# A MONOLITHIC FORMULATION OF FLUID-STRUCTURE INTERACTION FOR THE CO-SIMULATION : APPLICATION TO THE PISTON PROBLEM

**M. GIBERT[1,3], A. BEL-BRUNON[1], M. BRUN [2], A. GRAVOUIL[1] and M. ROCHETTE[3]**

[1] Contact and structural mechanic laboratory (LaMCos)
Université de Lyon, Insa Lyon, CNRS UMR 5259, F-69621, France
e-mail: {marie.gibert,aline.bel-brunon,anthony.gravouil}@insa-lyon.fr
Web page: http://lamcos.insa-lyon.fr

[2] Geomecanic, material and structure laboratory (GEOMAS)
Université de Lyon, Insa Lyon,F-69621, France
e-mail: michael.brun@insa-lyon.fr - Web page: http://geomas.insa-lyon.fr

[3] Ansys France SA
ANSYS, Bâtiment Einstein, 11 Avenue Albert Einstein, 69100 Villeurbanne, France
e-mail: {marie.gibert, michel.rochette}@ansys.com - Web page: https://www.ansys.com

**Key words:** Fluid-Structure Interaction, Finite element method, Finite volume method, Co-simulation, Energy conservation

**Abstract.** This extended abstract presents a coupling method for the Fluid-Structure Interaction (FSI) problem, which aim is to be conservative and to use classical spacio-temporal discretization methods on each sub-domains; fluid and structure. In this way, the coupling is based on a monolithic formulation and solved by a co-simulation algorithm. The structural sub-domain is spatially discretized by finite elements method and temporally discretized by Newmark implicit scheme. While the fluid sub-domain is discretized by finite volumes method and Runge-Kutta order 2 explicit scheme. The long-term objective is to couple two existing commercial softwares. The proposed method is validated using the one-dimension piston test case.

## 1 INTRODUCTION

There are various fields of application for the FSI simulation, aerospace [1], bio-medical [10], civil engineering [13]... The three past decades the research have been really active and numerous coupling methods have been developed. Several classifications and definitions are used, here the classification used is based on the formulation of the problem.

Thus, in term of formulation, the FSI problems are divided into two main categories; the partitioned coupling treats the problem in a uncoupled way while the monolithic coupling treats the sub-domain fluid, the sub-domain structure and the FSI-interface as a single system, see Fig.1. The partitioned methods have the advantage to be easy to implement but lacks accuracy due to the time-lag involved by the methods itself. The monolithic methods are better in term of accuracy and stability but also are difficult to implement and not easily generalizable.


**Figure 1**: Partitioned formulation versus Monolithic formulation

In order to guarantee the energy conservation, the method presented here is based on a monolithic formulation. Nevertheless, to avoid the difficulties of implementation and generalization, the algorithm used is based on the GC coupling method [4] [5], which has initially been introduced for the coupling problems of structural sub-domains. This method allows the construction of an algorithm, predictor/corrector like, without time lag between the two sub-domains, using the Lagrange multiplier method. This coupling method has been extended to the FSI problems, and has already shown promising results. As an instance, Casadei [3] has proposed an explicit coupling method for finite elements and finite volumes vertex centered. Li has coupled finite elements and SPH particles [7]. More recently, Meduri has proposed a coupling method fully Lagrangian of finite elements fluid and structure [8].

The aim here is to extend this coupling method to more classical spatio-temporal discretization methods for both physics in order to use an existing and efficient commercial solvers. Thus, the structural sub-domain uses a Lagrangian formulation and is discretized thanks to the finite elements in space and thanks to an implicit Newmark scheme in time. The fluid is written using an Arbitrary Lagrangian Eulerian (ALE) formulation, discretized by the means of finite volumes method (cell-centered and vertex-centered) and an explicit second order Runge-Kutta scheme. Finally each sub domain uses its own time step. In this way the global integration method is heterogeneous and asynchronous.

In a first step, the discretization methods of each sub-domain are briefly recalled. Then the coupling method is presented. Finally, this one is validated using the 1D piston test case.

## 2 CONSTITUTIVE EQUATIONS

The domain $\Omega \times [0, T]$ closed, spatially partitioned, without overlap, by a fluid sub-domain $\Omega_f$ and a structural one $\Omega_s$, such as $\Omega_f \cap \Omega_s = \emptyset$, is considered. The external fluid

sub-domain boundary is called $\Gamma_f$, respectively $\Gamma_s$ for the structural sub-domain. Finally $\Gamma_{FSI}$ is the boundary between the two sub-domain, see Fig. (2).

