602 - CFD ANALYSIS DURING THE DESIGN OF FUEL EQUIPMENT

Marine Robin, marine.robin@safrangroup.com
David Taieb, david.taieb@safrangroup.com

SAFRAN Helicopter Engines
417 avenue du Bearn, 78 200 Buchelay (France)

Abstract

Computational Fluid Dynamics (CFD) simulations are increasingly used to apprehend the hydraulic behaviour of fuel equipment in helicopter engines. However, most problematics at stake involve fluid-structure interactions and remain unreachable for traditional mesh-based CFD approaches. The present study investigates the capability of Lattice-Boltzmann methods to cope with the main fluid-structure applications encountered in fuel systems. The first case involves one-way interactions where the pressure generated by a low-pressure pump impeller is modelled. The second case study covers two-way interactions where the dynamic coupling between a deltaP constant poppet, subjected to pressure loads and a spring force, and a controlled metering valve is computed. In the last case, instabilities of a check valve are reproduced and oscillations eigenfrequencies are correlated with experimental data. XFlow Latice-Boltzmann solver shows good capability to handle all of those complex applications. Obtained results and reference data are in very good agreement with a significant improvement in computational time. Those methods open new perspectives to deal with a large panel of fuel system problematics like gear pumps or fire test scenarii.

1 INTRODUCTION

Computational Fluid Dynamics solvers, especially unsteady Reynolds-Averaged Navier-Stokes (RANS) turbulence approaches, have been widely used in the aeronautical industry over the past decade. They provide a powerful tool to address problematics encountered through the design process or for troubleshooting purpose. Yet, unsteady RANS methods are not adapted to simulate highly transient and turbulent flows. Alternative turbulence approaches like Large Eddy Simulation (LES) allow to capture transient phenomena with more accuracy. However, their application to fluid-structure interaction using Immersed Boundary Methods (IBM) remains complex. As such methods rely on a mesh-based approach, adapting the mesh according to the solid motion while ensuring a good quality mesh is very challenging. The main two techniques are on the one hand deforming meshing techniques wich preserve the mesh topology but are not suited for large structural displacements leading to low quality nodes. On the other hand, remeshing techniques allow larger deformations but their implementation remains difficult and requires more computational time and ressources.

Lattice-Boltzmann Methods (LBM) are breaking with those traditional mesh-based CFD approaches. They open new perspectives to address fluid-structure and structure-structure interactions with high transient flows. The resolution is based on a lattice structure generated inside the whole domain, both fluid and moving solid domains. The lattice nodes are updated and marked every time step to identify those included in the fluid or the solid domain which allows an automatic detection of the moving boundaries. The suppression of the complex and time-consuming mesh generation step has many benefits and opens new opportunites to solve accurately complex industrial cases with moving parts.

2 NOTATIONS

SAFRAN HE: SAFRAN Helicopter Engines
CFD: Computational Fluid Dynamics
LBM: Lattice Boltzmann Method
LES: Large Eddy Simulation
IBM: Immersed Boundary Methods
WALE: Wall-Adapting Local Eddy
FSI: Fluid-Structure Interaction
RANS: Reynolds-Averaged Navier-Stokes
LP: Low Pressure
HP: High Pressure
CPU: Central Processing Unit

3 GENERALITIES ABOUT FUEL SYSTEM

3.1 Fuel system architecture

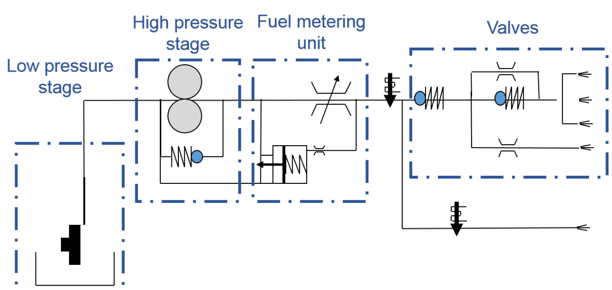

Figure 1: General fuel system architecture

All fuel systems in helicopter engines are almost based on the same architecture with the following sub-systems:

- A low pressure stage used to suck the fuel up in the reservoir and to ensure a sufficient pressure level upstream of the high pressure (HP) pump;
- A high pressure stage used to generate a fuel flow in excess to the fuel metering unit;
- A fuel metering unit (deltaP constant poppet combined with metering valve) used to control the fuel flow injected in the combustion chamber according to the actuator command;
- Various valves like check valve, pressure relief valve...

