Purdue University
Purdue e-Pubs

International Compressor Engineering Conference   School of Mechanical Engineering

2016

Fluid-Structure Interaction of a Reed Type Valve

Ignacio Gonzalez

Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya (UPC) - BarcelonaTech ETSEIAT, Colom 11, Terrassa (Barcelona) 08222, Spain, nacho@cttc.upc.edu

Oriol Lehmkuhl

Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya (UPC) - BarcelonaTech ETSEIAT, Colom 11, Terrassa (Barcelona) 08222, Spain / Termo Fluids S.L., Avda. Jacquard 97 1-E, Terrassa (Barcelona) 08222, Spain, oriol@cttc.upc.edu

Alireza Naseri

Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya (UPC) - BarcelonaTech ETSEIAT, Colom 11, Terrassa (Barcelona) 08222, Spain, anaseri@cttc.upc.edu

Joaquim Rigola

Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya (UPC) - BarcelonaTech ETSEIAT, Colom 11, Terrassa (Barcelona) 08222, Spain, quim@cttc.upc.edu

Assensi Oliva

Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya (UPC) - BarcelonaTech ETSEIAT, Colom 11, Terrassa (Barcelona) 08222, Spain, oliva@cttc.upc.edu

Follow this and additional works at: https://docs.lib.purdue.edu/icec

Fluid-structure interaction of a reed type valve

Ignacio GONZALEZ[1*], Oriol LEHMKUHL[1], Alireza NASERI[1], Joaquim RIGOLA[1], Assensi OLIVA[1]

[1] Heat and Mass Transfer Technological Center (CTTC)
Universitat Politècnica de Catalunya – BarcelonaTech (UPC), ESEIAAT,
Colom 11, 08222, Terrassa (Barcelona), Spain
Tel. +34 937398192
Email: cttc@cttc.upc.edu, http://www.cttc.upc.edu

* Corresponding Author

ABSTRACT

This paper presents a complete numerical procedure to study the fluid-structure interaction problem of incompressible flow through reed valves, typically employed in hermetic reciprocating compressors. A partitioned semi-implicit coupling scheme is implemented, which only strongly couples the added-mass-effect (pressure term) of the fluid to the structure hence, assuring numerical stability and avoiding excessive computational cost. The fluid is solved by a three-dimensional CFD solver using large eddy simulation closures to model the turbulent flow, while the reed valve is described with the classical plate theory and the normal mode summation method. To showcase the potentiality of the proposed methodology, a sensitivity analysis regarding valve thickness is carried out for a given velocity in the feeding channel. Considerable differences, mainly in valve lift and pressure drop, are appreciated between the considered configurations.

1. INTRODUCTION

High efficiency of hermetic reciprocating compressors has become an essential requirement in the domestic refrigeration field as a result of their increasing worldwide electricity consumption and the environmental awareness. According to Ribas et al. (2008), almost half of the compressor thermodynamic losses are produced during suction and discharge processes. Therefore, special attention should be given to the flexible reed valves that regulate the gas flow in both processes. In addition, valves are determining not only in terms of efficiency but also regarding installation reliability, since they have been identified to be the most common cause of compressor failures (Leonard, 1996; Woo et al., 2010).

The study and optimization of these reed valves are challenging tasks due to the complexity of the fluid-structure interaction (FSI) problem. Significant work has been carried out in order to predict the flow field by means of computational fluid dynamics (CFD). Matos et al. (2000) and Barbi et al. (2012) performed a three-dimensional simulation of a laminar flow through a valve, in the former solved with a one-dimensional model and in the latter with a predefined motion. Besides, the flow was studied in two dimensions by Matos et al. (2002), Kim et al. (2006) and Gasche et al. (2010) resulting from an axisymmetric problem arrangement or from simplifications. Mayer et al. (2014) used a loosely-coupled strategy to combine a three-dimensional turbulent model for the refrigerant with a strain-stress structural solver for the valve. Encouraging agreement with experimental data was obtained.

