Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30486

FULLY-COUPLED FLUID SOLID COMPUTATION FOR SIMULATION OF FLUTTER IN TUBES AND TUBE ARRAYS

E. Longatte
LaMSID UMR EDF-CNRS-CEA 2832
Avenue General de Gaulle
92141, Clamart, France
Email: elisabeth.longatte@edf.fr

ABSTRACT

This work is concerned with the modelling of the interaction of a fluid with a rigid or a flexible elastic cylinder in presence of axial or cross-flow. A partitioned procedure is involved to performe the computation of the fully-coupled fluid solid system. The fluid flow is governed by the incompressible Navier-Stokes equations and modeled by using a fractional step scheme combined with a co-located finite volume method for space discretisation. The motion of the fluid domain is accounted for by a moving mesh strategy through an Arbitrary Lagrangian-Eulerian (ALE) formulation. Solid dyncamics is modeled by descrete or beam elements in the linear elasticity framework and systems are solved through a finite element method. The resulting strongly coupled fluid solid set of non linear equations is solved by means of a partitioned solution procedure. A fixed point method combined with under-relaxation is involved to ensure the optimal convergence of the iterative procedure. In the present work two examples are presented to show the methodology robustness and efficiency. The purpose is to attempt to simulate a fluid structure interaction resulting in the development of a dynamic instability induced by a positive damping generation of the system. Both flutter of a flexible cylinder conveying an internal fluid and fluid-elastic instability of a tube array submitted to an external cross flow are investigated numerically. According to first results the partitioned procedure relies on consistant numerical methods ensuring energy conservation at the interface and describing with a sufficient accuracy the mechanical energy transfer between fluid and solid systems through the interface with a limited numerical diffusion. Therefore it seems to be qualitatively convenient for simulation of flutter. For a quantitative evaluation of the methodology further complementary simulations validating these developments from a physical point of view will be required in order to confirm these first trends.

INTRODUCTION

Fluid structure interaction occur in a wide range of industrial applications involving vibrations of mechanical structures excited by complex flows. In the nuclear domain, multi-physics problems such as those encountered in heat exchangers or fuel assembly in reactor cores are often investigated through global empiric modeling due to the large number of degrees of freedom of the systems that are considered. The purpose of the present work is to propose an optimization of these global modeling in terms of vibration risks by using a new class of local numerical approaches. The proposed methodology consists in building a fully-coupled fluid solid system and solving it through a partitioned procedure. The article outlines computational methods for calculating the full coupled system. Multi-physics computation is performed by using a devoted platform developed by EDF R&D teams called *Salomé*[1] ensuring the coupling between fluid and solid system computations by using *Code_Saturne*[2] and *Code_Aster*[3]. An implicit integration of the set of non linear

[1] *http://www.salome-platform.org*
[2] *http://www.code-saturne.org*
[3] *http://www.code-aster.org*

equations of motion is involved. Examples shown deal with simulation of flutter in a duct conveying fluid and a tube array submitted to cross flow featuring possible dynamically unstable behaviour.

PARTITIONED PROCEDURE FOR COMPUTATION OF FULLY-COUPLED FLUID SOLID SYSTEM

The purpose is to solve a fully-coupled fluid solid system by preserving the compatibility of the subdomain and the interface modeling in the full domain. Numerical methods are presented below.

Solid dynamics

In the solid domain the task is to determine deformations of solid bodies, which arise due to the action of various kinds of forces. From this, stresses in the body can be determined. For the different material properties there exist a large number of material laws, which together with the balance equations lead to diversified complex equation systems for the determination of deformations (or displacements). In principle, for structural mechanics problems one distinguishes between linear and non-linear models, where the non-linearity can be of geometrical and/or physical nature. Geometrically linear problems are characterized by the linear strain-displacements relation for the strain tensor ε_{ij} defined by :

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

with the displacement vector u_i, whereas physically linear problems are based on a material law involving a linear relation between strains and stresses. In the present work we restrict ourselves to the formulation of the equations for the linear elasticity theory but the coupling strategy has been developed in the general case. The theory of linear elasticity is a geometrically and physically linear one. In the following the spatial coordinates are denoted by x_i. The equations of the linear elasticity theory are obtained from the linearized strain-displacement relations and the momentum conservation law formulated for the displacements. In the framework of structural mechanics the equation of motion may be written :

$$\rho \frac{\partial^2 u_i}{\partial t^2} = \frac{\partial T_{ij}}{\partial x_j} + \rho f_i$$

and the assumption of a linear elastic material behavior is characterized by the constitutive equation known as the Hooke's law:

$$T_{ij} = \lambda \varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij}$$

λ and μ are the Lamé constants, which depend on the corresponding material. The elasticity modulus (or Young modulus) E and the Poisson ratio ν are often employed instead of the Lamé constants. The relations between these quantities are:

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} \quad \text{and} \quad \mu = \frac{E}{2(1+\nu)}$$

