POLITECNICO DI TORINO
Repository ISTITUZIONALE

A new CFD-Simulink based systems engineering approach applied to the modelling of a hydraulic safety relief valve

Original
A new CFD-Simulink based systems engineering approach applied to the modelling of a hydraulic safety relief valve / DALLA VEDOVA, M.D.L., Maggiore, P., Riva, G.. - In: INTERNATIONAL JOURNAL OF MECHANICS. - ISSN 1998-4448. - ELETTRONICO. - 11:(2017), pp. 43-50.

Availability:
This version is available at: 11583/2668611 since: 2017-04-05T13:11:04Z

Publisher:
North Atlantic University Union

Published
DOI:

Terms of use:

This article is made available under terms and conditions as specified in the corresponding bibliographic description in the repository

Publisher copyright

(Article begins on next page)

07 November 2026

A new CFD-Simulink based systems engineering approach applied to the modelling of a hydraulic safety relief valve

M. D. L. Dalla Vedova, P. Maggiore, and G. Riva

***Abstract*—**A safety relief valve is a simple hydro-mechanic device, needed to avoid overpressure transients inside hydraulic circuits. Such valves are a critical part of the hydraulic system of aircraft; hence their performances must be adapted to a specific nominal pressure level and design requirements. In the following paragraphs the authors will address the issue of designing and validating a safety valve through a hybrid CFD/MATLAB-Simulink® approach. The main constraints are the geometrical dimensions and the need to limit the weight of the device. A significant part of the work consists of gathering all the possible information available in the literature, dealing with the best design practices to achieve the performance objective. Thanks to a robust computational procedure, it should be possible to reduce the amount of "physical" prototypes required to validate the functionality of a safety relief valve. The process presented uses a numerical computational fluid dynamic (CFD) approach, to define the pressure field inside the valve and the forces acting on it; identifying the force distribution inside the valve is paramount to address the performance evaluation of the valve itself. The first step deals with the definition of a computer aided design (CAD) model of the valve. Then the CFD software uses the above-mentioned CAD model to define the domain of the problem. Once obtained the pressure field, it is possible to integrate it through the surface of the valve, thus obtaining the forces acting on the moving part (poppet). After the numerical scheme has been calibrated, some investigations are done to reduce the computational cost: the main aim is to run a complete simulation (meshing and solving) on a standard computer. Some of the positions (i.e. strokes) of the valve have been simulated as static, hence a steady-state calculation has been applied to solve the motion field. Another important result consists of creating a MATLAB-Simulink® model, capable to reach results comparable to the CFD simulation, but in shorter times. While the CFD model can provide high quality results, the MATLAB-Simulink® calculation should be used to create a "first guess" instrument, useful to address the very first valve geometry. The implementation of the Look-Up Tables (LUTs) links the MATLAB-Simulink® model to the CFD simulation, but increases the time required to obtain a solution: on the other hand, this reduces the amount of equation-modeled quantities, delivering a greater precision to the calculations.

P. Maggiore is with the Department of Mechanical and Aerospace Engineering (DIMEAS), Politecnico di Torino, Corso Duca degli Abruzzi, 24 - 10129 - Torino, ITALY. (e-mail: paolo.maggiore@polito.it).

G. Riva is with the Department of Mechanical and Aerospace Engineering (DIMEAS), Politecnico di Torino, Corso Duca degli Abruzzi, 24 - 10129 - Torino, ITALY. (e-mail: lorenzo.pace@polito.it).

M. D. L. Dalla Vedova is with the Department of Mechanical and Aerospace Engineering (DIMEAS), Politecnico di Torino, Corso Duca degli Abruzzi, 24 - 10129 - Torino, ITALY. (corresponding author phone: +390110906850; e-mail: matteo.dallavedova@polito.it).

***Keywords*—**Pressure relief valve (PRV), hydraulic system, lumped parameters numerical model, orifice flow, steady CFD, quasi-dynamic numerical simulation.

I. Introduction

A safety relief valve (also known as pressure relief valve, or PRV) is a simple device, needed to limit the level of pressure inside a hydraulic circuit, relieving the excess of flow to the oil tank through a mechanical system. This element is extremely important, since in many applications (i.e. not only purely aeronautical, but also mechanical or industrial), the pressure inside a system could build up past a desired threshold, damaging or even destroying the system itself. This is the case of a cold-start of an engine, when the oil is not viscous enough to create a protective layer of fluid around the mechanical components, or of an industrial machine, where an excessive amount of pressure could generate water hammers or damage the internal components; indeed, a burst of oil on the internal tools could damage the machine and the workpieces. Figure 1 represents the general structure of a hydraulic circuit, highlighting all the main components and its functioning.

