Preliminary Analysis and Optimization via CFD of a Liquid Hydrogen Pressure Regulating Piston Valve

Journal of Physics: Conference Series

**PAPER** • **OPEN ACCESS**

# Preliminary Analysis and Optimization via CFD of a Liquid Hydrogen Pressure Regulating Piston Valve

To cite this article: Arash Safaei *et al* 2024 *J. Phys.: Conf. Ser.* **2716** 012053

# Preliminary Analysis and Optimization via CFD of a Liquid Hydrogen Pressure Regulating Piston Valve:

**Arash Safaei, Matteo D.L. Dalla Vedova, Paolo Maggiore**

Department of Mechanics and Aerospace Engineering, Politecnico di Torino, Italy

Email: arash.safaei@polito.it

**Abstract**. Due to their high reliability and precision, piston valves are frequently used for pressure regulating applications. Particularly in the aerospace industry, where cryogenic fluids such as liquid hydrogen are frequently used, the design and operation of piston valves become crucial. The current state of advancement of this technology in the cryogenic field is still in its early stages, owing to the difficulties in designing such complex systems in harsh environments. This justifies the need for further in-depth studies and analysis using CFDs tools and predictive models. In order to ensure an optimal and efficient use of a piston pressure regulating valve in cryogenic environment, it is necessary to understand the strengths and limitations of this technology in an extreme thermal and mechanical condition. The presented work concentrates therefore on a preliminary analysis and optimization of a piston valve operating in liquid hydrogen flow field, for pressure regulating applications. Particular focus will be dedicated to the overall dynamics of the main body of the piston, in terms of robustness and controllability of the desired response of the system. The dynamics of the piston within an extremely low-viscous flow, as well as the thermodynamic and fluid dynamic aspects of the valve system, will be discussed. Simulations of the flow field will be performed through CFD tool, crossing the results with the dynamics of the simulated system response through and implemented Simulink model. The obtained results will be then critically analysed in order to suggest possible optimization of the valve in the locations where the system is most affected from a thermal and mechanical standpoint.

**Keywords**: Aerospace systems, Aeronautical systems, Pressure regulating valve, Liquid hydrogen, Low-viscous fluids, Cryogenic fluids, Non-linear modelling, Simplified fluid dynamic numerical models, CFD Analysis, Fluid dynamic optimization.

## 1. Introduction

Piston valves play a crucial role in pressure regulating mechanisms within several industries, such as aerospace, aeronautics and energy. Their ability to precisely control fluid pressure is essential, especially when working with cryogenic fluids such as liquid hydrogen, which is characterized by extremely low temperatures and unique fluid dynamic characteristics. However, the use of piston valves in such demanding environments presents a number of complexities and significant challenges, predominantly due to the complexity of manipulating cryogenic fluids.

Liquid hydrogen, which is commonly used as a rocket propellant in aerospace propulsion applications, behaves very differently compared to fluids at room temperature. At cryogenic temperatures, the gases transform into extremely dense liquids, and their physical properties such as density, viscosity, and thermal expansion, are drastically altered.

The presented work focuses therefore on performing a preliminary analysis and optimization via CFD of a piston valve operating with liquid hydrogen, to address the complex fluid dynamic challenges posed by the considered cryogenic fluid. A simplified model of the dynamic response of the piston in the optimized and non-optimized case is then presented, comparing the related solutions in terms of efficiency and performance of the valve's regulating mechanism.

## 2. Problem analysis and optimization

The analysis and optimization of the piston valve's simplified architecture (Fig. 1) performed in this study is based on fundamental fluid dynamic principles and the physics of flow crossing a narrow section. To optimize a conventional pressure regulating valve for cryogenic applications, it is crucial to be aware of its constituent elements and their respective weaknesses and strengths.

The occurrence of fluid recirculation zones and significant localized pressure reductions across a pressure regulating valve is a crucial issue with far-reaching thermal and dynamic consequences for the overall system and the performance of the regulating mechanism. These zones might arise for a variety of reasons, including improper valve design, inadequate flow conditions, or fluid properties that cause turbulence and separation [1]. Such phenomena can lead to localized heat transfer challenges within the system, which can affect structural integrity and functionality of the valve and its surrounding components [2].

For the reasons listed above, it is essential to design valve architectures and mechanisms that can mitigate the impact of the reported fluid dynamic criticalities.