Possible boundary conditions for linear elasticity problems are:

– Imposed displacement: $u_i = u_{bi}$ on Γ_u

– Imposed stress: $T_{ij}n_j = t_{bi}$ on Γ_T

Parts of boundary Γ_u and Γ_T must be disjointed and cover the whole domain boundary Γ, i.e., $\Gamma_u \cap \Gamma_T = \emptyset$ and $\Gamma_u \cup \Gamma_T = \Gamma$. In the case of a single-degree-of-freedom system, the solid is modeled through a discrete element where a mass, a damping and a stiffness are imposed. In the case of a multi-degree of freedom system, in the present work, only beams are investigated and they are modeled by beam elements in the framework of linear elasticity theory. Linear systems are solved by using a finite element method and time discretisation is ensured by an implicit Newmark algorithm of second order unconditionaly stable.

Computational fluid dynamics

For the description of fluid flows an Eulerian formulation is employed, because one is interested in the properties of the flow at certain locations in the flow domain. We restrict ourselves to the case of linear viscous isotropic fluids known as Newtonian fluids. Newtonian fluids are characterized by the following material law for the Cauchy stress tensor $\mathbf{T}$:

$$T_{ij} = \mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}\frac{\partial v_k}{\partial x_k}\delta_{ij}\right) - p\delta_{ij}$$

with the velocity vector v_i with respect to Cartesian coordinate x_i, the pressure p, the dynamic viscosity μ, and the Kronecker symbol δ_{ij}. In the framework of incompressible flow the conservation of mass yields to a divergence-free velocity vector:

$$\partial v_i / \partial x_i = 0$$

For incompressible flows the stress tensor becomes:

$$T_{ij} = \mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) - p\delta_{ij}$$

The conservation equations for mass, momentum, and energy then read:

$$\frac{\partial v_i}{\partial x_i} = 0$$
$$\frac{\partial(\rho v_i)}{\partial t} + \frac{\partial(\rho v_i v_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right)\right] - \frac{\partial p}{\partial x_i} + \rho f_i$$

For isothermal processes in the incompressible case the energy equation does not need to be taken into account. This equation system has to be completed by boundary conditions and, in the unsteady case, by initial conditions. As boundary conditions for the velocity, the velocity components can be explicitly prescribed:

$$v_i = v_{bi}$$

Here, $\mathbf{v}_b$ can be a known velocity profile at an inflow boundary or, in the case of an impermeable wall where a no-slip condition has to be fulfilled, a prescribed wall velocity ($v_i = 0$ for a fixed

wall). Attention has to be paid to the fact that the velocities cannot be prescribed completely arbitrarily on the whole boundary Γ of the problem domain, since the equation system only admits a solution, if the integral balance

$$\int_{\Gamma} v_{\mathrm{b}i} n_i \, \mathrm{d}\Gamma = 0$$

is fulfilled. This means that there flows as much mass into the problem domain as it flows out, which, of course, is physically evident for a "reasonably" formulated problem. At an outflow boundary, where usually the velocity is not known, a vanishing normal derivative for all velocity components can be prescribed. For incompressible flows, the pressure is uniquely determined only up to an additive constant.

For time discretisation a θ-scheme is involved. A second order Crank-Nicolson scheme with $\theta = 1/2$ is used. A fractional step scheme is used to solve the mass and momentum equations with a Chorin method [2]. The first step (the predictor step) provides predicted velocity components : they are determined sequentially and without coupling between each other. The mass equation is taken into account during the second step (corrector step) where a Poisson equation is solvedfor pressure and the mass fluxes at the cell faces are updated. Finally all variables are updated and another time step may start. During the flow computation the time step is constant in time and uniform in space. For space discretisation a finite volume method is involved with colocated variables. Within the framework of the finite volume approach, the equations are integrated over each cell of the mesh called control volumes. This yields to an equilibruim equation for fluxes through control cell faces and volumic sources. Convection and diffusion terms are thus integrated.

To deal with any fluid structure interaction problem the computation of coupled fluid and solid systems is required with convenient data interchange through the interface. Therefore boundary conditions must be compatible at the interface. From a numerical point of view solid and fluid boundaries Γ_s et Γ_f may be considered as standard boundaries. If Γ_i designates the interface boundary, velocity and stress tensor must satisfy the following conditions :

$$v_i = \frac{\partial u_i^{\mathrm{b}}}{\partial t} \quad \text{and} \quad \sigma_{ij} n_j = T_{ij} n_j$$

where u_i^{b} and $\partial u_i^{\mathrm{b}}/\partial t$ designate respectively the interface displacement and velocity. When the fluid domain changes in space and time during the computation due to solid wall motion or deformation, it is necessary to account for this motion in the fluid domain mesh. The mesh and domain evolution is accounted for by using a moving mesh method. An Arbitrary Lagrange Euler (ALE) formulation is involved. The method consists in introducing an arbitrary referential domain in which the set of equations is reformulated and solved. This domain is built to coincide with the Lagrangian referential near with moving boundary and with the Eulerian referential far from the interface. Using the Geometric Conservation Law (GCL) for an incompressible flow it can be shown that in the ALE context, Navier-Stokes equations can be reformulated as follows [8] [9]:

$$\nabla.(\mathbf{v}) = 0$$
$$\rho\{\tfrac{\partial \mathbf{v}}{\partial t} + (\mathbf{v}-\mathbf{w})\,\nabla\mathbf{v}\} = -\nabla p + \nabla.(\mu\,\nabla\mathbf{v})$$

where $\mathbf{w}$ designates the mesh node velocity in the reference domain. This velocity is updated at each time step of the computation. Several methods are possible for updating. Algebraic or elliptic methods are used in the configurations presented below.

Coupling strategy

As far as multi-physics systems are concerned, in terms of time integration, two approaches are possible. A monolithic strategy consists in considering the whole coupled system as a single system. Therefore both sub-systems are formulated in the same framework and solved with the same numerical methods, especially with the same time integration algorithms, by using interface matching discretizations. This leads to an algebraic system that can be inversed by any direct or iterative computation algorithm. In spite of its convergence properties, this approach features the main inconvenience to imply high price for the storage and the inversion of the Jacobian of the full system. As an alternative method, a partitioned approach enables the resolution of the fully-coupled system in case of different discretizations either on the space or on the time domain, with possible non-matching interfaces. Its efficiency has been pointed out in many previous works [22] [17] [15] [16] [23] [11]. It is used in the present work.

The interaction of motions of fluid (denoted by f) and solid (denoted by s) is considered. Both systems can be non-linear and time-dependent. In what follows, for a structure, $\mathbf{u}_s$ designates the displacement field, whereas $\mathbf{u}_f = (\mathbf{v}_f, p)$ takes into account both velocity and pressure for an incompressible flow. Each quantity u_i is associated with a strain F_i through an operator called the Neumann-Dirichlet operator and depending on chosen models, material properties, boundary conditions but also on discretization techniques, time integration schemes, inverse discrete equation solvers for each sub-system. Lets one denote by $\mathscr{F}_i$ and $\mathscr{U}_i$ the operators defined on convenient spaces such that:

$$F_i = \mathscr{F}_i(u_i) \text{ and } u_i = \mathscr{U}_i(F_i)$$

for $i = f$ or s. The interaction between both sub-systems is based on the two following principles : the continuity of velocities and the equilibrium of forces at the interface. In the continuum space interface fluid and solid velocities (v) and accelerations (a) are equal. However these equivalence relations are no longer ensured in the time discretized domain. The equilibrium equations correspond to the action-reaction principle at the interface :

$$\mathscr{F}_f(\mathbf{u}_f) + \mathscr{F}_s(\mathbf{u}_s) = 0$$

Therefore find the solution $\mathbf{u}$ of the coupled system (where u designates the value of the variable at the interface defined on $\Gamma \times [0,T]$ with Γ the fluid structure interface and $[0,T]$ the time domain) is equivalent to find the solution of the following problem :

$$\text{Find } \mathbf{u} \text{ on } \Gamma \times [0,T] \text{ so that : } \mathscr{F}_f(\mathbf{u}) + \mathscr{F}_s(\mathbf{u}) = 0$$

or using the inverse of the solid operator :

$$\text{Find } \mathbf{u} \text{ on } \Gamma \times [0,T] \text{ so that : } \mathbf{u} = \mathscr{F}_s^{-1}(-\mathscr{F}_f(\mathbf{u}))$$

Then several strategies are possible to solve this equation. It is possible to consider fixed point formulation with iterative schemes. Stability and convergence are thus conditional. With explicit coupling, the method consists in solving the coupled system in a window $[T_N, T_{N+1}]$ of size Δt (which corresponds to the time step). At each time step an iteration of the fixed-point equation is solved as follows :

$$\mathbf{u}_{N+1} = \mathscr{F}_s^{-1}(-\mathscr{F}_f(\mathbf{u}_N)).$$

This method features bad conservation properties at the interface. It can be improved either by the addition of a predictor of the interface displacement or by subcycling. At each subcycling, a predictor is defined : $\mathbf{u}^P_{N+1} = P(\mathbf{u}_N, \mathbf{v}_N, \mathbf{u}_{N-1}, ...)$. Solid and fluid systems are then solved by computation of equations :

$$F_{N+1} = \mathscr{F}_f(\mathbf{u}^P_{N+1})$$
$$\mathbf{u}_{N+1} = \mathscr{F}_s^{-1}(F_{N+1})$$