3.2 Scope of work

CFD simulations have become part of the standard design process of fuel equipment. Classical unsteady RANS solvers are used to assess pressure drops in equipment or to optimize conduct arrangements.
Most of fuel equipment are complex geometries with moving parts involving fluid-structure interactions. Fuel pumps are classified as one-way interaction applications as the solid motion is prescribed by the user. On the other hand, check valves involve two-way interactions as the solid dynamic is subjected to external hydraulic and mechanical forces. The fuel metering unit brings into play both approaches. As traditional CFD solvers based on finite volume methods can hardly capture the dynamic of those problematics, new methods needed to be investigated. A benchmark with a LBM solver has been carried out and is presented in this paper.

4 SIMULATION OF FUEL EQUIPMENT HYDRAULIC BEHAVIOUR

The aim of this work is to demonstrate the capability of the LBM approach implemented in XFlow software developed by Dassault Systemes to predict the hydraulic behavior of the main equipment of the fuel system, whether it be pressure generation, dynamic coupling or more complex phenomena like instability issues. The different test cases presented in the following have been selected to address all three FSI applications previously described.

4.1 Low pressure fuel pump

The first case study is aimed to assess the pressure generated by a low pressure (LP) fuel pump. This technology of pump is considered as half-centrifugal half-volumetric. The rotational speed of the impeller is prescribed and the housing allows the fuel circulation (Figure 2). Note that the impeller is not centered in the housing to ensure suction and compression of the fluid.

The hydraulic behaviour of this type of pump is currently modelled using a research CFD calculation code YALES2 [4] r1.4.2 based on a LES approach combined with an IBM solver. Test bench and numerical results show good agreement, but those simulations are time expensive and the mesh generation step as well as the setup process remain complex. The objective of this benchmark case is at first to show the non-regression of XFlow LBM solver to predict pressure generation and evaluate computational performances.

4.1.1 Numerical model of the LP fuel pump

This section describes the different setup parameters of the numerical model and the lattice structure.

Model description
The 3D CAD model is composed of the impeller included in the fluid domain as depicted in Figure 2.

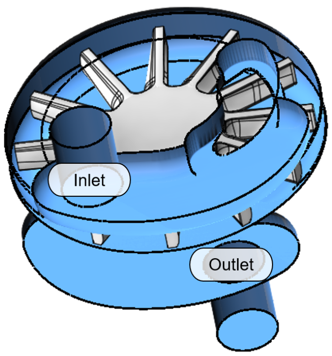

Figure 2: 3D CAD model of the LP fuel pump

This test case is a one-way FSI application as the rotation of the impeller is enforced and specified by the user.
A mass flow rate is prescribed at the inlet and a pressure condition is defined at the outlet of the domain.

Lattice structure

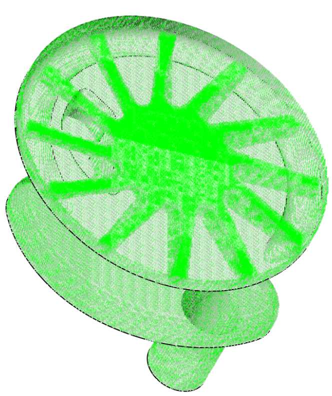

Figure 3: Uniform lattice structure in a cutting plane of the LP pump fluid domain

The space discretization is uniform and a distance of $dx = 5L_{ref}$ is set between each lattice node. L_{ref} defines the small clearance between the impeller and both inlet and outlet bodies. An equivalent resolution is used for YALES2 computation. The final lattice structure is composed of 1 600 000 elements (Figure 3). The lattice is generated in both the fluid and the solid domain which enables the resolution of the solid movement equations.

Computational parameters
The time step is automatically computed to statisfy the CFL condition based on a reference velocity V_{ref} specified by the user and the lattice discretization. V_{ref} is driven by the maximum velocity at the tip of the impeller blades. A time step of 4.0e-7 sec is used for the whole computation. 4 rotations of the impeller are simulated for two functional points (idle power and maximum take-off).
The turbulence model is based on a LES solver fully coupled with a WALE approach to model the sub-grid turbulence and a generalized law of the wall for the wall treatment [1].
The pressure generation of the pump is assessed with kerosene JET-A1 at 35°C:

JET-A1 $35°$C	
Density	786 kg.m^{-3}
Dynamic viscosity	0.00102 Pa.s

Table 1: Fluid physical properties

4.1.2 Pressure generation of the LP fuel pump

The pressure field distribution depicted in Figure 4 shows the suction area on the left side and the discharge area on the right side where pressure values are higher.

