

Methods for numerical study of tube bundle vibrations in cross-flows

E. Longatte^{a,*}, Z. Bendjeddou^b, M. Souli^b

^a *Fluid Mechanics and Heat Transfer Department, EDF-R&D Division, 6 Quai Watier, Chatou 78400, France*

^b *Mechanical Engineering Department, University of Lille, 1 Boulevard Paul Langevin, Lille 59655, France*

Received 14 March 2003; accepted 18 August 2003

Abstract

In many industrial applications, mechanical structures like heat exchanger tube bundles are subjected to complex flows causing possible vibrations and damage. Part of fluid forces are coupled with tube motion and the so-called fluid-elastic forces can affect the structure dynamic behaviour generating possible instabilities and leading to possible short term failures through high amplitude vibrations. Most classical fluid force identification methods rely on structure response experimental measurements associated with convenient data processes. Owing to recent improvements in Computational Fluid Dynamics, numerical simulation of flow-induced vibrations is now practicable for industrial purposes. The present paper is devoted to the numerical identification of fluid-elastic effects affecting tube bundle motion in presence of fluid at rest and one-phase cross-flows. What is the numerical process? When fluid-elastic effects are not significant and are restricted to added mass effects, there is no strong coupling between structure and fluid motions. The structure displacement is not supposed to affect flow patterns. Thus it is possible to solve flow and structure problems separately by using a fixed nonmoving mesh for the fluid dynamic computation. Power spectral density and time record of lift and drag forces acting on tube bundles can be computed numerically by using an unsteady fluid computation involving for example a large Eddy simulation. Fluid force spectra or time record can then be introduced as inlet conditions into the structure code providing the tube dynamic response generated by flow. Such a computation is not possible in presence of strong flow structure coupling. When fluid-elastic effects cannot be neglected, in presence of tube bundles subjected to cross-flows for example, a coupling between flow and structure computations is required. Appropriate numerical methods are investigated in the present work. The purpose is to be able to provide a numerical estimate of the critical flow velocity for the threshold of fluid-elastic instability of tube bundle without experimental investigation. The methodology consists in simulating in the same time thermohydraulics and mechanics problems by using an arbitrary Lagrange Euler (ALE) formulation for the fluid computation. A fully coupled numerical approach is suggested and applied to the numerical prediction of the vibration frequency of a flexible tube belonging to a fixed tube bundle in fluid at rest or in flow. Numerical results turn out to be consistent with available experimental data obtained in the same configuration. This work is a first step in the definition of a computational process for the full numerical prediction of tube bundle vibrations induced by flows.

© 2003 Elsevier Ltd. All rights reserved.

1. Introduction

In many industrial configurations, mechanical structures such as PWR components are subjected to complex flows causing possible vibrations and damage and as far as nuclear security is concerned, it is necessary to prevent wear

*Corresponding author.

E-mail addresses: elisabeth.longatte@edf.fr (E. Longatte), mhamed.souli@univ-lille.fr (M. Souli).

Nomenclature

C_f	added damping in flow per unit length (N s/m ²)
C_s	structural damping per unit length (N s/m ²)
C_v	added damping in still water per unit length (N s/m ²)
\overline{D}	fluid deformation tensor (s ⁻¹)
D_e	tube outer diameter (m)
D_o	tube outer diameter in test facility (m)
D_i	tube inner diameter (m)
f	tube frequency (Hz)
f_o	forced tube frequency (Hz)
f_f	fluid-elastic force per unit length (N/m)
f_t	turbulent force per unit length (N/m)
K_s	structural stiffness per unit length (N/m ²)
K_f	added stiffness in flow per unit length (N/m ²)
\overline{I}	identity tensor (dimensionless)
L	tube length (m)
M_a	added mass per unit length (kg/m)
M_s	structural mass per unit length (kg/m)
P	pitch in square tube bundle (m)
\overline{R}	Reynolds stress tensor (N m/kg)
St	Stokes number (dimensionless)
x	tube displacement (m)
x_o	forced tube displacement (m)
U	pitch flow velocity (m/s)
ξ_s	tube damping ratio in still air (%)
ξ_e	tube damping ratio in still water (%)
ξ_f	tube damping ratio in flow (%)
$\overline{\sigma_f}$	fluid stress tensor (N/m ²)
$\overline{\sigma_s}$	structure stress tensor (N/m ²)
ω	“in flow” circular frequency $\omega = 2\pi f$ (rad/s)
ω_d	forced frequency in still water (rad/s)
ω_w	circular frequency in still water (rad/s)
ω_s	circular frequency in still air (rad/s)

problems generated by vibration fatigue. In this context, many experiments are carried out at EDF in order to predict turbulent and fluid-elastic forces responsible for possible flow-induced vibration problems. These forces can sometimes be directly measured by transducers but with direct approaches it is often difficult to stand between the different physical mechanisms involved when a distributed external loading is considered. On the contrary, indirect experimental prediction methods have shown their ability to provide fluid force estimates. Most of them rely on force density analytical models depending themselves on unknown spectral scaled parameters (Corcos, 1963; Gagnon and Païdoussis, 1994; Axisa et al., 1990). They are thus not always reliable, especially in the presence of complex turbulent flows. An advanced indirect approach has also been developed by EDF since about 15 years (Granger and Perotin, 1997a, b). It relies on structural vibration response measurement. After convenient transfer function calculation and data processing, the method provides an estimate of fluid excitations acting on dynamic structures. A modal modelling of the mechanical system is used and a spatial orthonormal decomposition of force fields is combined with a regularization process ensuring the closure system. This approach is efficient and it has been applied to the prediction of turbulent and fluid-elastic forces acting on tubes (Granger and Perotin, 1997a, b) and on PWR components like rod cluster control assemblies (Longatte et al., 2000) and heat exchanger tube bundles (Adobes et al., 2001). However, this technique often involves high costs because it relies on modelling fitted with experimental data deduced from measurements carried out on specific devices. As far as tube bundle vibrations in cross-flows are concerned, it is also possible to use a semi-analytical quasi-unsteady modelling fitted with experimental, numerical or analytical data and providing tube response expressed in terms of drag and lift force coefficients (Granger and Païdoussis, 1995; Granger and Gay, 1996). However, this modelling does not describe all physical phenomena involved by flow-induced vibrations.

In order to reduce experiments and to be able to study many configurations involving complex flow-induced vibration problems, numerical methods are also considered. Owing to recent developments incorporated into Computational Fluid Dynamic (CFD) codes, numerical simulation of flow structure coupling is investigated. There are three cases to be considered, as follows.

(i) In presence of fluid at rest, without flow, mass and damping terms added by fluid can be identified numerically by using a moving mesh formulation for the fluid computation. An Arbitrary Lagrange Euler (ALE) formulation is required (Souli et al., 1999; Souli, 2000, 2001; Souli and Zolesio, 2001) and structure motion is introduced in the fluid calculation as a moving boundary conditions.