As a starting point of the presented analysis, a pressure drop of 23 bar is applied to the entire system using liquid hydrogen as operating fluid. The initial geometry of the piston valve, the pressure drops at the boundary and the no-slip wall condition allow the resolution and closure of the fluid dynamic problem. The most essential parameters are then computed, including the total force acting on the piston, $F$, as well as the pressure and velocity fields across the piston's body, which are derived from the CFD's analysis.

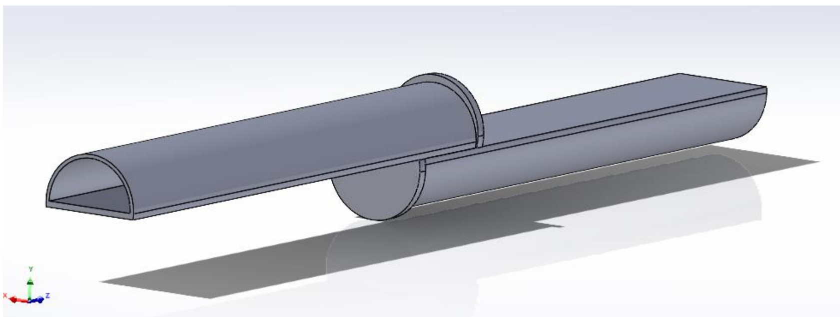

*Figure 1: 3D Model of the simplified valve's geometry.*

Determining the total force $F$ acting on the piston's body is essential for calculating the dynamic response of the system. This force parameter is the input of the Simulink-implemented dynamic model; therefore, its accurate estimation through fluid dynamic CFD simulations is crucial for ensuring the accuracy and reliability of the modeling results.

As far as concerns the body of the piston, an efficient fluid dynamic and mechanical optimization regards the thickness of its stem. One advantage in having a thick piston stem is that the traction force that the flow exercises on the piston is lower than that with a thinner stem, because the area of the circular crown on which the flow acts is smaller. Thus, the piston is less stressed and has a longer life.

This is why the presented simplified architecture is made of a central piston body conceived with a robust and well-designed geometry, in order to let the cryogenic fluid to cross it without encountering sharp edges or significant geometry variations, which could lead to separation or recirculating zones in the proximity of the crown orifice which governs the pressure reduction.

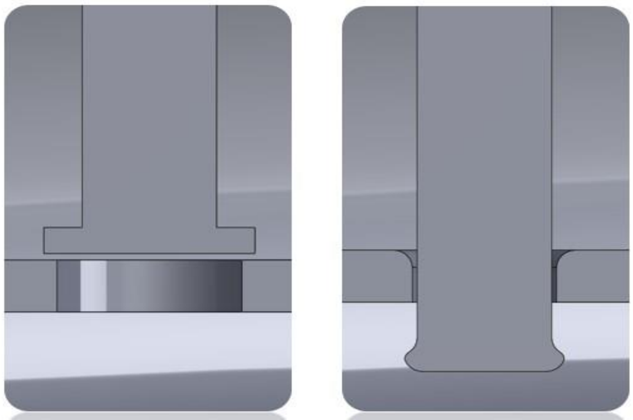

*Figure 2: Non-optimized piston body (on the left) and optimized one (on the right).*

As it will be shown by the results of the CFD analysis, the optimized solution of the piston with body passing through the circular crown orifice has a lower total force acting on it, yet much more stable over time than the total force acting on the classic configuration with the piston pushing toward the orifice as shown in Fig. 2 on the left. Having a constant force acting on the piston implies having less oscillations during the piston pressure regulating phase, and therefore better performances.

### *2.1. Numerical Model*

In valves design, it is not conventional to install dampers owing to the fact that the action of the flow itself, specifically the fluid dynamic resistance that is generated along with the viscous effect, can already be regarded as a damping action [3].

The simplified model of the optimized pressure regulating valve here presented can be idealized as a mass-damper-spring system, where the mass is represented by the piston body itself, the damper contribution comes from the viscous action of the flow that passes through the valve and the spring is represented by the physical spring which controls the position of the piston.

The overall system can be modeled therefore by a differential equation of the second order, with the classical expression:

$$M\ddot{x} + C\dot{x} + Kx = F_{tot} \tag{1}$$

where $M$ represents the mass of the piston, $C$ the viscous coefficient, $K$ the stiffness of the control spring, while the variable $x$ represents the position of the piston under the action of the total force $F_{tot}$.

