A Transient Multidimensional CFD Approach to the Analysis of a Control Valve Using Non-Newtonian Fluids

Fabrizio Grasselli, Massimo Milani, Luca Montorsi and Fabrizio Paltrinieri

Department of Sciences and Methods for Engineering, University of Modena and Reggio Emilia, Reggio Emilia 42122, Italy

Received: September 08, 2011 / Accepted: December 01, 2011 / Published: September 30, 2012.

Abstract: In this paper the flow through a control directional valve is studied by means of a CFD (computational fluid-dynamics) analysis under transient operating conditions. The mesh motion is resolved on a time basis as a function of the external actuation system. In the analysis, an open source fluid-dynamics code is used and both cavitation and turbulence are accounted for in the modeling. Moreover, the numerical model of the working fluid is modified in order to account also for the non-Newtonian fluids. The effects of the shear rate on the shear stress are accounted for both by using experimental measurements and correlations available in literature, such as the Herschel-Bulkley model. The analysis determines the performance of the control directional valve under different operating conditions when using either Newtonian or non-Newtonian fluids. In particular, the discharge coefficient, the recirculating regions, the flow acceleration angle and the pressure and velocity fields are investigated.

Key words: CFD, hydraulic valve, transient analysis, moving mesh, non-Newtonian fluid.

Nomenclature

\[ \eta \] Power law exponent [-]
\[ k \] Consistency \[ Pa \cdot s \]
\[ s \] Non-dimensional spool displacement [-]
\[ \tau_y \] Yield stress \[ Pa \]
\[ \tau_s \] Shear stress \[ Pa \]

1. Introduction

Numerical analysis is gaining an important role in the design of hydraulic components and systems. The investigation of the flow through the metering section of hydraulic components plays a fundamental role in the design and optimization processes. In particular, multidimensional simulation is increasingly applied to the investigation of the fluid dynamics behaviour of hydraulic component in order to broaden and complete the experimental campaigns. Nevertheless, in order to obtain a sufficient accuracy of the numerical results, particular care must be devoted to the operating conditions used in the simulation. In fact, the flow phenomena characterizing the hydraulic components are highly time dependent, while in the CFD (computational fluid-dynamics) analysis the conditions are generally approximate to steady state.

Many examples of CFD applications to hydraulic systems and component analysis are available. In Refs. [1-3] the CFD analysis was used to study the flow field and the flow-induced forces in hydraulic valves, as well as in Refs. [4-7] theoretical approaches and experimental investigations have been compared to CFD predictions for the hydraulic valves design and optimization.

This paper focuses on the development and the tailoring of an open source multidimensional CFD code to the analysis of the internal flow-field in hydraulic components. In this paper, the metering characteristics of a control valve of a load-sensing control system are investigated using the numerical simulation. A fully transient approach is used in the analysis; in particular, both the boundary conditions and the moving geometry are accounted for, therefore, the boundary conditions as well as the spool position,
and thus the CFD domain grid, are function of time in the numerical simulations. Furthermore, two different types of fluid are considered in the analysis. First, a standard hydraulic oil is used in the simulation and it is assumed as Newtonian fluid; afterward, a non-Newtonian fluid is accounted for, since the control valve can be used for applications to viscoelastic fluid.

The open source OpenFOAM [8] code is applied to predict the flow through the metering edge of the control valve under different operating conditions. The numerical simulations are carried out assuming incompressible flow, and therefore physical phenomena such as cavitation and gas absorption are neglected.

The hydraulic valve performance is addressed in terms of overall discharge coefficient, the recirculating regions, the flow acceleration angle and the pressure and velocity fields as well as the flow forces on the spool are investigated and confronted.

2. Numerical Model

2.1 The Studied Control Valve

The control valve studied in this paper is included in an electro-hydraulic load-sensing proportional control system, usually adopted in multi-slice blocks to control parallel actuations of industrial, agricultural and earthmoving applications. Furthermore, the component can be used in industrial applications where non-Newtonian fluids are employed.

Fig. 1 depicts the geometry of the valve analyzed. The inlet and outlet regions are outlined, as well as the coordinate system used in the paper for describing the results in the following.