This work arises from the effort made by Estruch et al. (2014), who focused on a three-dimensional turbulent flow through a simple rectangular reed valve. A newly developed FSI method allows strongly coupling of the fluid and structural solvers, assuring numerical stability with an acceptably moderate increase in computational cost. The CFD solver consists of a three-dimensional explicit finite volume model formulated in a second-order, conservative and collocated unstructured grid arrangement. The turbulent nature of the flow is modeled by the wall-adapting local eddy-viscosity (WALE) model. For the elastic reed valve, it is followed the classical flat plate theory based on the structure modal analysis in order to save considerable effort in time and cost. The geometric interaction between both dynamically moving media is managed with a fluid deformable mesh and the space conservation law, whereas the immerse boundary method is used to establish static bodies, i.e. the valve seat and the channel.

Two distinct objectives are pursued in this paper. The first one is to present a complete numerical procedure to analyze FSI applications, for instance the turbulent flow through reed valves. The second is to evaluate the sensitivity of these valves regarding their thickness. To do so, a generic reed valve is considered together with a feeding channel with a suction flow inlet condition.

2. MATHEMATICAL AND NUMERICAL MODEL

The developed FSI algorithm is built on a partitioned approach, which uses independent solvers for fluid and structural sub-problems and adopts a coupling scheme to account for the interaction of the domains. In order to precisely satisfy the coupling condition at the fluid-structure interface and to avoid numerical instabilities, a strongly coupled scheme is followed. It enforces the exact coupling condition by means of iterating between the fluid and structural solvers until the convergence criterion is met.

The change in the geometric domains –derived from the deformation of the structure– is taken into account by directly moving the solid mesh and updating the fluid grid according to the new solid interface position. The fluid mesh adaptation is governed by the equations of elasticity similarly to Stein and Tezduyar (2002) and Smith (2011). By introducing a stiffening in the smaller grid elements, large displacements can be accomplish preserving the mesh quality near the solid surfaces. Namely, not influencing the accurate calculation of the boundary layers.

Throughout the valve opening, new fluid elements will appear between the valve and the seat. Since there is no remeshing but the fluid mesh is just deforming, extra elements are needed under the valve, which first perform as rigid seat and later as fluid. The immerse boundary technique allows this change in the nature of an element thanks to a modification in its viscosity.

Firstly, the governing equations of both sub-domains, the fluid Ω_f and the structure Ω_s, are presented. Then, there is the description of the algorithm for solving the coupling problem on the interface $\Gamma = \Omega_f \cap \Omega_f$.

2.1 Governing equations

The governing equations for the fluid correspond with the incompressible Navier-Stokes and continuity equations within an arbitrary Lagrangian Eulerian formulation to consider dynamic mesh effects,

$$\nabla \cdot \vec{u} = 0 \tag{1}$$

$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} - \vec{u}_w) \cdot \nabla \vec{u} = -\frac{1}{\rho_f}\nabla p + \frac{\mu}{\rho_f}\Delta \vec{u} \tag{2}$$

where the unknowns p and $\vec{u}$ are the fluid pressure and velocity, respectively. The velocity of the control volume's faces, $\vec{u}_w$, is nonzero as long as the mesh moves. Its value is calculated by means of the space conservation law (Estruch et al., 2013). The fractional step projection method is adopted to decouple velocity and pressure and solve equations 1 and 2. The expected turbulence of the flow will generate small scales of motion that need high grid refinement to be properly solved. The influence of these scales of turbulence can be modeled by Large Eddy Simulation (LES) techniques so as not to require that refinement. In this work, LES has been performed using the wall-adapting local-eddy viscosity model (WALE), available in TermoFluids code (Lehmkuhl et al., 2009).

On the fluid-structure interface Γ, there is a Dirichlet boundary condition determined by the displacement of the structure $\vec{d}$

$$\vec{u} = \frac{\partial \vec{d}}{\partial t} . \tag{3}$$

By contrast, the structure has a traction boundary condition on the common interface. This external force is represented by $\vec{\sigma}_\Gamma$ and is a function of fluid velocity and pressure distribution on Γ, $\vec{\sigma}_\Gamma = \vec{\sigma}_\Gamma (\vec{u}, p)_\Gamma$. Therefore, having adapted the mesh and updated the boundary conditions (Equation 3), the fluid solver is able to find p and $\vec{u}$ in Ω_f and, in this manner, evaluate a new traction distribution on the solid surface

