FLUID STRUCTURE INTERACTION PROBLEMS MODELLING: FIRST APPLICATIONS AT CNES.

J.HERPE, S.PETITOT, Space Transport Systems, CNES Launcher Directorate, Evry, France,

ABSTRACT

This paper deals with the numerical methodologies used at CNES DLA to set up Fluid/Structure Interaction (FSI) calculations. The numerical strategy developed is based one the code coupling which allow the communication between two specific codes.

In fact, to model launcher's components, CNES uses various specialized codes (for thermal, mechanical, fluids, combustion applications…), to seek individual solutions of the highly complex problem. Then a third code allowing communication between CFD and FEA (interpolations between different meshes) and exchanging shared variables (mainly pressures and displacements) is required for FSI problems.

One of the major application fields identified concerns the solid propulsion. Two problems are investigated to perform coupled calculations with increasing complexity. The first one is about the Solid Rocket Motor (SRM) ignition. This transient phase can be decomposed by successive steady states with internal motor pressure increasing step by step. This allows performing unidirectional coupling with both a script developed at CNES and the MpCCI software. The second one concerns thrust oscillations of SRM. A bidirectional approach is used to simulate the behaviour of a flexible obstacle, made of elastomer, protruding in a cold gas flow of an experimental test bench developed to reproduce oscillating pressure phenomena at a reduced scale.

1. INTRODUCTION

Launcher design, development activities and post flight analysis often help pointing out problems that involve strong coupling between different physical behaviour. Several examples can be found among the three main domains of the propulsion :

- liquid propulsion: aero-elastic coupling between choc oscillations appearing inside the expanded nozzle and its wall deformations, aero-thermal solicitations leading to crack formations on injectors in cryogenic combustion chamber, aero-thermo-mechanical coupling involved on "side loads" phenomena with deformation of the cooling ducts along Vulcain 2 nozzle,
- cryogenics tanks: thermo-mechanical coupling for Internal Thermal Protections design, hydroelastic and sloshing vibrations which occur in stability studies ("pogo" effect, attitude control of launcher),
- solid propulsion: Aero-mechanics coupling for modelling “Front Thermal Protection” (PTF) behaviour (bending and vibrations) impacting on Pressure Oscillations phenomena, aero-mechanical coupling necessary for solid propellant mechanical diagnostic (initiation of cracks) during motor ignition (pressurisation),

Among these three domains, the solid propulsion focus our attention. In fact, for historical reasons, the Pressure Oscillations (OdP) appearing in Ariane 5 Solid Rocket Motors is a fundamental research subject at CNES/DLA, especially for a better understanding of the physical phenomena that lead to OdP. Moreover, from a numerical point of view the solid applications leads us to improve our simulation strategy due to the kind of the phenomena encountered in solid propulsion application.

To perform fluid and solid simulations, specific codes are usually employed at CNES. Then, in order to model the fluid structure interaction a third code allowing communication between CFD and FEA is required to exchange shared variables (mainly pressure and displacement values). In this paper Fluent [1] is used for the fluid simulation and Abaqus [2] is used for the structural analysis. Two methods are studied for coupling aspects. The first one is based on a script developed at CNES. The second one is based one the MpCCI [3] software that already includes adapters for the commercials codes Fluent and Abaqus..

In a general manner, when fluid structure interaction phenomenon occurs, the flow pressure bend the structure that can modify, in exchange, the flow field. According to the relevance of the impact of the structural bending on the flow pressure, it is possible to consider or ignore in the numerical exchange scheme (figure 1). In the case of a one a way coupling scheme the exchange is only done from the fluid to the solid (unidirectional coupling). In a two way coupling the exchange is done in both directions (bidirectional coupling).

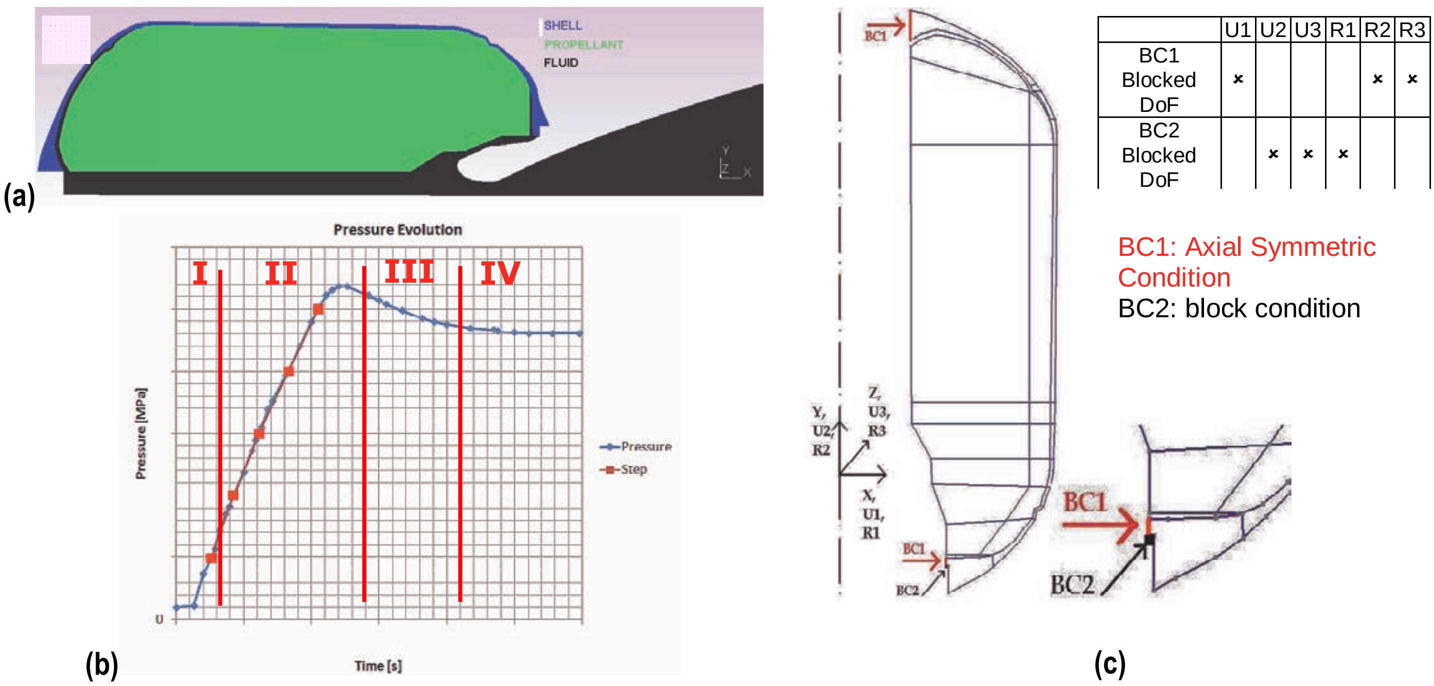
(a) (b)

