Viscous Heating in a Hydraulic Control Valve

Gavin Siow

Jeremy Leggoe
Department of Chemical Engineering
University of Western Australia

Jon Wright & David Cree
CEED Client: AME Pty Ltd

Abstract

The behaviour of hydraulic systems is strongly influenced by the properties of the fluid being used. Complex valves can transfer significant heat to this fluid during operation, which may affect fluid properties and thus how the valve then performs. In this study, the heating in a valve is investigated through CFD modeling, and the effect that a reduced viscosity which might result from temperature increase will have on the flow behaviour. The study finds approximately 17.74 kJ of heat being generated each period during flow cycle through the valve. At this rate, 100 kg of the hydraulic fluid used in the study can increase in temperature by almost 1 degree every 2 minutes if not dissipated to the surroundings. The implication of this result is lessened however by the finding that even a 50% reduction in viscosity of hydraulic fluid only alters the mass flow through the valve by 2%. Thus the heat generated in this valve is not likely to significantly affect the valve's performance over time even if the fluid increases in temperature and drops in viscosity. A study of the numerical error associated with discretization from the meshes used in this study yielded error bands between 3% and 10%.

1. Introduction

Hydraulic control valves are widely used to control the behaviour of hydraulic systems. Due to the complicated flows generated in hydraulic valves, the hydraulic fluid can be subjected to significant viscous heating – whereby friction energy losses within the fluid are converted into heat (Morini, 2013). If not dissipated, this heat generation can in turn affect the viscosity of the fluid, and potentially influence the behaviour of the hydraulic system. The consequences of such an event vary depending on the application of the hydraulic valve, but could include risks to safety and operations. Better understanding of the heating behaviour in such valves and their response to changes in fluid properties due to heat accumulation can thus provide safety and economic benefits.

In this study, the flow field and heat generation in a particular hydraulic control valve are modeled. The valve's sensitivity to reduced viscosity is then investigated by focusing on the volume flow rate permitted through the valve in a pressure driven flow. The effect of the reduced viscosity on continued viscous heating is also investigated. These results are achieved using the computational fluid dynamics (CFD) software, ANSYS Fluent. The limitations of this approach are considered in the form of discretization error, and a measure of discretization error is explored with regards to the confidence in measured quantities from this study.

2. Modeling

2.1 Geometry and Mesh

Geometry for the valve model was adapted in SolidWorks from an existing model. This process involved removing minor features which were not expected to strongly affect flow and which would have been computationally demanding to mesh and model. The geometry of the volume occupied by fluid was then imported into ANSYS and a mesh for this volume was created using ANSYS Mesher. The meshing included body of influence controls to reduce element sizes to 1 mm at the congestion created around the eight-way opening shown on the left of Figure 1 below. The same method was also applied downstream (to the right) to create intermediate sized elements (2 mm) at a restriction created by an orifice plate. Finally, the remaining body was limited to element sizes no greater than 3 mm. Inflation layers were implemented on all valve walls to better capture near-wall interactions. 10 layers were created at each wall with a growth rate of 1.5 between successive layers. The resulting mesh was composed of 729,687 elements and 275,789 nodes, having a maximum skewness of 0.9281 and an average skewness of 0.25485. The mesh is shown below in Figure 1.

