STUDY OF CAVITATION IN LIQUID SODIUM AND SIMULATION OF DYNAMIC CORE DEFORMATIONS

Sofiane HOUBAR[1], Antoine GERSCHENFELD[2], and Cyril PATRICOT[1]

[1]*DES - Service d'études des réacteurs et de mathématiques appliquées (SERMA)
[2]DES - Service de thermohydraulique et de mécanique des fluides (STMF)

CEA, Université Paris-Saclay, F-91191 Gif-sur-Yvette, France

sofiane.houbar@cea.fr, antoine.gerschenfeld@cea.fr, cyril.patricot@cea.fr

ABSTRACT

Due to the high confinement of the closed fuel assemblies in Sodium-cooled Fast Reactors (SFR), the liquid sodium filling the space between these assemblies may vaporize in case of an important mechanical excitation (cavitation process). That phenomenon would therefore induce a specific force feedback to the assemblies leading to a specific neutronic response of the core. Neutronic of SFR is indeed sensible to mechanical deformation. Hence the necessity to have a good prediction of the displacement of these mechanical structures in the framework of Fluid-Structure Interaction (FSI).
To study this kind of transient at a reactor scale, we propose in this paper a coupling procedure between the fluid and the structures. The fluid behaviour is investigated by Merkle's three-equations model and simulated within a coarse mesh. The structures are modelled by a mass-spring system subjected to the fluid forces.
In order to stabilize the numerical code coupling, a relaxation process is added and some results of this computational work are presented.

KEYWORDS: Deformed cores; Cavitation; Fluid-Structure Interaction; Code coupling

1. INTRODUCTION

Mechanical excitation of SFR fuel assemblies may appear for example in case of external events like earthquakes or internal initiators due to a mechanical default. Recent developments at CEA have proposed a homogeneized core modelling [1] to investigate such transients at the reactor scale. In [2] and [3], this modelling is applied to a core subjected to excitation of the assemblies and the neutronic effect (which can be important in case of mechanical deformation of SFR core) is investigated using the CNTT (Cast3M Neutronics Transient Tool) solver [4].
However the procedure to model cavitation in [2] and [3] requires an additional physical study and a numerical modelling has to demonstrate how vaporization of the fluid may appear during a mechanical transient of the core. We also show that the model in [1] has to be completed to take into account the effects of fluid convection which are not negligible regarding the pressure forces exerted on the mechanical structures.
The challenge in the context of our study is then to build a numerical tool able to:

- Reproduce the main effects of the cavitation phenomenon
- Couple Fluid and Structure behaviour
- Model a whole reactor core for a reasonable computational cost

The choice has been made to consider the flow model of the Navier-Stokes equation for the fluid flow. In order to model the two-phase flow behaviour, a mixture velocity field is used where the dynamics of each phase is governed by a mass transfer coefficient. This approach has been used using Merkle's model [5] which consists in a three-equations modelling.

2. GENERAL CONSIDERATIONS

2.1. Impact of Deformed Cores on Neutronics

2.1.1. Motivations and state-of-the art

Within the design of a SFR, two recent works [2] and [6] have used different softwares and methodologies in order to investigate the neutronic behaviour of a deformed core.
In this context, the work [3] presents different kind of mechanical transients and hence neutronic transients impacted by different thermal-hydraulic initiators.
Such sensitivity of fast reactor cores to deformations can be explained by the relative high free mean path λ of the neutrons regarding the core radius R: a very simple calculation indeed shows (see [2]) that λ varies relatively faster than R. Let us denote N the homogenized atomic density of an isolated core and V the volume. Then we have in case of deformation:

$$NV = \text{constant} \tag{1}$$

Since R evolves like $V^{1/3}$, Eq. (1) implies:

$$N \sim R^{-3} \tag{2}$$

And then:

$$\lambda \overset{def}{=} \frac{1}{N\sigma} \sim R^3 \tag{3}$$

These results show with very simple considerations that growing the size of a core exacerbates the neutron leakage.

2.1.2. An academic test case

In order to study mechanical effects on neutronics, specific displacements are assumed. Such academic displacements are described in [2]: the displacement of the assemblies does not depend on their position (see Fig. 1).

