Best Practices and Recommendations for CFD Simulation of Hydraulic Spool Valves

Authors : Jérémy Philippe, Lucien Baldas, Batoul Attar, Jean-Charles Mare

Abstract : The proposed communication deals with the research and development of a rotary direct-drive servo valve for aerospace applications. A key challenge of the project is to downsize the electromagnetic torque motor by reducing the torque required to drive the rotary spool. It is intended to optimize the spool and the sleeve geometries by combining a Computational Fluid Dynamics (CFD) approach with commercial optimization software. The present communication addresses an important phase of the project, which consists firstly of gaining confidence in the simulation results. It is well known that the force needed to pilot a sliding spool valve comes from several physical effects: hydraulic forces, friction and inertia/mass of the moving assembly. Among them, the flow force is usually a major contributor to the steady-state (or Root Mean Square) driving torque. In recent decades, CFD has gradually become a standard simulation tool for studying fluid-structure interactions. However, in the particular case of high-pressure valve design, the authors have experienced that the calculated overall hydraulic force depends on the parameterization and options used to build and run the CFD model. To solve this issue, the authors have selected the standard case of the linear spool valve, which is addressed in detail in numerous scientific references (analytical models, experiments, CFD simulations). The first CFD simulations run by the authors have shown that the evolution of the equivalent discharge coefficient vs. Reynolds number at the metering orifice corresponds well to the values that can be predicted by the classical analytical models. Oppositely, the simulated flow force was found to be quite different from the value calculated analytically. This drove the authors to investigate minutely the influence of the studied domain and the setting of the CFD simulation. It was firstly shown that the flow recirculates in the inlet and outlet channels if their length is not sufficient regarding their hydraulic diameter. The dead volume on the uncontrolled orifice side also plays a significant role. These examples highlight the influence of the geometry of the fluid domain considered. The second action was to investigate the influence of the type of mesh, the turbulence models and near-wall approaches, and the numerical solver and discretization scheme order. Two approaches were used to determine the overall hydraulic force acting on the moving spool. First, the force was deduced from the momentum balance on a control domain delimited by the valve inlet and outlet and the spool walls. Second, the overall hydraulic force was calculated from the integral of pressure and shear forces acting at the boundaries of the fluid domain. This underlined the significant contribution of the viscous forces acting on the spool between the inlet and outlet orifices, which are generally not considered in the literature. This also emphasized the influence of the choices made for the implementation of CFD calculation and results analysis. With the step-by-step process adopted to increase confidence in the CFD simulations, the authors propose a set of best practices and recommendations for the efficient use of CFD to design high-pressure spool valves.

Keywords : computational fluid dynamics, hydraulic forces, servovalve, rotary servovalve
Conference Title : ICFMHS 2025 : International Conference on Fluid Mechanics and Hydraulic Systems
Conference Location : London, United Kingdom
Conference Dates : May 24-25, 2025