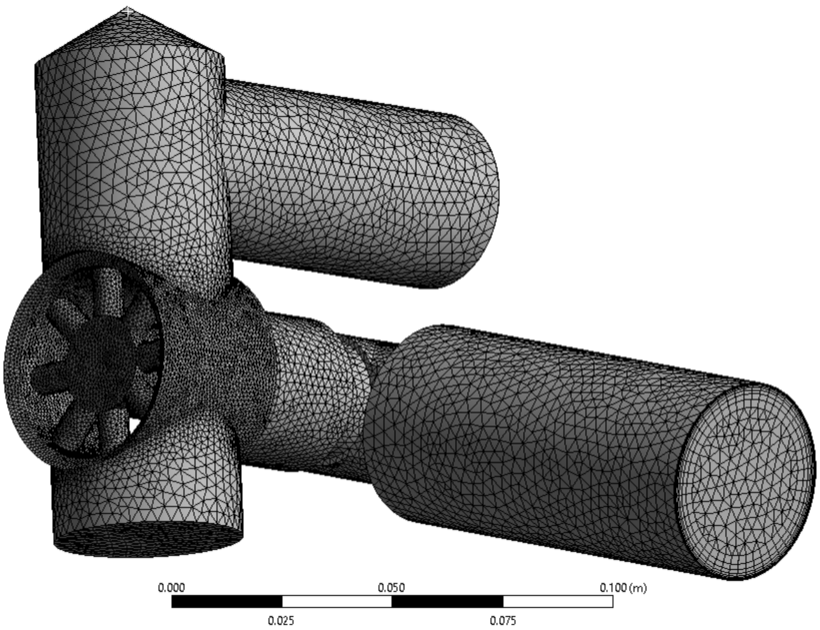

Figure 1 Mesh of hydraulic control valve, showing finer mesh areas and inflation layers.

2.2 Fluent Inputs and Turbulence Model

Velocity boundary conditions for the valve were determined by geometric consideration for the necessary inlet flow during typical operation – found to be 4.2419 m s^{-1}. Pressure driven flows were given an inlet to outlet pressure differential of 627,700 Pa as this was the pressure differential generated by the preceding velocity boundary condition. The hydraulic fluid was assigned constant density, viscosity, and specific heat of 1070 kg m^{-3}, 0.107 kg m^{-1} s^{-1}, and 2000 J kg^{-1} K^{-1}, respectively. The solution algorithm used was the “Coupled” solver, as this was found to best converge when solving for flow through the valve. Second order discretization schemes were also chosen over first order schemes in the interest of solution accuracy, particularly important for the tetrahedral mesh elements used here (SAS IP, Inc., n.d.).

The choice of turbulence closure model was based on literature, however literature on flows through control valves presents inconsistent conclusions on which model is best suited. The decision was thus influenced by studies which presented both validation against empirical data as well as justification in CFD theory. The model chosen was the re-normalization group (RNG) k–ε model coupled with the "Two-Layer Zonal Model". This approach considers flow near walls separately to central flow zones. The RNG k–ε model is used due to its reliability in modeling separated flows and recirculation (Del Vescovo et al., 2003). This is further supported by Amirante et al. (2014) who suggest that this method is best suited to dealing with both free shear flows (e.g. jet flows at the exits of restrictive regions) and wall bounded flows, the coexistence of which characterize the flow inside hydraulic valves. Implementation of the "Two-Layer Zonal Model" in Fluent was achieved through the provided "Enhanced Wall Treatment" option (SAS IP, Inc., n.d.).

3. Results and Discussion

3.1 Heat Generation

The model exploring heat generation was velocity driven and modeled at steady state, to later be compared with a transient simulation for model verification. The heat generation increased with increasing velocity, with losses peaking just below 3.5 kW at an inlet velocity of 4.2419 m s^{-1}. A cubic curve was then fitted on this data, as this is the relationship expected between velocity and power loss. This curve is show below in Figure 2.

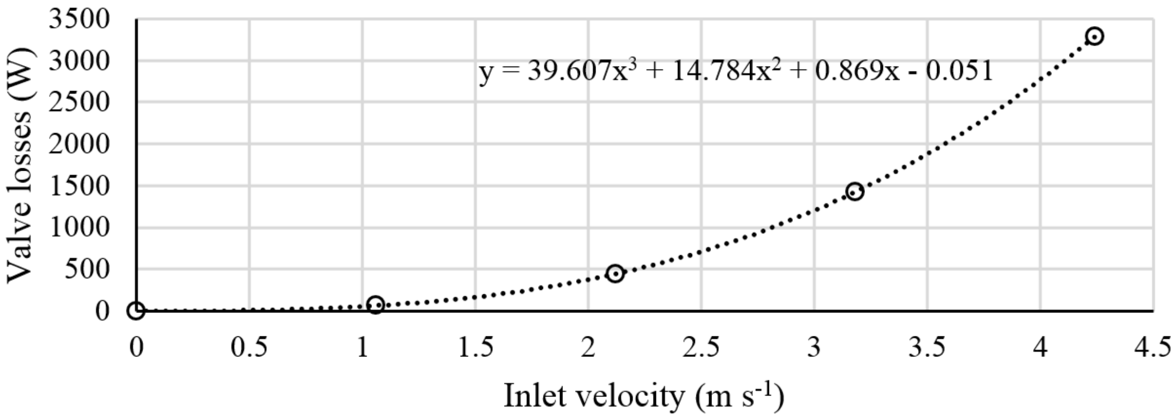

Figure 2 Plot of energy losses at varying inlet velocities, with cubic curve fitted to data.