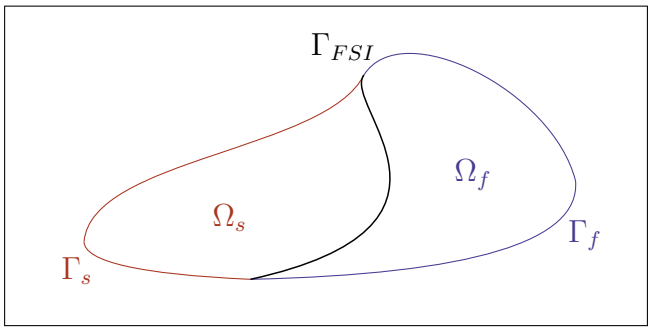


**Figure 2**: Geometry of the fluid-structure interaction problem

The fluid considered is compressible, inviscid and submit to the perfect gas law. The sub-domain structure is considered homogeneous, continuous and isotrope. It is considered linear elastic, under small displacement. The small perturbations hypothesis is assumed.

### 2.1 Structural sub-domain

The structural sub-domain $\Omega_s$ is spatially discretized because of the finite element method. This one is based on a Lagrangian formulation and the weak formulation of the equation of motion. Thus, under the hypotheses and the boundaries conditions, the discretized admissible state vector $\mathbf{U}_s(t) = \left[\mathbf{d}_s, \mathbf{v}_s, \mathbf{a}_s\right]^T(t)$ is sought. Where $\mathbf{d}_s$, $\mathbf{v}_s$, $\mathbf{a}_s$ are the discretized fields of displacement, velocity and acceleration respectively :

$$\mathbf{M}_s\mathbf{a}_s(t) + \mathbf{K}_s\mathbf{d}_s(t) = \mathbf{F}_s(t) \qquad \forall t \in [0, T] \tag{1}$$

Where in the semi-discretized equation (1), $\mathbf{M}_s$ is the mass matrix, $\mathbf{K}_s$ is the stiffness matrix and $\mathbf{F}_s$ is the vector of the external force.

Finally the implicit Newmark scheme is used as temporal discretizaton. This one is second order and unconditionally stable. Thus the discretized structural state vector at

the instant $t^{(n+1)}$ is written as :

$$\mathbf{a}_s^{(n+1)} = \tilde{\mathbf{M}}_s^{-1}\left(\mathbf{F}^{(n+1)} - \mathbf{K}_s\mathbf{d}_{pred}^{(n+1)}\right) \tag{2}$$

$$\mathbf{v}_s^{(n+1)} = \underbrace{\mathbf{v}_s^{(n)} + \frac{\Delta t}{2}\mathbf{a}_s^{(n)}}_{\mathbf{v}_{pred}^{(n+1)}} + \frac{\Delta t}{2}\mathbf{a}_s^{(n+1)} \tag{3}$$

$$\mathbf{d}_s^{(n+1)} = \underbrace{\mathbf{d}_s^{(n)} + \Delta t\mathbf{v}_s^{(n)} + \frac{\Delta t^2}{4}\mathbf{a}_s^{(n)}}_{\mathbf{d}_{pred}^{(n+1)}} + \frac{\Delta t^2}{4}\mathbf{a}_s^{(n+1)} \tag{4}$$

where $\tilde{\mathbf{M}}_s = (\mathbf{M}_s + \frac{\Delta t^2}{4}\mathbf{K}_s)$.

### 2.2 Fluid sub-domain

Regarding the hypotheses, the Euler equations (mass, momentum and energy conservation) are used over the fluid sub-domain. Thus, the quantity of interest is the vector of the conservative variables $\mathbf{U}_f = \left[\rho_f, \rho_f\underline{v}_f, E_f\right]^T$ admissible on $\Omega_f \times [0,T]$.

The Euler equations are traditionally written using the Eulerian Formulation for the fluid problem. In other words, the laboratory referential, fixed over the time, is used. Nevertheless, dealing here with FSI problems without overlap, a dynamic referential has to be used to fit at each instant at the boundary $\Gamma_{FSI}$. The Lagrangian formulation being not really well adapted for fast transient fluid problems, the ALE formulation is used. The ALE grid is defined thanks to its velocity $\underline{w}$. This one is Lagrangian at the domain's boundaries; defined by the boundaries conditions on $\Gamma_f$ and by the velocity continuity condition on $\Gamma_{FSI}$. On $]\Omega_f[$, the interior fluid domain, $\underline{w}$ is arbitrary defined using any desired law, under the CFL condition. Thus the continuous ALE formulation of the Euler equations is :

$$\frac{\partial J\mathbf{U}_f}{\partial t} + J\nabla.(\mathbf{F}_f - \underline{w}\mathbf{U}_f) = \mathbf{0} \tag{5}$$

