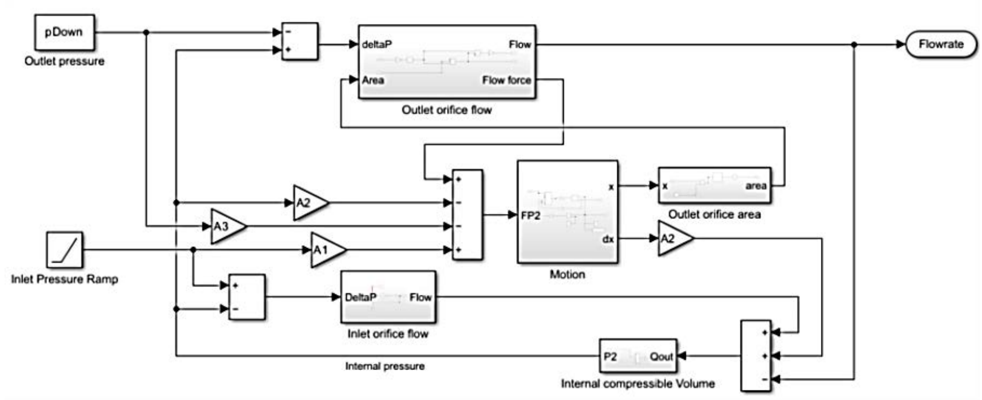


Figure 9: Matlab® Simulink block diagram of valve model

The pressure ramp values are converted to force acting on the sliding hat and the spool which pushes the spring. The displacement of the spool is derived with the equation

$$F = m\ddot{x} + c\dot{x} + kx \qquad (6)$$

Where, *m* is the mass of the spool and the sliding hat, *c* is the damping coefficient, *k* is the spring stiffness, *x* is the displacement of the spool and $\ddot{x}$ and $\ddot{x}$ represent the acceleration and the velocity of the spool. The inputs for the flow coefficient and the jet angle were taken from the CFD results and compiled into a 1-D table for different points of displacement and approximate area value for the variable orifice. The flow force is calculated using the equation (1) and is subtracted from the pressure force acting from equation (6). The flow across the orifice is calculated using the equation (4). A curve for the flow vs the pressure drop is obtained from the Simulink model shown in figure 10, for upward and downward pressure ramps.