Finally $\mathbf{u}_{N+1}$ and $\mathbf{u}^P_{N+1}$ are compared and at each time step, the current error is evaluated in order to continue or to stop the subcycling process. Additional relaxations on fluid forces may also be introduced in order to increase the convergence velocity and the coupling procedure order. In Figure 1 a schematic view of the iteration process is given. After the initializations the flow field is determined in the actual flow geometry. From this the friction and pressure forces on the interacting walls are computed. These are passed to the structural solver as boundary conditions. The structural solver computes the deformations, with which then the fluid mesh is modified. Afterwards the flow solver is started again. The fluid-structure interaction (FSI) iteration loop is repeated until a convergence criterion ε is reached. It is defined by the change of the mean displacements:

$$\Re^{FSI} = \frac{1}{N} \sum_{k=1}^{N} \frac{\|\mathbf{u}^{k,m-1} - \mathbf{u}^{k,m}\|_\infty}{\|\mathbf{u}^{k,m}\|_\infty} < \varepsilon\,, \qquad (1)$$

where m is the FSI iteration counter, N is the number of interface nodes, and $\|\cdot\|_\infty$ denotes the infinite norm. An explicit coupling method is obtained if only one FSI iteration is performed. The data transfer between flow and solid solvers within the partitioned solution procedure is performed via an interface that controls the data communication and also carries out the interpolations of the data from the fluid to the solid grid and conversely.

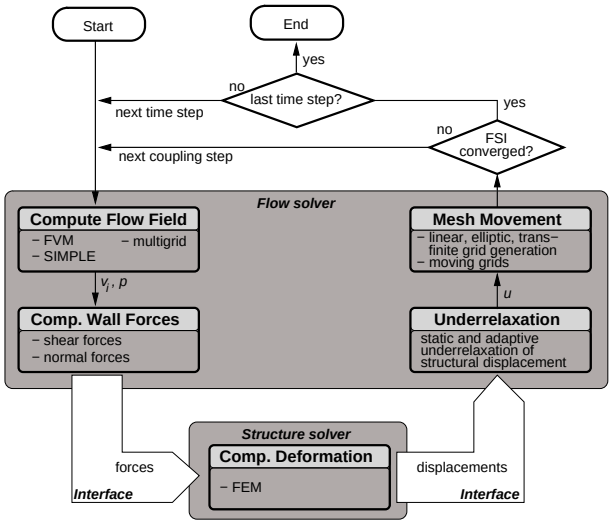

FIGURE 1. FLOW CHART OF COUPLED SOLUTION PROCEDURE

Various test computations have shown that the coupling scheme is rather sensitive with respect to the deformations especially in the first FSI iterations. Here, situations that are far away from the physical equilibrium can arise, which may lead to instabilities or even the divergence of the FSI iterations and an adaptive underrelaxation is sometimes unsufficient to ensure stability.

Space discretization

In terms of space discretization due to the presence of non-matching interface, an interpolation method has been chosen to ensure data projection between sub-system solvers [18]. One can notice that in the case of pipes and cylinders, the interfaces are non-matching from at least three points of view : in terms of interface mesh refinment, since fluid mesh refinment is greater than solid mesh one, in terms of geometry because fluid and solid interface models do not coincide and in terms of dimension because the solid model is represented by a beam element while the fluid domain is two-dimensional. A two-dimensional to one-dimensional condensation method is then required. Therefore an interpolation technique is combined with a condensation technique. The interpolation method relies on a consistant approach ensuring the equilibrium of forces evaluated in both sub-domains at the interface. The accuracy is equivalent to the accuracy of a method relying on Lagrange multiplicators, if the fluid mesh is sufficiently refined, which is the case in the present work. The fluid is viscous, therefore the meshes move simultaneously without any slip condition. Each discretization points of a subdomain can thus be connected to an element of the other one and be repered with the same barycentric coordinates during all the computation. The interface of the first mesh is called slave in-

terface while the other one is referred to as the master interface. There is then a connectivity table between discretization nodes of slave interface and elements of master interface and by using interpolation functions, it is possible to transfer fields from one mesh to the other one. The fluid interface can be considered as the slave and the solid one as the master interface. The projection procedure satisfies the two following functions : (1) by using an interpolation function, the force at any fluid discretization point is distributed on the discretization points belonging to the associated solid element ; (2) by using an interpolation function, the displacement at the fluid discretization point is deduced from the discretization points of the associated solid element. The choice of interpolation functions is free. In the present work, the fluid solver relies on a finite volume formulation and the solid solver on a finite element method. The interpolation functions chosen correspond to the shape functions of the solid finite element method. For field condensation a domain decomposition method is applied and fields are interpolated and spatially averaged over each sub-domain. From a geometrical point of view one sub-domain corresponds to a cylinder or tube cross-section extruded over one cell of the mesh.

NUMERICAL SIMULATION OF FLUTTER

Two examples of simulation of flutter are presented below. The first one deals with the motion of a flexible duct conveying fluid, the second one is related to the vibration of a tube embedded in a tube array under cross flow. In both cases a damping-governed dynamic instability is simulated numerically and first conclusions on the consistency of the partitioned procedure applied to this class of problems can be drawn.