Where $J$ is the Jacobian of the frame transformation of the Eulerian space to ALE space, and $\mathbf{F}_f$ the flux vector define as $\mathbf{F}_f = \left[\rho\underline{v}, \rho\underline{v}\otimes\underline{v} + p\underline{\underline{I}}, (E+p)\underline{v}\right]^T$. We also introduce the following notation $\tilde{\mathbf{F}}_f = \mathbf{F}_f - \underline{w}\mathbf{U}_f$
Using the integral form of the equation (5) and the finite volumes method cell centered, the semi-discretized Euler equations are written over one finite volumes $V_i$ with boundary $\Gamma_{V_i}$ as:

$$\frac{\partial \Delta V_i(t)\mathbf{U}_{f_i}(t)}{\partial t} = -\sum_{\forall\Gamma_{ih}\in\Gamma_{V_i}} s_{ih}(t)\tilde{\mathbf{F}}_{f_{ih}}(t) \qquad \forall i \in [1,...,n_{cell}], \quad \forall t \in [0,T] \tag{6}$$

Where $n_{cell}$ is the number of cells over the sub-domain $\Omega_f$. $\Delta V_i$ is the volume of the $i^{th}$ cell and $s_{ij}$ the area of the boundary between the $i^{th}$ and the $j^{th}$ neighbourhood cell,

these two values being variable due to the use of the ALE formulation. Finally, $\tilde{\mathbf{F}}_{ij}$ is the numerical flux between the actual cell $i$ and the adjacent cell $j$ defined here using the Roe flux difference splitting method [12].
The explicit second order Runge-Kutta scheme is used for the temporal discretization of the equation (6).

$$\Delta V_i^{(n+\frac{1}{2})}\mathbf{U}_{f_i}^{(n+\frac{1}{2})} = \Delta V_i^{(n)}\mathbf{U}_{f_i}^{(n)} - \frac{\Delta t}{2}\sum_{\forall \Gamma_{ih}\in\Gamma_{V_i}} s_{ih}^{(n)}\tilde{\mathbf{F}}_{f_{ih}}^{(n)} \tag{7}$$

$$\Delta V_i^{(n+1)}\mathbf{U}_{f_i}^{(n+1)} = \Delta V_i^{(n)}\mathbf{U}_{f_i}^{(n)} - \Delta t\sum_{\forall \Gamma_{ih}\in\Gamma_{V_i}} s_{ih}^{(n+\frac{1}{2})}\tilde{\mathbf{F}}_{f_{ih}}^{(n+\frac{1}{2})} \tag{8}$$

The finite volumes cell centered method, defines the vector of the conservative variables at the center of each cell. In this way, to compute the numerical flux at the boundary of the fluid sub-domain, it is necessary to use ghost cells, defined by extrapolation. Hence, the variables at the interface $\Gamma_{FSI}$ are not exactly known, which induces an approximation for the velocities continuity condition.

To avoid this problem, the fluid equations are also written using the finite volumes method vertex centred and the same temporal discretization. Thus the velocity at the boundaries will be computed instead of being extrapolated. The equations are written in the same way as the equations (7) and (8) except that they are written for $i \in [1, ..., n_{vertex}]$ where $n_{vertex}$ is the number of vertex.

## 3 COUPLING METHOD

### 3.1 Multi-time step monolithic formulation

The GC coupling method is based on a dual formulation of the Schur complement, that ensures the velocities continuity through the interface $\Gamma_{FSI}$. In this way, the Lagrangian multiplier method is used. Thus the cinematic condition is written as :

$$\mathbf{L}_s\mathbf{v}_s(t) + \mathbf{l}_f\mathbf{v}_{f_{vertex}}(t) = \mathbf{0} \tag{9}$$

Where $\mathbf{L}_s$ is the row selection vector of the finite elements nodes being at the fluid-structure interface, its size being the number of nodes. In the same way, the operator $\mathbf{l}_f$ is the row selection vector of the vertex of the finite volume included in $\Gamma_{FSI}$. When the discretized grids of the fluid and the structural sub-domain are compatible, these operators are Boolean.
Finally, based on the energy method, the semi-discretized balance equations of fluid and structural sub-domains are written using Lagrange multipliers, thus the condition of ve-

locities continuity is ensured :

$$\mathbf{M}_s\mathbf{a}_s(t) + \mathbf{K}_s\mathbf{d}_s(t) = \mathbf{F}_s(t) - \mathbf{L}_s^T\Lambda(t) \tag{10}$$

$$\frac{\partial \Delta V_i(t)\mathbf{U}_{f_i}(t)}{\partial t} = -\sum_{\forall \Gamma_{ih} \in \Gamma_{V_i}} \left(s_{ih}(t)\tilde{\mathbf{F}}_{f_{ih}}(t) + \mathbf{L}_{f_{ih}}^T\Lambda_{ih}(t)\right) \tag{11}$$