$$\vec{\sigma}_\Gamma = \vec{F}\left(\vec{d}\right) . \tag{4}$$

In regards to the reed structure, a solver based on the Kirchhoff-Love theory for isotropic plates under transverse loading is employed, hence just the normal component of $\vec{\sigma}_\Gamma$ is taken into account. Likewise, only the displacement in

the plate transverse direction is treated as unknown, since the rest can be derived easily from it,

$$d_x(\vec{x}) \quad = \quad -z\frac{\partial d_z}{\partial x} \tag{5}$$

$$d_y(\vec{x}) \quad = \quad -z\frac{\partial d_z}{\partial y} \tag{6}$$

$$d_z(\vec{x}) \quad = \quad d_z(x,y) \tag{7}$$

considering z as the plate normal direction and locating the mid-surface plane of the structure on $z = 0$.

Referring to Soedel (1992), the valve motion can be described from the superposition of its infinite normal modes of vibration. Each mode m has a natural radial frequency ω_m and a particular deformation pattern in z direction known as mode shape $\varphi_m(x,y)$. Thus, the transverse displacement is a linear combination of the normal modes of the system. For practical applications only the first few modes do actually participate and, consequently, only a finite number of modes should be computed:

$$d_z(x,y,t) = \sum_{m=1}^{\infty} q_m(t)\varphi_m(x,y) \tag{8}$$

where $\varphi_m(x,y)$ as well as ω_m of any given plate geometry can be found experimentally or numerically by a structural solver.

The combination of both assumptions, the plate theory and the assumed modes method, leads to one generalized momentum equation for each mode,

$$\ddot{q}_m(t) + 2\xi\omega_m\dot{q}_m(t) + \omega_m^2 q_m(t) = \frac{\int_\Gamma \varphi_m(x,y)\sigma_{\Gamma,z}(x,y,t)d\Gamma}{\rho_s h \int_\Gamma \varphi_m^2(x,y)d\Gamma} \tag{9}$$

where ξ is the damping ratio and h is the plate thickness. A Newmark time integration scheme is employed to find the generalized coordinates q_m, $\dot{q}_m$ and $\ddot{q}_m$.

Moreover, the impact between the valve and its seat is assumed to occur in such a short period of time that only the velocity is affected by the collision. Hence, a coefficient of restitution C_r has been advocated in order to model the impact. When the coefficient of restitution is set to one, all the kinetic energy is preserved after impact. Having solved Equation 9 at time step t^{n+1}, if contact is detected, the generalized velocity will be corrected with the approximation of Sheu and Hu (2000)

$$\dot{q}_m\left(t^{n+1}\right) = -C_r\dot{q}_m\left(t^n\right) . \tag{10}$$

Similarly to Equation 4 for the fluid solver, the structural solver can be summarized by

$$\vec{d} = \vec{S}\left(\vec{\sigma}_\Gamma\right) \tag{11}$$

in other words, given the current fluid traction on the solid, the solver $\vec{S}$ will obtain the deformation or displacement of the structure.

2.2 Fluid-Structure Interaction coupling scheme

Defining the fluid and structural equations concisely as Equations 4 and 11 reduces the coupled fluid-structure system of equations into an interface problem of the form:

