International Journal of Heat and Fluid Flow 64 (2017) 89–102

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Evaluation of steady flow torques and pressure losses in a rotary flow control valve by means of computational fluid dynamics

Ivan Okhotnikov[a,*], Siamak Noroozi[a], Philip Sewell[a], Philip Godfrey[b]

[a] Bournemouth University, Faculty of Science and Technology, BH12 5BB, United Kingdom
[b] Hydreco Hydraulics Limited, 32 Factory Road, Poole, Dorset, BH16 5SL, United Kingdom

ARTICLE INFO

Article history:
Received 4 July 2016
Revised 22 December 2016
Accepted 17 February 2017

Keywords:
Rotary flow control valve
Computational fluid dynamics
Pressure drop
Steady flow torques
Discharge coefficient
Jet angle

ABSTRACT

In this paper, a novel design of a rotary hydraulic flow control valve has been presented for high flow rate fluid power systems. High flow rates in these systems account for substantial flow forces acting on the throttling elements of the valves and cause the application of mechanically sophisticated multi-staged servo valves for flow regulation. The suggested design enables utilisation of single-stage valves in power hydraulics operating at high flow rates regimes. A spool driver and auxiliary mechanisms of the proposed valve design were discussed and selection criteria were suggested. Analytical expressions for metering characteristics as well as steady flow torques have been derived. Computational fluid dynamics (CFD) analysis of steady state flow regimes was conducted to evaluate the hydraulic behaviour of the proposed valve. This study represents a special case of an independent metering concept applied to the design of power hydraulic systems with direct proportional valve control operating at flow rates above 150 litres per minute. The result gained using parametric CFD simulations predicted the induced torque and the pressure drops due to a steady flow. Magnitudes of these values prove that by minimising the number of spool's mobile metering surfaces it is possible to reduce the flow-generated forces in the new generation of hydraulic valves proposed in this study. Calculation of the flow jet angles was analytically verified by measuring the deflection of the velocity vector using flow velocity field distribution, obtained during visualisation of the results of CFD simulations. The derived calculation formulas can predict metering characteristics, values of steady flow torques and jet angles for the specified design and geometry of the suggested valve. The proposed novel structure of the flow control valve promises to attain improved controllability, reliability and efficiency of the hydraulic control units of heavy mobile machinery operating at high flow rates regimes.

1. Introduction

Fluid power systems are a major element in the design and development of all heavy off road, earthmoving, agricultural and construction machinery. Their contribution to the overall performance of such machines is hard to overestimate. These industrial applications require hydraulics functioning at high-pressure and high-flow rates in order to operate multiple drives of a single mobile machine such as manipulator arms, wheels, crawlers, transmission and other appliances simultaneously. These large flow rates require a significant amount of energy to achieve and consequently result in a substantial loss of flow energy or pressure drops, which is quite common in such hydraulic systems (Merritt, 1968). These losses are usually due to viscous friction, swirls formation, sudden changes in flow direction and cross-section (Lisowski and Rajda, 2013). Large flow rates also account for considerable flow forces acting on the regulating elements of the control valves (Rajda and Lisowski, 2013).

In order to overcome these large resisting forces, an indirect pilot hydraulic actuation of the main sliding spool is employed. In this solution, the design of flow control valves lacks mechanical reliability due to the introduction of an additional hydraulic stage, what causes immense pressure losses, poor energy efficiency and rigorous requirements to the level of oil contamination due to sophisticated and narrow internal channelling (Filho and De Negri, 2013). Furthermore, production of such valves demands high-precision manufacturing processes. That increases the overall cost of these flow control units. Manual assembly of torque motor driven two-stage valves and their adjustment also motivates investigations of alternative technologies (Plummer, 2016).

* Corresponding author at: Department of Design and Engineering, Faculty of Science and Technology, Bournemouth University, Poole House, Talbot Campus, Fern Barrow, Poole, Dorset, BH12 5BB, United Kingdom.

E-mail addresses: iokhotnikov@bournemouth.ac.uk (I. Okhotnikov), snoroozi@bournemouth.ac.uk (S. Noroozi), psewell@bournemouth.ac.uk (P. Sewell), pgodfrey@hydreco.com (P. Godfrey).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2017.02.005
0142-727X/