(ii) In presence of turbulent flows, when turbulent forces are the most significant and fluid-elastic effects are reduced to added mass and damping effects, structural motion effects acting on turbulent flow patterns can be neglected. It is then possible to perform thermohydraulics and mechanics calculations separately (Benhamadouche and Laurence, 2002). Turbulent force spectrum and time record can be simulated by using large Eddy simulation (LES) and introduced as inlet conditions into the mechanical calculation providing the structure vibration response. LES has already been applied to the numerical prediction of turbulent loading acting on tubes subjected to turbulent mainly axial flows without confinement and numerical results turned out to be in good agreement with available analytical or experimental data (Moreno et al., 2000; Longatte et al., 2001).

(iii) Finally, in presence of flow with high fluid-elastic effects, it is necessary to use a specific numerical method. The configuration we are interested in is a flexible tube moving in a fixed tube bundle submitted to cross-flows (Fig. 1). The methodology consists in simulating in the same time thermohydraulics and mechanics problems by using an ALE formulation (Bendjeddou et al., 2002; Longatte et al., 2002). The purpose is to take into account the coupling between flow and structural motions. The fluid computational domain is distorted at each time step of the process to account for tube motion and associated strains and conversely fluid forces acting on structure walls are used to compute wall displacements.

The second configuration is not investigated here (Longatte et al., 2001). The present work is devoted to the study of the first and the third cases, i.e. to the identification of fluid-elastic parameters of tubes in fluid at rest and in cross-flows. The main objectives and methods are specified in the first section. The second part is devoted to the presentation of ALE formulations and of their application to the prediction of fluid-elastic parameters of tubes in fluid at rest. In the third part, the main flow-structure coupling processes are discussed and some results related to tube flow-induced vibrations are presented. According to the comparisons between numerical results and available experimental data, it is demonstrated that a full numerical simulation of tube bundle vibrations in cross-flows is now reachable.

2. Objectives

2.1. Physical problem

In nuclear power plants heat exchanger tube bundles carrying primary fluid are subjected to cross-flows of secondary fluid. External fluid forces may generate high magnitude vibrations of tubular structures causing possible dramatic damages in terms of nuclear safety. Vibrations result from four kinds of fluctuations (Pettigrew and Taylor, 2002a, b;

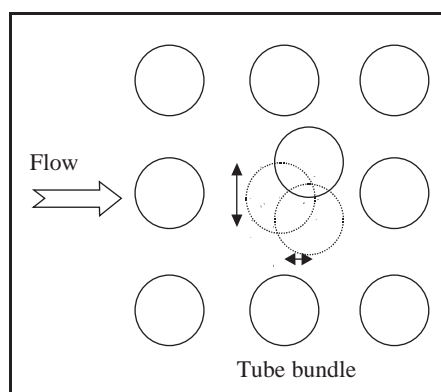


Fig. 1. Lift and drag force effects on a flexible tube belonging to a fixed tube bundle in cross-flow.

Chen, 1987; Price and Païdoussis, 1989): (i) random fluctuations generated by turbulence in fluid at large Reynolds numbers; (ii) fluctuations induced by structure-flow motion coupling due to fluid-elastic effects; (iii) resonance with flow periodicity due to vortex shedding; and (iv) possible acoustic excitation.

Fluid-elastic forces and resonance with flow periodicity resulting from this coupling can affect the structural dynamical behaviour, causing possible instabilities and leading to possible short term failures through high magnitude vibrations. For industrial concerns, it is necessary to be able to predict these fluid-elastic forces and their effects on tube bundle dynamic stability. In the present work we focus our attention on this kind of fluid forces.

Flow-structure interaction induces energy exchange between flow and structural motions and, up to a critical flow velocity, the flow-structure system may become unstable and high magnitude structure displacements occur. Fig. 2 features a typical response showing tube motion amplitude and cross-flow velocity.

Fluid-elastic instability development has been widely studied experimentally and many experiments were carried out to identify critical flow velocity in many different configurations. Experimental data resulting from these studies gave rise to available reference instability maps providing critical velocity thresholds in terms of tube bundle characteristic parameters. The purpose of the present work is to build a numerical method to retrieve numerically the laws between tube bundle parameters and critical flow velocity.

2.2. Fluid-elastic parameters definition

We are interested in the study of vibrations of a flexible tube belonging to a regular fixed tube bundle subjected to a fluid coupling, with or without flow. This configuration is defined by known parameters describing the system geometry and hydraulics (Adobes et al., 2001). For each configuration to be studied, for a given parameter gather, fluid-elastic forces may affect tube motion. As we will see below, fluid-elastic effects can be expressed and measured in terms of fluid-elastic parameters. These coefficients were previously identified experimentally in many different configurations (Chen, 1986; Price and Païdoussis, 1986). The purpose here is to show how to estimate numerically these fluid-elastic parameters.

Geometric parameters characterizing a regular tube bundle are the following: tube external and internal diameters $D = D_e$ and D_i ; tube gap P ; tube row angle θ ; and tube bundle length L . From a mechanical point of view, the flexible tube motion is characterized by: tube mass M_s ; tube stiffness K_s ; tube damping C_s ; and mass of the tube internal fluid M_f . In presence of one vibration mode, which may be a double one, and is denoted s , the equation of motion of the tube without fluid can be written as follows:

$$M_s \ddot{x} + C_s \dot{x} + K_s x = 0, \tag{1}$$

or equivalently,

$$\ddot{x} + 2\zeta_s \omega_s \dot{x} + \omega_s^2 x = 0, \tag{2}$$

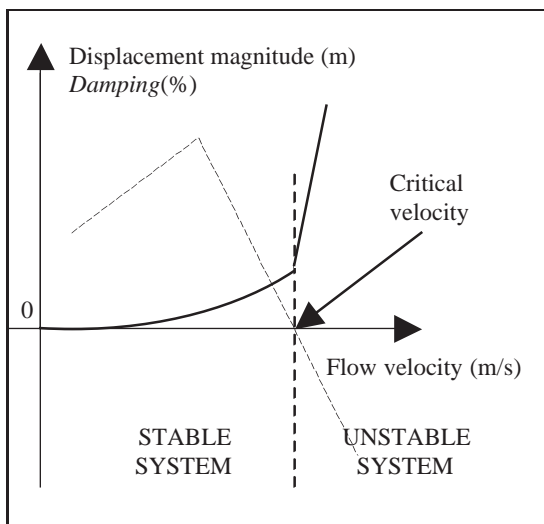


Fig. 2. Critical flow velocity generating structure instability development.

where ω_s and ξ_s designate pulsation and damping coefficients of the tube alone defined by

$$K_s = M_s \omega_s^2 \quad (3)$$

and

$$C_s = 2M_s \omega_s \xi_s. \quad (4)$$

Concerning the hydraulics parameters, the flow is supposed to cross-perpendicularly the tube bundle and the axial component is zero. The flow is totally defined by its Reynolds number,

$$\text{Re} = \frac{\rho D U}{\mu},$$

where $U = U_{\text{gap}} = [P/(P-D)]U_\infty$ designates the gap velocity, U_∞ the inlet flow velocity before crossing the tube bundle and μ the fluid dynamic viscosity. Concerning the structural motion in the fluid at rest, the apparent mode mass and damping are affected by added mass effects (represented by M_a), by internal fluid mass effects (represented by M_i) and by fluid viscosity (represented by C_v). The equation of motion becomes

$$(M_s + M_a + M_i)\ddot{x} + (C_s + C_v)\dot{x} + K_s x = 0, \quad (5)$$

or

$$\ddot{x} + 2\xi_e \omega_e \dot{x} + \omega_e^2 x = 0, \quad (6)$$

where ω_e and ξ_e designate the pulsation and damping coefficients of the tube in fluid at rest. Then one can use the following identification:

$$K_s = M_s \omega_s^2 = (M_s + M_a + M_i)\omega_e^2.$$

This yields

$$M_a = M_s \left(\frac{\omega_s^2}{\omega_e^2} - 1 \right) - M_i, \quad (7)$$

$$C_v = 2(M_s + M_a + M_i)\omega_e \xi_e - C_s. \quad (8)$$

If one knows the structural parameters in air, mass and damping terms added by the fluid are totally expressed in terms of tube frequency and damping in fluid at rest.