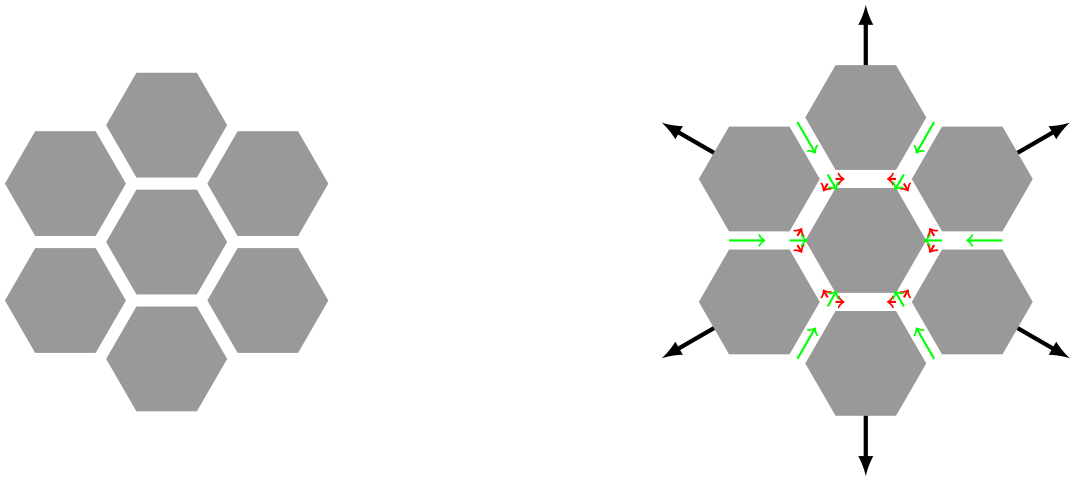

Figure 1: First ring of assemblies in steady state (left) and excited (right)

2.2. Fluid Motion Around The Assemblies

2.2.1. The one-phase Navier-Stokes equations

The local Navier-Stokes equations (4) are numerically assumed to solve the fluid Pressure field P. This one corresponds to an unstationary convection-diffusion equation-type with a constraint on the mass balance driven by P. To simulate the fluid in a domain with moving boundaries, the *Arbitrary Lagrangian-Euler* (ALE) method is used. A description of this technique can be found in [7].

$$\begin{cases} \underbrace{\nabla \cdot \mathbf{u}}_{mass\ balance} = 0 \\ \underbrace{\partial_t \rho \mathbf{u}}_{unsteadiness} + \underbrace{(\mathbf{u} - \mathbf{u_w}) \cdot \nabla (\rho \mathbf{u})}_{convection} = \underbrace{-\nabla P}_{driving\ source} + \underbrace{\mu \Delta \mathbf{u}}_{friction} \end{cases} \tag{4}$$

2.2.2. Modelling by means of domain homogenization

Modelling and simulating feedbacks of hydraulic effects on movements of assemblies are quite challenging: the convection term in the momentum equation is indeed non linear and may then lead to complex phenomena which would require a very fine mesh size. This is of course a high limitation in order to perform simulations at the whole reactor scale.
In Broc *et al.* [1] the so-called "UPϕ" model has been applied. It consists in homogenizing the whole core: the fluid part is not simulated but completely modelled.

2.3. Learnings From FSI Calculations in Homogenized Cases

2.3.1. Results of the UPϕ model in several transients

Results of chaining FSI calculations to neutronics within an homogeneized core in different scenarios leading to a core mechanical excitation are provided in [3]. In some cases, the FSI model can lead to negative pressure values. This suggests that the fluid may vaporize through a cavitation process. This one can not be taken into account by the model (4).
This finding is crucial: vapour apparition in the inter-assembly space may lead to different pressure forces exerted on the fuel assemblies. This vapour is characterized (in a two-phase equilibrium) by a constant pressure value P_{sat}. The linear fluid model in the $UP\phi$ model in such a case would lead to a wrong pressure estimation and then to a wrong FSI coupling.