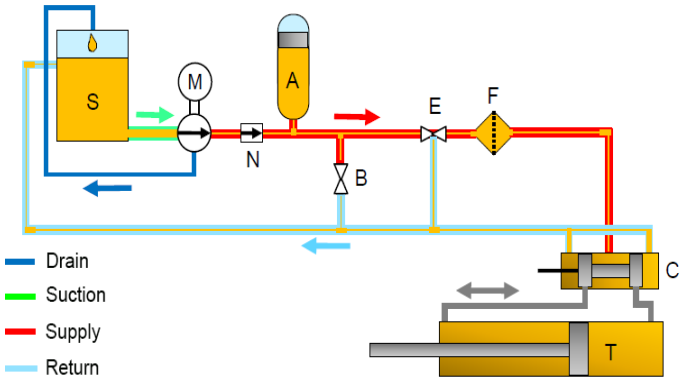
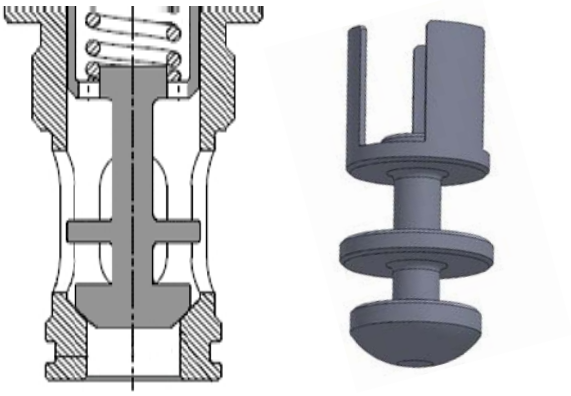


Fig. 1 general scheme of the considered hydraulic circuit

Oil is pushed from a tank (S) by a pump driven by a motor (M), then flows to a check valve (N). An accumulator (A) avoids pressure fluctuations and ensures a power backup in case of failure of the pump or potential water hammers. Oil flows through a pressure regulating valve (E) and a filter (F), but a way back to the tank (S) is ensured by the safety pressure relief valve (B) (shown in Fig. 2).



(a) (b)

Fig. 2 schematic of the pressure relief valve (a) and explanatory drawing of the PRV poppet (b)

Then the fluid moves into the spool valve (C) and is diverted in the desired chamber of the jack (T), or directly sprayed on the gearing of an engine/tool machine as a cooling lubricant. The final destination of the oil depends on the specific application analyzed, but any kind of circuit, using liquid or gaseous fluids, needs a relief valve to reduce the excess pressure (for instance, even a pressure cooker needs it, for safety reasons). The working principle of a pressure relief valve is quite simple, and the generic outline of this kind of valve can be found in Fig. 3.

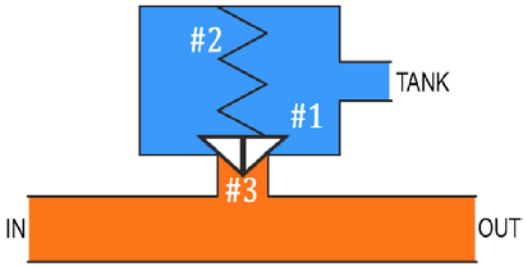


Fig. 3 schematic view of a pressure relief valve

In particular, the aforesaid PRV is schematically composed by a moving element, called poppet (#1), pushed down by a spring (#2) that holds it in position inside a seat (#3). When the pressure of the supply line (orange zone of Fig. 3) is lower than a defined value (cracking pressure) the preload of the spring prevents the movement of the poppet, so the oil cannot flow. As the cracking pressure is reached, the valve poppet slides back in a way which is proportional to the amount of force generated; this opening value can be set by preloading the spring with a desired degree of compression: usually this happens through a screw fitted over the spring itself.

II. Experimental Setup

In order to simulate a generic real-life hydraulic circuit, an experimental test bench has been properly developed: the prototype of a generic PRV has been created to generate a set of experimental data used to compare the simulations results.

The relationship between the stroke (x) of the poppet, the mass flow of the oil (Q) and the pressure drop (Δp) inside the valve orifice is calculated by the following equation:

$$Q = C_d A(x)\sqrt{\frac{2\Delta p}{\rho}} \tag{1}$$

where A(x) represents the area of the orifice available for the oil flow; it must be noted that this variable area can be calculated as follows:

$$A(x) = \pi d x \tag{2}$$

The main geometrical dimensions of the valve are reported in Fig. 4; in particular, d represents the inlet diameter of the abovementioned valve.

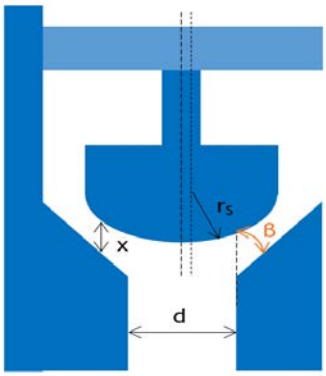


Fig. 4 PRV main characteristic dimensions

It is important to clarify the meaning of the coefficient indicated as C_d in (1): it is indeed the discharge coefficient, which represents the role of the geometry of the valve. Hence it indicates the capability of the device itself to allow a defined flow of oil in base to the interaction between pressure and density of the fluid (together with the available outflow area). This coefficient was first studied experimentally by Richard von Mises [1-2] and then evaluated analytically by Mikhail I. Gurevich [3]. Since the experimental test bench uses a fixed displacement pump, the oil flow regulation is made by a series of valves, making it possible to vary the flow rate. Then the manual movement of the poppet through a screw defines the pressure that will be measured across the valve. In this case, given that a fixed spacer is used instead of the spring, the position of the poppet is measured by means of proportion between the pitch of the screw and the amount of turns of the screw itself. This generates a setting which is as stationary as possible, in order to obtain an experimental setting consistent with the CFD simulations. As the screw is turned, the poppet rises from the closed position to the fully opened one, making it possible to record the amount of pressure created for the specific flow rate and stroke. The values are recorded in an Excel® table and plotted to obtain a Δp-Q characteristic, which allows the calculation of the discharge coefficient. Pressure is measured via digital manometers and flow rate is measured with a flow meter.