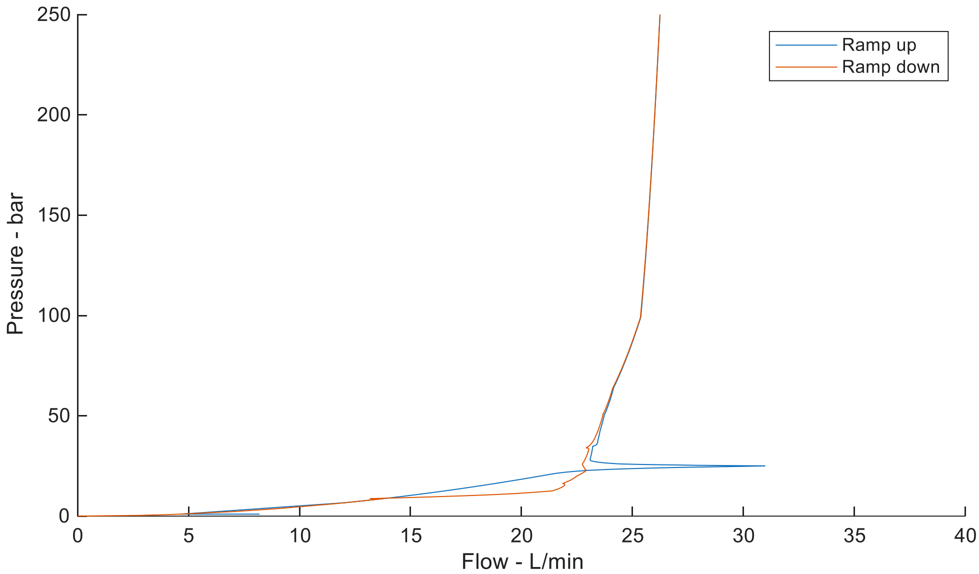


Figure 10 – predicted pressure-flow characteristics

From the figure, it is observed that for pressures above about 30 bar the valve meters the flow close to the setpoint of 25 L/min as desired, with a slight increase in flow with pressure. It was found that this variation was quite sensitive to the flow force. For pressures below about 30 bar there is insufficient pressure force to compress the spring, so the valve is fully open, and the flow is not controlled. There is some hysteresis apparent between about 10 bar and 30 bar, and for increasing pressure ("ramp up") a large spike is observed in the flow. This is because the predicted flow coefficient (figure 7) decreases with opening for large openings, such that as the spool starts to move in the direction to close the orifice, the flow resistance decreases resulting in a spike in flow for increasing pressure. This spike is not observed for decreasing pressure.

5. EXPERIMENTAL RESULTS

The experimental rig was setup at the premises of Hydraulic System Products Ltd. The rig was driven by an axial piston pump capable of achieving a maximum flow of 300 lpm and a maximum pressure of 350 bar. The measurements for the flow and the pressure were recorded using a Webtec HPM4000 data logger with a sampling rate of 60 Hz. The schematic of the test rig and assembly of the valve is shown in figure (11).

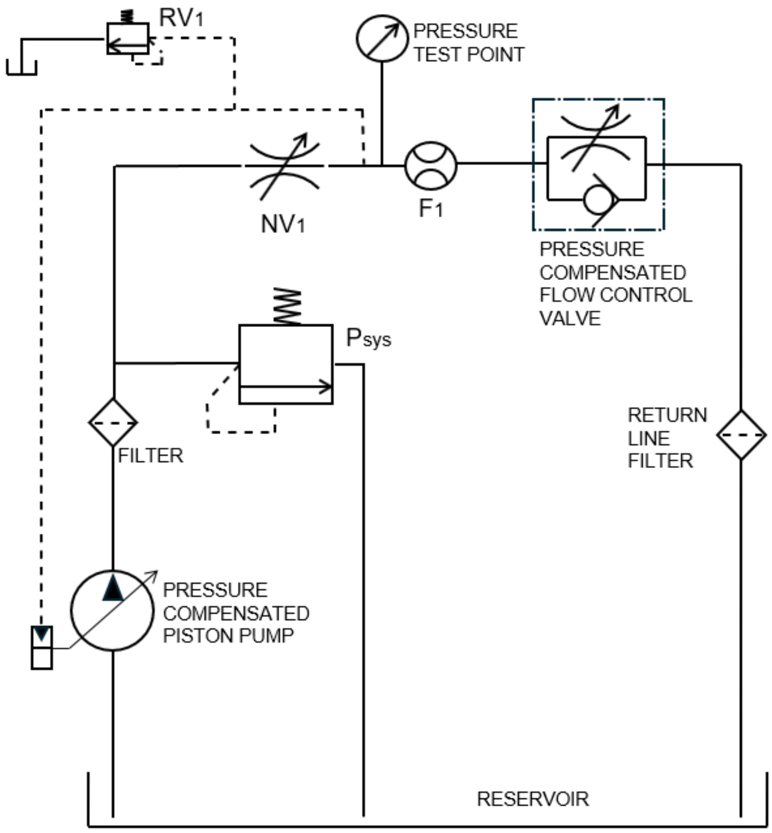


Figure 11: Schematic diagram of the hydraulic test rig

The oil is pumped from the reservoir by the piston pump through the system relief valve P_{sys} and a flowmeter F_1 into the valve. The flow is controlled by the needle valve NV_1 and the pressure through a restrictor valve RV_1. The outlet of the valve is connected back through the tank. The image of the valve on the test rig is shown in figure (12).