2.3.2. Phase change and the UPϕ model

A "made-by-hand" modelling of vaporization or gas presence in the inter-assembly space has been performed in [2]. If one wants to assess the behaviour of the assemblies, all the inter-assembly liquid sodium is removed from a certain time step during a simulation. However in such an approach, the gas dynamics is not simulated.

3. AN UPGRADED FLUID MODELLING

3.1. Hydraulic Investigations at a Local Scale

3.1.1. Case description and analytical model

In order to upgrade the modelling of the fluid behaviour in case of movement of the assemblies, a local case has been studied: it consists in the study of the fluid behaviour in a domain where two parallel plates are moving and driven by a constant pressure reference at the left/right borders. The configuration of such a case is presented in Fig. 2.
Solving Eq. (4) in the configuration of Fig. 2 leads us to formulate the analytical model Eq. (5) for

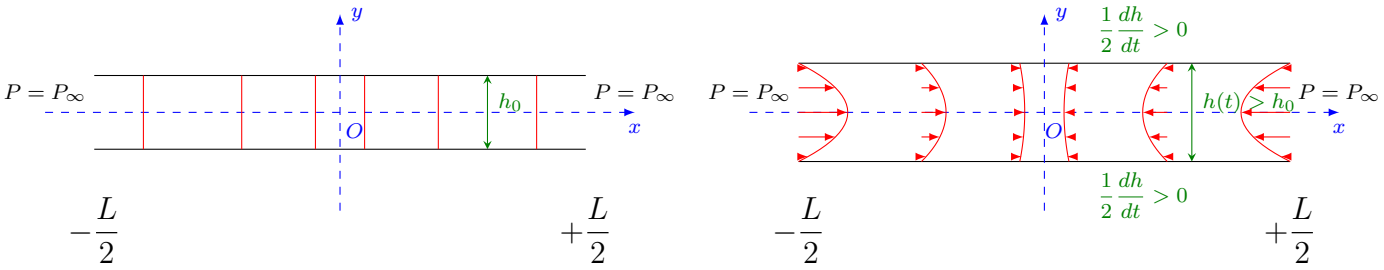

Figure 2: Fluid in steady state (left) and movement induced by moving boundaries (right)

the pressure term P. This has enabled us to predict which set of movement frequency and minimal gap distance h_0 can possibly lead to reach P_{sat}. The results of this analysis at the local scale has been presented in [8].

$$< P(x,y,t) >_{(x,t)} = P_\infty + \frac{\rho}{2h}\left[h'' - (1+A_n)\frac{h'^2}{h}\right]\left[x^2 - \left(\frac{L}{2}\right)^2\right] + \frac{\alpha\rho\nu^\beta}{2^{2+\beta}(3-\beta)}\left[h^{-3}|h'|^{(1-\beta)}h'\right]\left[|x|^{3-\beta} - \left(\frac{L}{2}\right)^{3-\beta}\right] \quad (5)$$

3.1.2. 1D-Homogenization in 2D geometries

In order to reduce the computational cost as mentioned in Sec. 1, the 2D whole core domain (inter-assembly space) is homogenized into a 1D domain: the meshed domain between too assembly edges (inter-assembly space) is then created by a $N_x \times 1$ mesh. With this technique only a mean effect on pressure is simulated and a correlation has to be used in order to re-build a correct head loss term (see Eq. (7)).

3.1.3. Modelling of the vaporization process under core excitation

In order to model the cavitation process, a set of equations has to be capable of reproducing mass exchanges between the liquid and gaseous phases in a physical manner. A large variety of models used to model cavitation processes can be found in [9]. In our case the mass exchanges are managed by the term Γ_v. To close the set of equations Eq. (7), this term is splitted into the vaporization term $\dot{m}^+$ and the condensation term $\dot{m}^-$ (Eq. (8)). The mixture equation model implemented in the CFD code *Code_Saturne* is used (see [10]). The mixture density ρ_m, velocity $\mathbf{u_m}$ and dynamic viscosity μ_m are then written as a function of the void fraction α such as:

$$\begin{cases} \rho_m &= <\alpha>\rho_v + (1-<\alpha>)\rho_l \\ <\mathbf{u_m}> &= <\alpha\mathbf{u_v} + (1-\alpha)\mathbf{u_l}> \\ \mu_m &= <\alpha>\mu_v + (1-<\alpha>)\mu_l \end{cases} \quad (6)$$

