Advanced numerical methods for uncertainty reduction when predicting heat exchanger dynamic stability limits. Review and perspectives.

Elisabeth Longatte, Franck Baj, Yannick Hoarau, Marianna Braza, Christian Canteneur, Daniel Ruiz

Advanced numerical methods for uncertainty reduction when predicting heat exchanger dynamic stability limits: Review and perspectives

E. Longatte [a,*], F. Baj [a], Y. Hoarau [b], M. Braza [c], D. Ruiz [d], C. Canteneur [e]

[a] LaMSID, UMR EDF-CNRS-CEA 2832, Computational Mechanics Group, 92141 Clamart, France
[b] IMFS, CNRS-UDS, Instabilité Turbulence Diphasique Group, 67000 Strasbourg, France
[c] IMFT, UMR 5502, Single-phase Transitional Turbulent flow Department, 31400 Toulouse, France
[d] IRIT, Computing Department, 31062 Toulouse, France
[e] AREVA, Steam Generator Department, 92400 Courbevoie, France

HIGHLIGHTS

- Proposal of hybrid computational methods for investigating dynamical system stability.
- Modeling turbulence disequilibrium due to interaction with moving solid boundaries.
- Providing computational procedure for large size system solution approximation through model reduction.

ABSTRACT

This article proposes a review of recent and current developments in the modeling and advanced numer-ical methods used to simulate large-size systems involving multi-physics in the field of mechanics. It addresses the complex issue of stability analysis of dynamical systems submitted to external turbulent flows and aims to establish accurate stability maps applicable to heat exchanger design. The purpose is to provide dimensionless stability limit modeling that is suitable for a variety of configurations and is as accu-rate as possible in spite of the large scale of the systems to be considered. The challenge lies in predicting local effects that may impact global systems. A combination of several strategies that are suited concur-rently to multi-physics, multi-scale and large-size system computation is therefore required. Based on empirical concepts, the heuristic models currently used in the framework of standard stability analysis suffer from a lack of predictive capabilities. On the other hand, numerical approaches based on fully-coupled fluid–solid dynamics system computation remain expensive due to the multi-physics patterns of physics and the large number of degrees of freedom involved. In this context, since experimentation cannot be achieved and numerical simulation is unavoidable but prohibitive, a hybrid strategy is proposed in order to take advantage of both numerical local solutions and empirical global solutions.

1. Introduction

This article addresses one of the major challenges related to increasing the reliability of nuclear power plant safety barriers. Under operating conditions, some critical components in Pressurized Water Reactors (PWR), such as heat exchangers, are submitted to complex flows making them vibrate and subsequently causing possible damages, inducing wear or vibratory fatigue (Fig. 1). Weaver (2008) recently addressed about fluid-elastic instability in cylinder arrangements: *in spite of more than 40 years of research, this mechanism is still not fully understood*. Moreover, as depicted in Fig. 1, fluid-elastic instability resulting from a Motion-Induced Vibration (MIV) process may be combined with Turbulence-Induced Vibration (TIV), Vortex-Induced Vibration (VIV) and Fluid Structure Interaction (FSI). Numerical simulations of these combined effects are required to better understand MIV. All developments proposed in this work contribute to this major evolution.

This article aims to address the High Performance Computing (HPC) of large scale multi-physics, multi-scale systems by introducing a new generation of algorithms. Our approach will take the two following issues into account: (1) firstly the fact that, with current capabilities, it is impossible to use the available multi-physics coupling platforms incorporated into efficient environments in a large, real-scale setting, and that will remain the case for the next twenty years, unless there is a major development in scientific computing algorithms and simulation resources; and (2) secondly, the fact

* Corresponding author. Tel.: +33 1 30 87 80 87; fax: +33 1 30 87 77 27.
E-mail address: elisabeth.longatte@edf.fr (E. Longatte).

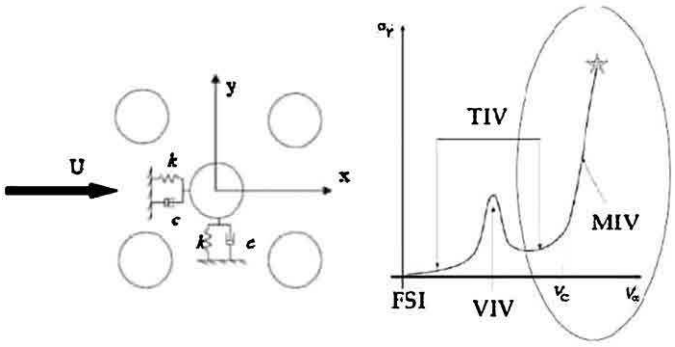

Fig. 1. Motion-Induced Vibration (MIV) leading to fluid-elastic instability below a critical reduced velocity threshold, combined with a sudden linear increase in cylinder vibration magnitude, stopped through non-linear effects like collision or breakdown which must be controlled for safety purposes. Possible combination with Turbulence-Induced Vibration (TIV), Vortex-Induced Vibration (VIV) and Fluid Structure Interaction (FSI) in cylinder arrangement submitted to cross flows.

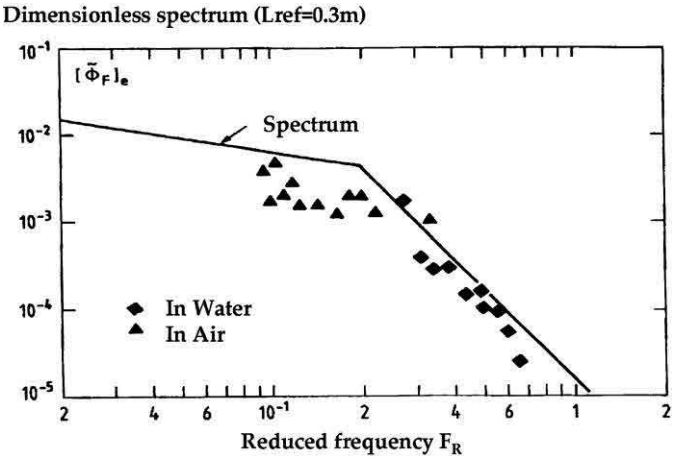

Fig. 3. Turbulent stress model deduced from empirical and experimental data enrichment method for cylinder arrangement under cross-flow.

that there is a lack of predictive capabilities in current pseudo-static models involved in the field of multi-physics, especially for stability analysis of dynamical systems. These currently tend to rely on small perturbation development and on Theodorsen (1935) theory through Dirac's response superposition methods and suffer from a lack of reliability for the design of industrial systems which require high-fidelity predictive modeling to be kept under safe operating conditions. An optimization of existing software platforms is proposed, making them efficient for very large number of degrees of freedom, through parallelization, model reduction and hybridization also useful in the framework of validation and systematic parameter-dependency analysis.

The challenge lies in defining a systematic procedure that minimizes errors on stability limit estimates and enables us to forecast whether or not, for a given configuration, the safety barrier is reliable. The aim is to develop currently missing algorithms and computational processes that may be applicable, regardless of the component type, age, cycle, operating conditions, conception or design, in order to undertake risk assessment. From an economic point of view, expected gains may be significant if one considers the potential impact in terms of improved power plant availability, greater efficiency of maintenance and controls and a possible decrease in radiation emissions thanks to the increased reliability of safety barrier controls. Figs. 2 and 3 feature examples of data, correlations and turbulent load spectra, that are currently used in the framework of the standard small perturbation development procedures involved.