In presence of flow, stiffness and damping terms are affected as follows:

$$(M_s + M_a + M_i)\ddot{x} + (C_s + C_f + C_v)\dot{x} + (K_s + K_f)x = F_t, \quad (9)$$

with F_t designating fluid forces independent on structure motion. Equivalently one gets

$$\ddot{x} + 2\xi_f \omega_f \dot{x} + \omega_f^2 x = 0, \quad (10)$$

and after identification

$$(K_s + K_f) = (M_s + M_a + M_i)\omega_f^2,$$

$$(C_s + C_f + C_v) = 2\xi_f \omega_f (M_s + M_a + M_i).$$

Finally, the stiffness and damping terms added by flow are given by

$$K_f = (M_s + M_a + M_i)\omega_f^2 - M_s \omega_s^2,$$

$$C_f + C_v = 2\omega_f \xi_f (M_s + M_i + M_a) - 2M_s \omega_s \xi_s.$$

They can be expressed in terms of tube frequency and damping in air and in flow as follows:

$$K_f = M_s \omega_s^2 \left(\frac{\omega_f^2}{\omega_e^2} - 1 \right), \quad (11)$$

$$C_f + C_v = 2M_s \left(\omega_f \xi_f \frac{\omega_s^2}{\omega_e^2} - \omega_s \xi_s \right). \quad (12)$$

From a practical point of view, the critical flow velocity is reached when a structure dynamic instability develops. That is the reason why it is important to be able to control these parameters. The fluid-elastic parameters M_a , C_f , C_v and K_f can be identified experimentally after measurement of tube frequency and damping in air, in fluid at rest and in

flow. The purpose of the present work is to introduce a method for the numerical evaluation of these parameters. This requires specific techniques (i) to perform the fluid computation, (ii) to perform the structural calculation, (iii) and finally to make the data-exchange possible between the two calculations by using a convenient moving boundary condition or an appropriate code coupling process. The computational process is described below.

2.3. Computational process

In continuum mechanics one can describe fluid motion with two classical formulations: (a) An Eulerian formulation: one focuses as his attention on a particular volume in space. This volume is fixed with respect to a laboratory frame, and one consider the fluid as it passes through the fixed volume. The fluid is continuously renewed inside the domain and a convective term is introduced in the basic equations of motion to express the material time derivative in the reference configuration; (b) A Lagrangian formulation: one identifies and follows a particular region of fluid. The volume of fluid changes in shape while the total mass remains constant. The computational domain mesh moves with the particle flow velocity and this may lead to an element entanglement. This formulation is not convenient in presence of high magnitude motions.

For problems involving moving wall boundaries, it is necessary to have a middle formulation following the boundary motion and preserving the element shape in the same time. An ALE formulation has been introduced to ensure these capabilities. Finite element ALE formulation for incompressible viscous flows has been introduced by Hughes et al. (1981), Liu and Huerta (1988), Belyschko et al. (1982), and the finite difference formulation by Noh (1964) and Hirt et al. (1974). The purpose of the ALE algorithm is to enable the computational mesh to remain regular even in presence of high magnitude structure displacements (Benson, 1989).

In the present work, an ALE formulation is used to study numerically the vibrations of a tube in fluid at rest and a tube in cross-flow. From a numerical point of view there are two cases to be considered:

(i) the tube motion can be introduced as a moving boundary condition and the fluid problem is solved by using an ALE formulation involving a time-dependent computational domain; or

(ii) a flow-structure code coupling is required to account for interactions between fluid and structure problems. At each time step, the fluid problem is solved on the reactualized computational domain and fluid forces acting on the flexible tube are estimated. These forces are then introduced as inlet conditions in the right-hand side of mechanical equation (1) providing tube displacement and velocity, the tube velocity is used to deform the computational mesh and to estimate the new problem geometry at each iteration.

Both cases are considered below. In each case, numerical methods are described and results are discussed. Finally, one gets a full computational process, making it possible to identify numerically all fluid-elastic parameters characterizing tube bundle vibrations in cross-flows.

3. Numerical methods

3.1. ALE formulation

The ALE formulation was previously used to solve defence problem and nowadays its application was extended to free surface problems, high velocity impact, offshore structures, multi-physics problems. It has also medical applications like modelling of blood vessel deformation. There are two ways to solve ALE equations: the first one which is used here corresponds of an Eulerian viewpoint, the fully coupled equations are solved in one step; this approach can only handle with one-phase flows; the second method described by Souli et al. (1999, 2000) is a split method: it uses two steps to solve ALE equations: (i) a Lagrangian step in which the mesh moves with material velocity, and (ii) an advection step where the mesh moves from its material position to its arbitrary position. This method is better to model two-phase flows for explosion modelling for instance.

Here the ALE method is investigated to evaluate numerically the fluid-elastic parameters of a flexible tube in tube bundle in fluid at rest or in flow. In the framework of Arbitrary Lagrangian Eulerian formulations the fluid dynamic problem is solved as follows. One defines three domains in space and associated mappings from one domain to another (Fig. 3). The first one is called the material domain Ω_m and follows the fluid particle motion X (Lagrangian formulation). The second domain called spatial domain Ω_s is fixed and occupies fixed positions in space x (Eulerian formulation). It is convenient to rely the Eulerian and Lagrangian space reference coordinates, respectively, x and X as follows:

$$x = x(X, t),$$

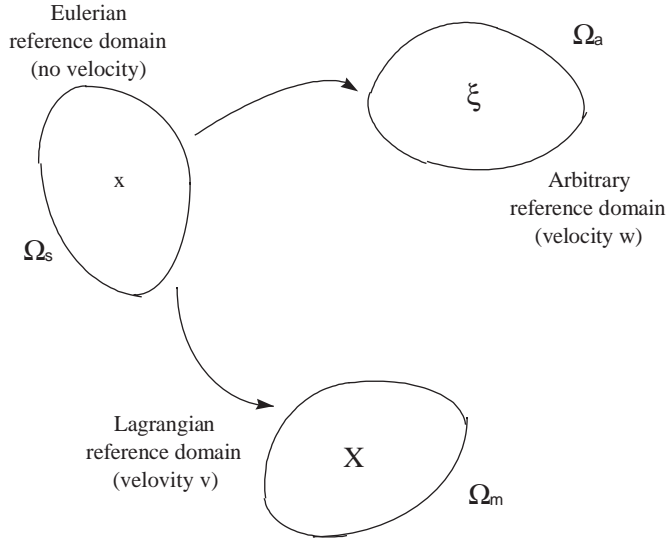


Fig. 3. Representation of Eulerian, Lagrangian and arbitrary reference domains Ω_s , Ω_m and Ω_a whose velocities are, respectively 0, v and w .