Oscillations of a flexible duct conveying an internal axial fluid flow

The dynamical system involving a fluid-conveying pipe has been mainly studied because of its applications. This is a system of great relevance in the field of fluid structure interaction as this is a model problem for a large variety of fluid structure interaction systems [19] [3] [5] [21]. The identification of the dynamical instability threshold in such a configuration has been extensively studied from an experimental point of view under many mechanical and hydraulical conditions. Analytical models have been derived and good agreement has been obtained between analytical predictions and experimental observations. Numerical simulations have although been performed by using boundary condition linearized formulations for example [12].

One considers a flexible cylinder of circular cross section of diameter D and length L conveying a fluid. The flow is incompressible and the velocity v_f^0 is uniform along the initial axis of the cylinder (Figure 2). The cylinder is fixed upstream in the flow direction and it is free downstream. This means that the motion of the downstream extremity of the duct depends on the interaction with the fluid flow and conversely. The cross section diameter is uniform and only small flexion motion is allowed along the cross direction. Gravity and external perturbations are neglected. Only fluid forces acting on the cylinder wall are taken into account. Reference analytical solutions are available [15] [19]. A two-dimensional modeling is used. Until a laminar Poiseuille flow profile has been reached, solid walls are fixed. Then the coupling process can be started. An initial impulsion is introduced in order to create an asymmetry by using the mode shape of the second mode of the structure. The second mode is chosen because it is expected to be unstable in the configuration that is considered according to the analytical theory [19]. The solid modeling is formulated along the mean axis of the cylinder. Using a two-dimensional modeling implies connecting solid walls at each time step of the computation so that they move in accordance. For fluid mesh deformation and node displacement computation an algebraic method combined with an adjacent cell tracking process is involved.

Examples of meshes used for fluid and solid modeling are shown in Figure 3. The configuration parameters are described in Table 1 and correspond to those of a previous work [12]. Results are compared to those obtained by a non-moving mesh method in a small displacement framework with a boundary condition linearization approach. Two instantaneous mode shapes are plotted in Figure 4 for reduced velocities *4,0* and *4,5* where the reduced velocity is defined by the ratio between the flow velocity and the product between the duct diameter and the frequency of the second mode of the solid. According to the Argand diagram, the critical velocity corresponds to the dynamic instability threshold where the damping of the system falls to zero. The critical threshold is comprised between *4.0* and *4.5* as shown in Figure 5. As shown on the Figure, numerical results are consistant with previous solutions established numerically [12] in terms of critical reduced velocity threshold estimate. Moreover the solution

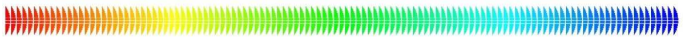

FIGURE 2. PRESSURE FIELD IN THE INITIAL STATE

obtained in the present work may be more consistant in the post-instability range since the solution resulting from the boundary linearization method is not reliable for large motion magnitudes. From these first results one can conclude that the partitioned procedure is convenient for simulation of flutter and identification of instability threshold. The full system damping estimate seems to be acceptable which tends to show that the numerical diffusion generated by the partitioned procedure does not affect the results significantly.

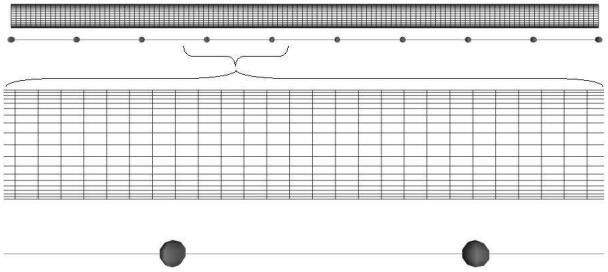

FIGURE 3. FLUID AND SOLID MESHES (TOP). ZOOM ON A SOLID ELEMENT (BOTTOM)

TABLE 1. PARAMETERS OF THE CONFIGURATION WITH A FLEXIBLE DUCT CONVEYING FLUID

– Tube length	L	=	$1,0$	m,
– Tube diameter	D	=	$4,0\ 10^{-2}$	m,
– Young modulus	E_s	=	$1,5.10^{9}$	Pa,
– Poisson coefficient	ν_s	=	$0,3$,	
– Solid density	ρ_s	=	160	kg.m^{-2},
– Fluid density	ρ_f	=	$1,0\ 10^{3}$	kg.m^{-3},
– Dynamic viscosity	η	=	$5,0\ 10^{-2}$	kg.m^{-1}.s^{-1},
– Reduced velocity	$\bar{v}_{red}$	$\in$	$[0,7]$,	
– Initial magnitude motion	$Ampl$	=	$1,84\ 10^{-6}$	m.s^{-1}.