The same results are searched in the CFD simulations and in the simplified model used in MATLAB-Simulink®, hence it is required to build a proper and reliable setup, capable to prove that the simulations can substitute the real-life prototypes.

III. CAD Geometry, Mesh Generation and CFD Simulation Settings

The geometry of the valve has been modelled in SolidWorks® 2015, used to generate an assembly made out by the valve body and the poppet. The assembly has been saved with the poppet located in four different positions[1] , in order to reproduce the situation experimented with the test rig (schematically shown in Fig. 1); it must be noted that the main geometrical dimensions used to parameterize the valve are highlighted in Fig. 4. An algorithm was used to extract the fluid volume from the solid geometry (shown in Fig. 5)

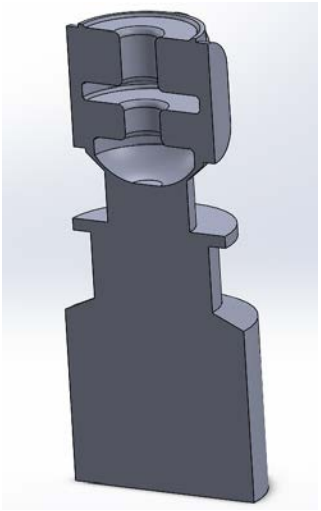


Fig. 5 example of fluid volume extracted from the solid geometry of the pressure relief valve

Once the model is defined, it is needed to generate a mesh that can be used by OpenFOAM® in order to execute the fluid dynamic simulations. A tetrahedral mesh has been chosen, since it is reliable and capable to generate the calculation grid without an excessive computational effort (Fig. 6).

The mesh diagnostic tool of OpenFOAM® confirms the good quality of the cells and of the overall grid, as no interpenetrating cells or misaligned normal face vectors are found. The valve is fairly compact, but in order to obtain a good precision while executing the fluid dynamic calculations the number of cells used, since the domain is three-dimensional, ranges from 3 to 6 million. As shown in Fig. 7, the grid is made finer in the area of the orifice, since the very limited stroke (the useful value of poppet displacement x is always below 1 [mm], even in maximum mass flow conditions) generates a very little outflow area.

1 It must be noted that the number of steps tested must be defined in base to the linearity of the considered phenomenon and to the amount of resolution required to optimally fit the experimental data, but a linear characteristic can just require a small amount of points.

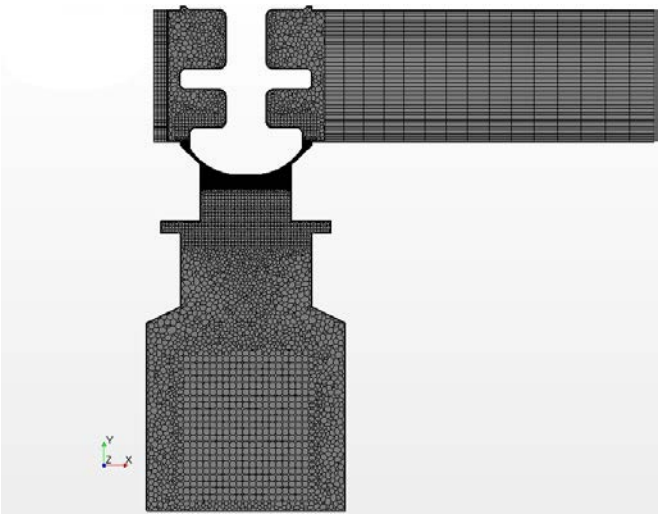


Fig. 6 example of result of the meshing process for PRV

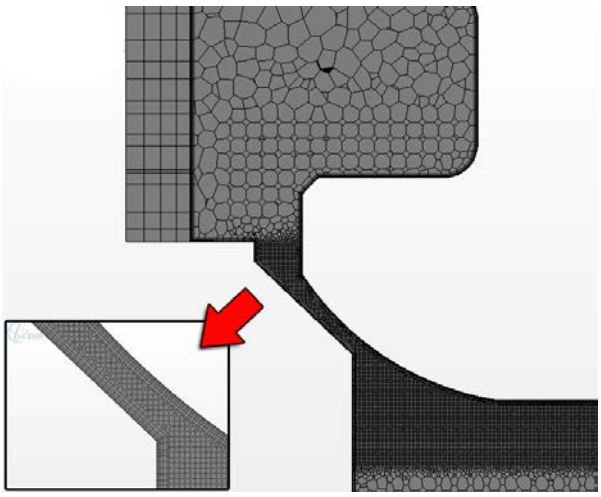


Fig. 7 detail of the fine mesh around the orifice and snapshot of the orifice on the upper right corner

Having enough cells is paramount to allow a proper calculation of the strong gradients occurring inside the orifice, but at the same time a too fine subdivision of the domain would increase too much the computational time, wasting precious resources. The simulations are set as follows:

1) Steady
2) Static
3) Single-phase
4) Turbulent and laminar models are tested
5) Three dimensional
6) Constant density
7) PIMPLE solver[2]

With the aforesaid conditions we assume that the valve is in a stable equilibrium condition, without incurring in the Hopf bifurcation phenomenon [16].