2.2 The Numerical Analysis

The CFD analysis of the pressure compensator for a load-sensing system is carried out by means of the open source, computational fluid dynamics code OpenFOAM. Bounded central differencing is used for the discretization of the momentum, second-order upwind for subgrid kinetic energy. Pressure-velocity coupling is achieved via a PISO (pressure implicit splitting of operator algorithm). The second-order implicit method is used for time integration scheme.

The numerical simulations are carried out assuming incompressible flow, and therefore physical phenomena such as cavitation and gas absorption are neglected. The effects of turbulence are included in the analysis by means of a two-equation approach using the k-ω SST (shear stress transport) model.

The CFD domain is plotted in Fig. 2. Due to the symmetry of both the geometry and the boundary conditions, it was possible to simulate only half of the real geometry using symmetry plane boundary conditions.

The mesh is created using an unstructured grid made mostly of hexahedra; the average mesh resolution is set to 0.1 mm with local refinement down to 0.01 mm close to the metering edge and in the spool volume. Moreover, wall cell layers are employed in order to have the proper wall cell height according to the adopted turbulence model. The grid resolution was the best tradeoff between results’ accuracy and computational effort.

In this paper, the numerical analysis of the control valve was carried out by means of a fully transient approach. In the simulations, the valve is actually moving as a function of time during the calculation. In order to realize the mesh motion of the spool, the mesh...
changer libraries of the OpenFOAM code were modified to account for the linear motion of a portion of the geometry sliding over a second portion. The mesh motion was resolved by using a GGI (generalized grid interface) approach [9], originally developed for turbomachinery applications and modified to include not only rotational motion of the moving grid but also the linear displacement of a valve spool. The spool was moved according to the curves depicted in Fig. 3. Two different actuations were considered in the analysis. A slow actuation that moves the spool from the closed position to the maximum displacement, then the spool holds in the maximum opening position for approximately 2 s and moves back to the complete closure. The total period of time is 5.36 s. In the fast actuation the same displacement profile is used, but the total period of time elapsed during the actuation is one tenth of the slow actuation one. The spool trajectory simulates a PWM (pulse width modulation) control to the spool motion. The control valve was tested under several operating conditions. First, the pressure drop across the valve was held constant and equal to the average pressure drop measured experimentally. Second, the pressure drop was modified according to the measured flow rate through the hydraulic component (Fig. 4). In each case, the slow and fast actuations were simulated. Finally the mentioned boundary conditions were used for testing the hydraulic component behaviour when operating with a non-Newtonian fluid. Table 1 lists the simulated cases, detailing the boundary conditions of the simulations. In order to simulate the non-Newtonian fluid, the viscosity model based on the Herschel-Bulkley correlation was used. In fact, this model demonstrated to have the best agreement with the rheological measurements carried out on the real fluid. In the Herschel-Bulkley correlation the following parameters were used:

\[
\tau = \tau_0 + k \times \gamma^n
\]

\[
\tau_0 = 1.01; k = 0.76; n = 0.85
\]

3. Results

The results of the numerical analysis of the control valve for a load-sensing system are discussed in terms of discharge coefficients, pressure and velocity fields, flow acceleration angles and flow forces acting on the
Table 1  Boundary conditions used for the different simulated cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>Operating fluid</th>
<th>Boundary conditions</th>
<th>Actuation</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>Newtonian</td>
<td>Constant pressure drop ($\Delta p = 6$ bar)</td>
<td>Slow (Fig. 3a)</td>
</tr>
<tr>
<td>#2</td>
<td>Newtonian</td>
<td>Constant pressure drop ($\Delta p = 6$ bar)</td>
<td>Fast (Fig. 3b)</td>
</tr>
<tr>
<td>#3</td>
<td>Newtonian</td>
<td>Measured flow rate (Fig. 4)</td>
<td>Slow (Fig. 3a)</td>
</tr>
<tr>
<td>#4</td>
<td>Newtonian</td>
<td>Measured flow rate (Fig. 4)</td>
<td>Fast (Fig. 3b)</td>
</tr>
<tr>
<td>#5</td>
<td>Non-Newtonian</td>
<td>Constant pressure drop ($\Delta p = 6$ bar)</td>
<td>Slow (Fig. 3a)</td>
</tr>
<tr>
<td>#6</td>
<td>Non-Newtonian</td>
<td>Constant pressure drop ($\Delta p = 6$ bar)</td>
<td>Fast (Fig. 3b)</td>
</tr>
<tr>
<td>#7</td>
<td>Non-Newtonian</td>
<td>Measured flow rate (Fig. 4)</td>
<td>Slow (Fig. 3a)</td>
</tr>
<tr>
<td>#8</td>
<td>Non-Newtonian</td>
<td>Measured flow rate (Fig. 4)</td>
<td>Fast (Fig. 3b)</td>
</tr>
</tbody>
</table>