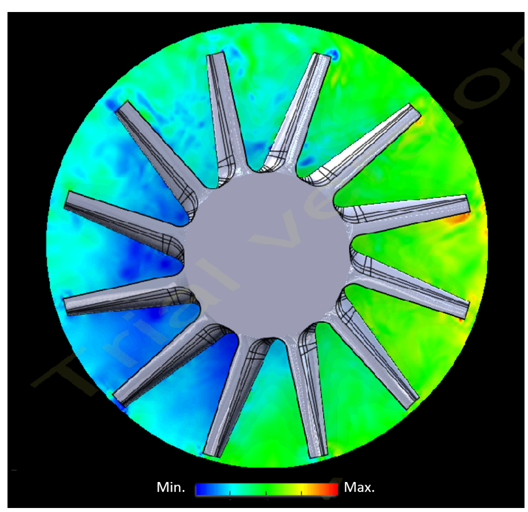

Figure 4: Instantaneous static pressure field distribution around the LP pump impeller

Figure 5 represents the velocity field distribution and points out high velocities in the small clearance at the tip of the impeller blades.

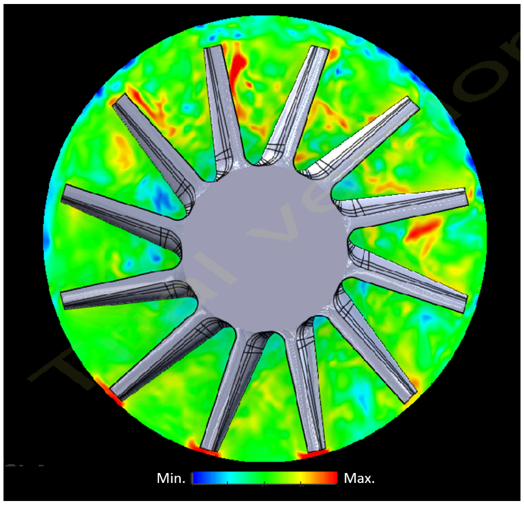

Figure 5: Instantaneous velocity field distribution around the LP pump impeller

XFlow and YALES2 results are in good agreement. The relative difference of pressure generation computed with XFlow and YALES2 is given in Table 2 for idle power and maximum take-off.

Engine speed	ΔP relative difference
Idle power	+ 3.0 %
Maximum take-off	+ 3.2 %

Table 2: Relative difference of pressure generation computed with XFlow and YALES2

Considering the same spatial resolution, results for both CFD codes match very well. Note that pressure generation values are overestimated by 20% compared to test bench results. Indeed, leakage flow is not modelled here as the spatial resolution is larger than the clearance between the impeller and the housing. A smaller lattice resolution close to the regions of interest would enable to get more accurate results.

4.1.3 Computational performances

Computational parameters and CPU time are compared in Table 3 for both simulations. CPU time is significantly lower with XFlow solver compared to YALES2 with limited ressources.

CFD code	Number of cores	CPU time
YALES2 (r1.4.2)	1 024	20 hours
XFlow	12	12 hours

Table 3: Comparison of computational performances between XFlow and YALES2 calculations

This first case study shows that XFlow solver is able to handle one-way FSI application with enforced solid motion and provides accurate results in the case of LP fuel pump with improved computational performances.

4.2 Fuel metering unit

This second case is built to demonstrate the capability of XFlow solver to model dynamic coupling of a whole system. The fuel metering unit is composed of a metering valve coupled with a deltaP constant poppet. Both equipment are constantly interacting with one another. The metering valve position is commanded by an actuator to provide the fuel flow required to the combustion chamber. The metering valve aperture is considered turbulent and the mass flow rate law, under a constant differential pressure ΔP, is defined as follows:

$$Q = CdS\sqrt{\frac{2\Delta P}{\rho}} \tag{1}$$

where Cd is the characteristic discharge coefficient of the metering valve geometry, S the opening area and ρ the fluid density.
The pressure difference is regulated by the deltaP constant poppet which recirculates the flow rate in excess provided by the HP pump. In this particular case, two-way fluid-structure interactions are involved as the deltaP valve displacement is driven by the differential pressure in the fluid domain while the valve motion impacts the fluid dynamics.

This challenging case can not be currently modelled with traditional mesh-based CFD approaches. Besides, thoses methods are not well suited to handle changes in the fluid domain topology as the valve can either be open or fully closed.

4.2.1 Numerical model of the fuel metering unit

Model description
A simplified 2D geometry of both equipment is built to assess the capability of XFlow to handle a dynamic system. The 2D CAD model of the simplified fuel metering unit is depicted in Figure 6.