2 The PIMPLE solver algorithm is contained in OpenFOAM software tools and calculates the pressure and the velocity fields of the simulated flow through following steps, correcting the values of such fields at every iteration steps [4].

The characteristics of the oil chosen are typical of an aeronautical (and to a certain degree are also close to that of an industrial machine) fluid:

1) $\rho = 900$ [kg/m3]
2) $\nu = 25$ [cSt]

The pressure field is integrated all over the poppet to obtain the distribution of forces required to size the spring and properly evaluate the stroke of the poppet itself.

IV. CDF Results

The results obtained through the CFD simulations, as shown for instance in Fig. 8-10, have been compared with experimental results in order to evaluate the accuracy of the proposed approach. Inlet pressure reaches the maximum value where the velocity of the fluid is nearly zero, while decreases as the velocity of oil increases.

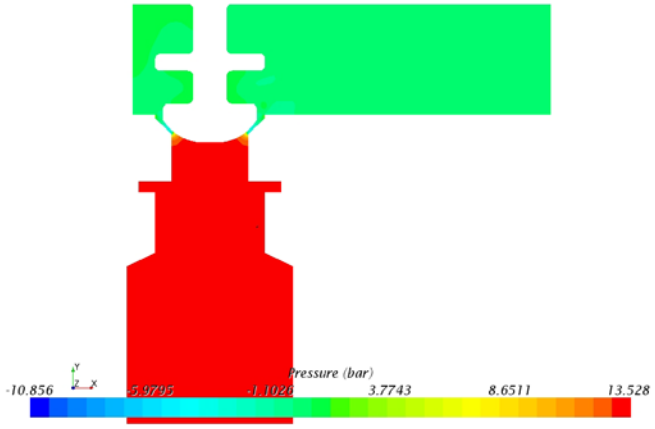


Fig. 8 pressure profiles calculated by CFD simulation:
Q = 64,97 [l/min] - Poppet Stroke x = 0,748 [mm]

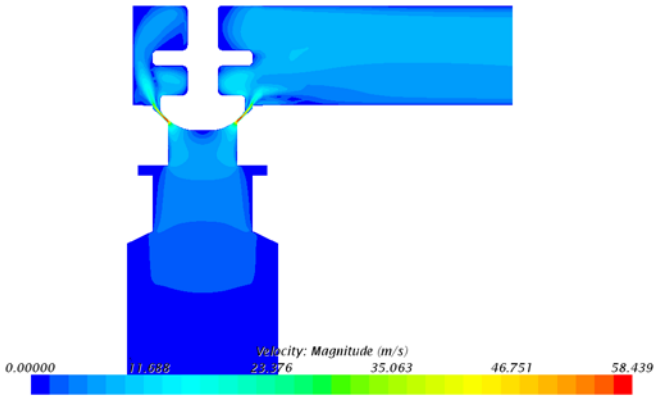


Fig. 9 velocity profiles obtained by CFD simulation:
Q = 64,97 [l/min] - Poppet Stroke x = 0,748 [mm]

It is important to point out the negative values of absolute pressure obtained in the initial simulations. Such results are due to the fact that simulations are run as single phase ones, so it is not possible to accurately capture the phenomenon of bubble formation in certain areas of the valve [5-8].

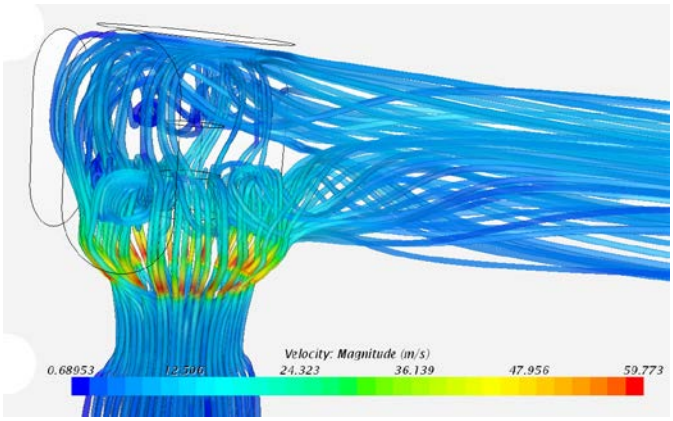


Fig. 10 detail of streamlines obtained by CFD simulation:
Q = 64,97 [l/min] - Poppet Stroke x = 0,748 [mm]

Given that the so obtained CDF simulations provide a detailed evaluation of pressure and force fields acting on the poppet taking into account also effects due to viscous and flow forces (as shown, for instance, in Fig. 11), it is possible to correlate the different PRV operating conditions (i.e. stroke of the poppet x, oil mass flow Q, pressure drop Δp, average oil temperature, etc.) with the respective fluid dynamic forces and, therefore, provide a more accurate modeling of these phenomena by means of a dedicated multi-input LUT.