By injecting Eq. (6) into Eq. (4) and by averaging (spatially) the variables according to the 1D-homogenization, an equation for $< \alpha >$ (void fraction transport) is then obtained to build the following averaged 3-equation model:

$$\begin{cases} \nabla\cdot<\mathbf{u_m}> &= \Gamma_v\left(\frac{1}{\rho_v} - \frac{1}{\rho_l}\right) \\ \partial_t(\rho_m<\mathbf{u_m}>) + (<\mathbf{u_m}> - <\mathbf{u_w}>)\cdot\nabla(\rho_m<\mathbf{u_m}>) &= -\nabla P - \frac{1}{2D_h(t)}f||<\mathbf{u_m}>||<\mathbf{u_m}> \\ \partial_t<\alpha> + [-<\mathbf{u_w}>\cdot\nabla(<\mathbf{u_m}>) + \nabla\cdot(<\alpha><\mathbf{u_m}>)] &= \frac{\Gamma_v}{\rho_v} \end{cases} \quad (7)$$

The exchange term $\Gamma_v = \dot{m}^+ + \dot{m}^-$ results from a local analysis for boiling sodium. Its application to cavitating sodium is justified due to the similitude between the two processes:

$$\begin{cases} \dot{m}^{+} &= \dfrac{\rho_l \max\left(P_{sat}-P,0\right) <\alpha> (1-<\alpha>)}{t_\infty P_{sat}} \\ \dot{m}^{-} &= \dfrac{\rho_v \max\left(P-P_{sat},0\right) <\alpha> (1-<\alpha>)}{t_\infty P_{sat}} \end{cases} \tag{8}$$

with t_∞ = 1 ms corresponding to a physical relaxation term.

3.1.4. Modelling of the assemblies displacement

The displacement of each assembly (indexed by i) in 2D is modelled by a damped mass-spring system submitted to an external force from the fluid part.

$$M\ddot{\mathbf{X}}_\mathbf{i} + C\dot{\mathbf{X}}_\mathbf{i} + K\mathbf{X}_\mathbf{i} = -\int_{\partial\Omega_i} P\mathbf{dS} \tag{9}$$

This displacement leads to a fluid volume update which requires a new calculation of the fluid variables in the framework of a FSI code coupling.

3.1.5. Numerical scheme for the FSI code coupling

A native coupling consisting at each step in calculating the pressure field and then the displacement field is numerically unstable due both to the high confinement of the assemblies and the important density of the liquid medium. Consequently an implicit time scheme with under-relaxation has been implemented and is presented in a nutshell in Eq. (10), $n+1$ depicting the current time step and k the local iteration step of the relaxation process.

$$M\ddot{\mathbf{X}}^{k}_{n+1} + C\dot{\mathbf{X}}^{\mathbf{k}}_{\mathbf{n+1}} + K\mathbf{X}^{k}_{n+1} = (1-\lambda)\mathbf{F}^{k-1}_{n+1} + \lambda\mathbf{F}^{k}_{n+1} \text{ until } \frac{||\mathbf{X}^{k-1}_{n+1}||}{||\mathbf{X}^{k}_{n+1}||} < \epsilon \tag{10}$$

The relaxation term λ is empirically determined when calculations become stable.

4. SOME RESULTS OF FSI AND NEUTRONICS CALCULATIONS

4.1. Comparison of The Pressure Fields in a 1-Ring Configuration

In the cases described here, the assemblies are initially moved outwards (Fig. 1). Results within a 2D configuration made of one ring of assemblies are depicted in Fig. 3 with or without cavitation modelling. Effects of cavitation are observable on the right picture: In the one hand the region around the central assembly where cavitation occurs is maintained at P_{sat}. In the other hand the distance of the ring assemblies to the central assembly is also different in the two cases.