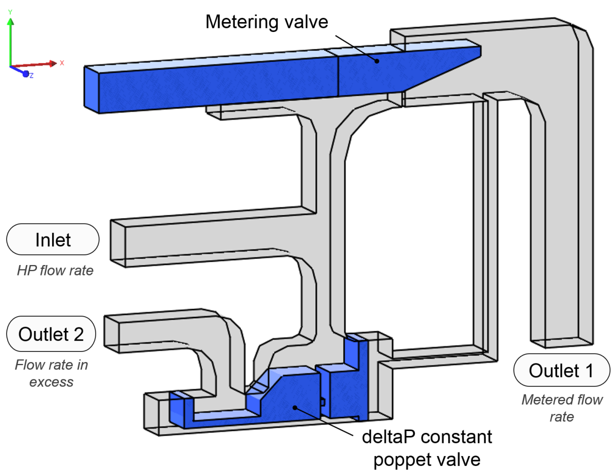

Figure 6: 2D CAD model of the fuel metering unit

The system is composed of the HP flow rate inlet and two outlets. Part of the HP flow rate is injected in the combustion chamber through the metering valve *Outlet 1*, and the other part is returned downstream of the LP fuel pump *Outlet 2* through the deltaP constant poppet valve.

The translation motion along the *x*-axis of the metering valve is enforced. The temporal evolution of the valve opening is detailed in Figure 7.

The metering valve position law is divided into 4 phases:

- **Phase 1:** valve initially closed
  The first phase enables to reach an established flow regime.
- **Phases 2 & 4:** 60% and 40% valve opening
  The injected flow rate is expected to increase and the deltaP valve to translate towards closing to decrease the HP flow rate recirculation.
- **Phase 3:** valve fully closed
  The injected flow rate is expected to stop and the deltaP valve to translate towards opening to increase the HP flow rate recirculation back to phase 1 state.

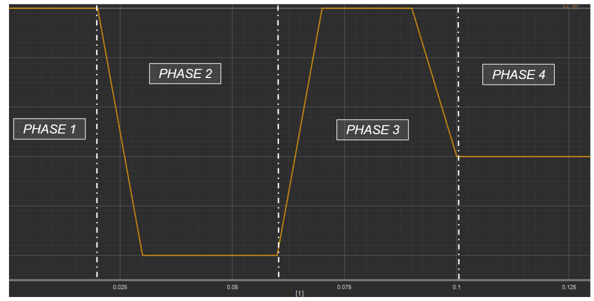

Figure 7: Prescribed position law of the metering valve against time (translation along *x*-axis)

The deltaP valve has a rigid body dynamics behaviour with one degree of freedom of translation along the valve longitudinal *x*-axis. The poppet is subjected to external hydraulic forces (pressure loads) and mechanical forces (spring force). The spring force applied on the valve is defined as follows :

$$F_{spring} = -k(x - X_0) - F_0 - d \tag{2}$$

where x and X_0 variables represent respectively the valve current and initial positions. The parameter k stands for the spring stifness, F_0 its preload and d a mechanical damping coefficient defined as a function of the valve displacement velocity ($d = 20\dot{x}$). Those parameters have been adjusted to work with a ΔP value close to operational conditions. The material characteristics are also defined to get the proper response of the valve.

Lattice structure
The space discretization is uniform and a distance of $dx = 5L_{ref}$ is set between each lattice node. L_{ref} defines the dimension of the smallest geometric detail of the domain which is in this case the deltaP valve pressure tap. The final lattice structure is composed of 333 000 elements.

Computational parameters
The time step is set to 5.0e-7 sec for the whole computation (0.13 sec of physical time). The turbulence parameters and fluid characteristics are the same as for the LP pump model.

4.2.2 Analysis of the system dynamics

Figure 8 represents the temporal response of the system. During phases 1 and 3, the mass flow rate drops to zero (Figure 8b) and the deltaP valve shows a larger

displacement towards opening (Figure 8c). On the contrary, during phases 2 and 4, the mass flow rate increases and the deltaP valve tends to move towards closing.
The overall dynamics of the 2D fuel metering system foreseen by XFlow is consistent with the real physics of the system detailed in section 4.2.1.

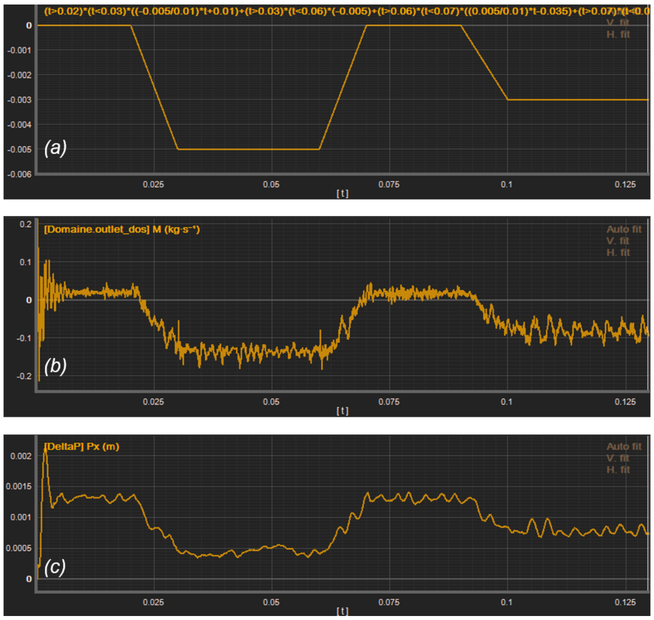

Figure 8: Temporal response of the metering unit *(a)* Valve opening command *(b)* Mass flow rate *(c)* DeltaP constant valve displacement