These charts result from a combination of empirical and experimental solutions, enabling the interpolation of stability thresholds

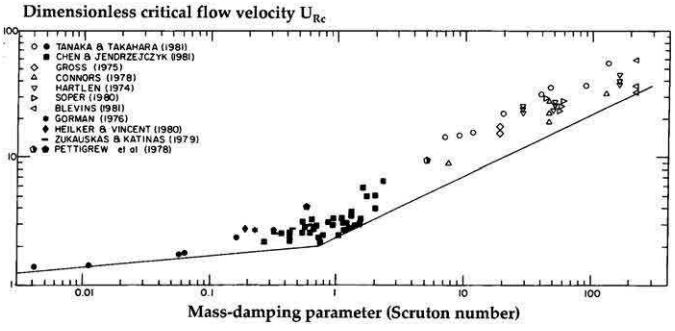

Fig. 2. Fluid-elastic instability map deduced from empirical and experimental data enrichment method for cylinder arrangement under cross-flow.

in real operating conditions. They therefore involve a degree of dispersion and may suffer from a lack of predictive capability and accuracy. In the framework of this work, these solutions may provide useful reference information to validating numerical solutions and enriching them for use in real scale settings. New stability maps of these systems will be established with great accuracy thanks to numerical simulations and HPC. Parameter sensitivity analysis will make it possible to cover a large range of parameters corresponding to real operating conditions. This will contribute to a real gain in terms of the accuracy of knowledge regarding safety barrier reliability.

The article addresses vibration risk assessment in heat exchanger design and it challenges major issues in the domains of turbulence modeling, scientific computing and HPC to enable numerical simulation of large-scale system dynamical behavior in order to improve their reliability and prevent instabilities. The objective is to enable numerical simulation of large, complex systems involving multi-physics in the field of fluid and solid mechanics. The industrial applications are significant. Beyond the context of tube arrays in heat exchangers, the proposed methodology is applicable to other systems such as electric pylons, nuclear submarine exchangers, oil platforms and risers, as well as fluid-elasticity in aeronautics and bridge design.

2. Physical framework

2.1. Stability analysis

As far as vibrations of cylinder arrangements under cross flows are concerned, a complete review describing the strengths and weaknesses of standard modeling is proposed in Weaver (2008). Possible dynamic fluid-elastic instability was identified about forty years ago. Many researchers tried to understand the reasons why the modal characteristics of cylinders inserted into bundles are modified as they are submitted to cross-flows. The fluid-elastic instability linked to a loss of damping of the coupled system has been especially investigated because of the damage it can generate (Fig. 1). Several semi-empirical models have been proposed by Connors (1970), Blevins (1990), Price and Padoussis (1984), Lever and Weaver (1986), Granger and Padoussis (1996), Adobes and Gaudin (2004), Adobes et al. (2006) relying on small perturbation development methods combined with empirical data enrichment. Each of these models takes into account a time delay between the solid response signal and the strain exerted by fluids on moving boundaries. When the sign of the phase lag changes,

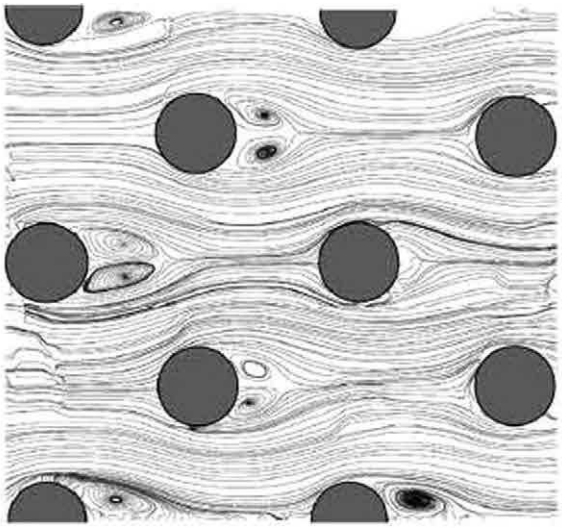

Fig. 4. Example of streamlines in cylinder array for Re = 9300.

the system becomes unstable. The challenge is to model this phase lag.

2.2. Turbulence modeling

From a physical point of view, the turbulence modeling challenge is to model the new generation stress–strain issued bye the flow-structure physics by the combined interaction of turbulence and body motion. Flow-induced vibration in tube bundles has been studied over the thirty last years. Many theoretical, experimental and numerical research programs have been undertaken. Mathematical modeling has been proposed by using several classes of useful assumptions and the conventional effects of fluid structure interactions have been characterized. Main fluid and flow effects have been formulated, either in terms of inertia, damping and stiffness or external turbulence stresses. In most studies performed so far, both effects, motion-dependent and motion-independent actions exerted by fluid, are modeled separately. A complete review of standard modeling is proposed in Weaver (2008). One particular aspect of moving cylinder arrangements is that the complexity of the flow regime map depends on the combination of the numerous hydraulics, geometric and mechanical parameters involved (Chen, 1987, Fig. 4). In many configurations, all cylinders shed Von Kármán vortices. A jet swing associated with vortex shedding is also possible. The free shear layer of a front cylinder can attach to the downstream cylinder and thus Von Kármán vortices cannot develop. Jet deflection may also occur. Vortex streets can be the same as those shed by isolated cylinders when the confinement decreases. The flow pattern is the same within a range of cylinder pitches and an abrupt change can be observed at the boundary of the domains separating the flow regimes. Many studies have been performed to identify the borderlines between flow regimes. However identifying these borderlines remains a complex subject as they depend on parameters and on their possible combinations like Reynolds and Stokes numbers. Turbulent modeling that is suitable for certain parameter ranges may, however, be inadequate for other close sets of parameters. In this context, hybrid modeling for turbulence and its interaction with moving solid boundaries is required.

Many visualization measurement results are available (Price and Padoussis, 1984; Chen, 1987; Yetisir and Weaver, 1993; Simonin and Barcouda, 1986) and numerical simulation can enable thorough study and classification of flow patterns depending on all dimensionless parameters. Other programs are required to focus on the application of Computational Fluid Dynamics (CFD) to identifying flow regimes. A review of CFD simulations performed in cylinder arrangements is proposed by Afgan et al. (2007). Large Eddy Simulation (LES) is performed by Rollet-Miet et al. (1999), Benhamadouche and Laurence (2003) to provide a representation of flow patterns in fixed, staggered tube arrays. Spectrum identification is proposed by Liang and Papadakis (2007). Comparisons between loading spectra identified numerically and experimentally are carried out by Moreno et al. (2000) for configurations involving single cylinders submitted to axial flows, by decoupling fluid force effects and solid motion dynamics.

The issue of evaluating local loads spectrum through numerical simulation by accounting for all coupling effects is investigated hereafter. The presence of fixed and moving wall affects flow patterns, especially turbulence that would be homogeneous isotropic without walls. Several effects have to be taken into account: (1) change of shear velocity, (2) change of near-wall viscosity, (3) breakpoint effect due to obstacles and the possible effects of confinement. Turbulence is a three-dimensional phenomenon featuring complex and irregular dissipative behavior. In turbulent flows the fluid velocity and pressure vary significantly and irregularly both in time and space. Therefore, from a numerical point of view, the most advanced CFD turbulence modeling approaches (statistical and hybrid ones) for unsteady flows are required to suitably capture unsteady loads in a fluid–structure interaction kernel.