$$\vec{S} \circ \vec{F}\left(\vec{d}\right) - \vec{d} = 0 \tag{12}$$

with $\vec{d}$ and functions $\vec{F}$ and $\vec{S}$, all in the new time step t^{n+1}. Instead of solving the interface problem 12 as in implicit schemes, here a semi-implicit approach is followed. This approach couples only the pressure stress term of the fluid, implicitly to the structure, while the rest of the fluid terms are coupled explicitly. It has been shown that pressure stress term is responsible for the added-mass effect, and coupling it explicitly will cause numerical instability (Causin et al., 2005). By coupling this term implicitly, the stability issue is overcome, while explicitly coupling the other terms saves a significant computational cost.
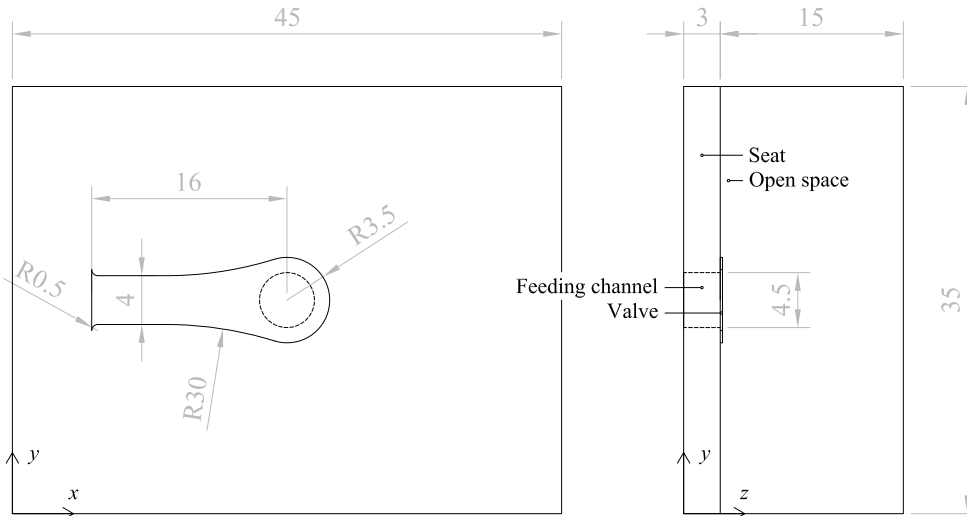
Figure 1: FSI problem domain. Dimensions in mm.

Thanks to the fractional step method, the pressure stress term can be split off from the fluid flow equations. At the beginning of a new time step, once the moving mesh function has updated the domain Ω_f and the velocities $\vec{u}_w$, a predictor velocity, which does not take into account the pressure gradient, is calculated. Then, a pressure field is derived from this intermediate velocity. The structure will deform facing this pressure field and, as a result, a new velocity boundary condition will arise in Γ (Equation 3). The implicit coupling is fixed for these two steps, the fluid pressure solver and the structural solver. So the new FSI interface problem is modified to

$$\vec{S} \circ \vec{f}(\vec{d}) - \vec{d} = 0 \tag{13}$$

where $\vec{f}$ is just the fluid pressure solver. To solve the nonlinear FSI interface problem, a fixed-point solver with a block Gauss-Seidel method and Aitken's relaxation has been chosen in this study. Aitken's dynamic relaxation mitigates possible convergence problems at the Gauss-Seidel method and reduces the number of iterations required. For further details of the FSI coupling scheme please refer to Naseri et al. (2016).

3. PROBLEM DEFINITION

Figure 1 shows the dimensions of the simulation domain and the reed valve chosen for the test. The domain contains a feeding channel, a rigid seat, a reed valve and an empty space downstream. Two different valve thicknesses have been considered: 0.18 mm and 0.20 mm.

The fluid properties are: ρ_f = 19 kg/m^3 and μ = 9.4 · 10^{-6} Pa·s. The valve material is steel and its properties are: ρ_s = 7870 kg/m^3, v = 0.29 and E = 2.05 · 10^{11} Pa. The damping ratio of Equation 9 is taken from Prater and Hnat (2003) ξ = 0.04 and the coefficient of restitution from Habing and Peters (2006) C_r = 0.3.

There is an inlet velocity boundary condition at the beginning of the channel. It is a time dependent mass flow that belongs to the suction process of a one-dimensional simulation of a hermetic reciprocating compressor, Figure 2 (Lopez, 2016). The average fluid velocity along the channel results in a Reynolds number close to $3 \cdot 10^4$, hence a turbulent flow is developed indeed.