The computed flow rates for phases 2 and 4 are compared to the expected values based on the opening section of the metering valve and the differential pressure. As the discharge coefficient varies depending on the opening section, an average value of 0.85 can be considered as a first approximation for large openings. Table 4 summarizes the relative difference between computed and expected values.

Valve opening	ΔQ relative difference
60%	+ 9%
40%	+ 5%

Table 4: Relative difference of flow rate between expected and computed values considering Cd=0.85

Computed results are overestimated by 5 to 10 % on average assuming a value of 0.85 for the discharge coefficient. However, to conduct a thorough study, a preliminary characterisation with a separate CFD simulation of the exact orifice geometry associated to the opening section considered would have been required.

The static pressure and velocity fields distributions in the fuel metering system are presented hereafter.

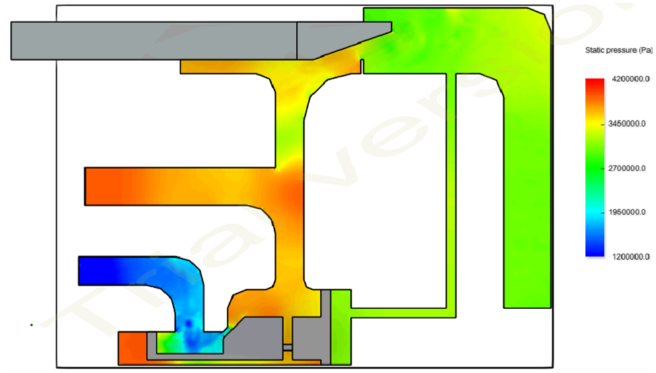

Figure 9: Instantaneous static pressure distribution in the fuel metering unit

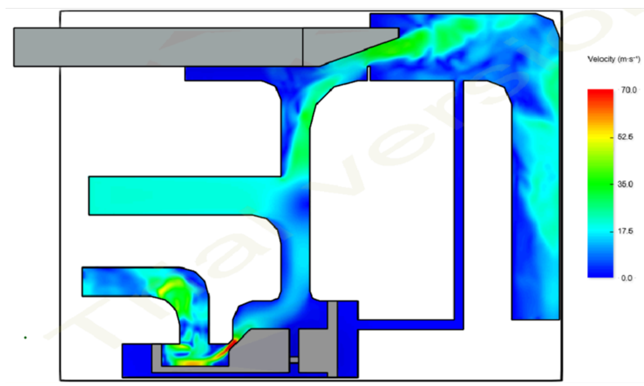

Figure 10: Instantaneous velocity distribution in the fuel metering unit

4.3 Check valve

This last case deals with a check valve whose design is known to present stability issues once integrated in the whole fuel hydromechanical unit. This case has been very challenging for SAFRAN HE for over a year as simulation tools available were not able to model and reproduce accurately the phenomenon observed on test bench. Thus investigations to find the appropriate design were difficult without requiring experimental tests to ensure a good stability of the new design.

The aim of this study is to investigate the capability of XFlow software to reproduce oscillations observed on test bench for different operational conditions. The eigenfrequency of the oscillations and the flow rates at which instabilities occur are correlated with experimental data. The initial design presenting stability issues is first modelled and the new design implemented is then also studied with the same numerical model to check its stability. This case study is thoroughly detailed in [3].

4.3.1 Pressure drop of the check valve

To set up and validate the numerical model, a preliminary study is carried out to correlate pressure drop values obtained through experimental tests and those computed with XFlow software for different functional points. Reference values are extracted from the technical specification of the check valve.

Model description
The 3D CAD model of the initial valve design depicted in Figure 11 is composed of 3 parts:

- **The valve:** With one degree of freedom along the longitudinal axis, the valve has a rigid body dynamics behaviour and is subjected to an external spring force and the pressure loads exerted by the surrounding flow.
  The spring is not physically modelled here but taken into account using a spring force law applied to the valve in the same way as for the constant deltaP poppet in section 4.2.1.
- **The spring seat:** Considered as a fixed body in the numerical model, it enables to be representative of the fluid domain topology.
- **The housing:** It corresponds to the sleeve geometry and is used to define the fluid domain topology.