Enormous effort has driven the development, verification and validation of Reynolds-Averaged Navier–Stokes (RANS) methods and the inherent turbulence models for a multitude of flow problems. The turbulence models under consideration are even applied to Unsteady Reynolds-Averaged Navier–Stokes (URANS) computations in cases where the flow is either forced to be unsteady because of body movement or where large separated flow areas naturally lead to large-scale unsteady flow behavior. In the context of advanced URANS, aiming at combining the robustness of this approach and its improvement in capturing non-equilibrium turbulence and in becoming less dissipative, the URANS Organized Eddy Simulation (URANS/OES) approach has been developed and tested for industrial use (Dervieux et al., 1998; Braza, 2000; Bouhadji et al., 2002; Braza et al., 2006). This approach distinguishes between the fluid structures to be resolved and those to be modeled using the criterion of the physical nature of the structures, organized or chaotic, and not their size, as in the case of LES. This provides robust and reliable prediction for the majority of unstable modes, as they occur in flutter instability and dynamic stall, in buffeting or when massive separation is taking place.

To combine the robustness of URANS (including advanced URANS) and LES benefits, a new class of hybrid turbulence modeling approaches has been developed and examined for industrial use. It is well known that the dominant, detached eddies are highly geometry-specific and have not much in common with the "standard" eddies of the thin shear flows that RANS models have been designed to model. In particular, the RANS modeling theories start off with a "local homogeneity" hypothesis. This is clearly not the case of turbulence interacting with vibration sources and with flutter instabilities. As a direct consequence, performing Reynolds averaging over the entire spectrum of the turbulent eddies, and trying to include those geometry-sensitive vortices that are typical in separated flows is still on the "wish-list" of industry. Of course, modeling in URANS has to be pursued by means of advanced URANS to capture non-equilibrium turbulence, but it has its counterpart in filtering approach used in LES, the latter assumed as being the only defensible tool that has a real promise for capturing higher-frequency vibrations, although very costly for industrial use.

A considerably high effort has been put into LES investigations thus far – and even on a more fundamental basis, namely

on Direct Numerical Simulations (DNS), but the resolution needed in boundary layers and wakes is making LES unaffordable in an industrial context. The recognition of the relative benefits between RANS/URANS and LES made it very tempting to create an approach that combines fine-tuned RANS technology in attached boundary layers with the power of LES in the separated regions. A general idea of such an approach was initially introduced by Spalart and Allmaras (1994) in interaction with Speziale et al. (1991). Finally, the so-called "detached" eddies provided the name for the Detached Eddy Simulation (DES, Spalart et al., 1997) initially based on a specific formulation of the Spalart and Allmaras (1994) turbulence model.

In the last decade, DES has provided some impressive results for complex aerodynamic applications. Unfortunately, standard DES (Spalart et al., 1997; Travin et al., 2000) introduces a significant grid dependency into the RANS part of the simulations, which requires grid spacing for the wall grid in both normal and tangential directions that is larger than the boundary layer thickness at that wall location. For this reason, Delayed DES (DDES) has been developed (Strelets, 2001), as well as the DES/OES (Bourguet et al., 2007a,b, 2008, 2009; Barbut et al., 2007, 2009, 2010; Haase et al., 2009). In the latter, the RANS part of DES has been reconsidered to reinforce the near-wall turbulence-stress anisotropy and modify the turbulence length and time-scales involved in capturing the non-linear interaction between organized and chaotic motion. These achievements are very promising for efficient prediction of flutter modes as well as for capturing the modification of the turbulence spectrum subjected to multiple sources of vibrations. It is noticeable that the DES approach does not need any interface for changing from URANS towards the LES region. This is achieved inherently by choosing the turbulence length scale between the URANS and the LES one. An efficient modification of the turbulence length scale is achieved by the Scale Adaptive Simulation (SAS, Menter et al., 2003) providing promising results for capturing non-equilibrium turbulence physics. Other hybrid RANS/LES approaches need to specifically define the interface, by using, among other techniques, synthetic turbulence (Sergent, 2002).

The progress beyond the state of the art in terms of CFD going to be proposed here is clearly based on the application of a limited number of the most efficient turbulence modeling methods: (1) advanced URANS and (2) hybrid turbulence modeling, especially DES, in the context of tube arrays. More precisely, (1) in URANS efficient and well-adapted Algebraic-Stress Modeling (ASM), as mostly appropriate to capture near-wall unsteadiness and non-equilibrium turbulence, and the tensorial eddy-viscosity concept in the present industry-oriented environment, to achieve more accurate prediction of turbulence-stress anisotropy for the unsteady loads in the near-wall region, and (2) in hybrid turbulence modeling: efficient DES approach that include improved statistical closures near the wall (the Anisotropic Organized Eddy Simulation (A/OES) and improved LES approaches (Benhamadouche and Laurence, 2003)). These improvements will ensure a high robustness in the DES, especially when switching from the statistical to the LES areas.

Most simulations concerning fluid-elastic instabilities use the LES approach for the fluid part, which is still limited to low Reynolds number ranges, as well as standard URANS approaches that are characterized by a high level of turbulence modeling diffusion rate which attenuates fluid-elastic instabilities (Barbut et al., 2007, 2009, 2010). It is therefore necessary to use improved URANS approaches that take into account the modification of the turbulence scales related to non-equilibrium turbulence (Bourguet et al., 2008; Barbut et al., 2009), as well as suitable hybrid turbulence modeling, especially using the DES approach (Travin et al., 2000; Spalart, 2000; Haase et al., 2009; Braza et al., 2010; Bourguet et al., 2008).

2.3. Interaction with solids

As far as fully-coupled fluid solid system computation is concerned, there are usually two different strategies: direct or iterative methods. Integration scheme stability, robustness, accuracy and performance issues must be considered and algorithm properties must be optimized in a way depending on the physics and associated mathematical modeling to be considered.

Direct procedures imply that the fluid and solid equation systems must be solved at the same time and in the same way by using interface-compatible formulations, discretization operators and numerical methods. To ensure interfacial compatibility of both sets of equations, direct methods may be either deduced directly from interface-compatible formulations or built by introducing interface-compatible numerical methods. Homogenization of added mass operators proposed by Planchard et al. (1994) results from an interface-compatible formulation of fluid and solid dynamics equations obtained by using a space averaging operator. Chandesris (2007) also considers space averaging methods to deal with porous media, investigating scale changing and scale interaction to formulate homogenized Navier–Stokes equations governing turbulent incompressible flows of fluid in a confined or unconfined domain without any energy exchange with solids. This direct porous method features the major advantage of enabling large-scale computation, but it is not compatible with local, small-scale analysis. Other numerical methods have therefore been developed to address fully-coupled fluid solid systems.

In the framework of finite element strategies, Morand and Ohayon (1995) proposes a coupled fluid–solid finite element method relying on a Ritz Galerkin projection and a finite element computation of the system formulated in the fluid domain, the solid domain and their interface. The formulation to be obtained is efficient in a linear context, especially for small perturbations of the system around an equilibrium state, without mean flow and with restrictive assumptions in the presence of solid large displacement motion, for example for reservoir balloting mode identification. In a finite volume context, a fully-coupled fluid finite volume–solid finite volume method is proposed by Papadakis (2008), introducing a pressure velocity formulation for solid computation. But turbulence modeling is not investigated in this formulation. In a finite element framework, a fully-coupled formulation is also proposed by Hachem et al. (2010) but system conditioning remains a major difficulty.