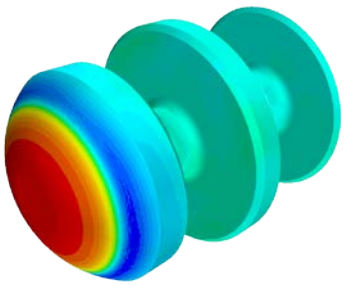


Fig. 11 example of poppet pressure distributions given by CFD:
Q = 64,97 [l/min] - Poppet Stroke x = 0,748 [mm]

V. MATLAB-Simulink® Model

Once defined the force and pressure fields acting on the poppet, it is important to analyze the dynamic behaviors of the mechanical components of the valve (i.e. the moving poppet driven by the spring). The properties of these components (poppet inertia, spring stiffness, viscous damping and dry friction) will determine the dynamic behavior of the valve, and its regulation field (e.g. acting on the spring preload the PRV's cracking pressure changes). To this purpose, a simplified numerical model has been developed in MATLAB-Simulink® simulation environment (shown in Fig. 12): it is able to simulate the dynamic behavior of the aforesaid PRV taking into account the effects of the mechanical properties of its components. It must be noted that it is a lumped parameter numerical model (i.e. spatially distributed entities of the examined physical system are condensed into a specific set of discrete elements, simplifying its real nature).

A data file is used to initialize the variables representing the geometry of the valve studied. At the same time two different calculation paths can be followed: indeed, forces used by the equations of dynamics can both be modeled (starting directly from physical or mathematical models given by literature) or interpolated by means of suitable look-up tables (LUT) containing the values calculated by the CFD simulations (e.g. the discharge coefficient C_d related to different values of poppet stroke x, oil mass flow Q and pressure drop Δp).

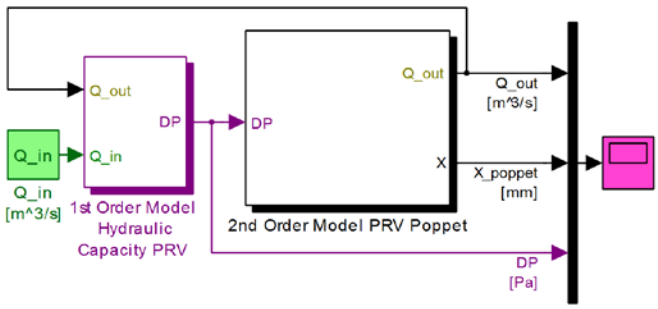


Fig. 12 MATLAB-Simulink block diagram of the PRV

The main forces used in the model are the flow force, the static pressure force, the dry friction (simulated by means of the Borello's model [9]) and the inertial force due to the mass of the poppet. The CFD is able to consider the contributions due to pressure as a single term (inclusive of the effects of static pressure, friction and flow forces), while the direct modelling needs to take into account static pressure and dynamic pressure as two different contributions. It must be noted that, according to [10-12], the abovementioned flow force represents the reduction in pressure operated by the motion of the fluid inside the oil, which decreases the overall value of pressure to an extent that acts as a recall force, limiting the stroke of the valve poppet.

Figure 13 represents the hydraulic capacity block, that it is required to tailor the simulation behavior to make it closer to reality; this block takes into account the amount of oil used and the overall compressibility of the oil itself (in fact some microscopic air bubbles could be present inside the fluid, reducing the global bulk modulus of the fluid).

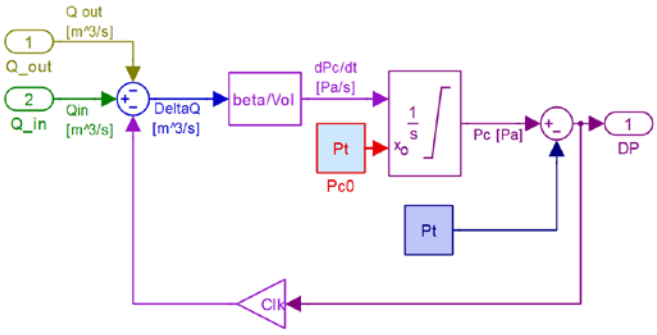


Fig. 13 MATLAB-Simulink block diagram of the "1st Order Hydraulic Capacity PRV" subsystem

Figure 14 shows the Simulink block diagram simulating the fluid-mechanic dynamic response of the PRV poppet; it is implemented by means of a second order nonlinear dynamic model that calculates the poppet displacement (x) and the oil flow drained across the valve (Q_{out}) as a function of differential pressure Δp, poppet spring preload (FRV0), geometrical characteristics of the valve and inlet oil flow (Q_{in}).

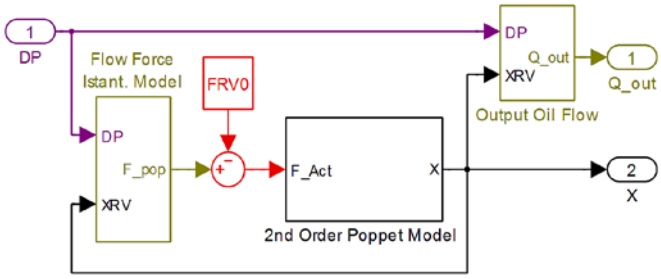


Fig. 14 MATLAB-Simulink block diagram of the "2nd Order Model PRV Poppet" subsystem

The "2nd Order Model PRV Poppet" subsystem is composed by four blocks shown in Fig. 15, 16, 17 and 18.

Figure 15 shows the "Flow Force Instantaneous Model": this simplified numerical model calculates the instantaneous value of the global fluid dynamic force acting on the sliding poppet F_{pop} as follows:

$$F_{pop} = F_p - F_{fl} \tag{3}$$

where F_p and F_{fl} represents, respectively, the static pressure force (due to differential pressure acting on the poppet effective area) and the said flow force.