The purpose of the numerical implemented model is thus computing the value of the displacement $x$ of the piston during the pressure regulating mechanism, solution of the differential equation reported in Eq. (1).

For the complete set up of the dynamic model, the mass $M$, the viscous coefficient $C$ and the spring stiffness $K$ must be properly set. As far as concerns the mass of the spring $M$, this parameter can be easily computed starting from the geometry and material the piston is made of, which in the case of cryogenic environment could be chosen between Inconel alloys and austenitic stainless steels [4][5].

The static model of the system gives the value of the stiffness $K$, by setting a starting suitable equilibrium position $x_{eq}$ of the piston:

$$x_{eq} = \frac{F_{tot}}{K} \quad \rightarrow \quad K = \frac{F_{tot}}{x_{eq}} \tag{2}$$

The last parameter that is needed to close the dynamic problem set up is the viscous coefficient $C$. This parameter can be determined with accuracy only experimentally or through CFD, as it depends on complex physical phenomena [6].

The Simulink implemented model is shown schematically in Fig. 3. It is important to emphasize that it is not necessary to have a complex model in order to describe in a preliminary analysis the dynamics of the piston within a liquid hydrogen flow, since the objective of the presented study is to provide a comparison in terms of performance and robustness of the valve between an optimized and not-optimized architecture from a fluid dynamic and dynamic point of view.

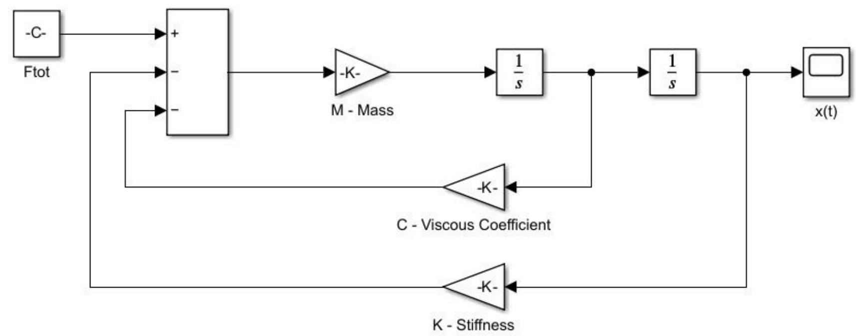


*Figure 3: Simulink simplified model of the dynamics of the piston.*

The complete dynamic model is then solved through waterfall integrations directly on Simulink, starting from the following differential equation obtained starting from Eq. (1):

$$\frac{d^2x}{dt^2} = \frac{1}{M}(F_{tot} - C\dot{x} - Kx) \tag{3}$$

## 3. CFD Simulation

The proposed CFD-based optimization involves thus modifying the traditional design of a piston regulating valve to enhance its efficiency and reliability in handling low-viscous fluids like liquid hydrogen.

Special attention is needed for the description of the boundary layer; since the capture of the viscous effect on the wall is fundamental to ensure the convergence and fluid dynamic accuracy of the simulations, a sufficient number of layers must be provided.

### *3.1. Geometry and Meshing*

3D's cross section model of the simplified main body of the valve is shown in Fig. 4. SimScale software is used for the CFD's simulations, from meshing phase to post processing. Patch independent and tetra dominant shell mesh is used as well as tetra/mixed volume meshing with tetrahedrons mesh method. The Prism mesh is used along the entire wall in order to ensure an accurate simulation of the boundary layer. The inflation method adopted is the smooth transition, extending from the first layer with a growth ratio of 1.3 for 20 layers.

Mesh quality values is used to assess the quality of the mesh and to ensure no negative volume exist. Minimum element quality is 0.03 with maximum mesh quality of 1, averaging at 0.88 for a total of 9727093 elements.

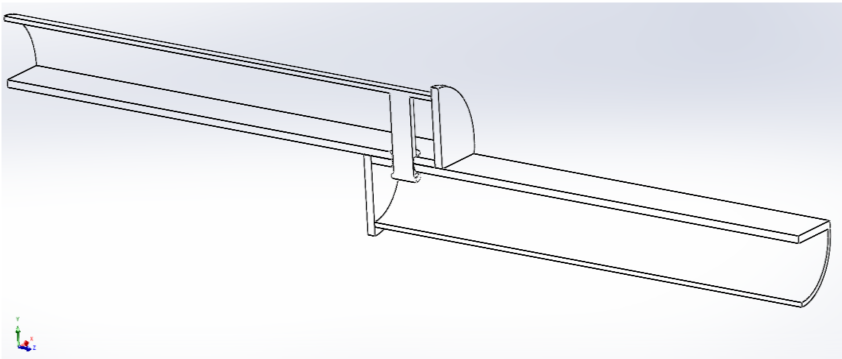

*Figure 4: Cross section of the CAD model.*

### *3.2. Solver Setting*

SimScale software is used to solve Reynolds-Averaged steady state Navier-Stokes equations with k-Omega SST turbulence model, with an inviscid flow. The wall is set to no-slip, with an inlet and outlet pressure driven boundary conditions.