The fitted curve could then be integrated over one 12 second period to estimate losses during a flow cycle, and resulted in energy losses of 17.02 kJ per cycle. A transient model was then run, where the inlet velocity was varied sinusoidally and losses were calculated from specific heat of the fluid, temperature, and mass flow rate at each 0.02 s time step. Only a quarter of the cycle was modeled as the energy loss in each of the remaining three quarters was expected to be identical. This resulted in an energy loss of 4,434 J over the simulation – or a total loss of 17.74 kJ over a period, which agrees closely with the initial approximation using steady state simulations. These losses are significant, and if operating with 100 kg of the hydraulic fluid used in this study and in the absence of heat dissipation, could raise the fluid temperature by almost 1 °C every 2 minutes. However these simulations have assumed a constant viscosity, and in reality the warming hydraulic fluid's viscosity would drop, which may then affect energy loss behaviour over time. The following study investigates the effect of varying viscosity.

3.2 Viscosity Sensitivity

The heating sensitivity study was conducted using a velocity driven flow while the mass flow sensitivity study used a pressure driven flow. The results of each study are shown in Figure 3.

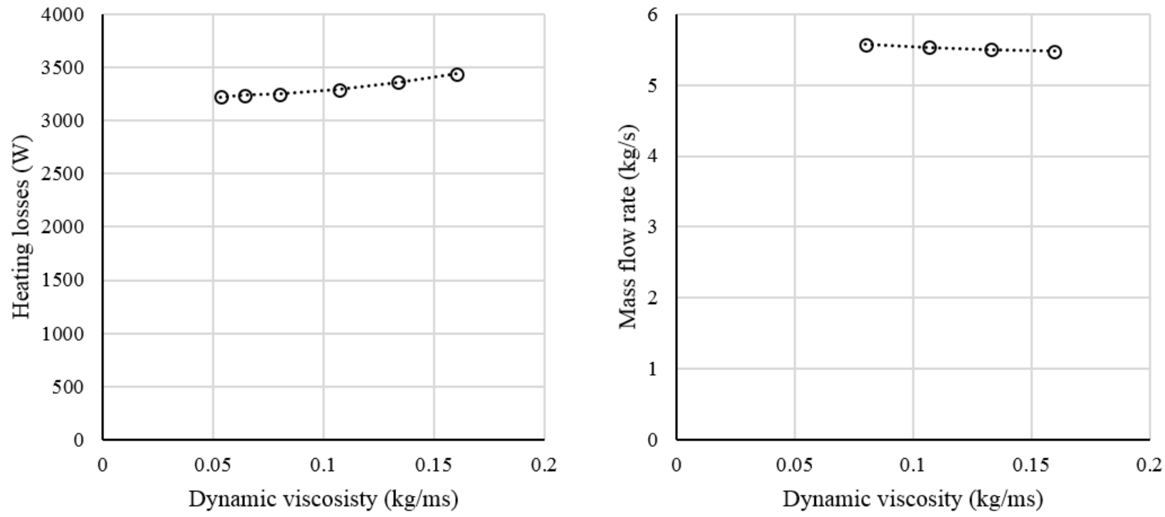

Figure 3 Plots of sensitivity of the valve's heating losses and permitted mass flow rate against varying viscosity.