spool walls. In particular, a comparison between the effects on the hydraulic component behaviour in case of slow and fast actuation is carried out. Moreover, the influence of the non-Newtonian operating fluid on the valve performance is depicted.

3.1 Steady State Analysis

First, for all the simulated cases detailed in Table 1 a preliminary steady state analysis is carried out in order to initialize the flow at the first time step of the transient simulation. This procedure is necessary to obtain accurate results also for the initial phase of the moving spool calculation, otherwise, the first part of the dynamic analysis would lead to non-realistic results due to the stabilizing flow field. Fig. 5 shows the convergence of the flow rate at the outlet boundary for Case #1. The converged steady state flow fields are therefore used as initial condition for the transient simulation.

3.2 Transient Analysis

3.2.1 Discharge Coefficient

Fig. 6 shows the results of the transient simulations in terms of discharge coefficients calculated on the basis of the total pressure drop across the valve. In Fig. 6a the results regarding the constant pressure drop calculations are summarized. It can be noticed that during the opening travel and the period of time when the spool is fully open, the slow actuation is characterized by larger discharge coefficients. This behaviour is more evident for the case in which a non-Newtonian operating fluid is adopted. The difference in terms of permeability between the two actuations reaches values up to 25% just after the opening travel for Cases #5 and #6.

This behaviour is due to the inertia effects of the flow that can be exploited in case of the slow actuation. Even though the metering area is not increasing due to the stand by phase of the spool motion, the inertia of the fluid contributes to increasing the flow rate through the notch. In the fast actuation the stand by phase is too short to take advantage of this phenomenon. This effect is more evident for high viscous flows as in Cases #5 and #6.
A Transient Multidimensional CFD Approach to the Analysis of a Control Valve Using Non-Newtonian Fluids

Fig. 6 Discharge coefficients as a function of the spool displacement for the simulations with (a) constant pressure drop across the valve and (b) measured inlet flow rate.

Fig. 7 shows the results in terms of discharge coefficients as a function of the square root of the Reynolds number for the constant pressure drop cases. In particular, the behaviours obtained by the slow and the fast actuations for the two considered operating fluids are compared. Furthermore, Fig. 7 outlines the discharge coefficients that characterize the opening and closure travels.

As mentioned before, the slow actuation enables larger discharge coefficient, in particularly during the stand by phase of the spool motion; similarly, during the closure travel the valve permeability demonstrated to be larger than during the opening travel. This behaviour is again due to the inertia effects that contribute to increase the flow rate during the reduction of the metering area, while during the opening phase, the acceleration of the flow is retarded with respect to the increasing metering section. In fact, the difference between the opening and closure phases is enhanced when adopting the fast actuation.

3.2.2 Flow Field and Efflux Angle

Figs. 8 and 9 show the results in terms of pressure, velocity and viscosity flow fields for different spool positions in the constant pressure drop cases. The results are plotted on a cut section through the first and third notches and on a cut plane through the outlet port axis. It is interesting to notice that when comparing the slow and the fast actuations, the former is characterized by larger values of pressure downstream the metering section. This result confirms what has been evidenced in terms of discharge coefficients. In Fig. 9, the contour plot of the viscosity is depicted for the cases employing the non-Newtonian fluid. The viscosity of the fluid demonstrated to decrease significantly, i.e., approximately by the half, through the metering area due to the flow acceleration and thus to the increase of the sheer stress.

Fig. 10a depicts the reference coordinate system used for calculating the acceleration of the flow through
the metering section of each notch. The reference axis is taken orthogonally to the metering area of each notch pointing outward from the spool. The efflux angle varies significantly with the distance from the notch metering area (Fig. 10b). In particular, it increases while moving far from the metering section, i.e., approximately by the 25%; this behaviour is likely due to effect of the edge of the outlet region located in front of the metering area; thus, the edge bends the flow at the spool exit increasing the acceleration angle. In the following, the considered efflux angles are referring to the reference plane at 0.5 mm from the notch exit, since the values for this plane are representative of the average behaviour.