The choice of the turbulent model is decisive for the quality of the results. In CFD problems, k-ω and k-ε are the predominant most used turbulent models. The main difference between these valid alternatives is that the k-Epsilon model resolves the viscous layer less precisely than the k-Omega model [7]. Moreover, changing to a different turbulence model cannot rectify a poor-quality mesh, particularly if the alternative model also employs wall functions.

Taking into consideration the benefits and drawbacks of the aforementioned models, the k-Omega SST model is eventually selected to perform simulations as it effectively combines the best of k-ω and k-ε models [8].

## 4. Results and discussion

All performed simulations satisfy the convergence criterion with less than 1000 iterations, as can be seen from Fig. 5. The net residual of the solutions ranges from the order of magnitude of $10^{-3}$ to $10^{-5}$, thus the results of the CFD can be considered practically steady with minor fluctuations.

The performed simulation shows and enhance how crucial is the importance of the optimization performed on the piston's body geometry on the fluid flow across the valve. The geometrical conformation of the piston and the circular crown that surrounds its central body through the orifice is able to maintain the flow in the form of a column-shaped jet, as can be noticed in Fig. 6. This is due to the fact that the fluid flow is forced to follow the tiny but long space between the wall of the orifice and the stem of the piston, from upstream to downstream. Therefore, the action of this particular geometrical configuration is to contribute to the laminarization of the flow, avoiding a substantial pressure recovery as usually happens after a flow through a single orifice [9], and to ensure a continuous and well distributed loss of pressure from the inlet to the outlet.

By performing the simulation through Simulink, and thus solving the dynamic differential model of the piston, repeated oscillations that asymptomatically approach the convergence value are then observed. This is because the only considered dampening contribution of the analyzed system is the viscous action of liquid hydrogen, which is not sufficient to dampen the motion of the piston.

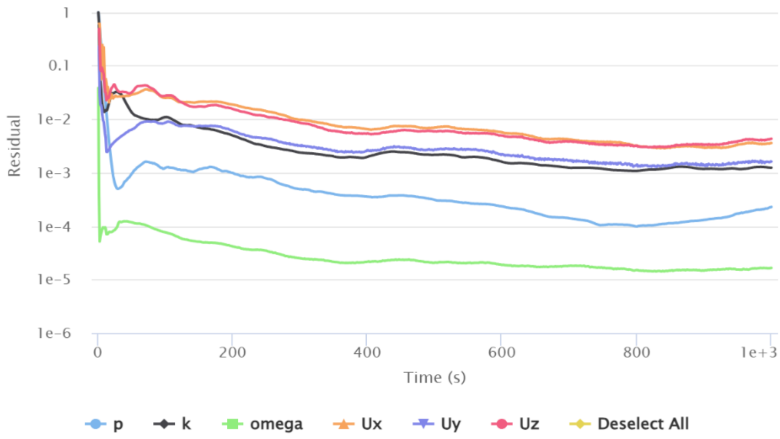


*Figure 5: Convergence Plot of the residuals of the simulation performed with SimScale.*