Figure 1: (a) One Way coupling scheme (b) Two way coupling scheme.

Then this article focuses on two pure numerical cases. The first one is related to a motor ignition problem. The influence of the structure on the fluid flow is neglected and a one way coupling approach is used. A specific script of exchange values developed at CNES is done. It allows to demonstrate the feasibility and to point out the main difficulties. Then a comparison is made with calculation via MpCCI. The second case is considering a two way coupling FSI problem. This numerical case is based on the MICAT bench test from Univ. Poitiers, France, that allows to investigate the coupling between a flexible obstacle (elastomer) movement and a fluid flow.

2. ONE WAY COUPLING APPLICATION

The first numerical application is related to the study of the ignition phase (phase II of figure 2b), or pressurisation of a solid rocket motor (figure 2a-b). The very fast increase of pressure in the motor induces stresses on the propellant block(s) and strain of the burning surfaces, modifying boundary limits for the fluid flow. The regression of the propellant block due to the combustion is not taken into account. In fact, the characteristic time of the phase II is very short compared with the characteristic time of the burning surface regression. Then, this study focuses on the strain due to the pressure loads.



	U1	U2	U3	R1	R2	R3
BC1 Blocked DoF	×				×	×
BC2 Blocked DoF		×	×	×		

Figure 2: **(a)** SRM configuration, **(b)** Pressure evolution in the SRM, **(c)** Boundary condition for the structural

A simplified 2D axisymmetric model is studied. The fluid and solid meshes, which are fitted at the fluid/structure interface, (figure 3) are done in the Gmsh software [4] and are exported into the corresponding software.

Hybrid elements are used to model propellant behaviour as it is fully incompressible, while incompatible mode elements are used to mesh shell part. The mechanical behaviour of propellant and shell are based on the hypothesis of a linear elastic law. The boundary conditions imposed on solid are plotted one figure 2c.

The fluid model is performed using a pressure based implicit solver with pressure-velocity coupled algorithm. 2nd order schemes are used for pressure, density, velocity and temperature discretization algorithms. The boundary conditions are obtained using the pressure curve of the figure 2b and the classical internal ballistic laws of the SRM. A mass flow inlet is imposed on the fluid/propellant interface. At the outlet of the nozzle the pressure is fixed. The other fluid boundary conditions are supposed to be adiabatic walls. We suppose that the flow is a non-reactive monophasic ideal gas flow. The viscosity, the thermal conductivity and the specific heat at constant pressure are supposed to be constant.

As a first approximation, a "step by step" approach is used to model the pressurization of the SRM. For each of the five operating points (red squares of the figure 2b), a one way co-simulation is done. From the pressure value of the operating point and with the classical ballistic laws of the SRM, the flow boundary conditions are obtained. A steady state simulation is done in Fluent. The surface forces obtained are only transferred once from the fluid model to the solid model (Abaqus), where they are applied as boundary conditions to compute the stress distribution. At the following operating point, the fluid model take into account the deforming surface of the propellant block and the new boundary conditions calculated with the internal ballistic laws. So the pressure load obtained is sent to the mechanical model. This procedure is done until the fifth step.

Two methodologies for the data exchange, from the fluid model to the solid model, are explained in the following paragraph. The first one is based on a script developed at CNES, while the second one is based on the MPCCI software.

2.1 Script methodology

For each point corresponding to a given motor pressure (figure 2b) the script procedure is the following:

1⇨ Boundary conditions: for a given operating pressure the inlet mass flux boundary conditions are calculated with the classical internal ballistic laws,
2⇨ Fluid simulation: A steady state calculation for the fluid flow is performed using Fluent,
3⇨ Exchange procedures: The pressure distribution along propellant surface(s) is read with a specific Fluent User Define Function (UDF) developed for this purpose. The specific Python script is used to match the CFD pressure values with finite elements and to generated the input file for structural analysis,
4⇨ Solid simulation: the deformed configuration of the propellant block(s) running the FE structural analysis and update the new geometry,
5⇨ Remeshing procedure: the fluid domain is manually remeshed with the "deformed" boundary limits
6⇨ The step 1 is run again with the following internal pressure value.