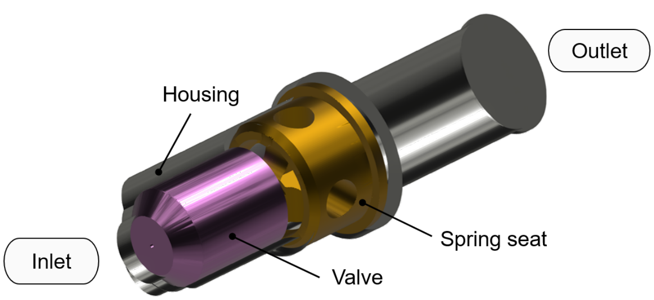

Figure 11: 3D CAD of the check valve initial design

Lattice structure
A uniform spatial resolution is used with a distance between each lattice node of $dx = 5L_{ref}$ where L_{ref} defines the valve orifice diameter. Different refinement levels may have been used to allow a finer resolution of the clearance between the valve and the sleeve as well as the small hole in the valve. A larger resolution would have been used elsewhere in the fluid domain. However, to foresee the acoustics analysis in section 4.3.2, a uniform discretization based on the smallest geometric detail is highly recommended. The final lattice structure is composed of 4.5 million of elements.

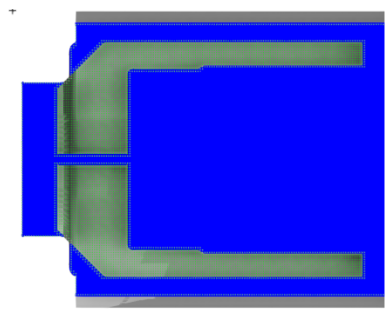

Figure 12: Uniform lattice structure of the check valve fluid domain (zoom close to the valve)

Computational parameters
The time step is set to $1.7e-7$ sec. The turbulence parameters and fluid characteristics are the same as for the previous cases.

Validation of the numerical model
To validate the numerical model, averaged pressure drop values computed with XFlow and experimental measurements are compared for different functional points. Those functional points correspond to different volumetric flow rates imposed at the inlet of the fluid domain. Table 5 compares averaged pressure drop values computed with XFlow with respect to the maximum accepted values extracted from the technical specification of the check valve and shows that the criteria are met for all functional points.

Flow rate (% Q_{max})	Experimental $\Delta P/P_0$	XFlow $\Delta P/P_0$
4.4	< 2.67	2.38
11.1	< 2.73	2.43
22.2	< 2.88	2.48
100	< 3.36	2.83

Table 5: Comparison of computed pressure drops values with the maximum expected values extracted from the technical specification

Besides, Figure 13 shows a very good agreement between XFlow results and experimental measurements with a similar trend of both curves which highlights a good capability of XFlow to predict the hydraulic behaviour of the check valve.

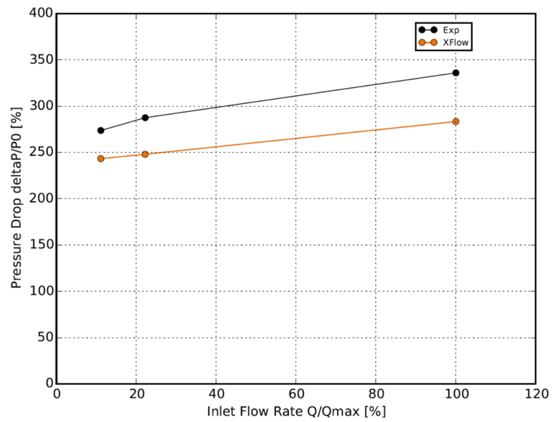

Figure 13: Comparison between XFlow results (orange) and experimental measurements (black) of the pressure drop evolution vs the inlet flow rate

Contrary to traditional CFD codes, LBM methods are well suited to deal with changes in the fluid domain topology. XFlow handles well the transition from a closed position of the valve to an open one.

The velocity field distribution around the valve for the maximum inlet flow rate is depicted in Figure 14. Note that the hole in the valve ensures communication between the upstream and downstream domains but is not used for damping purposes.

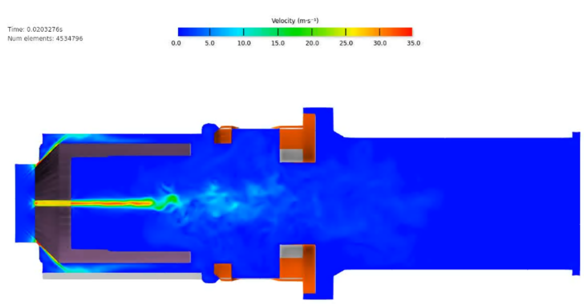

Figure 14: Velociy field distribution around the valve for the maximum inlet flow rate Q_{max}

4.3.2 Unstabilities of the check valve

The preliminary study enabled to validate the numerical model and showed a good correlation of the hydraulic behaviour of the check valve with measurements. A transient vibrations analysis is then conducted with the same design and lattice structure to reproduce instabilities observed on test bench.