$$\left. \frac{\partial x}{\partial t} \right|_X = v(X, t) = v(x, t), \tag{13}$$

with v the material velocity. Moreover, the material time derivative of a physical property ϕ is given by

$$\frac{\partial \phi}{\partial t} = \left. \frac{\partial \phi(X, t)}{\partial t} \right|_X = \left. \frac{\partial \phi(x, t)}{\partial t} \right|_x + \left. \frac{\partial \phi(x, t)}{\partial x_i} \right|_t v_i(x_i, t). \tag{14}$$

Finally, a third domain is introduced, called the arbitrary domain Ω_a , with an arbitrary motion different from the material domain motion. In the fixed spatial domain Ω_s , the arbitrary domain may be described by coordinates ξ expressed as follows:

$$x = x(\xi, t), \tag{15}$$

$$\left. \frac{\partial x}{\partial t} \right|_\xi = w(\xi, t) = w(x, t),$$

with the following time derivative expression:

$$\left. \frac{\partial \phi(\xi, t)}{\partial t} \right|_\xi = \left. \frac{\partial \phi(x, t)}{\partial t} \right|_x + \left. \frac{\partial \phi(x, t)}{\partial x_i} \right|_t w_i(x_i, t). \tag{16}$$

In presence of structural wall motion, it is then useful to choose the arbitrary domain Ω_a according to the moving structure boundaries. In this way, ξ and w designate location and velocity of the domain Ω_a . Thus, the governing equations in Ω_a are deduced from the ALE formulation. If one assumes the fluid is incompressible, the mass, momentum and energy conservation equations are formulated as follows:

$$\text{div}_x(v) = 0, \tag{17}$$

$$\rho \left\{ \left. \frac{\partial v}{\partial t} \right|_\xi + (v - w) \text{grad}_x(v) \right\} = - \text{grad}_x(p) + \text{div}_x(\mu \text{grad}_x(v)) - \rho \text{div}_x(R),$$

$$\rho \left\{ \left. \frac{\partial R_{ij}}{\partial t} \right|_\xi + (v - w) \text{grad}_x(R_{ij}) \right\} = - P_{ij} + \rho \Phi_{ij} - d_{ij} - \rho \varepsilon_{ij},$$

$$\rho \left\{ \left. \frac{\partial \varepsilon}{\partial t} \right|_\xi + (v - w) \text{grad}_x(\varepsilon) \right\} = \rho C_{\varepsilon 1} \frac{\varepsilon}{k} P - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \text{div}_x(B(\text{grad}_x(\varepsilon))).$$

Here R describes the Reynolds tensor defined by

$$R_{ij} = \overline{v_i v_j}.$$

In the framework of turbulence statistical approaches, each physical field ϕ can be split into two components: a mean part $\bar{\phi}$ and a fluctuating part ϕ' ; i.e.,

$$\phi = \bar{\phi} + \phi'.$$

In order to solve this problem it is necessary to introduce the mapping from one domain to the others. The relation between Eulerian x and Arbitrary Lagrangian ξ coordinates is ensured by

$$J = J(\xi, t) = \left| \frac{\partial x_i}{\partial \xi_j} \right| \quad \text{with} \quad J = Id \quad \text{due to the first-order discretization.}$$

Finally, all fields are expressed in terms of J in Ω_a and at each time step the solution is projected in the new spatial domain Ω_s . The point is that it is necessary to compute the velocity w or the displacement x of the arbitrary domain. A first-order approximation consists in relying velocity and displacement by

$$\xi_j^{n+1} = \xi_j^n + w_j^n \Delta t,$$

with Δt the time step, ξ_j^n and ξ_j^{n+1} the displacement of Ω_a at time steps n and $n+1$, and w_j^n the velocity at time step n .

In order to compute displacement, the domain Ω_a is represented as a continuum domain. The difficulty in the ALE formulation is to choose an appropriate arbitrary velocity in order to avoid element entanglement. There are many algorithms to get this velocity (Souli et al., 1999; Soli, 2000). In the general case, the mesh is considered as an elastic body and its distortion is solution of a classical “mechanical problem”. A stress tensor is defined for the mesh and this tensor can be a linear or nonlinear function of displacement or velocity.

In this computation the mesh domain corresponds to an incompressible fluid, it has thus a constant volume and the stress tensor is chosen as a linear function of velocity. For instance one can assume that the mesh is the most distorted near moving boundaries and the distortion propagates through the full domain, falling to zero far from the

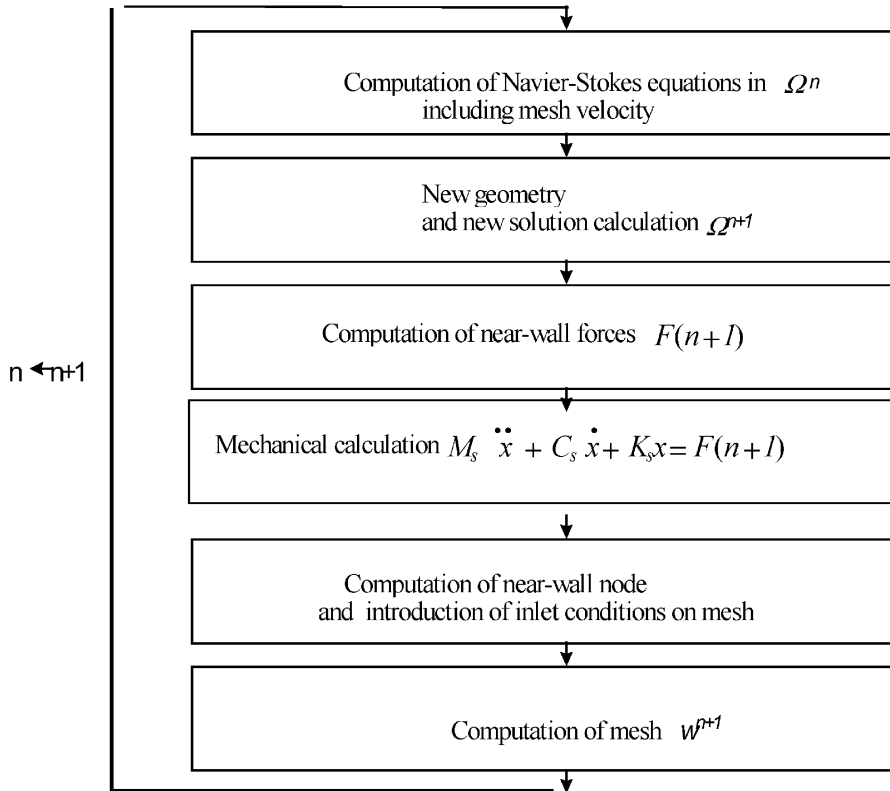


Fig. 4. Computational process for flow and structure motion coupling.

boundaries. This process can be described by a classical diffusion equation for w or ξ in Ω_a :

$$\text{div}(\lambda \text{grad } w) = 0 \tag{18}$$

associated with convenient boundary conditions as displacements are known on moving and nonmoving arbitrary domain boundaries. λ designates a specific viscosity to be defined. In the present work one puts $\lambda = 1$ far from the moving boundary and $\lambda = 10^6$ near the tube to keep the mesh distortion homogeneous near moving walls.