As an alternative in the framework of fully non-linear system computation, iterative methods have been extensively developed over the past decade, especially for computation in the field of fluid–structure interaction (Piperno and Farhat, 2001; Farhat and Lesoinne, 1997; Le Tallec et al., 2005; Schaefer et al., 2006; Huvelin, 2008; Longatte et al., 2009; Baj, 2002). Several algorithms have been proposed to enhance the stability and convergence properties of algorithms. For instance, an explicit, partitioned strategy can be made more robust and less restrictive in terms of time-step limitations by making it more implicit via a predictor-corrector iteration method. Another possibility is to combine some global direct solvers with an iterative procedure, acting as a smoother or a pre-conditioner. A flowchart describing the iteration loop involved in a fixed point method for coupled fluid–solid system computation is presented in Fig. 5. The coupling scheme is very sensitive in relation to the deformations especially in the initial FSI loop iterations. Situations that are far away from the physical equilibrium can arise, which may lead to instabilities or even divergence of the loop. In order to counteract this effect, adaptive under-relaxation may be employed. However these procedures are not unconditionally stable and under-relaxation may be insufficient to ensure solution stability and convergence. Hybrid direct iterative methods can therefore also be considered, either by splitting computational

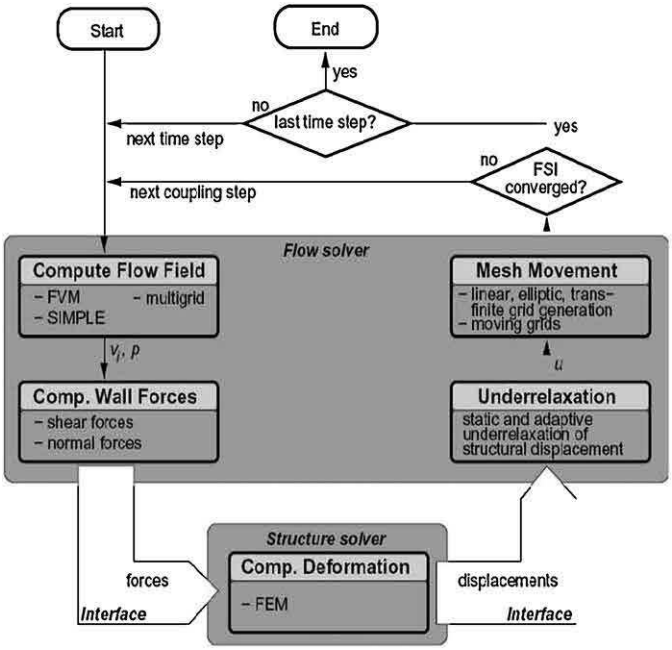

Fig. 5. Flow chart of iterative computation using a fixed point method.

domains or by isolating the coupling terms responsible for numerical stability. Developments of hybrid methods for time marching schemes as well as interfacial space discretization are proposed.

2.4. Dynamic instability limit

As far as correlation and stability threshold evaluation is concerned, the recent work of LAMSID contributes to the numerical identification of fluid-elastic instability analysis in configurations involving cylinder arrays under cross single-phase laminar flows (Fig. 6). The purpose is to establish new instability maps through numerical simulations and parametric sensitivity analysis is made possible thanks to an improved HPC environment, enabling real-scale challenge.

At each stage of development, validation of algorithms is proposed by using academic configurations. This work uses available experimental data to generate data bases for the validation of numerical solutions. In many cases, complementary data are required and new experimental programs are required to achieve validation. This is an efficient means of parametric study which allows the realization and analysis of the unstable regime birth,

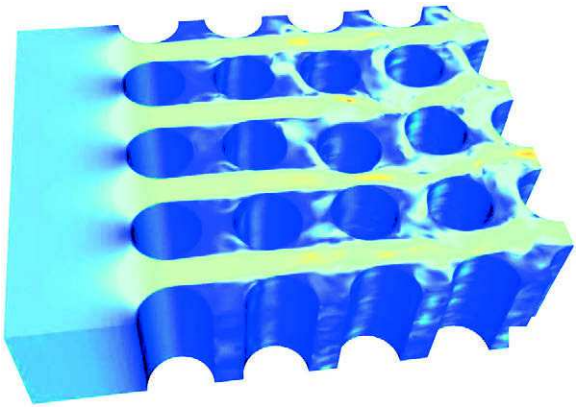

Fig. 6. Numerical simulation of cylinder arrangement vibration under sub-critical cross flows (Huvelin, 2008).

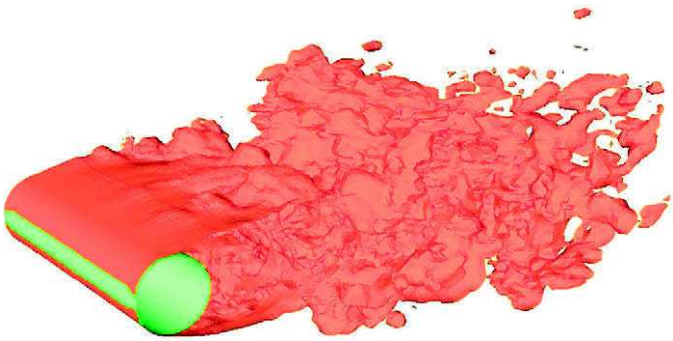

Fig. 7. Numerical simulation of single cylinder vibration under sub-critical cross flows (Jus, 2011).

beyond the critical couple of reduced velocity and Scruton number. This concept was initially suggested for VIV studies by Hover et al. (1997) as shown in Jus (2011) dealing numerically with single cylinders under sub-critical cross flows (Fig. 7) (Pomarede et al., 2010). It should be remembered that from an experimental point of view, in direct methods (Tanaka and Takahara, 1981) the structure motion is imposed (forced motion) and load is measured. As mentioned by Caillaud (1999), this approach was not widely used for tube bundles because of the dynamic response limit of the actuators. In indirect methods, vibration response is measured to deduce load, given the structural parameters. Thereafter a hybrid methodology is proposed.

Finally, this work aims to combine numerical simulation solutions to create a new formulation of stability maps for tube arrays in real-scale geometry. Up to now, semi-empirical and heuristic models have been used (Connors, 1970; Blevins, 1990; Price and Padoussis, 1984; Lever and Weaver, 1986) and have not taken the afore-mentioned effects of the non-linear interaction between turbulence and body motion into account accurately. Thanks to this work, it is estimated that the design efficiency will benefit greatly by using numerical uncertainties and approximation errors of solutions, providing bifurcation threshold probability in case of singularities.