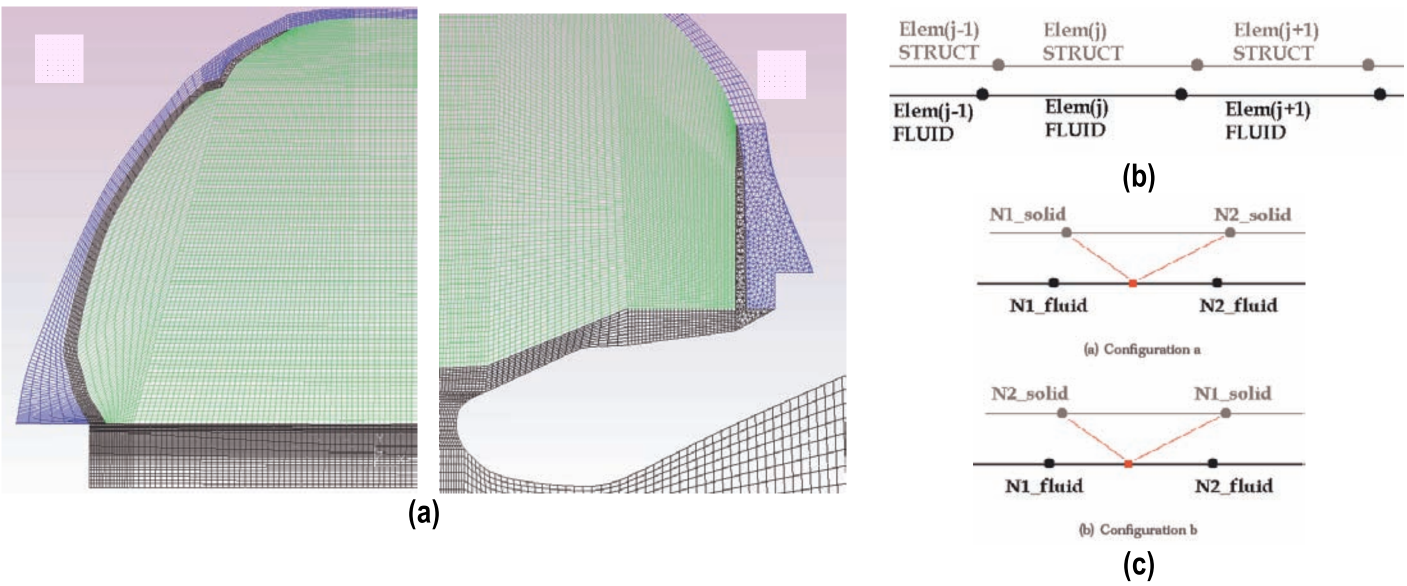


Figure 3: **(a)** Mesh details at the top and the bottom of the SRM, **(b)** Scheme of the discrepancy due to the round of error when importing the Gmsh mesh in Fluent and Abaqus **(c)** Central Point Distance Evaluation procedure.

The exchange procedure (step 3) is strongly linked to the mesh. If the fluid and propellant mesh are not conformal, it is necessary to develop interpolation algorithms to exchange values. Also, as a first simplified approach, it has been chosen to use a conformal mesh at the fluid/structure interface (figure 3a) to ensure the exact matching. But, even though the fluid and the propellant mesh have been both generated in Gmsh, the export operation in Fluent and Abaqus leads to some discrepancies (figure 3b)., probably due to round off error. This leads us to develop a script with a central point distance evaluation algorithm (figure 3c). The principle is based on the calculation of the distances between the central points of fluid face elements and the nodes of structural elements and to find the nearest structural element for a given fluid one (Fluent UDF lists also the fluid nodes coordinates while recording pressure values). It is not influenced by node configuration and doesn't need to reconstruct any mesh information ("low" computer memory cost). This algorithm can also be used for a 3D mesh, with the evaluation of three (triangular element at the surface) or four (hexahedral) distances.

2.2 MpCCI methodology

The same calculation is performed with the MpCCI code ensuring coupling between CFD and FEA. Data exchanges are then directly performed between the two conformal meshes by MpCCI, without any customer's intervention. The "step by step" process is slightly different from the one used with the script method due to the specificities of the "MpCCI" coupling procedure. The main differences are in the step 3 that include the step 3, 4 and 5 of the previous script methodology. This is done thanks to the automatic procedure of MpCCI.

1⇨ <u>Boundary conditions:</u> For the given functioning motor pressure, internal ballistic is evaluated. Inlet mass fluxes boundary conditions are calculated,
2⇨ <u>Fluid simulation:</u> A steady calculation for the fluid flow is performed using Fluent in order to evaluate the pressure distribution along propellant surface(s),
3⇨ <u>Exchange procedure, FE simulation and remeshing:</u> After the starting of the MpCCI server, the exchange is interactively done with the graphical user interface of Fluent. The solution is first initialized with the corresponding button in the "on demand init and exit" portion of the MpCCI control panel. The pressure is sent to Abaqus, and the node position is sent to Fluent by clicking on the "exchange" button of the panel. With the pressure load sent by Fluent UDF the FE simulation can be done and node displacement can be calculated and sent to Fluent. The "steady update mesh" button allows us to update the Fluent mesh. Eventually, the "Finalize" button is used to finish the simulation.
4⇨ The new Fluid mesh can be used for the next step.

At the step i+1, it is necessary for the structural analysis to take into account the solution, obtained at the step i, (not only the propellant node displacement but also the field of strain and stress). In order to do it, and due to the fact that the MpCCI procedure is not able to take into account the multi-step procedure of Abaqus, the restart procedure of Abaqus is used.

2.3 Results

Comparison between both coupling methodologies is based on both Von Mises Stress values and displacements obtained respectively at probe elements (central points) and probe nodes in shell and propellant grain.