Model description

As experimental tests showed the presence of oscillations for a given flow rate range, the inlet flow rate condition is defined with an increasing law to cover the operational range of the check valve from 22.2% Q_{max} to Q_{max}.

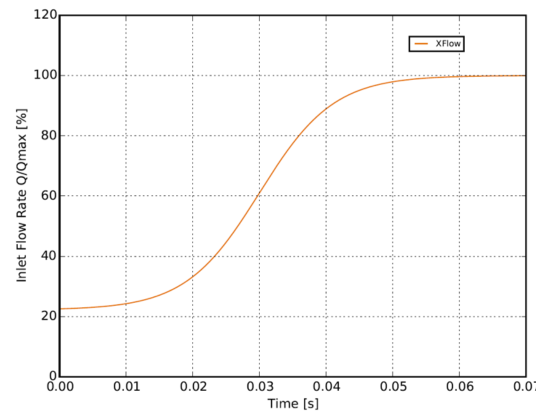

Figure 15: Increasing flow rate law defined at the inlet of the fluid domain from 22.2% Q_{max} to Q_{max}

Contrary to the previous computation in section 4.3.1, a transient analysis is performed in this part using a Direct Noise Computation method [2] to compute the acoustic field. The propagation of the pressure waves is captured by adjusting the time step according to the thermodynamic speed of sound in kerosene.

Transient response of the check valve

Figure 16 shows the relative displacement of the valve in response to the increasing flow rate law. It can be observed that instabilities occur at 51% Q_{max}.

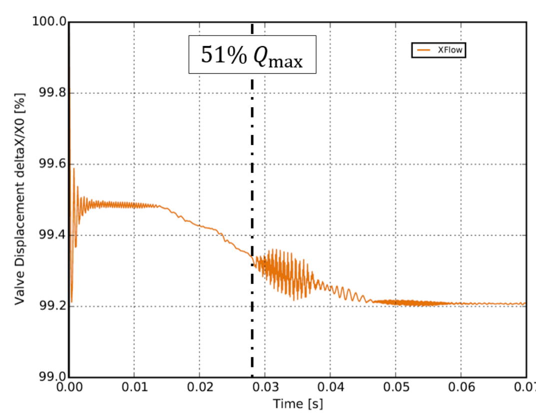

Figure 16: Relative displacement of the valve vs time in response to the increasing flow rate law

Experimental tests showed that oscillations start around 45% Q_{max}. which is close to results assessed with XFlow.

Oscillations of the valve displacement can be correlated with high instabilities of the pressure signal at the outlet of the domain as depicted in Figure 17.

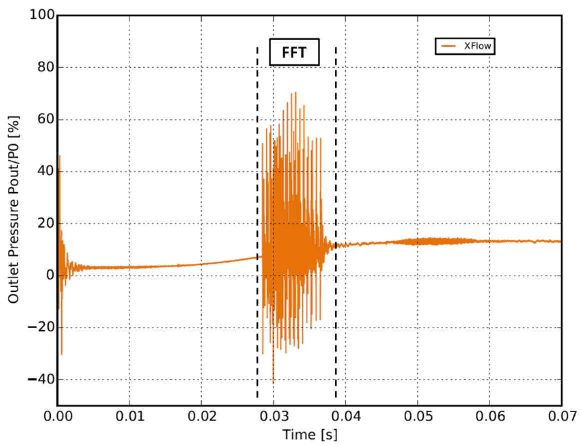

Figure 17: Outlet pressure signal vs time in response to the increasing flow rate law

A frequency analysis using a Fast Fourier Transform (FFT) of the pressure signal on the interval displaying high instabilities highlights one eigenfrequency F_0 as shown in Figure 18. The main frequency F_0 computed by XFlow is overestimated by 10% compared to experimental measurements.