By the way no experimental data are available related to the cavitation process under two moving boundaries. However models similar to Eq. (7) are compared to experimental data in [11] in case of cavitation on 2D foil sections and show a good agreement with them.

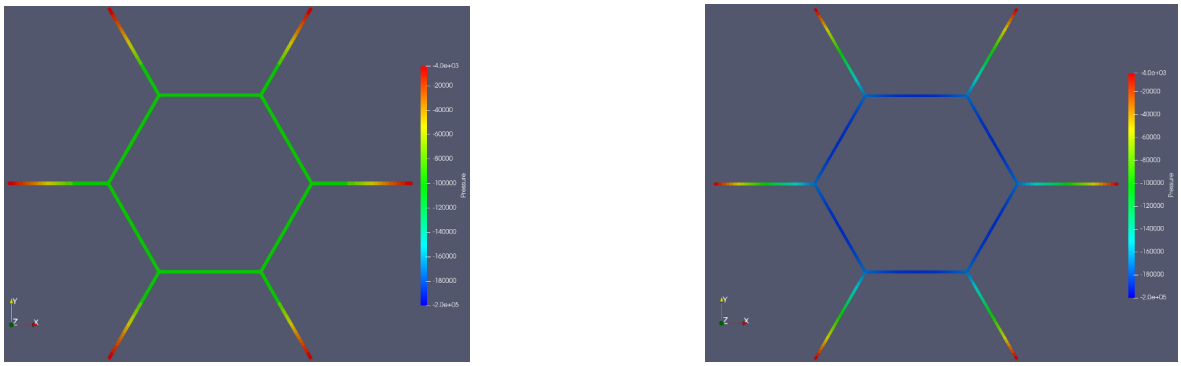

Figure 3: Pressure field for 1-phase fluid (left) and 2-phase fluid (right) at t = 0.013 s

4.2. Transient Reactivity Effects of Assemblies Displacement

The CNTT solver [4] has been used to compute the reactivity effect from deformation of the 1-Ring 3D core. The assemblies made of fissile fuel are assumed to be straight (without bending) and their movement is taken from the 2D FSI calculations described in Subsection 4.1. Influence of cavitation on the assembly displacement can be seen on the right side in Fig. 4. This behaviour is also observable on the curve depicting the reactivity effect. The high value of hte initial reactivity effect is due to the small size of the 1-Ring configuration.

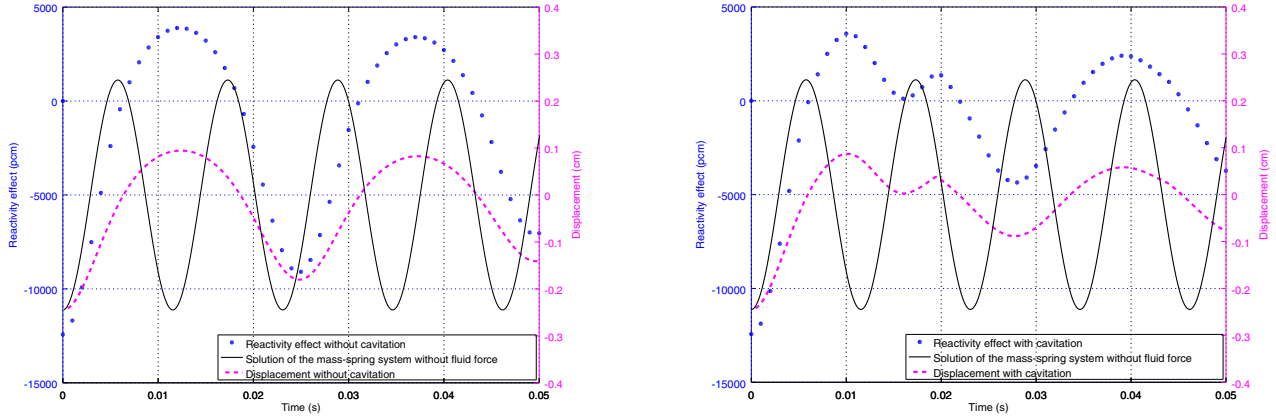

Figure 4: Assembly displacement and reactivity effect with (left) or without (right) cavitation

5. CONCLUSIONS

A numerical model enabling code coupling between thermal-hydraulics and mechanics with chaining to neutronics in the context of a SFR basic design has been presented here. This model is made

of a mixture modelling of the cavitation process within a 1D-homogenized geometry in order to perform calculations at the scale of a reactor.
In order to take into account the top-bottom fluid flowering in the inter-assembly space, an upcoming step consists in performing the same FSI coupling in 3D.