For displacement, a qualitative analysis can only be made as nodal displacements are little compared to the motor size. As shown on Figure 4, final position of the propellant grain surface on the rear part of the motor are quite similar for the two methodologies.

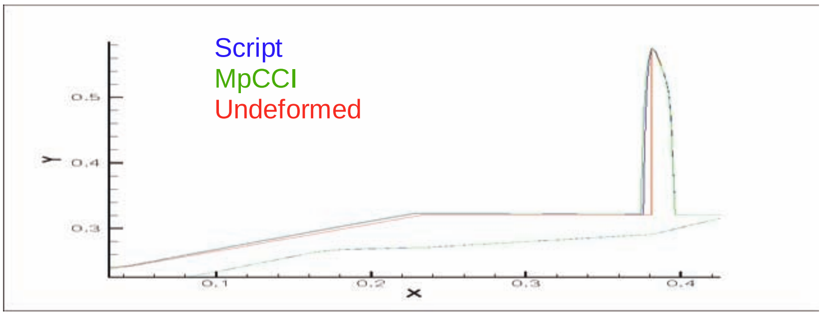


Figure 4: End surface position for both MpCCi and Script coupled calculations, zoom at the rear part of the motor where displacements are the most significant.

Also, an analysis of the Von Mises Stress field (figure 5) shows that the results from script and MpCCI methodology are comparable: good agreement between the two methodologies is found too considering Von Mises Stress Evolution with pressure increase (step) for points located inside the propellant, near the interface.

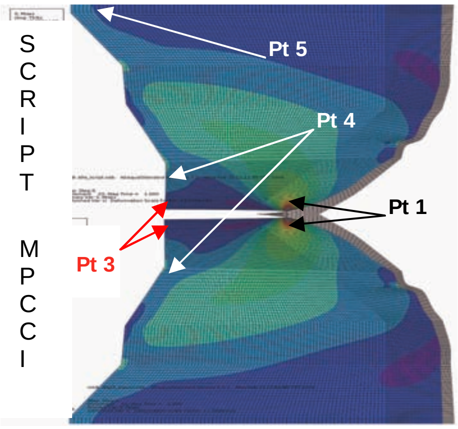
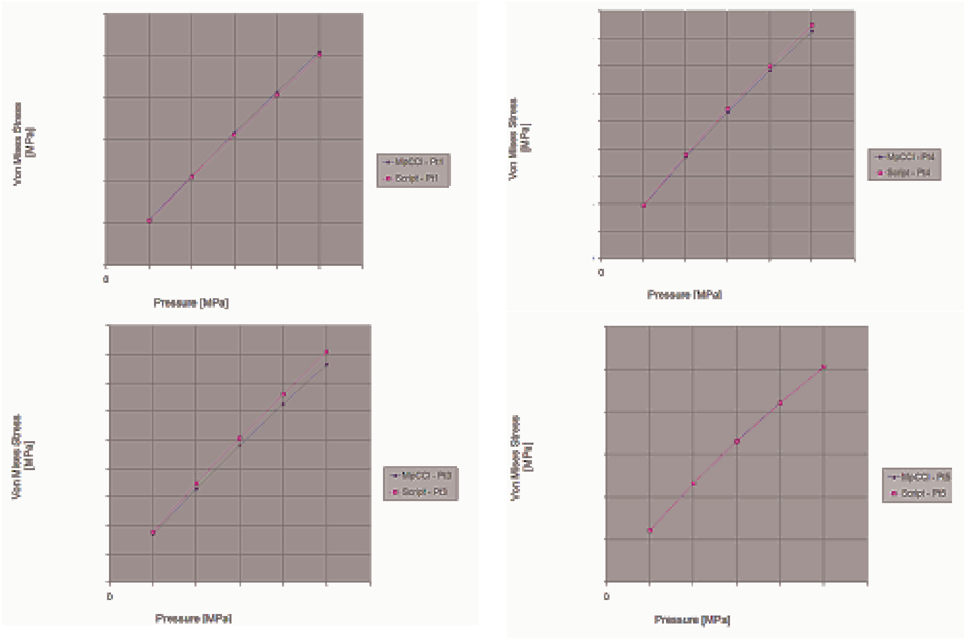


Figure 5: Comparison of Von Mises Stress field at rear part of the motor (above) and Von Misses Stress evolution with pressure rising (step) within some propellant located points.



So this first simplified "step by step" approach using MpCCI was validated with script's results comparison. Further works have to investigate a bidirectional coupling using unsteady calculations with MpCCI coupling only.

3. TWO WAY COUPLING

3.1 Description of the problem

The solid rocket motors of Ariane5 are made of three propellant blocks separated by thermal protections (PTS2 and PTS3). During the internal combustion, the fire front progresses and the thickness of the propellant blocks decreases. As a consequence, the thermal protection PTS3 protrudes and disturbs the gas flow, given rise of one of the major effects driving the pressure oscillations (figure 6). In fact, the vortex shedding obstacle (VSO) generated at the trailing edge of PTS3 impinges on the rocket motor nozzle. This interaction between the vortex and the nozzle is responsible for an acoustic feedback that excites the following detached VSOs.