The Lagrange multipliers represent the interaction force between the two sub-domains. $\mathbf{L}_f$ is a $(n_{vertex} \times 3)$ matrix, where the first and the last columns are zeros vector and the second columns is the transpose of the selection vertex vector $\mathbf{l}_f$. In this way, the matrix $\mathbf{L}_f = [\mathbf{0}, \mathbf{l}_f^T, \mathbf{0}]$, allows taking into account the interaction force for the cells with one or more vertex included inside $\Gamma_{FSI}$, in the momentum conservation equation. Thus the equations (9), (10) and (11) are the semi-discretized monolithic system.

Concerning the temporal discretization, the choice is to use a proper temporal scale on each sub-domain. Regarding the hypotheses and the temporal discretization schemes, the fluid sub-domain is discretized using a micro time scale while the structural sub-domain is discretized using a macro time scale. Let's calling $\Delta t$ the micro time step between the instants $t^{(j-1)}$ and $t^{(j)}$, and $\Delta T$ the macro-time step between the instants $t^{(0)}$ and $t^{(m)}$ such as $\Delta T = m\Delta t$. Finally, applying these time-scales to the discretization methods presented in the previous sections (equations (2), (3), (4), (7) and (8)) for the equations (10) and (11) and selecting the velocity continuity at the micro-time step for the equation (9), the following discretized system is obtained :

$$\begin{cases} (\mathbf{M}_s + \frac{\Delta T^2}{4}\mathbf{K}_s)\mathbf{a}_s^{(m)} = -\mathbf{K}_s\mathbf{d}_{pred}^{(m)} - \mathbf{L}_s^T\Lambda^{(m)} \\ \Delta V_i^{(j-\frac{1}{2})}\mathbf{U}_{f_i}^{(j-\frac{1}{2})} = \Delta V_i^{(j-1)}\mathbf{U}_{f_i}^{(j-1)} - \frac{\Delta t}{2}\sum\left(s_{ih}^{(j-1)}\tilde{\mathbf{F}}_{f_{ih}}^{(j-1)} + \mathbf{L}_{f_{ih}}^T\Lambda_{ih}^{(j-\frac{1}{2})}\right) \\ \Delta V_i^{(j)}\mathbf{U}_{f_i}^{(j)} = \Delta V_i^{(j-1)}\mathbf{U}_{f_i}^{(j-1)} - \Delta t\sum\left(s_{ih}^{(j-\frac{1}{2})}\tilde{\mathbf{F}}_{f_{ih}}^{(j-\frac{1}{2})} + \mathbf{L}_{f_{ih}}^T\Lambda_{ih}^{(j)}\right) \qquad j = 1, .., m \\ \mathbf{L}_s\mathbf{v}_s^{(j)} + \mathbf{l}_f\mathbf{v}_{f_{vertex}}^{(j)} = \mathbf{0} \end{cases} \tag{12}$$

### 3.2 Co-simulation algorithm

To solve the system (12), several tricks are used. First, the discretized structural velocities at the interface are defined at the Runge-Kutta mid-step and at the micro-time step. A simple linear interpolation is used, such as :

$$\mathbf{L}_s\mathbf{v}_s^{(j-\frac{1}{2})} = (1 - \frac{j-\frac{1}{2}}{m})\mathbf{L}_s\mathbf{v}_s^{(0)} + \frac{j-\frac{1}{2}}{m}\mathbf{L}_s\mathbf{v}_s^{(m)} \tag{13}$$

$$\mathbf{L}_s\mathbf{v}_s^{(j)} = (1 - \frac{j}{m})\mathbf{L}_s\mathbf{v}_s^{(0)} + \frac{j}{m}\mathbf{L}_s\mathbf{v}_s^{(m)} \tag{14}$$

The same kind of interpolation is used to define the Lagrange multipliers.