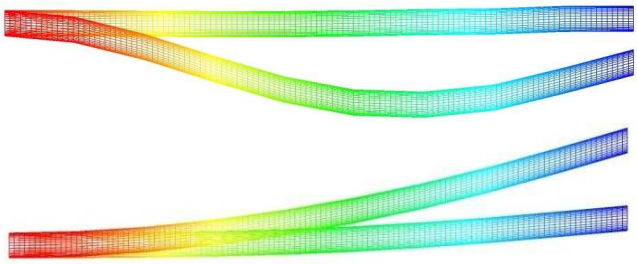

FIGURE 4. INSTANTANEOUS CYLINDER MODE SHAPES FOR REDUCED VELOCITY $4,0$ AND $4,5$.

Instability threshold in tube array under cross flow

Many studies on vibrations in tube arrays under cross flows have been performed over the last four decades. Experimental campaigns as well as theoretical and numerical developments have been published. The configuration that is investigated in the present section concerns a fictitious device modeled by using periodic boundary conditions to simulate an infinite tube array [4] [7] [15] [6]. The configuration is represented by a periodic square tube array involving 9 plain or cut cylinders (Figure 6). The pitch ratio is set to 1.44 and the modeling is two-dimensional. To avoid any three-dimensional effect induced by turbulence the Stokes number and the Reynolds number are chosen sufficiently small. Periodic conditions are introduced at inlet and outlet of the computational domain to ensure the convenient flowrate. Only the middle tube is supposed to have a rigid motion. The neighbour tubes are fixed. Fluid properties are described in Table 2. The properties of the moving tube are presented in Table 3. The modeling involves a single-degree-of-freedom system. Examples of velocity fields are shown in Figure 7. Simulations of tube motion have been performed for several reduced velocity in the low Reynolds number range (in this case the reduced velocity is defined by the ratio between the gap flow velocity and the product between the tube diameter and the tube frequency. The gap flow velocity is deduced from the inlet velocity and from the pitch ratio of the configuration). After post-processing the critical flow velocity of instability threshold can be identified. Displacement time history is illustrated in Figures 8 and 9 in stable and unstable cases. Figures 10 and 11 provide the evolution of frequency and damping with respect to inlet velocity. The frequency slightly decreases until a critical flow velocity has been reached. Above this critical threshold the damping is positive. Below it becomes negative and the frequency increases significantly. Accordingly an energy transfer occurs between kinetic energy terms and dissipative terms leading to a non conservative process.

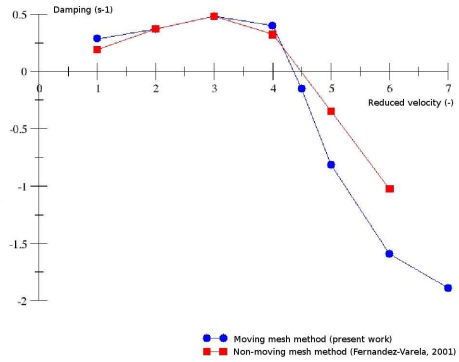

FIGURE 5. EVOLUTION OF DAMPING WITH RESPECT TO REDUCED VELOCITY

In this configuration there is no available experimental results because of the choice of the parameters enabling two-dimensional computations without any turbulence modeling. From an analytical point of view it is possible to formulate an exact expression of the fully-coupled fluid solid system and to identify its eigenvalues only in the case of a potential flow which

is not the case here. Therefore an interesting analysis consists in comparing the numerical solution obtained with the partitioned procedure to a predictive solution deduced from the Connors theory. According to Connors theory, the critical reduced velocity in the tube array under cross flow is expected to be provided by the following formula [5] :

$$\frac{U_{RC}}{f_n D} = C_{Connors}\left(\frac{m2\pi\xi}{\rho_f D^2}\right)^{1/2} \quad (2)$$

where f_n, m and ξ designate the solid frequency, mass and damping modified by added mass effects. ρ_f is the fluid density and D the tube diameter. $C_{Connors}$ designates the Connors constant taken to be equal to 4 according to standard convention in the framework of studies on in-line square tube arrays under single-phase cross flows. This relation provides an evaluation of the critical reduced velocity of $2,68$ which is consistant with numerical results displayed in Figures 10 and 11. The critical reduced velocity corresponds to a damping value falling to 0 obtained for an inlet velocity of $0,0187 m/s$ corresponding to a gap velocity of $0,61 m/s$ and leading to a critical value of $2,44$ for the reduced velocity. Numerical solutions are then in good agreement with expectations. These results show the capability of the partitioned coupling procedure to reproduce the non conservative fluid structure interaction in tube array under cross flow leading to a possible exponontial increase of vibration magnitude according to the rigid body linear theory.

Further works will now be developed in order to retrieve or to complete stability charts of tube arrays by reproducing operating conditions of heat exchangers [14] [19] [20] [10] [24].