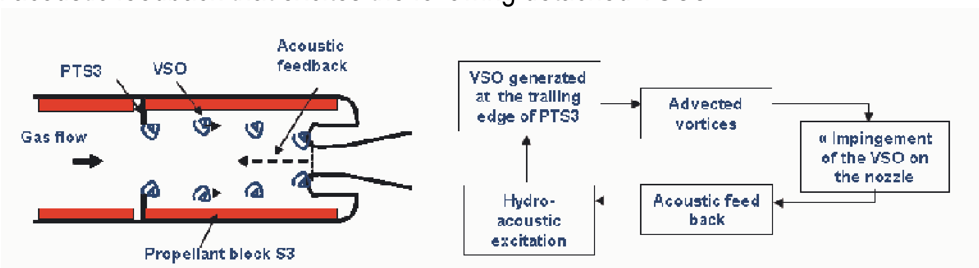
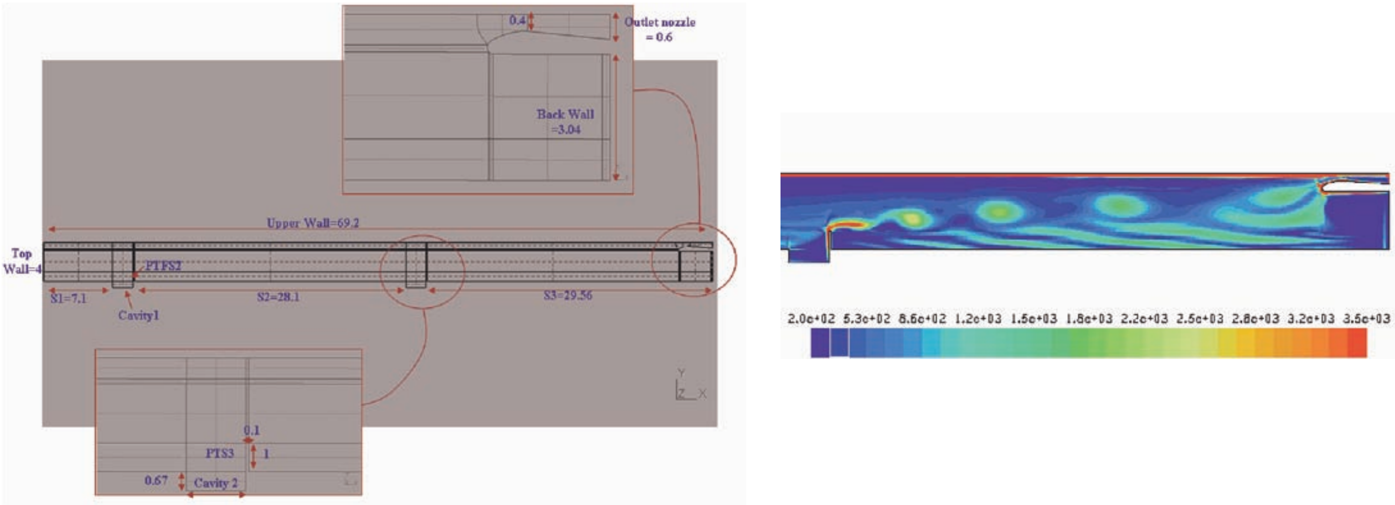


Figure 6: Pressure oscillations loop.

Cold-flow experiments [5-8] and numerical simulations [9-11] have been dedicated to study the influential parameters on pressure oscillations. Among them, the bending of the thermal protection PTS3, which is made of a flexible inhibitor, has been identified. But the quantification of the thermal protection bending by numerical simulation is still challenging despite the improvements of numerical analysis and computing power. In order to evaluate the ability of the approach, based on the Fluent/MpCCI/Abaqus coupling codes, to simulate this fluid/structure interaction (FSI) phenomenon, we focus this paper on the MICAT configuration test rig described in the following paragraph.

3.2 VSO study: the cold flow test bench MICAT

The MICAT configuration (7a) is an experimental cold air test bench developed to reproduced oscillating pressure phenomena at a reduced scale. The propellant block combustion is simulated by a parietal gas injection through a porous material. The rig is constituted by three propellant blocks, two thermal protections (sheet of a specific material) and a nozzle. The rig is adjustable in order to define different operating points corresponding to the different combustion times of a solid rocket motor.



(a) **(b)**

Figure 7: (a) Cold flow test bench MICAT (b) Contour of vorticity [s^{-1}] downstream the PTS3.

An operating point, near the end of combustion, also called resonant configuration, is numerically investigated afterwards (figure 7b). At this operating point, the propellant block S1 has completely burnt. Only the combustion of the propellant block S2 and S3 are taken into account. All the other surfaces of the rig are adiabatic walls. The PTS2 is supposed fixed and perpendicular to the axial flow direction. Due to its material characteristics the PTS3 could bend under the influence of gas flow pressure.

The acoustic response of the cavity MICAT is analysed with a pressure sensor set up at the top wall of the MICAT configuration (figure 7a).

3.2 Numerical FSI models

The fully coupled physical phenomena, summarized in the §3.1, lead to use the bidirectional approach available in the MpCCI software [3]. This co-simulation software linked the Fluent CFD code [1] (gas flow model) and the Abaqus Finite Elements code [2] (thermal protection PTS3 model). The models and the schemes used in each code, are described below.

3.2.1 Gas flow model

A 2D numerical model is used. The gas flow is a single phase, compressible (ideal gas) and laminar. The combustion reactions are not taken into account in the fluid model. The viscosity, the thermal conductivity and the specific heat at constant pressure are supposed to be constant.

The boundary conditions are the following: (i) on the adiabatic walls, the flux is set to zero and the no-slip shear condition is imposed, (ii) the mass flux is fixed on the injection side, (iii) the static pressure is fixed at the outlet of the nozzle (It is taken into account when the flow is subsonic. After the nozzle shock the pressure is extrapolated from the inlet motor condition).

The density based solver is used with a Roe scheme (Third order MUSCL discretization scheme). A Runge Kutta explicit time scheme is used. The Courant Friedrich Levy number is equal to 1.6. The Green Gauss gradient cell based option is used.