The natural modes of vibration of the reed valve ($\varphi_m(x, y)$ and ω_m) should be studied previously to the FSI simulation. A commercial structural solver has been used defining a clamped end, like in a cantilevered beam. For this problem only the first four modes has been considered. The normal mode shapes are illustrated for the thinnest valve in Figure 3 –the deformation is pretty similar for the thicker valve– whereas the natural frequencies are presented in Table 1.
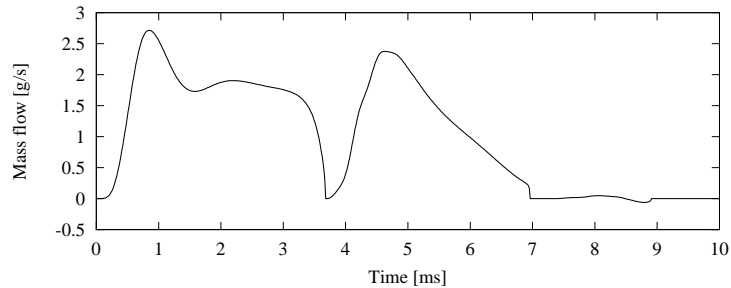
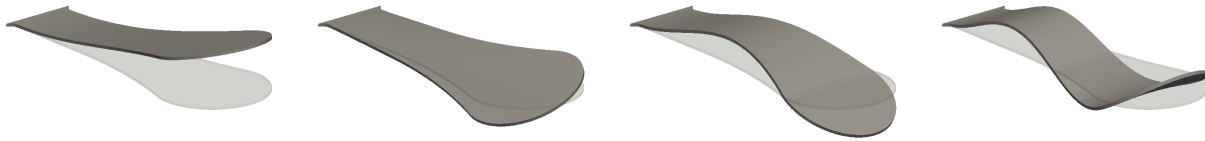**Figure 2: Suction mass flow profile extracted from Lopez (2016).**

Figure 3: Normal mode shapes of the 0.18 mm-thick valve. From left to right: mode 1, 2, 3 and 4.

4. NUMERICAL RESULTS

The results presented in this section have been obtained with a coarse computational mesh of $4 \cdot 10^5$ control volumes. It would be appropriate to perform a mesh refinement analysis to ensure that the FSI solution has converged. Even so, the flow phenomena obtained is consistent and accomplishes a qualitative accurate transient simulation of the turbulent flow through the reed valve.

Figure 5 outlines the valve lift throughout the suction process, which is equivalent here to the displacement of the valve point coincident with the channel axis. The numerical results show that the dynamic action of the reed valve is definitely dependent on the thickness. Although the first peak force exerted by the fluid while impacting with the valve is almost equal in both cases, the maximum lift is 18 % lower in the thickest design. It is clearly due to the bending stiffness of the valve, which rises with the thickness. This rigidity is at the same time affecting the oscillation period, as was also derived from the modal analysis of the valves (Table 1). A higher thickness implies a major number of opening oscillations and impacts, what can eventually influence to structural and acoustic issues. Nevertheless, neither of them seems to be suitable for the suction process proposed, as their stiffness make them oscillate in excess for a regular refrigerant flow.

Since it is the mass flow that is imposed in the boundary condition, the parameter to compare suction process efficiency will be the pressure drop of the flow through the valve. Thus, the smaller is the pressure drop, the more fluid mass will be suctioned and placed in the compression chamber. The transient evolution of the pressure on the inlet orifice is presented in Figure 6. It seems to be generally higher with h = 0.20 mm. It has been observed that this pressure is qualitatively identical to the fluid pressure on the valve, specifically around the channel axis. As can be seen, there are five pressure peaks that emerge from those times when the valve is going to close and there is still a feeding flow along the channel. In fact, the last bounces of the valve, those beyond 7 ms, would not provide any benefit for the suction cycle because there is no feeding flow anymore.

Table 1: Natural frequencies of the valves.