3. Computing framework

3.1. Modal reduction

Reduced Order Model (ROM) development allows fast and reliable FSI models to be built in order to design cylindrical arrangements. It consists of a Galerkin projection of the Navier–Stokes complete system and the URANS/DES system of equations onto a basis of orthogonal modes derived by a Proper Orthogonal Decomposition (POD). The most energetic instability modes responsible of the flutter phenomenon can then be taken into account. The success and predictive ability of the ROM is based on the "richness" of the POD basis. For this reason, to cover the objectives of this work, POD modes are derived: (1) from DNS (at low Reynolds numbers), (2) from LES (at intermediate Reynolds numbers) and (3) from advanced URANS (OES) and from hybrid approaches (DES), the two last standing for high Reynolds regimes. The ROM produced can therefore cover different critical regimes, in turn covering the principal mechanisms for the occurrence of the hydro-elastic flutter. Furthermore, the reliable ROM created can be used as an outcome to perform optimum design of what have up to now been cylindrical bundles. The ROM is elaborated here for the full fluid–solid system computation, the principal unstable mode in FSI for a single-degree-of-freedom system being the coupled FSI vibration. The advantage of using POD for the Galerkin projection in this ROM elaboration is that POD considerably reduces the number of degrees

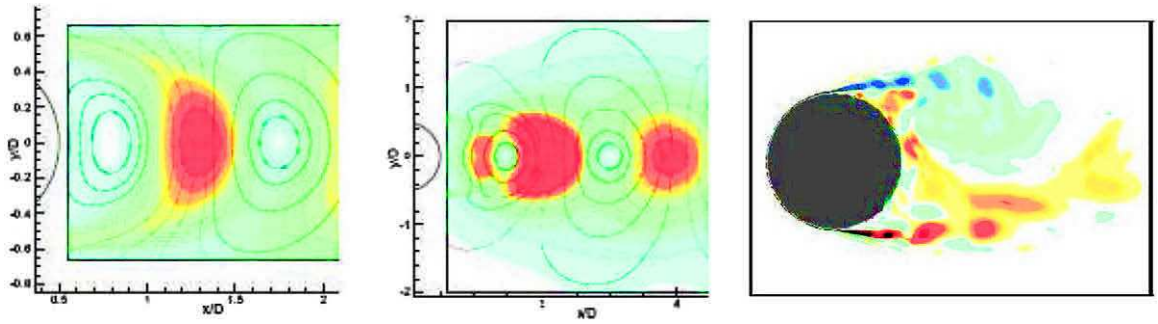

Fig. 8. Mode identification by using ROM technique. Second POD mode by TRPIV (left). Second POD mode by DES (middle). POD reconstruction with 49 modes, von-Kármán and Kelvin–Helmholtz eddies by DES/OES (right, Haase et al., 2009).

of freedom concerning the fluid motion, as mentioned by Dowell and Hall (2001). This dictates the choice of "POD–Galerkin projection".

The low-order modeling provides a comprehensive approximation of the physical model and leads to modeling of unsteady phenomena and at the same time as preserving a high level of physical relevance. The class of physics-driven methods based on the projection of the high-order model onto a reduced basis is particularly promising in the perspective of surrogate optimal design. In addition to data-driven approaches such as polynomial interpolations which are built on a database generated by resolution of the complex model, these methodologies enable parametric analysis. Indeed, the underlying high-order model enables changes in state system parameters to be introduced, such as the flow incidence, the Mach number or the shape design. Many base functions can be considered. The POD (Berkooz et al., 1993), also known as Karhunen–Loeve expansion of the flow variables allows to extract the main fluid energetic properties (Fig. 8).

The low-order models, based on a Galerkin projection of the Navier–Stokes system onto a POD basis consist of Ordinary Differential Equation (ODE) systems of considerably reduced dimensions. Many studies achieved accurate model reduction based on the incompressible Navier–Stokes system, in 2D, in the laminar regime, in the transitional regime and, in the 3D laminar case (Ma and Karniadakis, 2002a,b; Buffoni et al., 2006) with databases issued from DNS. The 3D transitional case was studied on the basis of LES, for example, on a backward-facing step flow (Couplet et al., 2005). POD–Galerkin models present instability properties induced by POD mode truncation, which leads to a lack of dissipation. Stabilization procedures have been developed, ranging from the addition of a global, artificial viscosity term to the introduction of optimal calibration coefficients, which implies a perfect prediction of the high-order reference dynamics (Couplet et al., 2005). Such POD–Galerkin ROMs were integrated in optimal control loops, for example, to control the laminar wake of an oscillating cylinder (Graham et al., 1999). Extensions were developed to increase the robustness of the empirical basis in relation to changes in the flow configuration (Arian et al., 2000; Bui-Thanh et al., 2003) and to adapt the POD basis to domain deformations with a view to design optimization. The addition of other types of modes, such as global instability modes (Noack et al., 2003) in the reduced basis seems to be a promising enrichment approach. A valid ROM from very low (incompressible) to high Mach number flows was derived on the basis of an isentropic scalar product (Rowley et al., 2004). A simple variable change in the high-order system enables a quadratic polynomial low-order model to be built without flow configuration assumptions (Vigo et al., 1998; Vigo, 2000; Bourguet et al., 2007a,b). Progress beyond the state-of-the-art for ROM is therefore aligned with the afore-mentioned models, because they are proven to predict the complex unsteadiness induced by global instability effects at a very moderate numerical cost compared with high-order models (Bourguet et al., 2007a,b), as displayed in Fig. 8.

3.2. High Performance Computing (HPC)

Numerical simulation of fluid-elastic instabilities using coupled systems combining Navier–Stokes equations with their different variants: URANS, LES and DES, together with vibration structure analysis has been widely developed in the last ten years, thanks to the increased efficiency of HPC (Ribes and Caremoli, 2007; Ribes, 2008). However, the non-linearity of the physical process and the complexity of turbulence transfers in the FSI need a high number of degrees of freedom to accurately predict the unsteady dynamic load on the cylinder barrier. The efficient prediction of fluid-elastic instabilities needs efficient MPI algorithms and numerical optimization, especially in the framework of FSI coupling strategies (Farhat and Lesoinne, 1997; Piperno and Farhat, 2001; Le Tallec et al., 2005; Schaefer et al., 2006; Longatte et al., 2003a,b, 2009). Regarding turbulence modeling people have tried to solve turbulence in theory for centuries. However, due to the complexity of non-linear effects in turbulence, it is very difficult to understand turbulence in theory, especially for non-equilibrium. People have therefore become more and more interested in solving it through computer simulation. Gottlieb and Orszag (1977) performed the first DNS on a 323 cell mesh at NCAR on a CDC7600 computer with only 50 MB of memory. DNS of turbulence using a Fourier Spectral Method on the Earth Simulator (5.120 vectorial processors) has been reported on SC2002 with a 20.483 cell mesh (Yokokawa et al., 2002). Wylie et al. (2007) demonstrated the linear scalability of the Navier–Stokes finite element solver XEN up to 4.096 BlueGene/P CPUs. Wolf et al. (2009) computed LES of reactive flow on a 93M point grid on 4096 BlueGene/P CPUs. Gotz et al. (2010) reported simulation of particle laden flows, a large scale coupled fluid structure interaction with up to 37 million geometrically-modeled moving objects incorporated in the flow on 8.192 processors. The present computational strategy proposal aims to reach a scaling gain factor of about 100 by using systems involving 1 billion degrees of freedom to enable several reference simulations, ROMs and parametric sensitivity analysis.

3.3. Open source software coupling platform distribution

For multi-physics computations, an open-source software platform[1] for numerical simulation is involved. It is based on an open and flexible architecture, made of reusable components. It

[1] Salome, www.salome-platform.org.

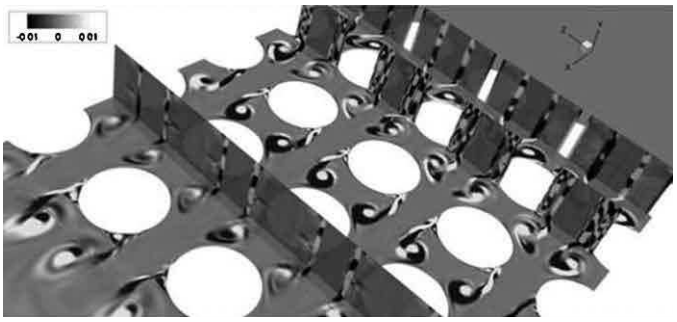

Fig. 9. Numerical simulation of flow-induced vibration of a large size cylinder arrangement for high Reynolds number range with $k-\varepsilon$ OES model (Barbut et al., 2010).

can be used as standalone application for generation of models, their preparation for numerical calculations and post-processing of calculation results. It can also be used as a platform for integration of external third-party numerical codes to produce a new application for the full life-cycle management of models (Ribes and Caremoli, 2007; Ribes, 2008; Bergeaud and Tajchman, 2007).