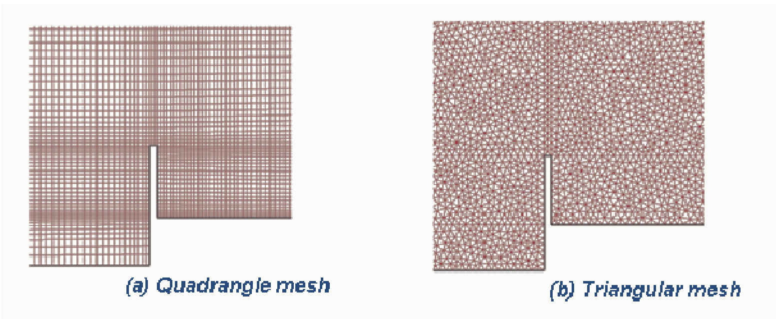


Figure 8: Gas flow meshes.

Two of the dynamic mesh methods available in Fluent are used in the FSI model. The first one is taken into account for the vibrating phenomena and the second one for the high strain. The first one called "the smoothing method" is based on the Hook's Law. The edges between any two mesh nodes are simulated as a network of interconnected springs characterized by their stiffness. A displacement at a given node will generate a force proportional to the displacement along all the springs connected to the node. This approach is useful for small deforming mesh. But in the case of high deformation it is necessary to use a remeshing method. The strategy of this second method consists in packing together the cells if specific criteria defined by the user (cells skewness …) are not respected. This approach only works with triangular meshes which constrains the mesh choice (figure 8b). This mesh constraint will be analysed by comparing the triangular mesh solution to the quadrangle mesh solution (figure 8a) in the case of a straight thermal protection.

3.2.2 Structural model

The thermal protection PTS3 is made of an arbitrary incompressible material (the Poisson coefficient is equal to 0.49). The mechanical behaviour is based on the hypothesis of a linear elastic law.

The boundary conditions for the PTS3 are the following: the foot of the PTS3 is fixed (U1=U2=U3=UR1= UR2=UR3=0).The implicit solver of Abaqus is used.

The mesh is made of the hybrid-reduced elements CPE4RH. If the structural mesh is fitted to the triangular fluid mesh (figure 9a), Hourglass problems can appear. Then, we use the mesh defined on the figure 9b.

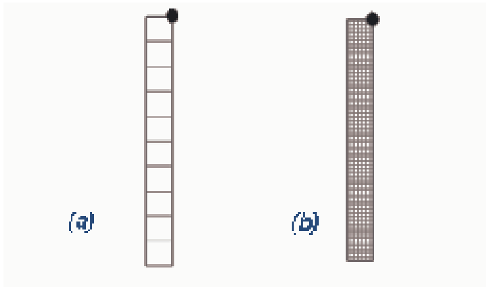


Figure9: PTS3 meshes.

The behaviour of the thermal protection is investigated at the black tip node of the figure 9.

3.2.3 Coupling model in MpCCI

As explained in previous paragraphs we use an explicit scheme for the fluid model and an implicit scheme for structural model. In such a case, bibliography [2] recommends to use a serial coupling scheme where the code using the explicit time integration is the leader code. In this case the code using explicit time integration computes the solution at t_{n+1} based on the solution at t_n and passes the forcing quantity to the implicit solver, which computes the solution at t_{n+1} (figure 10).

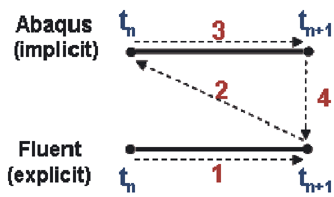


Figure10: Serial scheme.

In the fluid model the time step is not constant due to the explicit scheme used (CFL is imposed). Then, we don't use a constant coupling time step to define the rendezvous between Abaqus and Fluent. Nevertheless, the rendezvousing scheme could be defined in MpCCI specifying a variable coupling step size. In this case it is necessary to choose between Fluent or Abaqus to send or receive the time increment size. In our case, it is supposed that the sender is Fluent. Moreover it is enforced the exact target time by using the subcycling approach in Abaqus. The initial quantities transfer by Fluent is exchanged and received by Abaqus.

The values exchanged between the CFD code and the FE code during this process are respectively the pressure (Overpressure) and the node position (Nposition).

As the mechanical and the fluid meshes are not fitted, the exchange values are based on meshes interpolation using the minimal distance algorithm.

3.3 Flow model validation

The validation of the Fluent model is based on a numerical comparison with the CFD code CPS_P [12] dedicated to the solid rocket motor applications.

In this validation case the thermal protection PTS3 is not deformable and remains perpendicular to the axial flow direction. The CPS_P model is the same as Fluent. Both of the previous meshes are studied (figure 8). The nozzle is shocked and the pressure signal obtained in the sensor is established and periodic (figure 1**Erreur ! Source du renvoi introuvable.**1). The simulation is stopped when a sufficient number of periods are obtained to do a precise Fast Fourier Transform (FFT).