The second trick is to split the fluid and the solid variables into a part called *"free"*, which is equivalent to the predictor state, without interaction at the interface, and a part

called *"link"* representing the corrected state taking into account the interaction force from the interface. Thus the structural sub-domain is re-written as :

$$\begin{aligned}
\mathbf{a}_{s_{free}}^{(m)} &= \tilde{\mathbf{M}}_s^{-1}\big(\mathbf{F}^{(m)} - \mathbf{K}_s\mathbf{d}_{pred}^{(m)}\big) & \mathbf{a}_{s_{link}}^{(m)} &= -\tilde{\mathbf{M}}_s^{-1}\mathbf{L}_s\Lambda^{(m)} \\
\mathbf{v}_{s_{free}}^{(m)} &= \mathbf{v}_{pred}^{(m)} + \frac{\Delta T}{2}\mathbf{a}_{s_{free}}^{(m)} & \mathbf{v}_{s_{link}}^{(m)} &= \frac{\Delta T}{2}\mathbf{a}_{s_{link}}^{(m)} \\
\mathbf{d}_{s_{free}}^{(m)} &= \mathbf{d}_{pred}^{(m)} + \frac{\Delta T^2}{4}\mathbf{a}_{s_{free}}^{(m)} & \mathbf{d}_{s_{link}}^{(m)} &= \frac{\Delta T^2}{4}\mathbf{a}_{s_{link}}^{(m)}
\end{aligned} \tag{15}$$

Following the same idea on $\Omega_f$ :

$$\begin{aligned}
\big(\Delta V\mathbf{U}\big)_{f_{free_i}}^{(j-\frac{1}{2})} &= \Delta V_i^{(j-1)}\mathbf{U}_{f_i}^{(j-1)} - \frac{\Delta t}{2}\sum s_{ih}^{(j-1)}\tilde{\mathbf{F}}_{f_{ih}}^{(j-1)} \\
\big(\Delta V\mathbf{U}\big)_{f_{free_i}}^{(j)} &= \Delta V_i^{(j-1)}\mathbf{U}_{f_i}^{(j-\frac{1}{2})} - \Delta t\sum s_{ih}^{(j)}\tilde{\mathbf{F}}_{f_{ih}}^{(j-\frac{1}{2})} \\
\big(\Delta V\mathbf{U}\big)_{f_{link_i}}^{(j-\frac{1}{2})} &= -\frac{\Delta t}{2}\sum \mathbf{L}_{f_{ih}}^T\Lambda_{ih}^{(j-\frac{1}{2})} \\
\big(\Delta V\mathbf{U}\big)_{f_{link_i}}^{(j)} &= -\Delta t\sum \mathbf{L}_{f_{ih}}^T\Lambda_{ih}^{(j)}
\end{aligned} \tag{16}$$

Lets recall that the first and the last columns of the selection operator are zero vectors, in this way $\big(\Delta V\rho\big)_{f_{link}} = \mathbf{0}$ and $\big(\Delta VE\big)_{f_{link}} = \mathbf{0}$.

Using the relation $\mathbf{U}_s = \mathbf{U}_{s_{free}} + \mathbf{U}_{s_{link}}$ respectively $\mathbf{U}_f = \mathbf{U}_{f_{free}} + \mathbf{U}_{f_{link}}$, $\Lambda$ is defined as :

$$\big(\mathbf{H}_s + \mathbf{H}_f^{(j)}\big)\Lambda^{(j)} = \mathbf{L}_s\mathbf{v}_{s_{free}}^{(j)} + \mathbf{l}_f\mathbf{v}_{vertex}{}_{f_{free}}^{(j)} \tag{17}$$

The equation (17) is written in the same way at the half micro-time step $t^{(j-\frac{1}{2})}$.

Then, we define:

$$\begin{aligned}
\mathbf{H}_s &= \frac{1}{2}\Delta T\mathbf{L}_s\tilde{\mathbf{M}}_s^{-1}\mathbf{L}_s^T \\
\mathbf{H}_f^{(j-\frac{1}{2})} &= \frac{1}{2}\Delta t\mathbf{l}_f\mathbf{M}_f^{(j-\frac{1}{2})-1}\mathbf{l}_f^T \\
\mathbf{H}_f^{(j)} &= \Delta t\mathbf{l}_f\mathbf{M}_f^{(j)-1}\mathbf{l}_f^T
\end{aligned}$$

Where the mass fluid matrix $\mathbf{M}_f$ is defined as a diagonal matrix $(n_{vertex} \times n_{vertex})$ with $\mathbf{M}_{f_{ii}} = \big(\Delta V\rho\big)_{f_i}$.