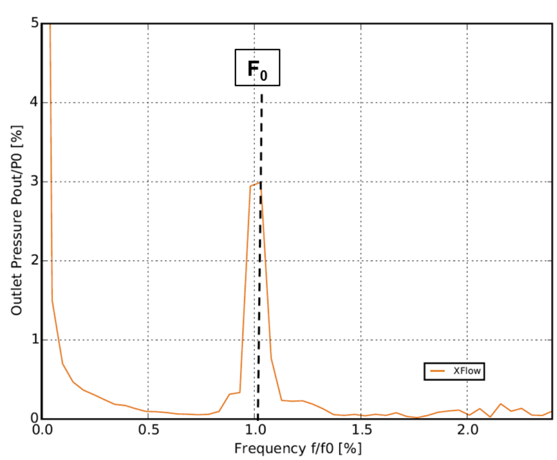

Figure 18: FFT transform of the pressure output signal

This analysis shows that XFlow is not only able to predict the stability of a valve but also to give a good evaluation of the flow rate range and the main frequency displaying instabilities.

4.3.3 Stability analysis of the valve optimized design

A similar analysis is performed on the new design of the check valve depicted in Figure 19. Experimental feedback demonstrates a good stability of this optimized design once integrated in the hydromechanical unit.

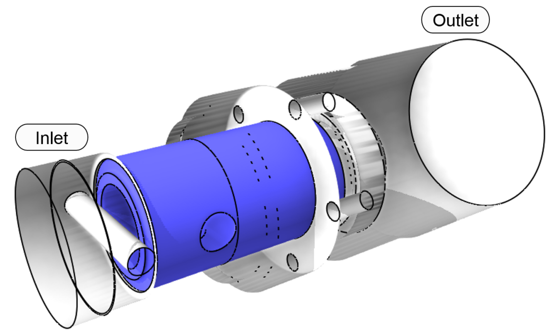

Figure 19: CAD of the countermesure design of the check valve

An increasing flow rate law is imposed at the inlet of the fluid domain as done previously. Figure 20 represents the relative position of the valve with respect to time. The evolution of the valve position does not display any instabilities which is in good agreement with experimental observations.

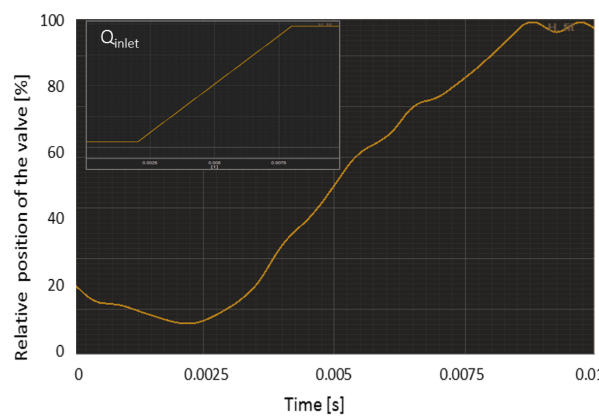

Figure 20: Relation position of the valve vs time in response to the increasing flow rate law for the optimized valve design

5 CONCLUSION

Case studies presented in this paper were for most of them unreachable for unsteady-RANS turbulence or LES solvers. Lattice-Boltzmann methods are a promising alternative to deal with FSI problematics. XFlow LBM solver demonstrated a good capability to handle one-way and two-way FSI applications encountered in fuel equipement whether it be with enforced body motions or rigid body dynamics behaviour subjected to external forces.

The application to the LP fuel pump shows that XFlow evaluates the pressure generation with the same accuracy as the mesh-based LES solver but with a significant gain in pre-processing and computational time. It also demonstrates wide prospects to deal with dynamic systems composed of multi-parts. The overall dynamics of the 2D fuel metering unit foreseen by XFlow is consistant with the real physics of the system. Finally the transient analysis of the check valve demonstrates that XFlow solver is a relevant tool to assess accurately the stability of an equipment through the design process.

XFlow software opens a wide roadmap to cope with other fuel system problematics which have never yet been investigated using CFD simulations such as the study of suction ability of a LP fuel pump or the prediction of cavitation areas in HP gear pumps. Studies are also currently in progress to use XFlow to identify the main hot spots on a fuel equipement during fire test scenarii.

References

[1] F. Ducros, F.Nicoud, and T.Poinsot. Wall-adapting local eddy-viscosity models for simulations in complex geometries. In *Proceedings of 6th ICFD Conference on Numerical Methods for Fluid Dynamics*, 1998.

[2] D. Holman, R.Brionnaud, G.Trapani, and M.Chavez. Direct Noise Computation with a Lattice-Boltzmann Method and Application to Industrial Test Cases. In *22nd AIAA/CEAS Aeroacoustics*, 2016.

[3] D. Holman, Z.Abiza, D.Taieb, and M.Robin. CFD analysis of a check valve of SAFRAN Helicopter Engines using a Lattice-Boltzmann solution. In *NAFEMS World Congress*, 2017.

[4] V. Moureau, P. Domingo, and L. Vervisch. Design of a Massively Parallel CFD Code for Complex Geometries. *Comptes Rendus Mecanique*, 2011.