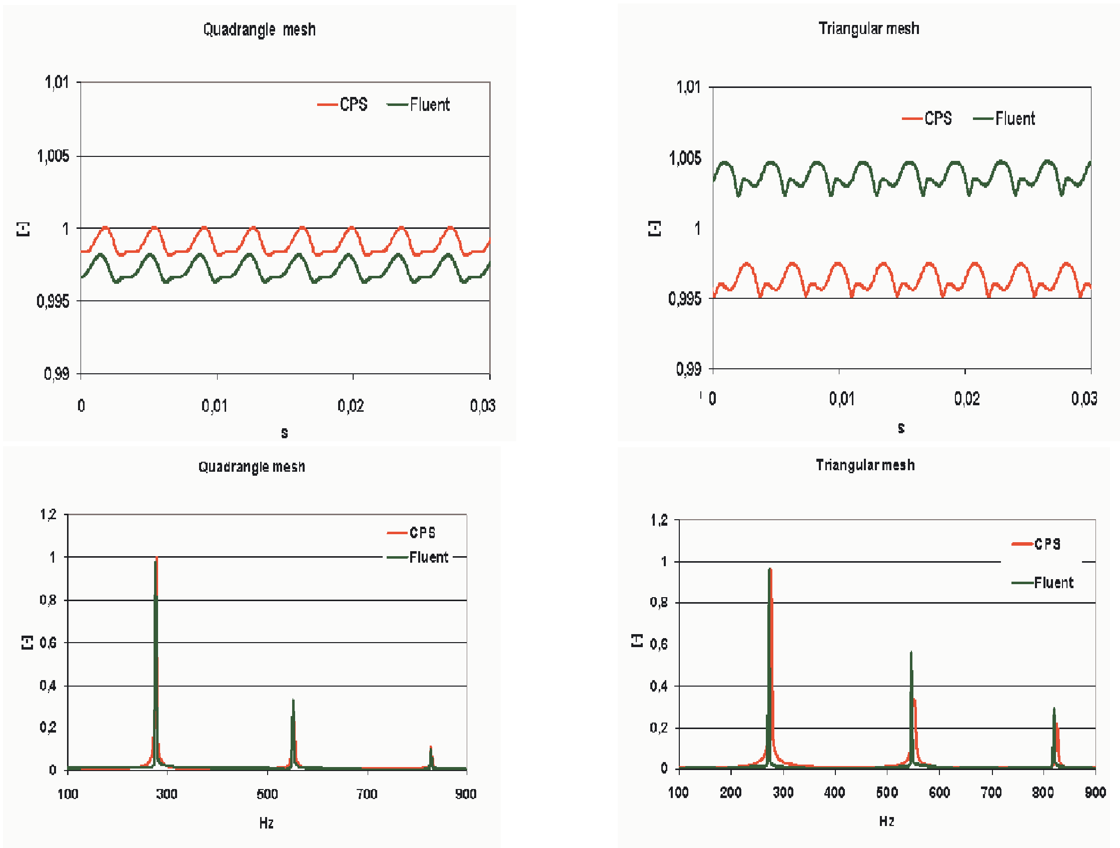


Figure 11: FFT of the pressure signal for the quadrangle and triangle mesh.

The results show a good agreement between CPS_P and Fluent in the case of the quadrangular mesh. The frequencies of acoustic modes are the same for the triangular and the quadrangular meshes and also for CPS_P and Fluent. There are some differences on the amplitude between quadrangular mesh and triangular mesh for both CFD codes. These differences are focused on the second and the third acoustic modes. The main difference between both CFD codes is on the second mode. As a first approximation, it is supposed that the Fluent model and the triangular mesh used are good enough. In fact, the pressure signal and the fundamental acoustic mode of the FFT are nearly the same for both meshes and both codes.

3.4 Co-simulation analysis.

Two strategies can be used for the co-simulation analysis. The first one consists in using the previous gas flow solution (§4) and to directly run the unsteady co-simulation. The second one consists in running a static co-simulation to obtain a first bending of the PTS3 and then to run an unsteady simulation from a converged gas flow solution. Due to the fluid time step, we have introduced the static stage to save CPU time.

For the static stage, the steady solver of Fluent and the static solver of Abaqus are used. In MpCCI, the exchange is manually done with the MpCCI control panel of the Fluent software, and no rendezvousing scheme are used. The quantities are still initially exchanged by Fluent and received by Abaqus. After this step, the unsteady co-simulation can be started by loading the corresponding Fluent case and using the restart procedure of Abaqus.

A middle analysis, at the end of the steady co-simulation, confirms the influence of the bending to the pressure signal (figure 12). The first mode amplitude of the Straight PTS3 is 20% higher than the amplitude of the Bending PTS3.

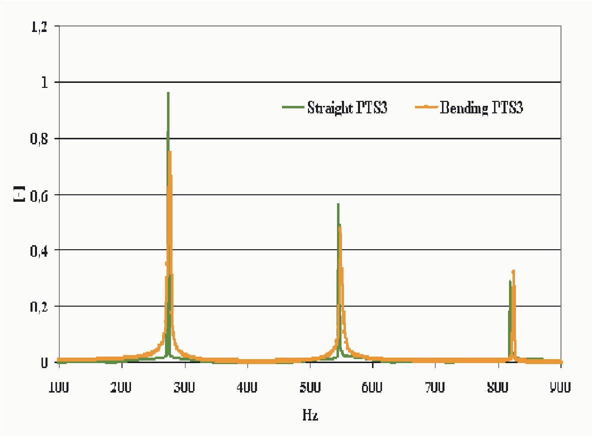
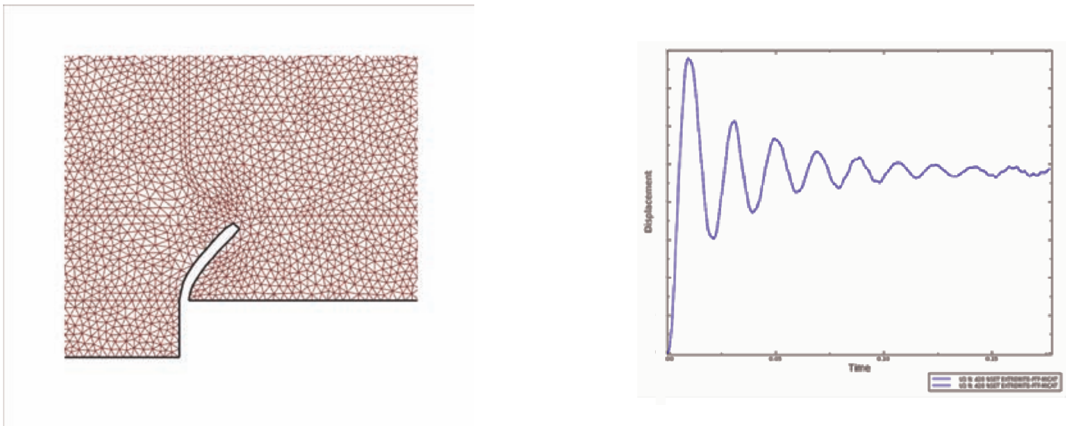