4. Proposal of a new hybrid modeling strategy

4.1. Modeling turbulence interacting with moving solids

The state-of-the-art in coupled Computational Fluid Dynamics and Computational Solid Mechanics (CFD–CSM) can be divided into two parts with increasing complexity. Firstly, linear or weak coupling has been widely used. This is used where static aero- or hydro-elasticity is concerned. There is no dynamic coupling for medium- and high-frequency responses. Nevertheless, the predictive abilities of this approach can be significantly improved, when using Explicit Algebraic Stress Models (EARSM) in the CFD part. Secondly, weak, non-linear schemes can be employed to model fluid–structure interaction (Farhat and Lesoinne, 1997) using iterative solvers. The non-linear equilibrium states obtained in this context allows the evaluation of free-vibration mode shapes for the structural models. These mode shapes are then transferred by geometrical coupling to the CFD volume-mesh. This method, which accuracy has already been proven, can be used in this proposal to accelerate structural-linear, small-deformation analysis such as flutter analysis.

For the CFD part, it is proposed to consider advanced URANS approaches in the context of OES modeling to enable efficient instabilities prediction by means of cost-effective grids (Fig. 9). Specific attention must be paid to so-called anisotropic OES modeling, based on a tensor eddy-viscosity concept and on projection of the URANS/DRSM modeling on specific principal directions of the strain-rate tensor. This takes the non-equilibrium turbulence phenomena, which are due to stress–strain misalignment around a lifting structure at high Reynolds number turbulent flows, efficiently into account. These developments allow regions where negative turbulence is produced to be evaluated combined with the inverse turbulence cascade and ensure a considerable improvement in the evaluation of the unsteady global coefficients and their fluctuations which are a crucial aspect of the design. These developments should enable upscale turbulence modeling in OES to be taken into account and therefore capture the transfer of energy from the intermediate wave-number ranges to the most energetic low wave-number range more effectively.

4.2. Modeling non-matching interfaces through hybrid interpolation

A large number of fluid flow problems of practical interest occur in geometrically complex domains. A common difficulty in simulating such flows is that not every geometry can be well represented using a single, contiguous, structured grid. Building a good quality single-block, structured curvilinear grid for a complex geometry is often a very challenging and time-consuming operation. Unstructured grids provide an alternative for simulating flows in or around arbitrary complex geometries and have been applied successfully to a large number of complex flows. The unstructured approach facilitates grid generation for complex geometries considerably; it is also very difficult to control the grid refinement in specific area with unstructured grids. However, it is well known that such methods can be memory and time-consuming, compared with structured grid methods. This applies especially to high-resolution spatial discretization (Wang, 2007; Mavriplis et al., 2009). Furthermore, generating a fully unstructured grid near solid boundaries for high Reynolds-number flow simulations is a difficult task (Thompson et al., 1999). An alternative to unstructured grids is to decompose the domain into a set of simpler sub-domains, each designed so that it can be easily discretized by a curvilinear block-structured grid. If these sub-domains are allowed to overlap, each geometry feature of interest can be meshed individually and the grid generation process is considerably facilitated (Benek et al., 1986; Rogers et al., 2003). An additional motivation to allow grids to overlap is to enable the simulation of bodies in relative movement (Meakin, 1993; Dougherty and Kuan, 1989; Deloze et al., 2010, 2012). This motivation applies to both structured and unstructured overlapping grids. Techniques utilizing a set of overlapping grids to discretize the solution domain are referred to as overset grid or Chimera methods. It is clear that in overlapping regions some kind of ranking for the grids has to be defined, to establish which grid is used for the calculation of the flow field. Moreover, connectivity between sub-domains has to be generated to allow the solutions from one domain to be imposed onto the others and inversely. These steps are also known as Chimera grid set up. For complex configurations, where it may consist of multiple overlapping grids, the Chimera grid set-up can become a complex and tedious task, because most of the existing set-up techniques work in an iterative (trial and error) manner and require a high degree of user input. A detailed overview and explanation of the Chimera method, restricted to structured grids, is given by Meakin (1993) for example. In addition, it should be pointed out that the early development of the Chimera method was focused mainly on structured overset grids. However, several unstructured implementations are now available (Lohner et al., 2001; Sitaraman et al., 2008).

For a cell-centred finite-volume discretization, the common Chimera algorithm can be structured into three main steps: 1. First, the status of every cell in the overlapping grids has to be declared. Normally, three different types of cells exist: (a) Hole cells, which are blanked or masked cells, that are neither updated by the discretization scheme nor interpolated. The determination of hole cells is called hole-cutting or blanking, and represents a critical element of the overset-grid assembly process. The non-trivial task of the hole-cutting procedure is to determine whether a computing cell is lying inside (hole cell) or outside (not a hole cell) of a specified region. Such regions are typically interiors of solid bodies, where cells have to be blanked in those overlapping grid blocks, where the solid body is not meshed. Many blanking methods have been proposed (Nakahashi et al., 2000). All of these methods work, but they have some restrictions such as requiring a moderate to high degree of user input, which can be difficult and/or time-consuming to provide. (b) Interpolated cells, which are interpolated from data of other overlapping grids. The interpolated cells are composed

of two groups of cells. i. Cells which are bordered by hole cells, also designated as hole fringe cells. ii. Cells adjacent to 'outer' grid boundaries, which are not described by physical (e.g. noslip wall surface) or conventional (e.g. symmetry plane, inflow/outflow) boundaries. (c) Finally, there are calculated cells, which are computed by the numerical discretization scheme of the flow solver. With the declaration of all cell types the final overlapping grid system is completely defined. 2. The second step is to find donor cells for all interpolated cells. In fact, for every interpolated cell, a cell in the dual donor grid, which contains the cell midpoint of the interpolated cell, has to be identified. For this purpose, various search algorithms, such as stencil walking or an alternating-digital-tree (ADT) based method (Bonet and Peraire, 1991) can be used. 3. In the third step, data have to be interpolated from the nodes of the dual grid to the interpolated cells. As for the donor search method, many interpolation methods exist (Chesshire and Henshaw, 1990; Sherer and Scott, 2005). The commonly-used interpolation method is a trilinear method, based on the relative position of the interpolated cell midpoint in the respective dual-grid donor cell.

4.3. Integrating coupled systems through hybrid direct iterative solvers

The purpose of building a numerical modeling of such multi-physics problems is to solve simultaneously and in a coupled manner different partial differential equations which are defined on different sub-domains. Single physics models which are the given fluid and solid models may be more or less compatible through the interface depending on the involved formulation, discretization and numerical methods. For coupled system computation, a strategy must be defined to ensure time marching as well as field transfer between models: interface kinematics and stress distribution. The approximated solution thereby provides information on fluid–solid system dynamical stability. When fluid–structure interaction is considered, we are interested in an energy exchange between fluid and solid systems. In the framework of a multi-physics approach without any multi-scale consideration, we concentrate on mechanical energy exchange, with potential, kinetic and deformation energy exchange. The energy quantity transferred from one system to the other one is directly defined by the product of the stress applied on the solid by the velocity of the interface motion. To be consistent, a fluid structure interface model must therefore satisfy three conditions through the interface: the geometric continuity of both system interfaces, the kinematic consistency and the action reaction principle corresponding to the mechanical stress continuity. Boundary conditions cannot, however, be explicitly known at the interface because fluid velocity at the moving boundary depends on solid dynamics and deformation, which in turn depend on the action exerted by fluid which is directly related to fluid velocity and pressure. Interface modeling therefore leads to the formulation of a fully, non-linear problem. The proposed method consists in investigating computation of this non-linear system through combined iterative and direct solvers.