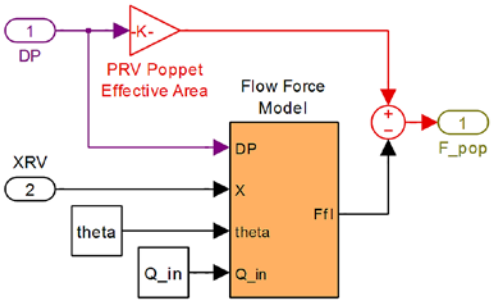


Fig. 15 MATLAB-Simulink block diagram of the "Flow Force Instantaneous Model" subsystem

Figure 16 shows the implemented flow force model: it is possible to notice the possibility to use the directly modelled or LUT-based setup for the calculations. It must be noted that flow forces are strongly dependent on the shape of the valve and on the mass flow; once defined a variation law for poppet stroke (together with a relation expressing the equivalent orifice area) and having identified the differential pressure Δp, it is possible to outline a different range of forces for big or small strokes. According to [10], in case of small stroke (in this case < 0,2 [mm]) the flow force is modeled as:

$$F_{flow} = 2 \cdot C_d(x) \cdot \Delta p \cdot \cos\vartheta \cdot A(x) \quad (4)$$

where C_d is the discharge coefficient, θ is the outflow angle of the jet produced inside the orifice (seat-poppet area), Δp is the pressure drop through the valve orifice, A(x) is the area of the orifice and x is the stroke of the poppet. Vice versa, as reported in [10-11], in case of larger stroke x the flow force acting on the PRV poppet is modeled as:

$$F_{flow} = \rho \cdot C_d{}^2(x) \cdot Q^2 \cdot \cos\vartheta / A(x) \quad (5)$$

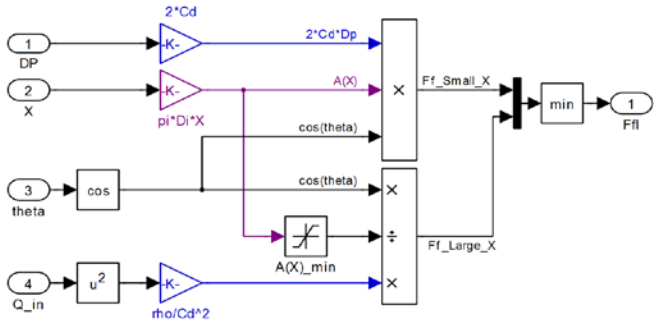


Fig. 16 MATLAB-Simulink block diagram of the "Flow Force Model" subsystem

Figure 17 represents the second order nonlinear dynamic model simulating the behavior of the PRV poppet: this algorithm, developed by Borello and Dalla Vedova in [9], implements a simplified dry friction model which can be fine-tuned to reproduce a result which is coherent to the values obtained by the real-life simulations.

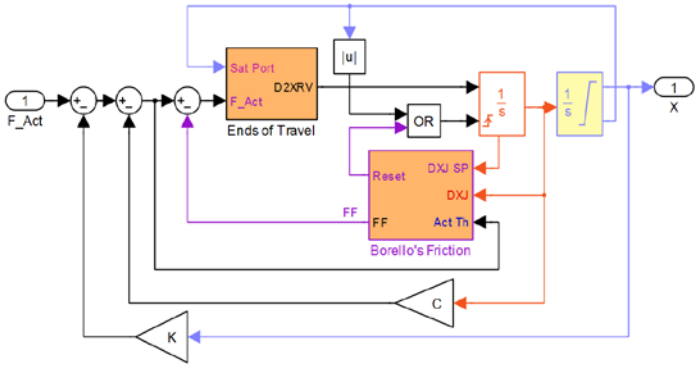


Fig. 17 MATLAB-Simulink block diagram of the "2nd Order Poppet Model" subsystem

It must be noted that friction represents only a very limited contribution, since the poppet is fully submerged in oil, but, in adverse conditions, can determine a jamming of the poppet in a certain position. Operatively speaking, this computational routine, schematically shown in Fig. 18, implements the classical Coulomb friction model and it is able to correctly describing many of its typical effects (breakaway, stop and stick-slip of the poppet, position resolution) as well as their interactions with the eventual mechanical ends of travel.

It must be noted that, if the only saturation port is enabled, the integration will create artifact velocity values, resulting in undesired non-zero values of velocity once the stroke limit is reached. For more clarity, it is necessary to remember that the Coulomb friction model can be generally represented by the following relationships, taking into account the difference between sticking and slipping conditions:

$$FF = \begin{cases} h & if \quad v = 0 \wedge |h| \le FS \\ FS \cdot \mathrm{sgn}(h) & if \quad v = 0 \wedge |h| > FS \\ FD \cdot \mathrm{sgn}(v) & if \quad v \ne 0 \end{cases} \quad (6)$$

where FS and FD represent the friction force in sticking and slipping conditions respectively, h is the active force (i.e. Act Th in Fig. 17) and v represents the relative slipping velocity (i.e. DXJ in Fig. 17).