Figure 12: FFT of the pressure signal for the bending PTS3 and the straight PTS3

The results of unsteady co-simulation show (figure 13) that the PTS3 oscillates around an average position. The oscillations of the PTS3 decrease leading to small vibrations. After a numerical transition, the pressure signal (figure 14) is seen to stabilize.



(a) **(b)**

Figure 13: (a) Maximum bending mesh of the PTS3. (b) PTS3 tip spatial displacement of the x coordinate.

In order to know if the PTS3 oscillation has an impact on the pressure signal we stop the co-simulation and proceed to a single Fluent simulation based on the last position (obtained at the end of the unsteady co-simulation) of the PTS3.The figure 14 shows that the pressure signal reaches another value which demonstrates the vibrations impact on the pressure signal.

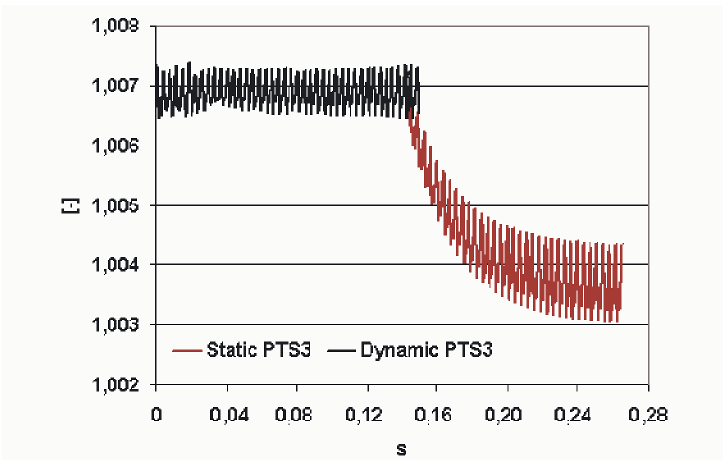


Figure 14: Pressure signal.

4. CONCLUSIONS

Two weak coupling strategies have been detailed in this paper. The first one, untitled one way coupling, can be used if the strain of the solid moving surface doesn't modify the flow field. This strong hypothesis has been made for the solid motor ignition case. This simplified approach is useful to quickly obtained first results necessary for a preliminary analysis. Moreover these results have been obtained by developing our script to exchange values that give us a freedom with regard to the commercial coupling software. But, in order to take into account the influence of the propellant surface displacement on the flow field, complementary development need to be done. Then, the second strategy, untitled two way coupling, based one the MpCCI software, has been used in the case of SRM pressure oscillation phenomena. The study established the influence of the bending of the PTS3 on the pressure signal and also the influence of the vibrations. But other numerical studies should be done and completed by experimental campaigns to confirm that the results obtained clearly quantified the real physical phenomena and are not disturbed by numerical artefact. Moreover the two way coupling strategy used is still prohibitive in CPU time and other algorithms should be tested.

5. REFERENCES

[1] Fluent 6.3. User Guide.

[2] Abaqus 6.7. User Guide.

[3] MpCCI version 3.0.6. User Guide.

[4] Gmsh Reference Manual, C.Geuzaine&J.F.Remacle

[5] Interaction des mécanismes tourbillonnaires en espaces confiné. Effet de la conicité. Chi Cong Nguyen (2002). Thèse de doctorat Université de Poitiers.

[6] Interaction des structures pariétales sur le développement instationnaire d'écoulements cisaillés en milieu confine - rôle de l'injection différentielle. Jérôme Vetel (2001). Thèse de doctorat Université de Poitiers.

[7] Analyse des transferts d'énergie cisaillée par injection pariétale - aspect instationnaire. Dominique Couton (1996). Thèse de doctorat Université de Poitiers.

[8] Structure turbulentes d'un écoulement segmenté à injection pariétale. Frédéric Plourde (1994). Thèse de doctorat Université de Poitiers.

[9] Simulations of slumping propellant and flexing inhibitors in solid rocket motors. R. A. Fiedler, M. S. Breitenfeld, X. Jiao, A. Haselbacher, P. Geubelle, D. Guoy, and M. Brandyberry. AIAA-2002-4341

[10] 3-D Coupled Simulations of Flexible Inhibitors in the RSRM. Bono Wasistho, Robert A. Fiedler,Alireza Namazifar and Mark D. Brandyberry. AIAA-2005-3996

[11] Numerical investigation of fluid-structure interaction in a model of solid propellant motors. *C. Rey, V. Froment, M.-P. Errera, B. Truffart, A. Langlois.* 2nd European Conference for Aerospace Sciences EUCASS 2007.

[12] Projet CPS : code Aérothermochimie propulsion spatiale CPS1.3 : Manuel utilisateur / dossier de modélisation.