4.4. Model reduction for coupled Navier–Stokes and elastodynamics equations

Reduced Order Modeling (ROM) can be used to simplify the complete system of equations of Computational Fluid Dynamics and Solid Mechanics CFD–CSM), by using a reduced number of degrees of freedom. These include the most energetic modes governing the equation system. The fluid-elastic instabilities correspond to low frequencies with high amplitudes, clearly distinct from the random turbulence background. Selecting a reduced number of (the most energetic) modes only to perform the Galerkin projection of the complete partial-differential equations system and hence to obtain an Ordinary Difference Equations (ODE) system is therefore a very promising approach; it can save time for the design and parametric study of the cylinders array in nuclear engineering applications. Hence, the ROM can be used as predictor and interpolator, to assess critical bifurcation points, after having first of all studied its intervals of confidence, to ensure an efficient, robust and reliable ROM approach. The basis of the most-energetic modes can be issued from Proper Orthogonal Decomposition (POD) and the system of equations is based on the complete set of state variables (Bourguet et al., 2007a,b, Bourguet et al., 2011). The ROM is especially advantageous, because of the strongly time-dependent CFD–CSM kernel that is to be introduced in the design procedures. The ROM actually offers a considerable time reduction in the computational cost.

An efficient ROM approach must be able to predict not only quasi-periodic phenomena, certainly characterizing the majority of the governing fluid-elastic instabilities, but also to capture 'irregular' (or 'erratic') phenomena that occur near the critical points of the fluid-elastic flutter. Furthermore, efficient ROM approaches for the present context have to take into account movement or deformation of the solid boundaries based on proper orthogonal modes. Whenever a POD basis is used, this is possible without changing the methodology, just by adding a source term and taking into account the modification of the space through use of Hadamard's approach (as described by Bourguet et al., 2011), which allows a frozen boundary condition pattern to be employed. This is a stringent requirement from a mathematical point of view, required from the POD approach to build the ROM. Of course, other orthogonal bases can be used such as wavelets or balanced truncation techniques, the latter option being limited at the moment to linearized systems, mainly used in flow control. Therefore, the POD basis is proposed here. Moreover, the ROM system must be stable and robust. It is well known that the mathematical form of the ROM system is, by nature, not conservative and it therefore produces a progressive divergence from the physical reality as a function of time. For this reason, appropriate calibration terms must be employed.

Finally, an important aspect of the ROM approach is to establish the confidence levels in which the ROM approach remains reliable as an interpolator of the complete system across a specific range of the sensitive parameter (e.g. the Scruton number, the structural damping and/or the stiffness). These confidence levels have to be studied in the vicinity of each bifurcation point by performing a reduced number of complete simulations at the extrema of the intervals of the specific sensitive parameter. The POD basis is suitably enriched to capture the appearance and evolution of the fluid-elastic instability, before performing the Galerkin projection. The ROM approach is therefore highly suitable for studying propagation of uncertainties in the fluid-elastic system, at a given interval covering an important critical behavior, by enriching the POD basis issued from a 'randomization' process of the boundary conditions.

4.5. Matrix transfer for parallel computing with distributed memory architectures

Since large data flows must be processed to deal with the ultimate goal of real-scale configurations (for 5.000 cylinder configurations with very long spanwise dimensions, representative of real exchanger cases, for instance), optimization algorithms are clearly required. Up to now, an order of 4D only is taken into account as elongation for numerical simulations, in a domain of 20 cylinders which correspond to a 100 million points grid. It is intended to achieve up to 12D span-wise dimensions, thanks to the newly proposed optimized numerical algorithms. Moreover, identifying the system stability threshold and criteria involves a large number of parameter sensitivity analyzes and therefore an efficient computing capacity. To achieve realistic numerical simulations of this

fluid–structure unsteady phenomenon, several techniques must be combined, such as moving grids of different scales superimposed on another one (Chimera techniques, for example) and coupled systems of PDE equations with many variables of different types, also including techniques to treat the non-linear terms. The resulting systems that will be solved may imply anything from 25 million nodes (for 1D spanwise) up to potentially 1 billion nodes (for more realistic 3D simulations) in mesh grids. The discretization in time, to monitor this evolution phenomenon appropriately, implies that these systems be solved efficiently many times successively. Obviously, to reach such large-scale and complex numerical simulations, parallelism with distributed memory architectures are mandatory both to hold the problems themselves and to achieve sufficient speed in the computations. The way that the problems will be modeled and discretized implies that part of the operators (after discretization) will be fixed throughout the simulation and another part will be updated at each time step, and one must take advantage from these numerical structural properties. Additionally, a solution involving several systems in sequence, with changing right-hand sides but with the same matrix, can also be achieved more efficiently with several techniques (Glolub et al., 2007). The purpose is to design efficient and parallel algorithms to be implemented into solvers, before incorporating them into the multi-physics platform involved.

4.6. Uncertainty on solutions to evolution problems with singularities

Dealing with fluid–solid coupled system in the space-time continuum domain, there are several possible strategies for discretization and computation. Most common approaches rely on small perturbation development methods providing a linear relation between kinematics and stress distribution. Consistent in a fully-linear case, these methods provide an approximation of the solution as the solution to an eigenvalue problem. They are currently used within the framework of dynamical system stability analysis and are often combined with a superposition method, enabling the separate computation of different linear effects and their superposition to a certain extent. In the framework of fully linear model problem formulation, small perturbation development procedures lead to a linearization of boundary conditions combined with the linearization of mass and momentum conservation equations, which lead to an expression of small pressure fluctuations exerted by fluids on solid walls. The action exerted by fluids on solids can be expressed by using the normal modes of the structure without fluid as the sum of three terms: added mass terms, quasi-steady terms and damping terms, potentially positive or negative and responsible for possible dynamic instability. Such formulation is for instance very efficient for considering potential flow near moving small magnitude solid boundary. Now considering a mechanical system made of a cylinder arrangement submitted to a potential cross flow, the stability of the system can be deduced from this analysis. The dynamical system can be expressed as a first order non linear differential equation whose solution is stable if all eigenvalues of the corresponding system have negative real parts. These eigenvalues continuously depend on mean flow velocity. Therefore the study of the family of solutions can exhibit series of critical reduced velocity values corresponding to Hopf bifurcations. In the presence of fully non-linear problems, with non-linearity in fluid, solid domain or through the interface, in presence of large magnitude motion or turbulence, such approaches are no longer valid and it is not possible to formulate implicitly boundary conditions at the interface. The question is: how to identity such bifurcation thresholds in fully non-linear configurations, out of small perturbation development procedure validity domain ? The present proposal enables the accurate identification of each terms involved in fluid solid coupled systems. Added mass, damping and stiffness terms and also external action of turbulence as well as non linear effects can be identified numerically by using numerical simulations. Uncertainty propagation are performed in order to get reliable stability maps and evaluate associated error intervals.