Finally the last trick is to keep the conservative variable and the cell volumes grouped. After the computation of $\big(\Delta V\mathbf{U}\big)_f$, the fluid velocity at the interface taking into account the structural interaction is known using the following relation :

$$\mathbf{v}_{f_{vertex}} = \frac{\big(\Delta V\rho\mathbf{v}\big)_{f_{vertex}}}{\big(\Delta V\rho\big)_{f_{vertex}}}$$

The monolithic coupling algorithm for finite elements / finite volumes vertex centred, is then completely defined. For the coupling method, using the finite volume cell centered, we need to define the fluid velocity at the boundary $\Gamma_{FSI}$. A simple first order extrapolation from the interior cells is used.
The complete algorithm is defined on the fig. (3). The global idea is the following; the *free* structural variable are computed at the macro time-step, the *free* fluid variables are computed at the half micro time-step, then the structural velocity at the interface is interpolated at the half micro-time step. Using the *free* velocities (fluid and solid) the Lagrange multiplier is computed, using it the *link* fluid state is known and finally the total fluid state at the half-micro time step. The process is repeated at each micro-time step until $j = m$. Then, at the macro-time step, the *link* structural state is computed in addition to the fluid one.

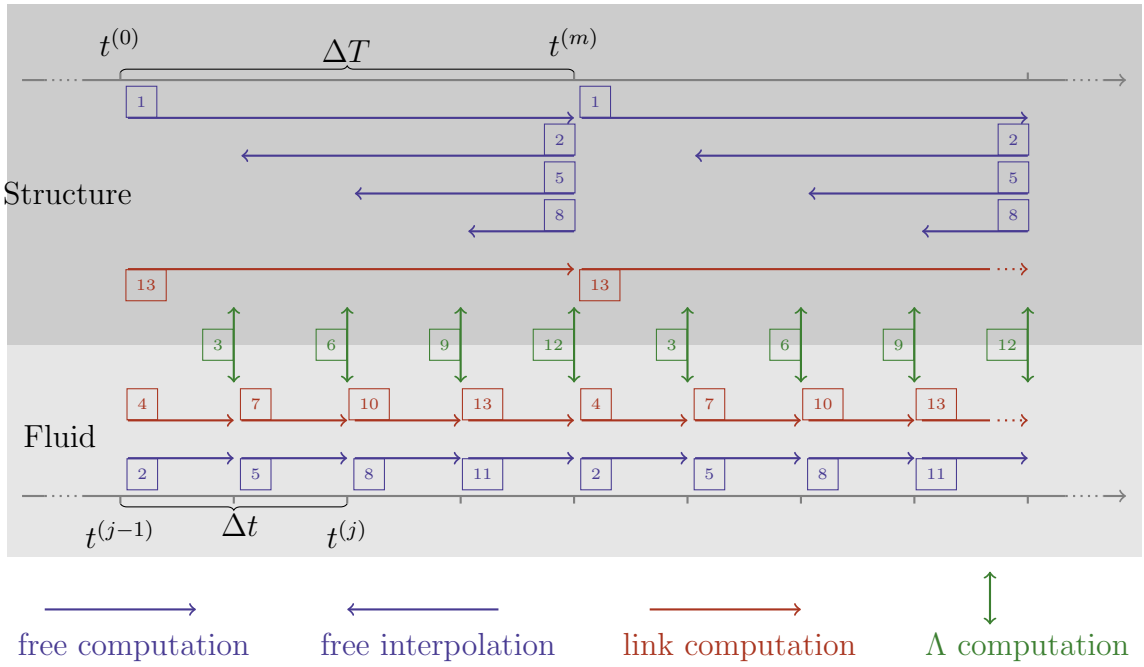


**Figure 3**: Multi time-step algorithm

## 4 NUMERICAL TEST CASE

To validate the method presented below, the piston test case is used. This one has been introduced by Piperno in 1995 [11]. Later, it has often been used to validate new FSI methods [2], [6]. It is really simple, an aero-elastic problem in one dimension. The set of parameters used here are the same as those proposed by Michler [9].

### 4.1 Presentation

The problem is considered in 1D. A gas is enclosed inside a cavity closed at the left side by a wall and at the right side by a mass-spring system. Its length is $1m$.
The fluid domain $\Omega_f$ is composed of air (specific heat ratio $\gamma_{air} = 1.4$ and celerity $c_{air} = 340m.s^{-1}$). Thus the fluid is considered a perfect gas compressible and inviscid. Moreover the thermal exchanges are neglected. The structure domain $\Omega_s$ is a simple mass-spring system, with a mass of $m_s = 0.8Kg$ and a stiffness of $k_s = 7991N.m^{-1}$.
At the beginning of the simulation, the piston is at rest. The pressure inside the cavity is $p_f^{(0)} = 10^5 Pa$, the atmospheric pressure. The velocity $v_f^{(0)}$ of the fluid is zero and its density is $\rho_f^{(0)} = 1.3Kg.m^{-3}$. The velocity of the structure $v_s^{(0)}$ system is initially zero and a displacement of $d_s^{(0)} = 0.01m$ is imposed on it.