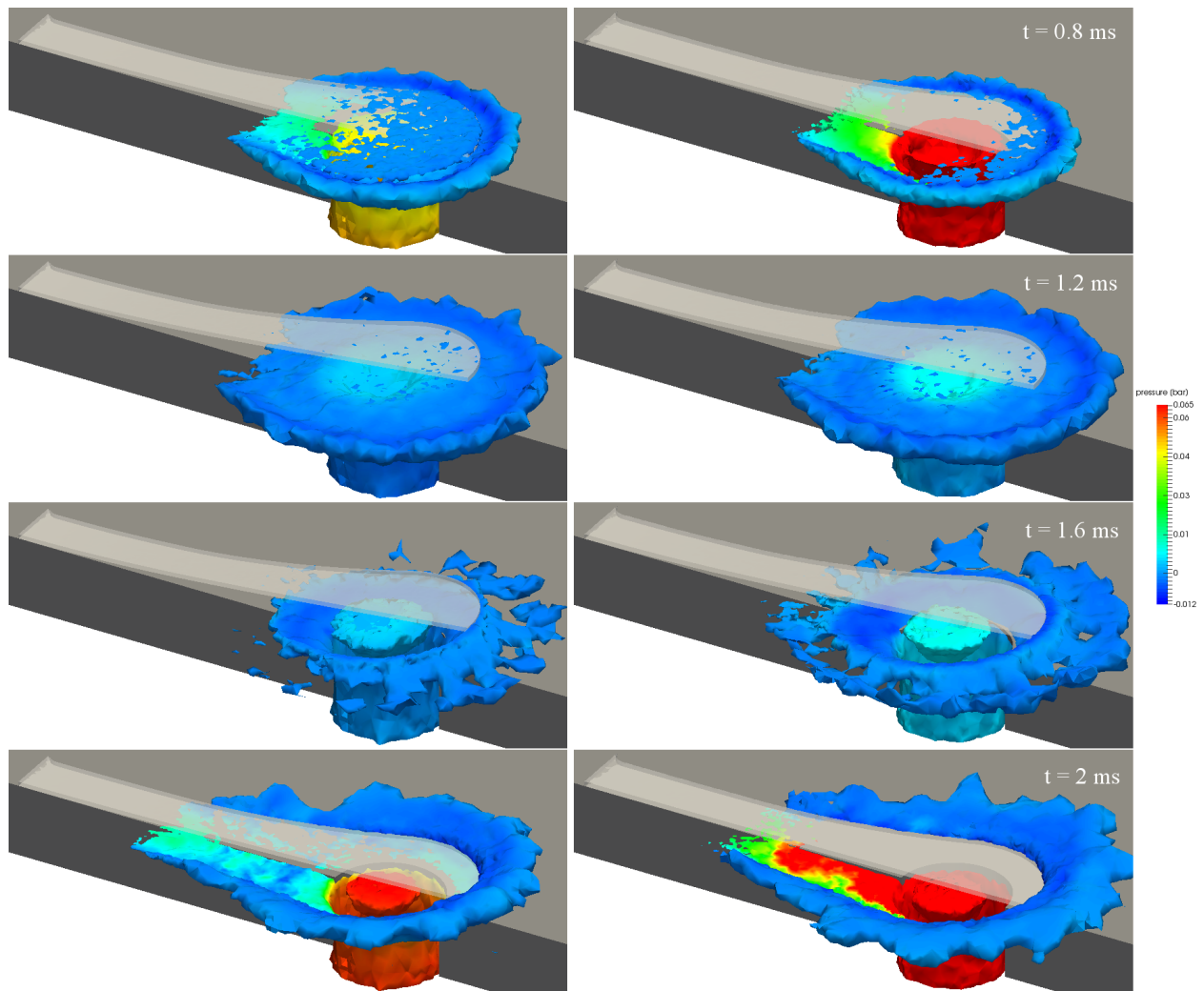|	mode 1	mode 2	mode 3	mode 4
h = 0.18 mm	339.7 Hz	2349 Hz	2499 Hz	7234 Hz
h = 0.20 mm	377.4 Hz	2606 Hz	2777 Hz	8035 Hz