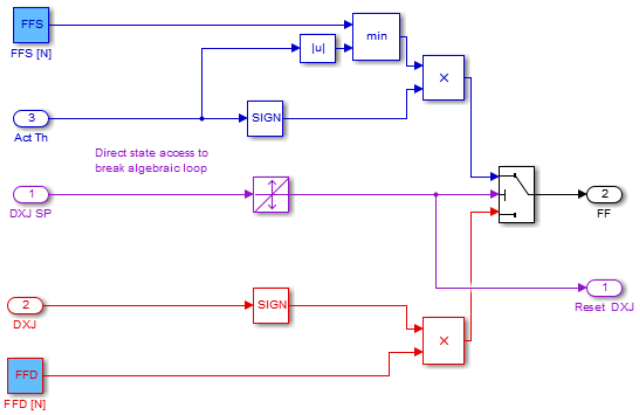


Fig. 18 MATLAB-Simulink block diagram of the "Borello's Friction" subsystem

In order to simulate the behavior of the valve, a quasi-dynamic scheme was chosen: this approach, proposed by Borello and Dalla Vedova in [13], is capable to handle complex systems of non-linear equations with reasonable ease, identifying by means of numerical calculation the corresponding stationary conditions. The system uses the same equation as the dynamics, but the real physical meaning of the damping constants is neglected, in fact they are only used to allow a fast and numerically stable convergence to the regime condition of the system studied. It must be noted that, as shown in [14], such numerical scheme can find application also in other disciplines, like structural calculations. This kind of algorithm hence ensures fast convergence times (thanks to a suitable configuration of the constants), but also a high quality of the results, even if the transient loses its physical meaning.

VI. MATLAB-Simulink® Results

Figure 19 shows, for instance, the dynamic response of the proposed lumped numerical model: it is calculated for a mass flow Q (that flows through the valve) of 64,97 [l/min]. The parameters of the MS numerical model taken into account for the simulations are reported in Table 1.

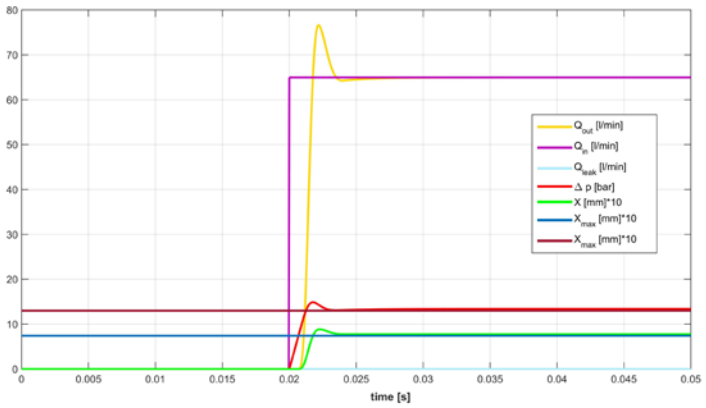


Fig. 19 MATLAB-Simulink simulation results for Q = 64,97 [l/min]

Table 1: Main parameters of the MATLAB-Simulink model

Parameter	Physical variable	Value	Units
K	spring stiffness	9150	[N/m]
C	damping coefficient	50	[N·s/m]
m	poppet's mass	0,01	[kg]
Dt	integration time step	1e-6	[s]
t	total simulation time	0,1	[s]

The color scheme adopted for the simulations performed in Simulink (according to Fig. 19) is shown in Table 2.

Table 2: Color scheme of MATLAB-Simulink model

Color	Physical variable	Units
Yellow	output mass flow	[l/min]
Magenta	input mass flow	[l/min]
Cyan	leakage mass flow	[l/min]
Green	stroke of the poppet	[mm]
Red	pressure at the inlet	[bar]
Blue	cracking pressure	[bar]
Dark Pink (horizontal line)	Related experimental stroke of the poppet	[mm]

As reported in Table 3, all the considered cases (i.e. all the considered mass flows sets tested by authors) are able to fit well the experimental results obtained. This means that the direct model of the flow force implemented in MATLAB-Simulink® is able to synthetically capture the behavior of the fluid inside the valve.

Table 3: Comparison between experimental results and corresponding MATLAB-Simulink® simulations

Q [l/min]	p_{sim} [bar]	x_{sim} [mm]	p_{exp} [bar]	x_{exp} [mm]	%p	%x
64,97	13,41	0,7757	13,56	0,748	1,11	3,70
49,6	11,89	0,6483	12,09	0,629	1,65	3,07
29,96	10,19	0,4765	10,22	0,464	0,29	2,69
9,96	8,22	0,2912	8,4	0,2903	2,14	0,31

A following work will consist in comparing the MATLAB-Simulink® model enriched with the LUTs with the equation-based simulation. This will make it possible not only to compare the results, but also to define which is the best and most effective method to tackle the problem.

VII. Conclusions

A 3D model of the full geometry of the valve has been developed to predict the distribution of pressures (hence forces) inside it to make an optimization process possible. The CFD calculations have been run with an open source software, OpenFOAM®. The mesh has been refined to take into account different strokes and various levels of accuracy of the results, in order to obtain a representation of the domain as accurate as possible. The meshing procedure was based on a trials and errors procedure, driven by previous examples found in literature [15]. The results show a good behavior of the MATLAB-Simulink® model for both the fully CFD modelled and CFD/Simulink interactive model. As a more accurate representation of the valve through the CFD will be reached, the sole MATLAB-Simulink® model could be enough to drive the very initial stages of the design, providing the preliminary information necessary to aim the successive detailed optimization procedure based upon extensive use of CFD-driven calculations. Clearly a full and accurate comprehension of the best geometry fitting the characteristics of flow rate and pressure can be reached only via the CFD, as a precious and cost-effective alternative to a purely practical approach.