Moreover, the fluid dynamic code used in the present work relies on a finite volume formulation with nonstructured meshes. Hence physical mean fields are computed on element volumes and normal gradients are estimated on each volume boundary. In the case of a nonstructured mesh, gradients may be rebuilt. Here meshes are cell-centred and fields are explicitly expressed in volume centres and in face centres. Finally, a semi-implicit method for pressure linked equations (SIMPLE algorithm) is used and one solves a linear system by using an iterative method, like the preconditioned conjugate gradient for pressure and Jacobi or Gauss–Seidel method for velocity. The mechanical equation is solved by using a second order centred scheme or a Newmark scheme. The mesh velocity w is also cell-centred and the velocity of each node is explained as a simple average of velocities cells including this node.

3.2. Fluid force modelling

The structure response is directly generated by near-wall fluid forces. At each time step of the fluid calculation, lift and drag forces acting on the flexible tube and responsible for its motion are estimated. These forces are expressed in terms of the stress tensor $\bar{\sigma}$ by

$$\vec{F} = \vec{T}S, \tag{19}$$

where S designates the wall surface and \vec{T} is defined by

$$\vec{T} = \bar{\sigma} \cdot \vec{n}, \tag{20}$$

with \vec{n} the unitary normal vector of the structure wall. According to the Stokes approximation, the stress tensor expression is given by

$$\bar{\sigma} = -p\bar{I} + 2\mu\bar{D} \quad \text{for laminar flows,}$$

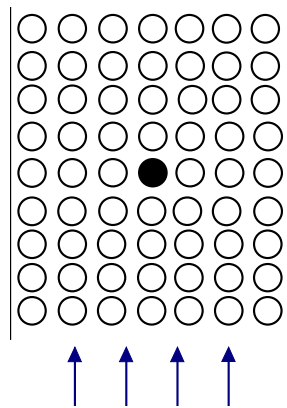


Fig. 5. Experimental set-up section: in-line tube bundle including 7×9 fixed tubes except the middle tube that is moving in presence of cross-flow.

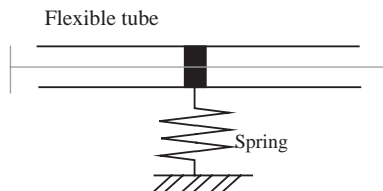


Fig. 6. Supporting process of the middle flexible moving tube.

Table 1
Flexible tube characteristic parameters

Tube modal mass M_s (kg)	0.223
Internal fluid modal mass M_i (kg)	0.061
Tube length L (m)	0.250
Tube external diameter D_e (m)	0.022
Tube modal stiffness K_s (N/m)	13185
Tube modal damping ξ_s (%)	0.13

Table 2
Flexible tube characteristic parameters

Tube vibration frequency f_s (Hz)	38.7
-------------------------------------	------

$$\bar{\sigma} = -p\bar{I} + 2\mu\bar{D} - \rho\bar{R} \quad \text{with a Reynolds stress-model for turbulent flows.}$$

The vortex shedding frequency is characterized by the Strouhal number: $St = fD/U$, with f the lift force frequency. Vortex shedding is a well-known phenomenon for a single tube in cross-flow but it is much more complicated for tube bundle and in this case the determination of the Strouhal number is difficult and depends on Reynolds number and tube confinement.

3.3. Coupling process

To account for fluid or flow structure coupling it may be necessary to use a code coupling to simulate both physical problems in the same time. The full computational process may be described as follows (Fig. 4). It relies on an ALE formulation for the fluid computation. At each step n of the fluid calculation, Navier–Stokes equations are solved in the fluid computational domain Ω^n and the system is modified to take into account a mesh velocity in the momentum conservation equation for the convective term. Then the new geometry and the new solution are calculated in domain Ω^{n+1} . Near-wall forces F^{n+1} acting on the moving tube are deduced from stress tensor at step $n+1$. These forces are introduced in the right-hand side of tube motion equations (1). According to Eq. (9), they can be identified to added flow effects as follows:

$$M_s\ddot{x} + C_s\dot{x} + K_sx = F = -(M_a + M_i)\ddot{x} - (C_f + C_v)\dot{x} - K_fx. \quad (21)$$

The mechanical calculation provides displacement x describing tube motion. Tube node velocity is then deduced from near-wall node displacement and it is introduced as inlet conditions on mesh velocity. Finally, the mesh velocity w^{n+1} is computed in the full domain Ω^{n+1} at step $n+1$ by solving the mesh diffusion equation.

In this way, one takes into account both fluid effects on tube displacements and conversely tube motion effects on fluid patterns. The previously mentioned method relies on a first-order explicit staggered coupling scheme. The main disadvantage of this approach is that the energy conservation equation for the full system is not satisfied because structure or fluid energy is numerically dissipated or created at the fluid–structure boundary.

To avoid this unwanted property another method is also considered. It consists in using other kinds of staggered synchronous or asynchronous schemes minimizing errors on numerical energy dissipation (Piperno and Farhat, 2001). It is also possible to use implicit code coupling process introducing sub-cycling until convergence of fluid calculation at each time step (Hermann and Steindorf, 1999).

4. Application

4.1. Test case

The previously mentioned numerical methods were used to build a complete numerical tool devoted to the prediction of tube vibrations in fluid at rest and in cross-flows. Numerical simulations were carried out on a specific test case corresponding to an experimental device (Granger et al., 1993). In this configuration many experimental data are available providing tube vibration frequency, damping ratio and r.m.s. vibration in terms of gap velocity. The

test-section is described in Fig. 5. An in-line tube bundle includes 7×9 fixed tubes, except the middle tube that is moving under the action of cross-flow. In the cross-direction the bundle is limited by two parallel walls simulating symmetry conditions. The flexible tube support is depicted in Fig. 6. The mockup was so built that the tube can only move in the cross-direction without any deformation. The tube displacement is called x and it is governed by a mechanism whose characteristic parameters are an apparent spring stiffness K_s and an apparent damping ratio ζ_s .

The main tube mechanical and geometrical parameters are reported in Table 1. According to these data, the theoretical tube motion frequency without fluid is deduced from Eq. (3), (Table 2).

4.2. Computation of tube vibration frequency in still water

The purpose is to evaluate numerically fluid-elastic parameters of a flexible tube in fluid at rest, i.e. added mass and damping terms induced by the fluid: M_a and C_v . An ALE formulation is used and there is no fluid–structure coupling, as the structure is supposed not to be affected by the fluid and its motion is imposed.