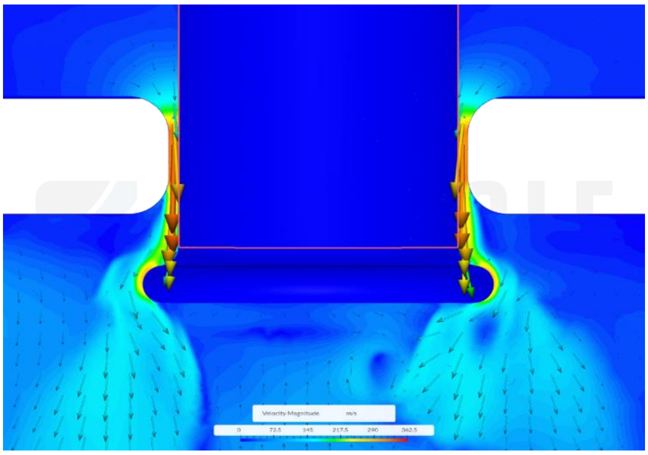

*Figure 6: Velocity field of the $LH_2$ flow around the optimized piston body.*

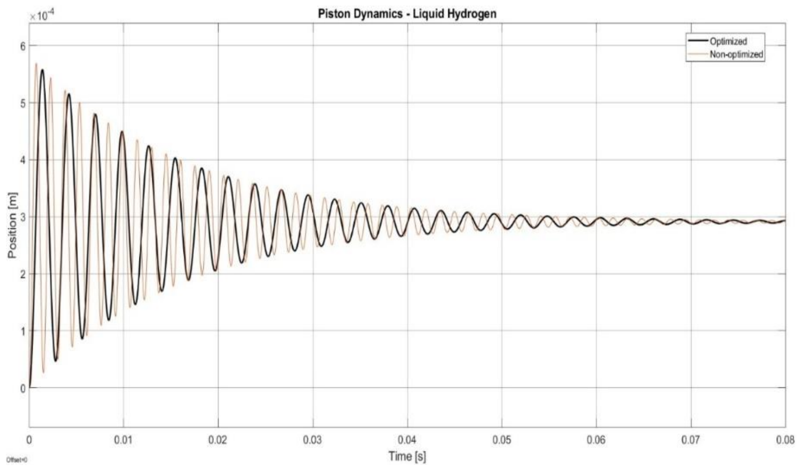


*Figure 7: Comparison between the optimized and non-optimized piston's motion with LH2.*

## 5. Conclusion

A preliminary analysis and optimization via CFD of a liquid hydrogen pressure regulating simplified valve is presented and discussed. The reported fluid dynamic results highlight and stress the critical aspects of the optimized piston's geometry, in terms of pressure drops, recirculating zones and dynamic response of the mechanism. The most important parameters of the fluid dynamic problem are faced and implemented in Simulink through a simplified dynamic model capable of forecasting the response of the piston as a function of its geometry.

The presented simplified valve pressure regulating mechanism shows how the liquid hydrogen flow impacts the dynamic behavior of the piston. The extremely low viscous property of such cryogenic fluid leads to a damping response characterized by many oscillations, which can be managed only through an in-depth optimization of the geometry and dynamics of the system. Furthermore, the performed analysis can be easily adapted to various fluids with different viscosities and pressure ranges, making the presented simplified valve a versatile option for a variety of fluid dynamic applications and scenarios.

## References

[1] Tabrizi A.S., Asadi M., Xie G., Lorenzini G. and Biserni C., "Computational fluid-dynamics-based analysis of a ball valve performance in the presence of cavitation", *Journal of Engineering Thermophysics,* pp. 23, 27-38, February 2014.

[2] Jin H., Zheng Z., Ou G., Zhang L., Rao J., Shu G. and Wang C., "*Failure Analysis of a high pressure differential regulating valve in coal liquefaction*", Engineering Failure Analysis, Vol. 55, pp. 115-130, September 2015.

[3] Turgut S.S., "*Wave Forces on Offshore Structures: Hydrodynamic Damping*", Cambridge University Press, pp. 265-284, July 2014.

[4] Kendall E.G., "*Metals and Alloys for Crygenic Applications – A review*", Defense Documentation Center for Scientific and Technical Information, United States Air Force, January 1964.

[5] MIL-HDBK-5J, "*Metallic Materials and Elements for Aerospace Vehicle Structures*", Department of Defense Handbook, United States of America, January 2003.

[6] Lesieutre G.A., "*Damping in FE Models*", Encyclopedia of Vibration, pp. 321-327, 2001.

[7] Minsheng Z., Decheng Wa, and Yangyang G., "*Comparative Study of Different Turbulence Models for Cavitational Flows around NACA0012 Hydrofoil*", Journal of Marine Science and Engineering, MDPI, 2021.

[8] Menter, F. R., "*Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*", AIAA Journal, vol. 32, no 8. pp. 1598-1605, 1994.

[9] Shivarudrappa K.B., Sargunar E.V.M. and Sakthivadivel R., "*An Experimental Investigation of Pressure Recovery Factor in Discharging Manifolds*", 6th Australasian Hydraulics and Fluid Mechanics Conference Adelaide, Australia, 5-9 December 1977.