The data suggests slight trends of increased losses and diminished mass flow with increasing viscosity. However these changes are small, and only vary the relevant outputs by 5% and 2% respectively for viscosity changes of 50%. These results indicate that while a reduction in viscosity of a hydraulic fluid will not noticeably reduce losses, the reduction in viscosity also does not drastically alter the behaviour of the control valve as far as permitted mass flow rate. These results are consistent with one another, as the valve's inhibition of mass flow should manifest as energy losses to the fluid due to the the fluid performing work to pass through the valve. Therefore a fairly consistent mass flow rate over a certain viscosity range must exhibit correspondingly stable losses at those viscosities. This finding also agrees with experimental research by Michael et al. (2018) who studied how viscosity reduction related to hydraulic system performance by measuring leakage flow in various hydraulic components. The study found no statistically significant change in flow losses as fluid viscosities were reduced.

The fairly unintuitive result that a reduced viscosity can provide similar fluid flow restriction to the case with an unaltered fluid might be explained by the increased presence of turbulence in low viscosity fluids. Thus, while fluids of reduced viscosity may experience weaker shear forces and lose less energy this way, they compensate somewhat by losing energy to the increased turbulence generated. Further, specific viscous losses ($\hat{E}_v$) are known to relate to averaged velocity $\bar{u}$ and a friction loss factor k according to the equation

$$\hat{E}_v = \frac{1}{2}\bar{u}^2 k \qquad \text{(Bird et al., 2002)}$$

The friction loss factor is a function of Reynolds number and flow geometry. However flow geometry often has a much stronger influence than viscosity on this value. This is increasingly true at higher Reynolds numbers where friction factors stabilize towards some value. As such, for a defined flow velocity, viscous losses experience little variation with changing viscosity.

3.3 Grid convergence

Due to the discretization involved in any CFD study, results possess an inherent uncertainty associated with the degree of discretization. A method of selecting suitable element number or size based on numerical error is proposed by Roache (1997). Suitability of meshes can be determined based on this error, and upon demonstration of monotonic reduction of discretization error in the parameter over successive levels of mesh refinement. This study has been conducted on the valve geometry, modeling for pressure losses over a velocity driven flow in three varying levels of mesh quality. As required by the method, each mesh was created such that the reduction ratio of element numbers between successive grades of mesh were roughly equal – here made to be approximately 1.93. The two coarser meshes used in the study are shown below in Figure 4.

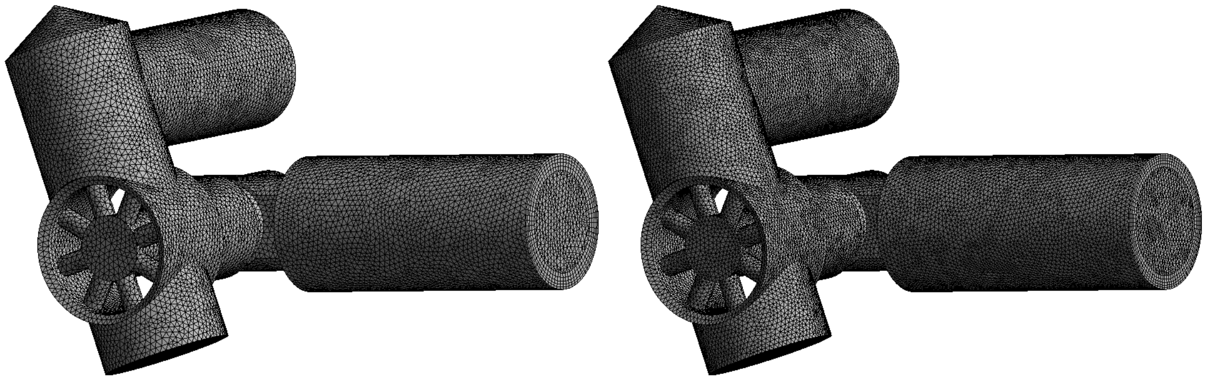

Figure 4 Coarse mesh (left) and medium mesh (right) used for grid convergence study.

The method proposed by Roache first requires a calculation of the order of truncation rate decay from the three pressure outputs of each simulation and the reduction ratio between meshes. The truncation error reduction rate was found to be approximately -1.79. Discretization error of each simulation can then be calculated. These only represent estimates of error however, and the grid convergence indicator proposed by Roache incorporates a safety factor to better capture the full numerical error associated with CFD simulations. A reduced safety factor of 1.25 is recommended by Versteeg et al. (2007) for studies of two or more levels of mesh refinement. This results in error bands of approximately 10% for the coarse mesh, 3% for the medium mesh, and 1% for the fine mesh. These results are summarized in Table 1 below.