Figure 4: Fluid flow during the first valve oscillation for the 0.18 mm-thick valve (left) and for the 0.20 mm-thick valve (right). Contour surfaces of velocity at $|\vec{u}|$ = 5 m/s.
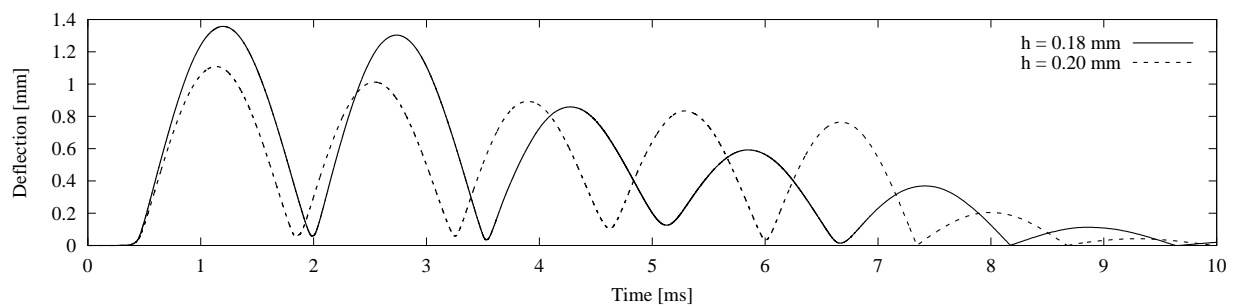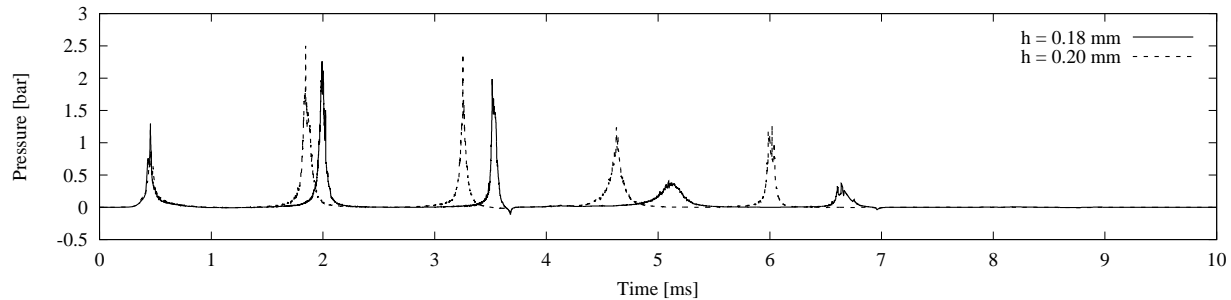

Figure 5: Valve lift.

Figure 6: Pressure drop measured on the channel inlet orifice.

Finally, the flow behavior during the first opening oscillation is depicted in Figure 4. Just prior to having a wide outflow aperture (at t = 0.8 ms), a considerable increase of velocity is appreciated between the valve and the seat. Therefore, large velocity gradients and high vorticity appears in this area. Later, the velocity tends to slow down along with the feeding velocity (Figure 2) until the valve moves towards the seat again narrowing the passage.

5. CONCLUSIONS

A semi-implicit FSI coupling scheme, that only couples the pressure stress term of the fluid implicitly to the structure, is used to solve the fluid flow through a reed valve. The geometric arrangement of the different elements is treated with a moving mesh technique, for the flexible valve, and with the immerse boundary method, for the rigid seat. The flow is studied by means of a three-dimensional CFD solver with LES model whereas the valve response is evaluated according to the plate theory and the assumed modes method. The proposed methodology shows a good numerical performance and is proven to be appropriate for FSI applications as this one. All together is used to carry out a thickness sensitivity analysis considering a generic reed valve and a particular suction mass flow configuration. The results demonstrate that this parameter is a crucial aspect for the design of reed valves since it has a big influence in the valve and fluid dynamics during the suction process. In the particular case analyzed, the thinner plate of 0.18 mm seems to be better in terms of pressure drop through the valve, though it is clear that an even less-rigid valve would be more suitable for the suction conditions considered.

NOMENCLATURE

C_r	coefficient of restitution
$\vec{d}$	displacement vector of the structure
E	Young's modulus
h	valve thickness
p	pressure
$\vec{u}$	velocity vector
t	time
μ	dynamic viscosity
$\varphi_m(x,y)$	m^{th} normal mode shape function
v	Poisson's ratio
ξ	damping ratio
ρ	density
$\vec{\sigma}_\Gamma$	fluid load on the structure
ω_m	m^{th} normal natural frequency

Subscript

f   fluid
s   structure

REFERENCES

Barbi, F., Gasche, J. L., and Villar, M. M. (2012). An efficient immersed boundary method for solving the unsteady flow through actual geometries of reed valves. In *International Compressor Engineering Conference*. Paper 2088.