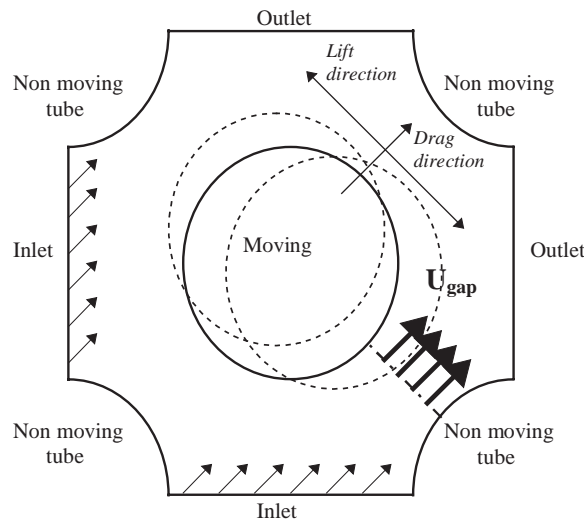


Fig. 7. Fluid computational domain providing a simplified representation of the experimental set-up depicted in Fig. 5. The tube bundle is supposed to be infinite. With periodic inlet and outlet conditions, each flexible tube neighbour is fixed and the tube motion is not expected to affect the other flexible tube motion. Inlet flow rates is introduced to get the convenient gap velocity U_{gap} . The tube is expected to move in the drag direction and to oscillate in the lift direction.

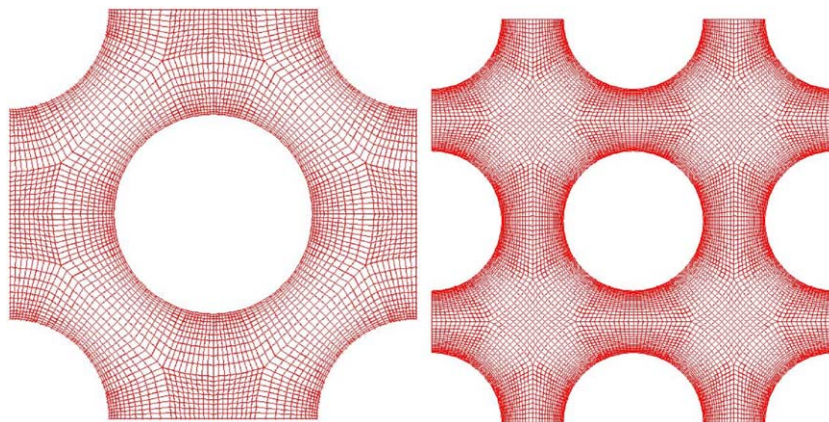


Fig. 8. Fluid computational domain mesh section at time step $n = 0$ in 5-tubes (left) and 9-tubes (right) configurations.

4.2.1. Fluid computational domain

Numerical simulation of the previous experiment relies on simplified assumptions making the calculation easier and shorter. The fluid computational domain describing the experimental set-up is represented in Fig. 7. It is restricted to the smallest tube bundle cell featuring periodicity involving 5 or 9 tubes as shown on Fig. 8, showing also the associated meshes. The tube bundle is supposed to be infinite in the lift and drag directions. This assumption is checked if one only considers vibrations of the middle tube in the tube bundle (Fig. 5). The fluid flow is supposed to be two-dimensional (2-D) and each tube is supposed to have infinite length in the length direction.

4.2.2. Boundary conditions

The computational domain is limited by periodic inlet and outlet conditions on the flow. In this way each near neighbour of the flexible tube is fixed and interactions between several flexible tubes are neglected. Periodic and free outlet boundary conditions have been tested. To model periodic boundary conditions, periodic cells are added to the real mesh and a term source is defined and introduced in the right-hand side of Navier–Stokes equations to keep a constant flow rate in case of fluid motion. One uses a second-order Newton method to compute the term source (Benhamadouche and Laurence, 2002).

There is no fluid flow, so turbulence effects and added stiffness decrease to null if displacements remain small. Here the fluid motion is only due to structural displacements and one keeps small tube displacements in order to keep laminar flow and neglect the turbulence forces.

Furthermore, the structural motion is supposed to be linear and a modal base is defined to compute displacement. To model correctly the fluid forces, the boundary condition between fluid and structure must be defined properly and precisely. Here the continuity of velocity and normal stress tensor at the fluid–structure interface is satisfied. No wall law is used on the moving boundary.

4.2.3. Identification of tube vibration frequency in fluid at rest

The tube motion is introduced as an imposed harmonic boundary condition solution of the homogeneous structure problem. For this calculation a periodic tube motion is imposed as fluid and mesh inlet conditions with a fixed frequency $\omega_d = 2\pi f_d$. At each time step fluid forces acting on the tube are estimated. According to Eq. (5) these forces are expressed in terms of added mass and viscosity damping terms as follows:

$$(M_s + M_f)\ddot{x} + C_s\dot{x} + K_s x = F = -M_a\ddot{x} - C_v\dot{x}. \quad (22)$$

Table 3

Inlet data for the calculations in fluid at rest : tube imposed harmonic frequency f_d (Hz) and tube vibration magnitude x_o (m)

	Inlet data 9 tubes	Inlet data 5 tubes
Harmonic frequency f_d (Hz)	29	29
Tube displacement magnitude x_o (m)	10^{-4}	10^{-4}

Table 4

Numerical results for the calculation in fluid at rest in terms of force magnitudes F_o (N) and phase φ between forces and tube displacement

	Results 9 tubes	Results 5 tubes
Force magnitude F_o (N)	2.7×10^{-3}	1.9×10^{-3}
Force phase φ (rad)	0.06	0.06

Table 5

Comparison between added mass term M_a deduced numerically from Eq. (24) and experimentally from Eq. (7). Good agreement between tube frequency in the fluid at rest deduced from Eq. (5) numerically and experimentally

	Num. 9 tubes	Num. 5 tubes	Exp.
Added mass M_a (kg)	0.104	0.057	0.116
Frequency in fluid at rest f_e (Hz)	29.3	31.3	29.0

For an harmonic tube motion $x = x_o e^{i\omega_d t}$, fluid forces are also periodic $F = F_o e^{i(\omega_d t + \varphi)}$, with φ the phase between fluid forces and tube displacement, and they can be split into two parts:

$$F_o e^{i\varphi} e^{i\omega_d t} = -M_a \ddot{x} - C_v \dot{x} = (M_a \omega_d^2 x_o - i C_v \omega_d x_o) e^{i\omega_d t}.$$

Then coefficients are given by

$$M_a = \frac{F_o \cos \varphi}{\omega_d^2 x_o}, \tag{23}$$

$$C_v = - \frac{F_o \sin \varphi}{\omega_d x_o}. \tag{24}$$

Accuracy on damping estimate depends on the Stokes number as the phase φ decreases with St. High Stokes number configurations requires small time steps. Inlet data used for these calculations are reported in Table 3. Fluid force and tube displacement histories are plotted in Fig. 9. Corresponding force magnitude F_o and phase φ are reported in Table 4. They are compared to experimental data deduced from Eqs. (7) and (8). Results are in good agreement. Added mass and viscosity damping terms are of the right order. The tube motion frequency and damping ratio in fluid at rest are finally deduced from Eqs. (5) and (6). Numerical values for M_a and f_e are reported in Table 5 and compared to experimental data. A good agreement is observed. It is interesting to notice that the ALE formulation enables the estimation of added mass effects. The 9-tubes mesh and periodic condition provide better results for added mass and added viscosity than the 5-tube configuration. These results tend to show that the assumption of an infinite tube bundle is not quite validated. Fluid–structure interactions depend on the confinement and in the 5-tube mesh the periodic boundary conditions are not sufficient to describe the effect of fixed tubes located near the moving tube, which tends to underestimate fluid–structure coefficients. Damping ratio is expressed in terms of $\sin \varphi$; hence results are very much dependent on mesh refinement and time step at high Stokes number. Added mass terms are sufficient to estimate vibration frequency characterizing tube motion in fluid at rest.