TABLE 2. FLUID PROPERTIES FOR THE CONFIGURATION WITH A TUBE ARRAY UNDER CROSS FLOW

Fluid density	Kinematic viscosity
$1,0\ 10^3$ kg.m^{-3}	$1,0\ 10^{-6}$ m^2.s^{-1}

TABLE 3. SOLID PROPERTIES FOR THE CONFIGURATION WITH A TUBE ARRAY UNDER CROSS FLOW

Frequency	Reduced damping	Mass	Diameter
$2,5$ Hz	$0,0437$ %	$0,298$ kg	$10,0$ mm

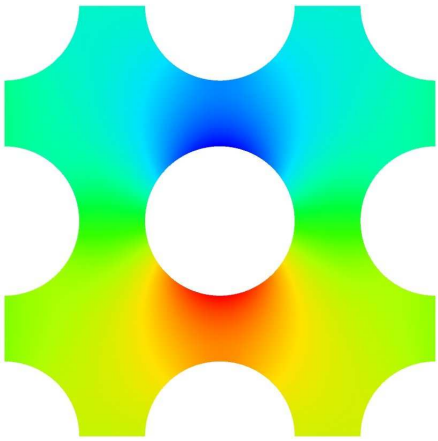

FIGURE 6. PRESSURE FIELD IN A MOVING TUBE ARRAY EMBEDDED BY STILL WATER

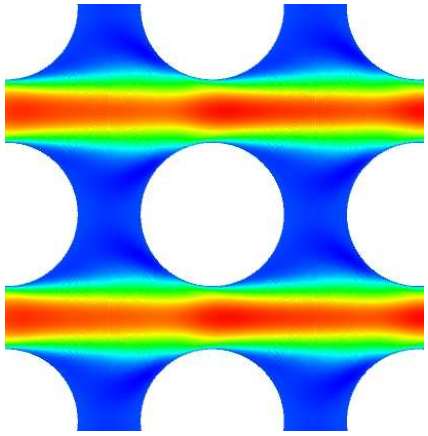

FIGURE 7. VELOCITY FIELD IN A MOVING TUBE ARRAY UNDER CROSS FLOW

CONCLUSION

The present work is related to simulation of vibrations of cylinders and cylinder arrays submitted to axial and cross flows. It is especially devoted to the identifiation of dynamic instability threshold. The application of staggered partitioned time integration to study of dynamic system stability is investigated. Further works are forecast on the extension of partitioned coupling procedure to strong non-linear effects, such as solid large deformations or turbulence in fluid flows. Vibrations of solids submitted to two-phase flows should although be investigated.

ACKNOWLEDGMENT

Thanks go to F. Baj and F. Huvelin for their helpful contribution in the present work. Authors also would like to

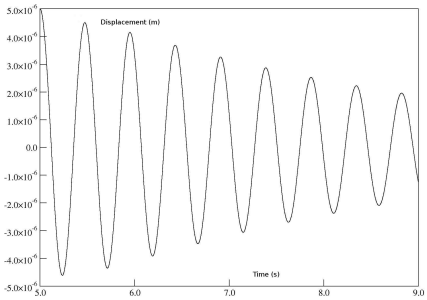

FIGURE 8. TUBE DISPLACEMENT FOR SMALL REDUCED VELOCITY

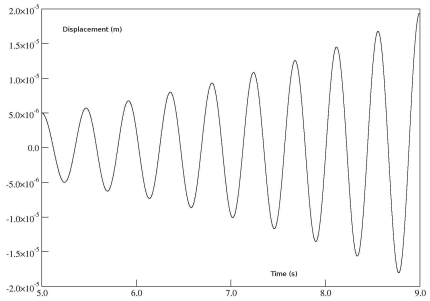

FIGURE 9. TUBE DISPLACEMENT FOR HIGH REDUCED VELOCITY

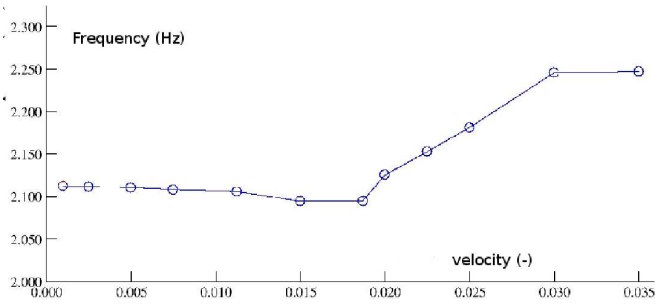

FIGURE 10. EVOLUTION OF TUBE FREQUENCY WITH INLET VELOCITY

thank the EDF R&D development teams of solvers and platform *Code_Saturne*, *Code_Aster* and *Salomé*.