NOMENCLATURE

$\sim$	*proportional to*	$\nabla.(\bullet)$	Divergence operator
ρ	Medium density	$\nabla(\bullet)$	Gradient operator
P	Pressure field	$\Delta(\bullet)$	Laplacian operator
P_{sat}	Saturation Pressure	$\mathbf{u}$	Fluid velocity field
$h(t)$	Local inter-assembly gap distance	$\mathbf{u_w}$	Grid velocity field
$D_h(t)$	Hydraulic diameter	$\mathbf{u_m}$	Mixture velocity field
α	Void fraction field	$< \bullet >$	Spanwise-averaging operator
t_∞	Cavitation relaxation term	$\mathbf{X}$	Assembly displacement field
$\partial\Omega_i$	Fluid - Assembly i interface	$\mathbf{F}$	Fluid pressure force

REFERENCES

[1] D. Broc and J.-F. Sigrist. "Fluid Structure Interaction for tube bundles: physical meaning of a Rayleigh damping." In *Proceedings of the ASME 2010 Pressure Vessels and Piping Conference*. Bellevue, Washington, USA (2010).

[2] C. Patricot. *Couplages multiphysiques : évaluation des impacts méthodologiques lors de simulations de couplages neutronique / thermique / mécanique*. Ph.D. thesis, École Polytechnique - Université Paris-Saclay (2016).

[3] G. Kepisty, C. Patricot, D. Broc, and G. Campioni. "SFR mechanical scenarios and neutron transport transients with CAST3M code." *Annals of Nuclear Energy*, **volume 101**, p. 226–236 (2017).

[4] C. Patricot, A. M. Baudron, O. Fandeur, and D. Broc. "Neutronic calculation of deformed cores: development of a time-dependent diffusion solver in Cast3M, a mechanics dedicated finite element code." In *PHYSOR 2016 – Unifying Theory and Experiments in the 21st Century*. Sun Valley Resort, Sun Valley, Idaho, USA (May 1 – 5, 2016).

[5] C. Merkle, J. Feng, and P. Buelow. "Computation modeling of the dynamics of sheet cavitation." In *3rd International Symposium on Cavitation CAV1998*. Grenoble, France (1998).

[6] M. Gentili. *Aspects Neutroniques Associés à des Réseaux Irréguliers d'Assemblages dans un Cœur de RNR-Na*. Ph.D. thesis, Université d'Aix-Marseille (2015).

[7] F. Duarte, R. Gormaz, and S. Natesan. "Arbitrary Lagrangian–Eulerian method for Navier–Stokes equations with moving boundaries." *Computer Methods in Applied Mechanics and Engineering*, **volume 193**, p. 4819–4836 (2004).

[8] S. Houbar and A. Gerschenfeld. "Towards numerical modelling of cavitating inter-wrapper liquid sodium in support of physical studies of Sodium-Cooled Fast Reactors." In *ECCM – ECFD*. Glasgow, UK (2018). URL http://www.eccm-ecfd2018.org/frontal/ProgSesion.asp?id=760.

[9] J. Decaix. *Modélisation et simulation de la turbulence compressible en milieu diphasique : application aux écoulements cavitants instationnaires*. Ph.D. thesis (2012).

[10] "Code Saturne 5.0.0 Theory Guide." Technical report (2002).

[11] S. Frikha, O. Coutier-Delgosha, and J. Astolfi. "Investigation of Cavitation Models for Steady and Unsteady Cavitating Flow Simulation." *International Journal of Rotating Machinery*, **volume 2008**, p. 240–251.