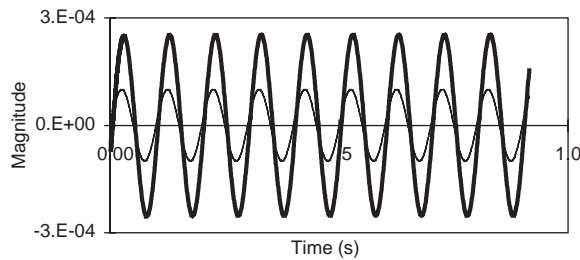


Fig. 9. Fluid force F (heavy line) and tube displacement x (thin line) time history. Estimation of force phase φ and force magnitude F_o .

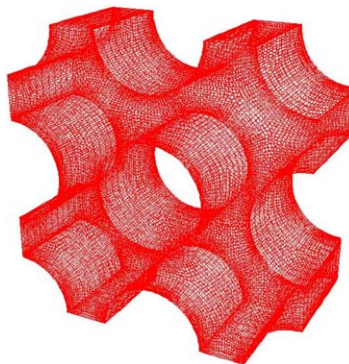


Fig. 10. 3-D mesh of the fluid computational domain at time step $n = 1$ with a 9-tubes configurations.

4.3. Simulation of flow-induced vibrations

In what follows, the purpose is to use the ALE method and a coupling with the mechanical calculation previously described in order to simulate the tube behaviour in presence of cross-flows. A first estimation of tube frequency in flow is provided for several gap velocities.

4.3.1. Fluid computation

Inlet flow rates are introduced to get the convenient gap velocity U_{gap} . The mechanical modelling is made up such that the tube is expected to deflect in the drag direction and to oscillate in the lift direction, as observed experimentally (Fig. 9). Three-dimensional (3-D) representation of the computational domain mesh used for the calculation is shown in Fig. 10. It is near-wall refined in the region near flexible tube where fluid forces must be estimated with accuracy.

4.3.2. Structure computation

For the coupling with the mechanical calculation, a staggered explicit time scheme is applied. At each time step, the tube motion is reactualized according to the coupling process described in Section 2.3. Tube displacement is solution of Eq. (21) solved by using appropriate numerical scheme. An Euler explicit scheme, a centred second order scheme and a Newmark scheme were compared, and the last one provided the best results in terms of numerical damping reduction.

4.3.3. Identification of tube vibration frequency in flow

Numerical results are described below. They are compared to experimental data obtained with the previously described experimental set-up. Calculations are performed in presence of turbulent flows in order to identify fluid-elastic parameters. Three-dimensional calculations were carried out for different gap velocity values corresponding to a Reynolds number range of 1 to 4×10^4 . Turbulence modelling was introduced to describe the flows. Several models were tested and, for 3-D calculations, it was shown that a $R_{ij} - \varepsilon$ modelling provides good results in terms of drag and lift forces near the tube (Benhamadouche and Laurence, 2002). In the present work a $R_{ij} - \varepsilon$ modelling involving an appropriate near-wall treatment or a DNS was used. In practice fluid-elastic forces are supposed to be independent of turbulence effects and the turbulence model may not affect numerical results in terms of tube vibration frequency.

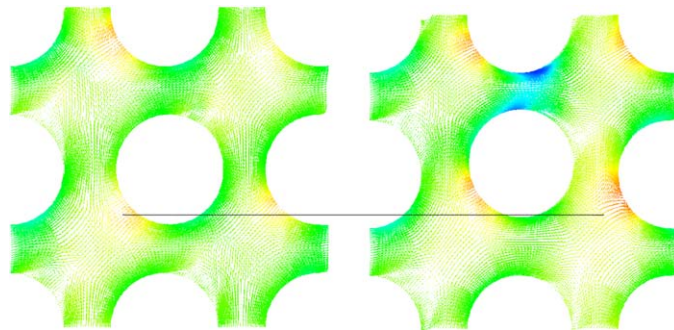


Fig. 11. Velocity fields coloured by pressure at two time steps by numerical simulation involving an ALE formulation and a flow structure coupling process.

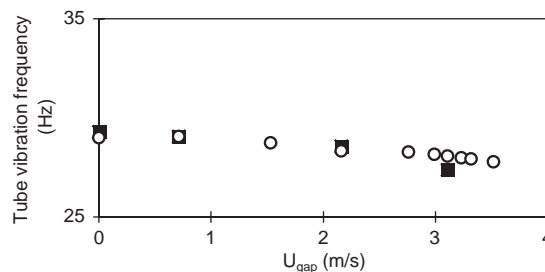


Fig. 12. Tube vibration frequency in flow f (Hz) in terms of gap velocity U_{gap} (m/s) estimated numerically (black points) and experimentally by Granger et al. (1993) (white points).

Table 6

Comparison between tube frequency f_f (Hz) of tube in flow estimated numerically in 2-D and in 3-D and experimentally for several gap velocity values U_{gap}

Frequency in flow f_f (Hz)	Numerical estimates 3-D	Numerical estimates 2-D	Experimental estimates
$U_{\text{gap}} = 0$ m/s		29.30	29.00
$U_{\text{gap}} = 0.71$ m/s	29.06	29.85	29.05
$U_{\text{gap}} = 2.16$ m/s		28.57	28.32
$U_{\text{gap}} = 3.11$ m/s	27.39		28.09

Fig. 11 provides flow fields simulated by using the full computational process for a given flow velocity. The time history of tube displacement induced by flow is illustrated. Finally, in-flow tube vibration frequency estimated numerically and experimentally are reported in Fig. 12 for several gap velocity values. Experimental and numerical results are compared and the expected trend is retrieved. Tube frequency numerical estimate is reasonable. This tends to show that the computational process involved in the present work enables the numerical prediction of flexible tube behaviour in cross-flows.

These first computations provide a first validation of the computational process applied in this article to the numerical prediction of a flexible tube vibrations in a tube bundle in cross-flow.

Results are reported in Table 6 for several gap velocity values. Finally, one can deduce from previous calculations fluid-elastic coefficients by using appropriate data processing.

These computations tend to show that a computational process for the numerical prediction of flexible tube vibrations in cross-flows is now reachable. Required CPU time with VPP5000 is 16/100 000 s per iteration per cell per processor with about 50 000 iterations without ALE until fluid calculation convergence and 15 000 iterations with ALE.

5. Conclusion

A flow-induced vibration prediction numerical method is presented in this paper. The fluid problem is solved by using an arbitrary Lagrangian Eulerian (ALE) formulation and a coupling process between fluid and structure computations is involved in order to account for flow structure coupling and fluid-elastic effects. Finally, the approach is applied to the numerical prediction of flexible tube bundle vibration frequency in cross-flows. Numerical results are consistent with experimental predictions and feature the expected tendency.