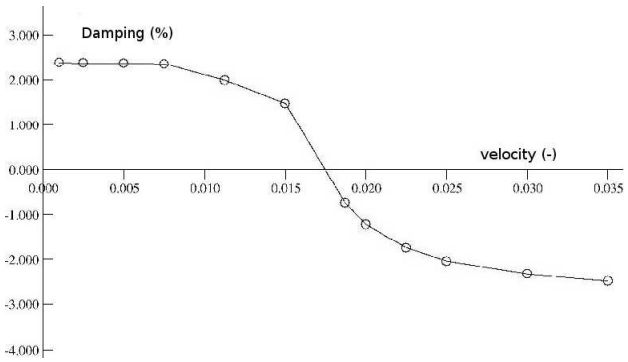

FIGURE 11. EVOLUTION OF TUBE DAMPING WITH INLET VELOCITY

REFERENCES

[1] Abouri, D., Parry, A., Hamdouni, A., Longatte, E., 2006. A stable fluid structure interaction algorithm : application to industrial problems. *Journal of Pressure Vessel Technology,* ***128***(4), pp. 516-524.

[2] Archambeau, F., Méchouita, N., Sakiz, M., 2004. A finite volume method for the computation of turbulent incompressible flows - Industrial applications. *International Journal on Finite Volumes,* ***1***.

[3] Axisa, F., 2001. Modélisation des systèmes mécaniques. *Hermès*.

[4] Baj, F., De Langre, E., 2003. Scaling of damping induced by bubbly flow across tubes. *Journal of Fluids and Structures,* ***17***, pp. 351-364.

[5] Blevins, R.D., 1977. Flow-induced vibrations. *Van Nostrand Reinhold*.

[6] Brandle De Motta, J.C., Baj, F., 2009. Numerical study of tube array vibrations under cross-flow. *Proceedings of AME Pressure Vessels and Piping Division Conference*, Prague, Czech Republic.

[7] De Morais, M.V.G., Baj, F., Gibert, R.J., Magnaud, J.P., 2007. Numerical inertia and damping coefficients determination of a tube-bundle in incompressible viscous laminar fluid. *Latin American Journal of Solids and Structures,* ***4***, pp. 179-203.

[8] Donea, J., Giuliani, S., Halleux, J.P., 1982. An Arbitrary Lagrangian Eulerian finite element method for transient dynamic fluid structure interactions. *Computer in Applied Mechanics and Engineering's,* ***33***(1-3), pp. 689-723.

[9] Farhat, C., Geuzaine, P., Grandmont, C., 2001. The discrete geometric law and the non linear stability of ALE schemes for the solution of flow problems on moving grids. *Journal of Computational Physics,* ***174***(2), pp. 669-694.

[10] Feenstra, M., Weaver, D.S., 1982. Modeling two-phase flow excited damping and fluidelastic instability in tube arrays. *Journal of Fluids and Structures,* ***16***(6), pp. 811-840.

[11] Fernandez, M.A., Gerbeau, J.F., Grandmont, C., 2007. A projection semi-implicit scheme for the coupling of an elastic structure with an incompressible fluid. *International Journal for Numerical Methods in Engineering,* ***69***(4), 794-821.

[12] Fernandez-Varela, 2001. Simple models for fluid structure

interaction. *PhD report*.
[13] Gibert, J.R., 1988. Vibrations des structures. *Eyrolles*.
[14] Granger, S., Paidoussis, M., 1996. An improvement of quasi-steady model with application to cross-flow induced vibration of tube arrays. *Journal of Fluid Mechanics,* ***320***, pp. 163-184.
[15] Huvelin, F., 2008. Code coupling for fluid structure interaction and application to fluid-elastic instability. *PhD Thesis*.
[16] Kassiotis, C., 2009. Non linear fluid structure interaction : a partitioned approach and its application through component technology. *PhD Thesis*.
[17] Longatte, E., Verreman, V., Souli, M., 2009. TIme marching for simulation of fluid structure interaction problems. *Journal of Fluids and Structures,* ***25***(1), pp. 95-111.
[18] Maman, N., Farhat, C., 1995. Matching fluid and structure meshes for aero-elastic computations : a parallel approach. *Computers and Structures,* ***54***(4), pp. 779-785.
[19] Paidoussis, M., 2003. Fluid structure interaction : slender structures and axial flow. Volume 1. *Academic Press Inc*.
[20] Pettigrew, M.J., Taylor, C.E., 1991. Fluidelastic instability of heat-exchanger tube bundles : review and design recommendations. *Journal of Pressure Vessel Technology,* ***113***, pp. 242-256.
[21] Price, S.J., Paidoussis, M.P., 1984. An improved mathematical model for the stability of cylinder rows subjected to cross flow. *Journal of Sound and Vibration,* ***97***, pp. 615-640.
[22] Piperno, S., 1997. Explicit fluid structure staggered procedures with a structure predictor and fluid subcycling for two-dimensional inviscid aeroelastic simulations. *International Journal of Numerical Methods in Fluids,* ***25***, pp. 1197-1116.
[23] Schaefer, M., 2006. Computational engineering, introduction to numerical methods. *Springer.*
[24] Weaver, D.S., 2008. Some thougths on the elusive mechanims of fluid-elastic instabilty in heat exchanger tube arrays. *Proceedings of Flow-induced Vibrations Conference,* , Prague,Czech Republic.