The results obtained by the initial CFD setting require a deeper analysis to clarify the need to use a more complex and rich simulation, with the use of a multiphase model to describe the dynamic instability phenomenon that usually incurs a lower flow rates and small lifts of the poppet [17].

In particular, the multiphase survey could also raise the need to introduce a cavitation model, thus making the model much more detailed and complex. To this purpose, authors are intended to continue this research developing more accurate numerical simulations based upon Matlab-SIMULINK® models enriched with look-up tables and, then, comparing them with a pure CFD dynamic simulation, created taking into account a multiphase fluid.

REFERENCES

[1] R. von Mises, "Berechnung von ausfluß und ueberfallzahlen," *Zeitschrift des Vereine Deutscher Ingenieure*, 1917 (in German).

[2] R. Von Mises, "The calculation of flow coefficient for nozzle and orifice", VDA, vol. 61, pp. 21–23, 1917.

[3] M. I. Gurevic, *Theory of jets in ideal fluids*. Academic Press, 1965.

[4] Nekropolistov Society. *OpenFOAM User Guide – Online*, 2016.

[5] S. I. Bernad, and R. Susan-Resiga, "Numerical model for cavitational flow in hydraulic poppet valves," *Modelling and Simulation in Engineering*, 2012.

[6] P. Zwart, A. G. Gerber, and T. Belamri., "A Two-Phase Model for Predicting Cavitation Dynamics," in *ICMF 2004 International Conf. on Multiphase Flow*, Yokohama, Japan, 2004.

[7] J. Sauer, G. H. Schnerr, and W. Yuan, "Modeling and computation of unsteady cavitation flows in injection nozzles," *Mecanique et Industries*, vol. 2, no. 5, pp. 383–394, 2001.

[8] A. C. Bayeul-Lainé, S. Simonet, D. Dutheil, and G. Caignaert, "Numerical study of cavitating flow inside a flush valve," *Mecanique et Industries*, vol. 12, no. 6, pp. 421–431, 2011.

[9] L. Borello, and M. D. L. Dalla Vedova, "A Dry Friction Model and Robust Computational Algorithm for Reversible or Irreversible Motion Transmission," *International Journal of Mechanics and Control (JoMaC)*, vol. 13, no. 2, pp. 37–48, 2012. ISSN: 1590-8844.

[10] N. Herakovič, "Flow-force analysis in a hydraulic sliding-spool valve," Strojarstvo, vol. 49, no. 3, pp. 117–126, 2007.

[11] M. Šimic, and N. Herakovič, "Reduction of the flow forces in a small hydraulic seat valve as alternative approach to improve the valve characteristics," *Energy Conversion and Management*, vol. 89, pp. 708–718, 2015.

[12] N. Herakovič, J. Duhovnik, and M. Šimic, "CFD simulation of flow force reduction in hydraulic valves," *Tehnicki Vjesnik*, vol. 22, no. 2, pp. 453-463, 2015.

[13] L. Borello, and M. D. L. Dalla Vedova, "Pseudo-dynamic simulation method solving hydraulic stationary problems," *International Journal of Mechanics and Control (JoMaC)*, vol. 8, no. 2, pp. 43–52, 2007. ISSN: 1590-8844.

[14] L. Borello, and M. D. L. Dalla Vedova, "A pseudo-dynamic approach to a simple example of structural static problem," in *XX AIDAA National Congress*, Milano (Italy), 2009.

[15] L. Pace, M. Ferro, F. Fraternale, A. Caimano, P. Maggiore, and M. D. L. Dalla Vedova, "Comparative analysis of a hydraulic servo-valve," *International Journal of Fluid Power*, vol. 14, pp. 53–62, 2013. ISSN: 1439-9776.

[16] C. Bazsó, and C. J. Hős, "An experimental study on the stability of a direct spring loaded poppet relief valve," *Journal of Fluids and Structures*, vol. 42, pp. 456–465, 2013.

[17] P. Moussou, R. Gibert, G. Brasseur, C. Teygeman, J. Ferrari, and J. Rit, "Instability of pressure relief valves in water pipes," *Journal of Pressure Vessel Technology*, vol. 132, 2010.

Matteo D. L. Dalla Vedova received the M.Sc. and the Ph.D. from the Politecnico di Torino in 2003 and 2007, respectively. He is currently assistant researcher at the Department of Mechanics and Aerospace Engineering. His research activity is mainly focused on the aeronautical systems engineering and, in particular, is dedicated to design, analysis and numerical simulation of on board systems, study of secondary flight control system and conception of related monitoring strategies and developing of prognostic algorithms for aerospace servomechanism.

Paolo Maggiore is a professor at the Mechanical and Aerospace Engineering Department of Politecnico di Torino, that joined in 1992, where he teaches aerospace general systems engineering. Currently his students are involved in projects ranging from hydrogen fuel cell powered airplanes and UAVs, and health monitoring of flight controls, to multi-disciplinary design optimization of aerospace systems design.

Giorgio Riva received the M.Sc. from the Politecnico di Torino and Milano in 2016. He is currently CEO in Camar S.p.A., an Italian company working in the field of engineering consulting for industrial workholding and hydraulic systems. He occasionally cooperates with the Politecnico di Torino inside the Department of Mechanics and Aerospace Engineering. His work deals with many Aeronautical and Mechanical industries all over Italy, whose plants require specific solutions to enhance and optimize the productivity of their CNCs, presses and tools.