In the present work, small flow velocities are involved and no instability development is observed. Other simulations will be performed in order to study the tube bundle behaviour below, near and above the critical flow velocity.

This will require a new validation of turbulence modelling in tube bundles. Further developments will be carried out in order to improve the coupling process and the flow modelling in presence of moving boundaries.

Acknowledgements

The authors want to acknowledge Messrs Caruso, Adobes, Fournier, Archambeau and Benhamadouche for their helpful contribution in the present work.

References

- Adobes, A., Gosse, A., Baratte, C., 2001. Qualification of motion dependent fluid forces coefficients. ASME PVP Conference, Atlanta.
- Axisa, F., Antunes, J., Villard, B., 1990. Random excitation of heat exchanger tubes by cross-flow. *Journal of Fluids and Structures* 4, 321–342.
- Belyschko, T., Flanagan, D.F., Kennedy, J.M., 1982. Finite element method with user-controlled meshes for fluid–structure interactions. *Computational Methods in Applied Mechanics and Engineering* 33, 689–723.
- Bendjeddou, Z., Longatte, E., Souli, M., 2002. Numerical simulation of tube bundle vibrations in cross flows. ASME PVP Conference on Flow-Induced Vibrations, Vancouver.
- Benhamadouche, S., Laurence, D., 2002. LES, coarse LES and transient RANS comparisons on the flow across a tube bundle. In: *Engineering Turbulence Modeling and Measurements Conference*, Vol. 5. Elsevier, Amsterdam.

- Benson, D., 1989. An efficient accurate, simple ale method for non linear finite element programs. *Computational Methods in Applied Mechanics and Engineering* 33, 689–723.
- Chen, S.S., 1986. *Flow-induced Vibration of Circular Cylindrical Structures*. Hemisphere Publishing Corporation, Washington.
- Corcos, G.M., 1963. Resolution of pressure in turbulence. *Journal of Acoustic Society of America* 35, 192–199.
- Gagnon, J.O., Paidoussis, M.P., 1994. Fluid coupling characteristic and vibration of cylinder clusters in axial flow. Part 1: Theory. *Journal of Fluids and Structures* 8, 257–291.
- Granger, S., Campistron, R., Leuret, J., 1993. Motion-dependent excitation mechanisms in square in-line tube bundle subject to water cross-flow: an experimental modal analysis. *Journal of Fluids and Structures* 7, 521–550.
- Granger, S., Paidoussis, N., 1995. An improvement to the quasi-steady model with application to cross-flow-induced vibration of tube arrays. *Journal of Fluid Mechanics* 320, 163–184.
- Granger, S., Gay, N., 1996. An unsteady semi-analytical model for cross-flow induced vibration of the tube bundle: comparison with experiment. In: Bearman, P.W. (Ed.), *Flow Induced Vibration*. Balkema, Rotterdam, pp. 327–338.
- Granger, S., Perotin, L., 1997a. An inverse method for the identification of a distributed random excitation acting on a vibrating structure: Part 1: Theory. ASME Conference on Fluid–Structure Interactions. Dallas, TX, USA.
- Granger, S., Perotin, L., 1997b. An inverse method for the identification of a distributed random excitation acting on a vibrating structure: Part 2: Flow-induced vibration application. ASME Conference on Fluid–Structure Interactions. Dallas, TX, USA.
- Hermann, G., Steindorf, J., 1999. Efficient partitioned procedures for the computation of fluid–structure interaction on parallel computers. In: *Developments in Computational Mechanics High Performance Computing*. Civil-Comp Press, Edinburgh, pp. 127–136.
- Hirt, C.W., Amsdern, A.A., Cook, H.K., 1974. An arbitrary Eulerian Lagrangian method for all flow speeds. *Journal of Computational Physics* 14, 229–253.
- Hughes, T.J.R., Liu, W.K., Zimmerman, T.K., 1981. Lagrangian eulerian finite elements formulation for viscous flows. *Journal of Computational Methods in Applied Mechanics and Engineering* 21, 329–349.
- Liu, W.K., Huerta, A., 1988. Viscous flows for large free surface motion. *Journal of Computational Methods in Applied Mechanics and Engineering* 69, 277–324.
- Longatte, E., Nhili, R., Weiss, T., 2000. Prediction of flow induced vibrations of drive line assembly. ASME PVP Conference, Seattle, WA, USA.
- Longatte, E., Laurence, D., Barré, F., Ledac, P., 2001. Application of large Eddy simulation to a flow-induced vibration problem. ASME PVP Conference on Flow-Induced Vibrations, Atlanta, GA, USA.
- Longatte, E., Bendjeddou, Z., Souli, M., 2002. Numerical simulation of tube bundle vibrations in cross flows. ASME IMECE Conference, New Orleans, LA, USA.
- Moreno, M.M., De Langre, E., Le Quere, P., 2000. A large eddy simulation of the turbulent forcing spectrum induced by axial flow on a rod. Seventh International Conference on Flow-Induced Vibrations, Lucerne, Switzerland.
- Noh, W.F., 1964. A time dependent two space dimensional coupled Eulerian Lagrangian code. In: Fenbarch, A.E., Rosenberg (Eds.), *Methods for Computational Physics* 3, 117–179.
- Pettigrew, M.J., Taylor, C.E., 2002a. Vibration analysis of steam generators and heat exchangers: an overview—Part I: Flow, damping, fluid-elastic instability, ASME IMECE, Conference on Flow-Induced Vibrations, New Orleans, LA, USA.
- Pettigrew, M.J., Taylor, C.E., 2002b. Vibration analysis of steam generators and heat exchangers: an overview—Part II: Vibration, response, fretting-wear, guidelines, ASME IMECE, Conference on Flow-Induced Vibrations, New Orleans, LA, USA.
- Piperno, S., Farhat, C., 2001. Partitioned procedures for the transient solution of coupled aeroelastic problems, part II: Energy transfer analysis and three-dimensional applications. *Journal of Fluids and Structures* 3, 61–82.
- Price, S.J., Paidoussis, M.P., 1986. A single-flexible-cylinder analysis for the fluidelastic instability of an array of flexible cylinders in cross-flow. *Journal of Sound and Vibrations* 108, 193–199.
- Price, S.J., Paidoussis, N., 1989. The flow-induced response of a single flexible cylinder in an in-line array of rigid cylinders. *Journal of Fluids and Structures* 3, 61–82.
- Souli, M., 2000. ALE and fluid structure interaction capabilities in LS-Dyna, ASME PVP Conference, Seattle.
- Souli, M., 2001. Euler–Lagrange Formulation for fluid structure interactions problems. ASME PVP Conference, Atlanta, GA, USA.
- Souli, M., Zolesio, J.P., 2001. Arbitrary Lagrangian-Eulerian and free surface methods in fluid mechanics, *Journal of Computational Methods in Applied Mechanics and Engineering* 191, 451–466.
- Souli, M., Ouahsine, A., Lewin, L., 1999. Ale formulation for fluid–structure interaction problems. *Journal of Computational Methods in Applied Mechanics and Engineering* 190, 659–675.