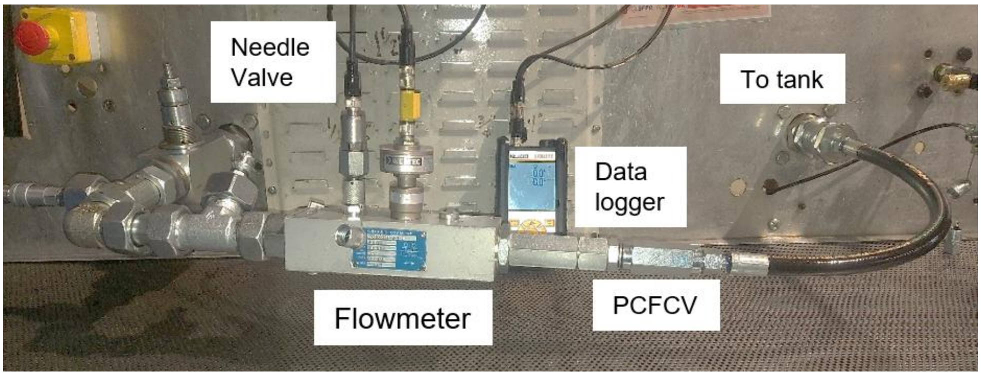


Figure 12: Hydraulic test rig at Hydraulic System Products LTD

The pressure was increased in small increments from 0 to 250 bar while recording the values of the pressure and flow. The recorded data form the data logger was used to plot the graph of flow versus the pressure as seen in figure (13).

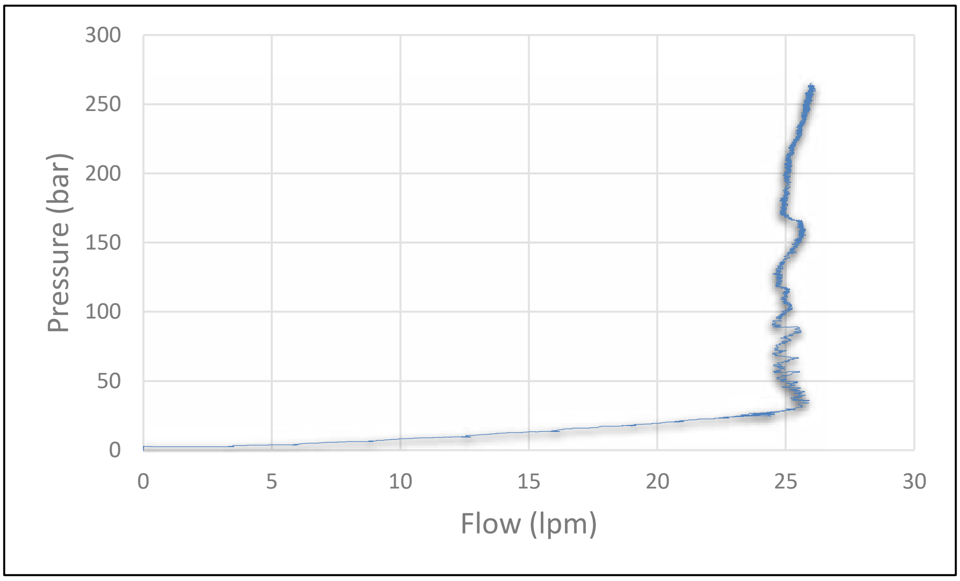


Figure 33: Experimental results - flow v/s pressure.

From the experimental results, it is observed that the behaviour of the valve follows closely with the results of the mathematical model. The flow is found to increase as the pressure increases and a slight overshoot at the set flow. The slight increase in the flow until the maximum pressure of 250 bar is observed. The spike in flow and hysteresis in the 10-30 bar region were not observed in the experimental results. The deviation from the set constant flow suggests the flow forces impacting the performance of the valve at higher inlet pressures and lower area of the exit orifice.