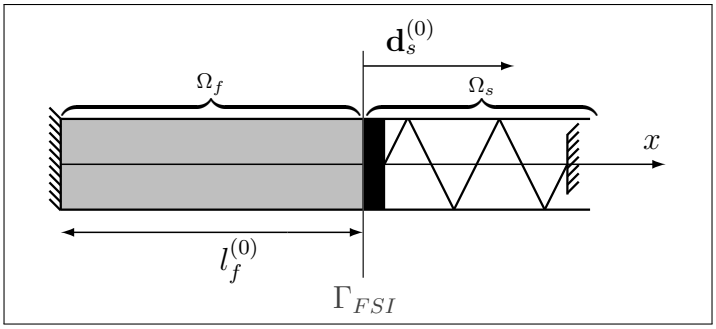


**Figure 4**: Piston au pas de temps initial

### 4.2 Results

First, the results obtained for the monolithic coupling method mono time step for finite element/finite volumes vertex centered are presented.
The Fig. (4.2) shows the position of the interface $\Gamma_{FSI}$ during time. The amplitude is globally constant, it seems that there is no significant loss or gain of energy. However a slight loss of amplitude, around $0,01\%$ after 20 cycle, is noticed. Nevertheless, this error could be correlated to the error induced by the simulation of the same fluid domain alone, where the right side is closed by a moving wall governing by the same mass-spring system (this simulation is equivalent to the one-way coupling of the piston problem). In this way the error is induced by the fluid discretization methods, not by the coupling method.

Moreover the relative error to the velocities continuity is also studied. We define $err = \frac{|\mathbf{L}_s \mathbf{v}_s^{(n)} - \mathbf{L}_f \mathbf{v}_{f_{vertex}}^{(n)}|}{max(\mathbf{v}_{FSI})}$. This one is of the order of $10^{-16}$ what is satisfying, see Fig.(6).

Considering the coupling finite elements / finite volumes cell centered, the interface position is globally the same, nevertheless a little gain of amplitude is remarked over $0,03\%$ after 20 cycles. More over this error cannot be correlated to the fluid model alone. The relative velocity error between fluid and structure is here more important

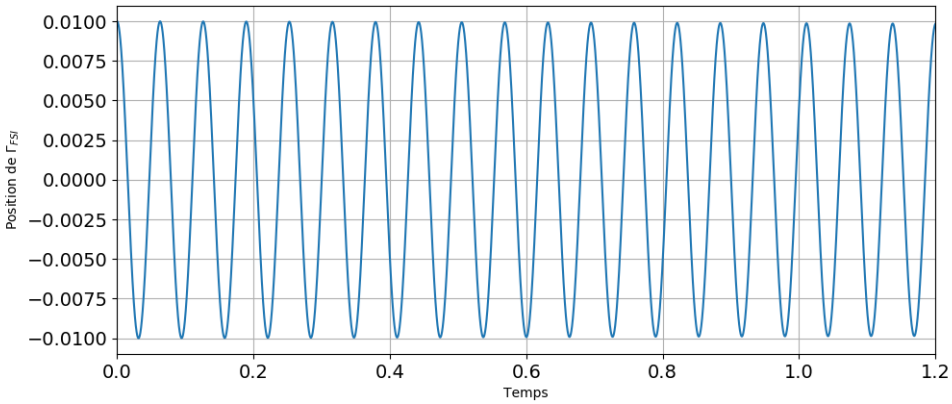


**Figure 5**: Interface position according time

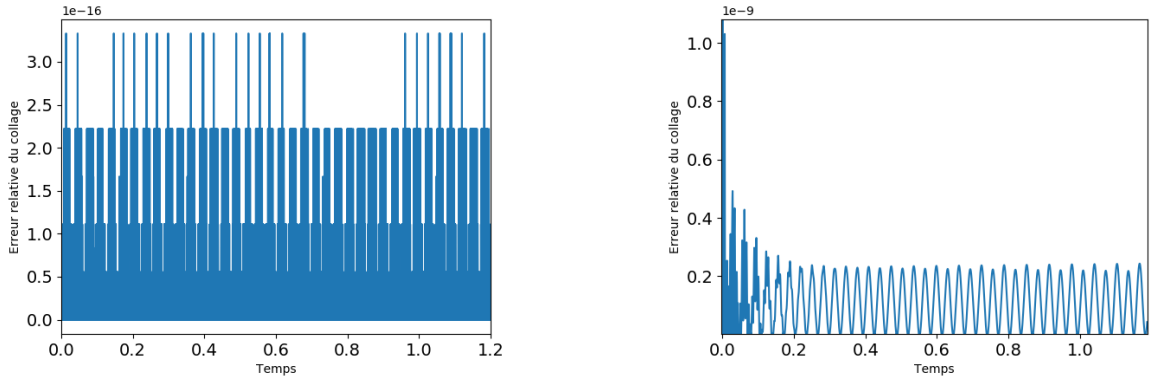


**Figure 6**: Relative error of the velocities continuity at the interface, left coupling of finite volume vertex centred, right coupling of finite volume cell centered

around $10^{-9}$. Let's recall that this coupling method introduced an approximation; the *free* fluid velocities at the interface are extrapolated. In this way, we can make the following hypothesis ; improving this extrapolation would improve the FSI results too.