In Fig. 11 the acceleration angle for all the simulated cases is plotted as a function of the spool displacement. The values obtained for the four different notches are included. It can be seen that the efflux angle increases during the opening travel of the spool, fluctuates slightly during the stand by phase and decreases during
Fig. 9  Pressure and velocity magnitude contour section plots at spool positions: (a) Case #5 and (b) Case #6.

Fig. 10  (a) Reference coordinate system used for the efflux angle calculations and (b) efflux angle as a function of spool displacement for the different considered planes (Notch #1, Case #1).
Fig. 11  Efflux angle as a function of the spool displacement for the different simulated cases: (a) Case #1; (b) Case #2; (c) Case #3; (d) Case #4; (e) Case #5; (f) Case #6; (g) Case #7; (h) Case #8.

the closure travel. Nevertheless, the efflux angles that characterize the closure phase are always slightly larger than the ones obtained for the opening travel. In fact, during the spool closure, the motion is opposing to the flow direction creating larger recirculation regions downstream the metering area that tend to increase the acceleration angle.

No significant differences were outlined when
comparing the efflux angles obtained for the slow and fast actuations in case of Newtonian fluid. Conversely, when the non-Newtonian fluid is employed by the flow acceleration angle resulted to be remarkably lower during the opening phase of the fast actuation, while it demonstrated to comparable during the closure travel. The difference is due to the kinematic viscosity behaviour; the value of the viscosity remains slightly larger during the opening travel of the fast actuation profile leading to a more evident coanda effect with respect to the walls in the notch outlet region.

3.2.3 Flow Forces

Fig. 12 details the flow forces calculated for all the simulated cases. The flow force in the $x$ direction is acting parallel to the spool motion and it opposes or enhances the actuation according to its sign (positive axial force is opposing to the spool motion). The radial forces are defined according to the global coordinate system as depicted in Fig. 1. As far as the axial force is concerned, the value results to be quite similar during the entire spool motion; nevertheless, during the opening and closure travels small oscillations appear. Moreover, the difference between the slow and fast actuations in terms of axial force demonstrates to be remarkably small, even though in the case of the fast actuation the oscillations during the opening and closure phases are more evident, in particular for the closure travel.

The radial forces, demonstrates to be definitely smaller than the axial one; nonetheless, the force in the $z$ direction approaches the values of the axial force during the last part of the closure phase.

4. Conclusions

In this paper the metering characteristics of a control valve for a load-sensing control system have been simulated by using a fully transient approach, including both the dynamic boundary conditions and the moving grid due to the spool motion. In the simulations a standard oil has been considered as an operating fluid as well as the non-Newtonian fluid. The dependency of the viscosity on the shear stress gradient has been accounted for by means of the Herschel-Bulkley correlation. Furthermore, a slow and a fast actuation for the spool motion were simulated and compared.
The numerical results demonstrated that during the opening travel and the period of time when the spool is fully open, the slow actuation was characterized by larger discharge coefficients than the fast actuation; the difference resulted to be more remarkable when employing a non-Newtonian fluid. This behaviour is likely due to the inertia effects of the flow that can be exploited in case of the slow actuation, while in the fast actuation the time becomes too short to take advantage of the inertia. Furthermore, during the spool closure travel, the valve permeability demonstrated to be larger than during the opening travel.

The efflux angle resulted to increase during the opening travel of the spool, to be quite stable during the stand by phase and to decrease during the closure travel. Nevertheless, the efflux angles that characterize the closure phase were calculated to be always slightly larger than the ones obtained for the opening travel. The acceleration angle proved to be scarcely influenced by the actuation speed, while significant effects were seen when considering a Newtonian of a non-Newtonian operating fluid, the efflux angle was smaller during the opening travel.

Finally, the axial force resulted to be quite similar during the entire spool motion, even though small oscillations were estimated during the opening and closure travels. Moreover, the difference between the slow and fast actuation in terms of axial force demonstrated to be remarkably small, except for the more evident oscillations that characterized the opening and closure phases.

References