Mesh quality	Mesh element number	Inlet pressure (Pa)	Discretization error (Pa)	Grid convergence indicator (Pa)	Percent error (%)
Coarse	498,658	707,472	-57,787	-72,233	10.21
Medium	961,887	667,479	-17,793	-22,241	3.33
Fine	1,857,478	655,164	-5,479	-6,848	1.05

Table 1 Results from the grid convergence study

Although solutions calculated from the fine mesh yielded greater accuracy, simulations of increased elements required exponentially greater time to solve. Simulations presented in this study have therefore used meshes between medium and coarse for calculating results as a compromise between solution accuracy and computational speed.

4. Conclusions and Future Work

Initial calculations of viscous heating in a hydraulic control valve yielded energy losses near 42 kJ per cycle. Considerations of the cyclic nature of the valve's operation reduced this estimation to under 18 kJ per cycle. While this is still a considerable energy loss, the studies conducted here have also shown that the valve does not exhibit significantly different behaviour when operating with reduced viscosity – with 50% changes in viscosity only changing energy loss and mass flow outputs by 5% and 2% respectively. This was suggested as being a result of the lower viscosity fluid allowing for increased turbulence, and the energy lost through this mechanism offsetting the reduction in energy losses to viscosity-driven shear stresses. Additionally, the influence of flow geometry was expected to dominate that of viscosity in the friction loss factor found in the equation for specific viscous losses. Thus the viscous losses in a flow of defined geometry and average velocity are expected to remain relatively constant. Finally, the error associated with discretization in this study was estimated using the grid convergence indicator proposed by Roache. This error was found to be between 3% and 10% for the mesh primarily used in simulations.

Areas which will need further investigation are transient simulations with variable viscosity to verify the steady state studies of viscosity sensitivity. Additional work continues to focus on acquiring better convergence of simulation solutions and behaviour at low viscosities so that the valve's exact response to viscosity changes can be confirmed with greater confidence. This might be achieved with finer meshing in the valve and more careful meshing around its critical areas. Finally, similar studies of viscous heating and viscosity sensitivity will be conducted on separate hydraulic systems to characterize a variety of geometries and their effect on functions of hydraulic valves.

5. References

Amirante, R., Catalano, L.A., & Tamburrano, P. (2014) The importance of a full 3D fluid dynamic analysis to evaluate the flow forces in a hydraulic directional proportional valve, *Engineering Computations*, **31** (5) pp. 898-922.

Bird, R.B., Stewart, W.E., & Lightfoot, E.N. (2002) Transport Phenomena, 2nd edn, John Wiley & Sons, Inc., New York.

Del Vescovo, G. & Lippolis, A. (2003) Three-Dimensional Analysis of Flow Forces on Directional Control Valves, *International Journal of Fluid Power*, **4** (2) pp. 15-24.

Michael, P., Cheekolu, M., Panwar, P., Devlin, M., Davidson, R., Johnson, D., & Martini, A. (2018) Temporary and Permanent Viscosity Loss Correlated to Hydraulic System Performance, *Tribology Transactions*, DOI: 10.1080/10402004.2018.1439210.

Morini, G.L. (2013) Viscous Heating. In: Li D. (eds) *Encyclopedia of Microfluidics and Nanofluidics*. Springer, Boston, MA.

Roache, P.J. (1997) Quantification of Uncertainty in Computational Fluid Dynamics, *Annual Review of Fluid Mechanics*, **29** pp. 123-160.

SAS IP, Inc. (n.d.) 4.16.5. Enhanced Wall Treatment ε-Equation (EWT-ε). Available from: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_th/flu_th_sec_ewt_eps.html. [15 August 2018].

SAS IP, Inc. (n.d.) 28.2.1. First-Order Accuracy vs. Second-Order Accuracy. Available from: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/flu_ug/x1-174200027.2.1.html. [15 August 2018].

Versteeg, H.K. & Malalasekera, W. (2007) An Introduction to Computational Fluid Dynamics, 2nd edn, Pearson/Prentice Hall, Sydney.