Finally, the multi time-step method has also been studied for the finite elements / finite volumes vertex centred coupling. The results seem to be really promising because the quality is a few degraded even when the ratio between the micro and the macro time-step increased. For instance, we have remarked previously that for $m = 1$ (mono time-step case) the relative error of velocities was around $10^{-16}$. For $m = 100$, this error remains is the same magnitude .

## 5 CONCLUSION

To conclude, the presented method allows a co-simulation coupling and ensures the energy conservation and the velocity continuity through the interface, for a mono time step coupling of finite volumes vertex centered and finite elements. The results are weakly defaced for a coupling using finite volumes cell centered instead of vertex centered which is involved by the extrapolation of the fluid velocity from the cell center to the fluid-structure interface, useful for the coupling condition. Nevertheless this error remains limited. Finally the multi time-step coupling finite volumes vertex centred / finite elements have been studied. This one shows good results, the results deterioration is limited despite the ratio increase between the fluid and the structural time step.

The next step is to study the coupling of finite volumes cell centered and finite elements in 2D and using multi time-step. Even if the coupling using finite volumes vertex centered has shown better results, we will use finite volumes cell centered because the aim is to use the commercial software Fluent to compute the fluid sub-domain, this one is based one finite volumes cell centered method. The velocity continuity condition at the interface could be improved, using more complex extrapolation of the fluid variables.

## REFERENCES

[1] Beckert, Wendland. *Multivariate interpolation for fluid-structure-interaction problems using radial basis functions*, Aerospace Science and Technology,5(2), 125–134, 2001.

[2] Blom. *A monolithical fluid-structure interaction algorithm applied to the piston problem*,Computer Methods in Applied Mechanics and Engineering, 167(3), 369–391, 1998.

[3] Casadei, Leconte. *Coupling finite elements and finite volumes by Lagrange multipliers for explicit dynamic fluidstructure interaction*, International Journal for Numerical Methods in Engineering, 86(1), 1–17, 2011

[4] Gravouil, Combescure. *Multi-time-step explicitimplicit method for non-linear structural dynamics*, nternational Journal for Numerical Methods in Engineering, 50(1), 199-225, 2001.

[5] Gravouil, Combescure, Brun. *Heterogeneous asynchronous time integrators for computational structural dynamics*, Int. J. Numer. Meth. Engng, 102(3), 2015.

[6] Ischinger, Anthonissen, Koren. *A monolithic fluid-structure interaction method, application to a piston problem*, ECCOMAS congress, 2016.

[7] Li, Leduc, Nunez-Ramirez, Combescure, Marongiu. *A non-intrusive partitioned approach to couple smoothed particle hydrodynamics and finite element methods for*

*transient fluid-structure interaction problems with large interface motion*, Computational Mechanics, 55(4), 697–718, 2015.

[8] Meduri, Cremonesi, Perego, Bettinotti,Kurkchubasche ,Oancea . *Tpartitioned fully explicit Lagrangian Finite Element Method for highly nonlinear Fluid-Structure-Interaction problems*, Numerical Methods in Engineering, 113(1), 43–64, 2017.

[9] Michler, Hulshoff, van Brummelen, De Borst. *monolithic approach to fluidstructure interaction*, Computers Fluids, 33(5), 839–848, 2004

[10] Nobile. *Coupling strategies for the numerical simulation of blood flow in deformable arteries by 3D and 1D models* , Mathematical and Computer Modelling, 49(11), 2152–2160, 2009.

[11] Piperno, Farhat, Larrouturou. *Partitioned procedures for the transient solution of coupled aroelastic problems Part I: Model problem, theory and two-dimensional application*, Computer Methods in Applied Mechanics and Engineering,124(1), 79–112, 1995.

[12] Roe, *Characteristic-based schemes for the Euler equations. Annual review of fluid mechanics*, 18(1), 337-365, 1996.

[13] Ross, Sprague, Felippa, Park. *Treatment of acoustic fluid structure interaction by localized Lagrange multipliers and comparison to alternative interface-coupling methods*, Computer Methods in Applied Mechanics and Engineering, 198(9),968–1005, 2009.