6. CONCLUSION

A method to integrate Computational Fluid dynamics (CFD) and mathematical modelling approach for analysing and optimising the performance for a pressure compensated flow control valve is presented. The study demonstrates that combining fluid parameters form the CFD simulations with a mathematical model provides an accurate and reliable method of predicting the valve behaviour. The CFD analysis quantified the forces acting on the spool which allowed for the calculations of the flow force and the jet angle. The results show that the flow force increases as the orifice area reduces but may reverse the direction at very small openings due to an increase in the jet angle. The calculated jet angles and the flow coefficients were integrated into the Simulink based mathematical model which produced the compensation behaviour of the valve, predicting the gradual increase in flow with an increase in the pressure. Experimental testing confirmed the accuracy of the model with the measured characteristics of the flow v/s pressure curve closely matching the results of the mathematical model. The findings show that the jet angles and the flow forces were the core contributors to the deviations from the ideal constant flows at higher pressures, and the predicted valve performance was very sensitive to the flow force.

The mathematical modelling was a key part of the design process. It highlighted the importance of flow force on the valve performance and helped to produce a design in which the flow force was reduced resulting in a good pressure-flow characteristic. Overall, this methodology offers a design framework for prediction of flow performance with the jet angle, flow force and the flow coefficient that would aid in optimising the valve geometry for future designs of the pressure compensated flow control valve. Future work will expand on the current methodology to parametrise the design and optimise the orifice geometry.

7. REFERENCES

[1] Weber, S.T., Johnston, D.N., and Edge, K.A., Mathematical modelling of a load control valve, *3 Iternationales Fluidtechnisches Kolloquium 2002*, Aachen, March 2002

[2] Weber S.T., Johnston D.N., Edge K.A. Recent Research into Cartridge-Style Hydraulic Valves. Bath Workshop on Power Transmission and Motion Control, PTMC 2000, Bath, Sept 2000

[3] Faulkner, J., Johnston, D.N., Weber, S.T., Computer Modeling of a Pressure Relief Cartridge Valve, Paper presented at *International Fluid Power Exposition*, Las Vegas, March 2011

[4] Borghi M., Milani M., Paoluzzi R. (2000). "Stationary axial flow-force analysis on compensated spool valves." *International Journal of Fluid Power*, 1(1), 17–25. DOI: 10.1080/14399776.2000.10781079.

[5] Cheng M. (2005). "Modeling and Analysis of a Pressure Compensated Flow Control Valve." *ASME FEDSM 2005 Proceedings.* DOI: 10.1115/FEDSM2005-77057.

[6] Heraković N., Dular M., et al. (2019). “Experimental and CFD analysis of flow in hydraulic valves.” *Energy Conversion and Management*, 196, 16–27. DOI: 10.1016/j.enconman.2019.05.010.

[7] Lisowski E., Filo G. (2016). “CFD analysis of the characteristics of a proportional flow control valve with an innovative opening shape.” *Energy Conversion and Management*, 123, 15–28. DOI: 10.1016/j.enconman.2016.06.025.

[8] Mao Q., Jia X., Liu Z., Li G., Cao Y., Yang Q. (2024). “Study on Steady Flow Force of a Bidirectional Throttling Slide Valve and Its Compensation Optimization.” *Applied Sciences*, 14, 11037. DOI: 10.3390/app142311037.

[9] Altare G., Rundo M., Olivetti M. (2016). “3D Dynamic Simulation of a Flow Force Compensated Pressure Relief Valve.” *ASME IMECE 2016 Proceedings.* DOI: 10.1115/IMECE2016-65624.

[10] Amirante R., Moscatelli P.G., Catalano L.A. (2007). “Evaluation of the flow forces on a direct (single-stage) proportional valve by means of CFD analysis.” *Energy Conversion and Management*, 48, 942–953. DOI: 10.1016/j.enconman.2006.08.024.

[11] ANSYS/Fluent 2025 R2: Users guide