Causin, P., Gerbeau, J. F., and Nobile, F. (2005). Added-mass effect in the design of partitioned algorithms for fluid–structure problems. *Computer Methods in Applied Mechanics and Engineering*, 194(42–44):4506 -- 4527.

Estruch, O., Lehmkuhl, O., Borrell, R., Pérez-Segarra, C. D., and Oliva, A. (2013). A parallel radial basis function interpolation method for unstructured dynamic meshes. *Computers & Fluids*, 80:44 -- 54.

Estruch, O., Lehmkuhl, O., Rigola, J., and Pérez-Segarra, C. D. (2014). Fluid-structure interaction of a reed type valve subjected to piston displacement. In *International Compressor Engineering Conference*. Paper 2380.

Gasche, J. L., Rodrigues, T. T., and Militzer, J. (2010). Flow simulation through moving hermetic compressor valves using the immersed boundary method. In *International Compressor Engineering Conference*. Paper 1999.

Habing, R. A. and Peters, M. C. A. M. (2006). An experimental method for validating compressor valve vibration theory. *Journal of Fluids and Structures*, 22(5):683 -- 697.

Kim, J., Wang, S., Park, S., Ryu, K., and La, J. (2006). Valve dynamic analysis of a hermetic reciprocating compressor. In *International Compressor Engineering Conference*. Paper 1805.

Lehmkuhl, O., Perez-Segarra, C. D., Borrell, R., Soria, M., and Oliva, A. (2009). Termofluids: A new parallel unstructured cfd code for the simulation of turbulent industrial problems on low cost pc cluster. In *Parallel Computational Fluid Dynamics 2007*, volume 67, pages 275 -- 282.

Leonard, S. M. (1996). Increase reliability of reciprocating hydrogen compressors. *Hydrocarbon Processing*, 75(1):67 -- 74.

Lopez, J. (2016). *Parallel object-oriented algorithms for simulation of multiphysics. Application to thermal systems*. PhD thesis, Universitat Politècnica de Catalunya.

Matos, F. F. S., Prata, A. T., and Deschamps, C. J. (2000). A numerical methodology for the analysis of valve dynamics. In *International Compressor Engineering Conference*. Paper 1411.

Matos, F. F. S., Prata, A. T., and Deschamps, C. J. (2002). Numerical simulation of the dynamics of reed type valve. In *International Compressor Engineering Conference*. Paper 1563.

Mayer, J., Bjerre, P., and Brune, F. (2014). A comparative study of different numerical models for flapper valve motion. In *International Compressor Engineering Conference*. Paper 2309.

Naseri, A., Lehmkuhl, O., Gonzalez, I., and Oliva, A. (2016). Partitioned semi-implicit methods for simulation of biomechanical fluid-structure interaction problems. In *7th European Thermal-Sciences Conference*.

Prater, G. and Hnat, W. P. (2003). Optical measurement of discharge valve modal parameters for a rolling piston refrigeration compressor. *Measurement*, 33(1):75 -- 84.

Ribas, F. A., Deschamps, C. J., Fagotti, F., Morriesen, A., and Dutra, T. (2008). Thermal analysis of reciprocating compressors - a critical review. In *International Compressor Engineering Conference*. Paper 1907.

Sheu, H. C. and Hu, Y. Z. R. (2000). Nonlinear vibration analysis of reed valves. In *International Compressor Engineering Conference*. Paper 1419.

Smith, R. (2011). A pde-based mesh update method for moving and deforming high reynolds number meshes. In *49th AIAA Aerospace Sciences Meeting*, Orlando. AIAA 2011-472.

Soedel, W. (1992). *Mechanics, Simulation and Design of Compressor Valves, Gas Passages and Pulsation Mufflers*. Ray W. Herrick Laboratories, School of Mechanical Engineering, Purdue University.

Stein, K. and Tezduyar, T. (2002). Advanced mesh update techniques for problems involving large displacements. In *Proceedings of the Fifth World Congress on Computational Mechanics*, Vienna, Austria.

Woo, S., O'Neal, D. L., and Pecht, M. (2010). Reliability design of a reciprocating compressor suction reed valve in a common refrigerator subjected to repetitive pressure loads. *Engineering Failure Analysis*, 17(4):979 -- 991.

ACKNOWLEDGEMENT