Nomenclature

Latin symbols

$A(\varphi)$	Function of the opening area, m^2
A_v	Van Driest coefficient
$C_{\varepsilon 1}, C_{\varepsilon 2}, C_\mu, C_B$	Constant empirical closure coefficients in the $k-\varepsilon$ turbulence model
C_d	Discharge coefficient
C_v	Velocity coefficient
F_{fl}	Steady flow force, N
f_μ	Turbulent viscosity factor
f_1, f_2	Lam and Bremhorst's damping functions
g_i	The component of gravitational acceleration in direction of i, m/s^2
K	Karman constant
k	Turbulent kinetic energy, m^2/s^2
$\dot{m}$	Fluid mass flow rate, kg/s
P_B	Buoyancy-generated turbulence production term in the $k-\varepsilon$ turbulence model, $1/s^2$
Δp	Pressure differential, Pa
Q	Volume flow rate, litres/minute
R_y, R_T	Turbulent Reynolds numbers
$R_{sl.ext}$	Sleeve external diameter, m
$R_{sp.ext}, R_{sp.in}$	Spool external and internal diameters respectively, m
T_{fl}	Steady state flow torque, N·m
t	Time, s
u_i	The ith component of the fluid velocity, m/s
u^+	Dimensionless longitudinal wall velocity
v_1, v_2	Average inlet and outlet velocities of a control volume respectively, m/s
x_i	The ith component of the Cartesian coordinate system
y^+	Dimensionless distance from wall surface

Greek symbols

α	Angle, occupied by the spool window in the outflow section, °
β	Angle, occupied by the sleeve window in the outflow section, °
γ	Backlash angle, °
δ_{ij}	The ijth component of the Kronecker delta function
ε	Turbulent dissipation rate, m^2/s^2
η	General variable
θ	Jet angle, °
μ	Dynamic viscosity coefficient, kg/(m·s)
μ_t	Turbulent eddy viscosity coefficient, kg/(m·s)
ρ	Fluid density, kg/m^3
$\sigma_\epsilon, \sigma_k, \sigma_B$	Constant empirical closure coefficients in the $k-\varepsilon$ turbulence model
τ_w	Wall shear stress, Pa
τ_{ij}^R	The ijth component of the Reynolds-stress tensor, Pa
φ	Spool angular position, °

Subscripts and superscripts

fl	Steady state flow
fl.tan	Tangential component of the flow force
i, j, k	Directions of the Cartesian coordinate system
w	At the wall
single	Single orifice parameter
total	Entire valve parameter

Abbreviations

+	Dimensionless wall parameter
CFD	Computational fluid dynamics
DC	Direct current
LPM	Litres per minute

One way to reduce losses in such valves is to optimise the flow paths through them in order to lessen flow disturbances (Simic and Herakovic, 2015). One promising method to enhance the valve's performance in terms of losses, flow disturbance reduction, and simplification of the valve geometry for manufacture is to utilise specially profiled sliding spools with special geometrical features. A number of studies have shown the positive impact of sleek spool geometry, especially smooth change of diametrical sizes of a spool along its length, on the fluid's rate of momentum change, hence reducing the flow forces experienced by the spool's driver. The introduction of a compensation profile on the sliding spool's shaft diminishes the flow forces by creating a pressure drop in the downstream cavity (Amirante et al., 2007). It was proven experimentally, that geometrical optimisation of central conical surfaces on the spool shank can provide higher axial velocities at the inlet of the meter-in chamber and the outlet of the meter-out chamber of the spool, which alleviates a net flow force, and lowers dynamic overshoot in step response (Amirante et al., 2016). Cone surfaces on the spool's control edges and returning oil jet back in spool's cavity on the meter-out edges enables application of direct actuation of the spool for larger nominal valve sizes (Herakovič, 2009).

Adding a supplementary parallel channel to the return line in the valve's body allows extension of the valve's operational flow range by improving the carrying capacity or conductivity of the drain line without resorting to more powerful solenoids (Lisowski et al., 2013). The geometrical optimisation of flow regulating parts has a notable effect on flow forces in seat valves as well (Simic and Herakovic, 2015).

Other viable ways to lessen the effect of flow forces are improvement of electromagnetic actuator's performance (Reichert, 2010), advanced regulation of spring rates of the return mechanism and geometrical optimisation of the spool's ambient parts and channels, inlet and outlet spool chambers (Abdalla et al., 2011) to make them less prone to formation of eddies, vortices and flow disturbances.

Advanced architectural approaches to design and control high-flow rate hydraulic systems are demonstrated in concepts of independent metering (Shenouda, 2006; Choi et al., 2015) and digital hydraulics (Linjama, 2011). The former utilises separate control of flows into and from actuator's chambers. The latter consists in incremental modulation of the flows in hydraulic lines by switching on and off individual valves connected in a parallel layout. Generally, these concepts rely on two-way valve setups. These two new concepts have had limited application due to the low flow rates used to date. Nevertheless, these concepts need further advancement in order to be effectively implemented within high flow rate systems as these approaches are considered promising in performance improvement of current hydraulic systems.

Despite the vast number of design studies all looking to minimise the flow forces and static pressure losses, the application of conceptually alternative construction of throttling elements and their arrangements are still rare in current literature. The use of valves with rotary spools to solve the problems associated with large flow forces that exist in high flow rate regimes has been studied (Yu et al., 2014, 2015; Yang et al., 2010; Wang et al., 2016). Unlike conventional designs, a rotating spool configuration creates a much smaller net area of surfaces subjected to the flow forces. So