4.7. Stability maps of real large scale systems

From a theoretical point of view an interfacial coupling is considered. A dimensionless analysis of the fully-coupled system indicates that characteristic parameters must be taken into account to describe the dynamical interaction between both sub-systems and these parameters can be obtained in different manners by combining fluid and solid single-physics parameters in order to get mixed coupling parameters. Most common coupling parameters are mass ratio, Cauchy number CY and also reduced velocity UR describing the ratio between fluid and solid motion characteristic times. However other parameters can be defined and in any case, the physics of the interaction and therefore the numerical methods to be chosen for computation of the fully-coupled system directly depend on the values of these parameters. For stability analysis of mechanical systems, De Langre (2001) proposes a new referential domain in the Cauchy number, reduced velocity plane pointing out static, pseudo-static and dynamic instability conditions. For example when reduced velocity is close to 1, a great accuracy of interfacial coupling modeling is required and when mass ratio is close to 1, stability properties of numerical methods must be enhanced. The purpose is to perform a model of a part of the tube bundle of the stream generator of a nuclear power plant. Fluid-elastic instability of tube bundles of steam generator is characterized by a critical cross-flow velocity beyond which the vibration amplitudes increase rapidly leading to possible rapid damaging of tubes. Expected result is a stability function proving system damping evolution with respect to the reduced velocity based on the cross flow velocity. The model must include several rows and columns, modeling the lower part of the tube bundle. The cross flow velocity as well as other parameters like Scruton number, Reynolds number, pitch ratio constitute input data. A complete dimensionless analysis must be provided in order to get a generalist modeling (Barbut et al., 2010). The results in terms of instability threshold can be compared to standard reference data deduced from practice guidelines and be useful in the framework of definition of the margin for design.

4.8. Extension to two-phase flows

In Pressurized Water Reactors (PWR) heat exchangers are submitted to two-phase flows of steam and water. The U-bend region of nuclear steam generators is particularly vulnerable to vibration due to two-phase cross-flows. Fluid-elastic coupling effect between the vibrating structure and the flowing fluid may lead to fluid-elastic instability generated by a loss of damping. Fluid-elastic instability limit criteria may be expressed in terms of dimensionless parameters like a dimensionless flow velocity and a dimensionless mass-damping parameter (in which the damping is taken in still flow). A classical method for calculation of damping suggests that damping in two-phase flow is due to a specific two-phase mechanism in addition to classical viscous damping. Yet, some difficulty may arise for example in the definition of the viscosity of the two-phase mixture. In that sense, it can be proposed to consider the fluid damping as a global entity. Such an approach can be extended to take intro account the effect of two-phase flow velocity. Comparison of the evolution of damping with flow velocity between two-phase flow and single-phase flow can be investigated in that framework. It can be shown that fluid-elastic

instability in two-phase flow can be formulated in terms of dimensionless flow velocity and void fraction (Baj, 1998; Baj and De Langre, 2003).

5. Conclusion

This article deals with very large scale computing to determine vibration response of components such as heat exchanger tubes subjected to real flows which includes turbulence response as well as that due to vortex shedding and fluid-elastic instability. It gives a review of standard methods and proposes new computational strategies to enable numerical simulation of large complex systems involving multi-physics in the field of fluid and solid mechanics so as to provide stability analysis of cylinder arrangements like those encountered in heat exchangers with an optimal accuracy in spite of the large size of the systems to be considered. To deal with computation of large size fluid solid coupled systems, the combination of several approaches is proposed and model hybridation is investigated. Turbulence modeling is addressed in flows around moving obstacles for stability analysis of dynamical systems. Analyzing and understanding physical mechanisms responsible for energy transfer between fluid and solid, explaining interaction between turbulence and moving walls and its impact on energy transfer by taking into account turbulence inhomogeneous non-equilibrium patterns induced by solid motion lead to proposing a generalist dimensionless modeling for coupled fluid stress and solid induced dynamical response.

Numerical methods are although developed including all aspects on stability, consistency and robustness improvement of algorithms for computation of fully-coupled fluid solid systems. Hybrid field transfer methods relying on projection methods or Lagrange multiplier coupled with XFEM-like approaches are proposed in order to deal with fluid solid interface modeling by optimizing interfacial energy flux transfer consistency, accuracy and minimizing constraints on frontier element topology. Hybrid direct iterative solvers are formulated in order to improve stability and robustness of time integrators. Hybrid moving fixed grid methods are established in the context of mixed Eulerian Lagrangian formulations in order to investigate in the same time small and large magnitude frontier deformations. Finally model reduction using full time-space solutions of evolution problems with singularity is involved in order to reduce significantly the number of useful degrees of freedom of numerical simulations. To go to real scale algorithm performance improvements are proposed with developments in generalist multi-physics software coupling platform on parallelism and scaling for coupled computations. The purpose is to manage sequential coupled computation, implicit solvers and field matrix transfer in a parallel context. In this framework numerical simulations of singularity and bifurcation in mechanical systems can be performed. Stability limits can be identified numerically and new instability maps and flow regimes in moving cylinder arrangements submitted to external turbulent flows can be established. New dynamical stability analysis are then possible providing a better knowledge on energy exchanges between fluids and solids in these systems. Challenging real scale is therefore possible by using interpolation and hybridation between numerical and empirical solutions although suitable for validation purposes. Analysis of numerical solutions can then be proposed in order to estimate uncertainty and approximation errors on real scale stability thresholds. All development are proposed in the framework of an Opensource software platform distribution. Algorithms to be developed on hybrid solvers, hybrid dynamic grid formulations, hybrid turbulence modeling, model reduction and field interpolation may be useful as far as multi-physics, multi-scale, large size computations are concerned.

Acknowledgments

Authors want to thank MM. *C. Caremoli*, A. Adobes, J. Berland, A. Ribes, D. Laurence, D. Monfort, L. Borsoi, J. Cardolaccia, J.P. Magnaud, G. Harran, E. Deri, S. Ahzi, J. Dusek, D. George, R. Guivarch, Y. Remond for their helpful contribution in the present work carried out in the framework of the *BARESAFE* project and supported by the French National Research Agency (ANR) and the Free and Open Source Software Group (GTLL) of the Systematic Competitiveness Pole.

References

Adobes, A., Gaudin, M., 2004. Numerical study of tube bundle vibrations in an N4 standardized nuclear plant series steam generator. In: Flow Induced-Vibrations Conference, Palaiseau, France.

Adobes, A., Pillet, J., David, F., Gaudin, M., 2006. Influence of steam generator tube bundle vibrations on the operating diagram of a nuclear plan during strech out. Flow-Induced Vibrations Conference, ICPVT-11 93239, pp. 1–10.

Afgan, I., Moulinec, C., Prosser, R., Laurence, D., 2007. Large Eddy Simulation of turbulent flow for wall mounted cantilever cylinders of aspect ratio 6 and 10. Int. J. Heat Fluid Flow 28 (4), 561–574.

Arian, E., Fahl, M., Sachs, E.W., 2000. Trust-Region Proper Orthogonal Decomposition for Flow Control. Icase Report 25.

Baj, F., 1998. Damping and fluid-elastic instability of a tube bundle submitted to two-phase flow. PhD Thesis.

Baj, F., 2002. Adimensionnement des forces de couplage fluide-structure, CEA-DM2S report, SEMT/DYN/RT/022-033/A.

Baj, F., De Langre, 2003. Scaling of damping induced by bubbly flow across tubes. J. Fluids Struct. 17, 351–364.

Barbut, G., Marcel, T., Braza, M., Harran, G., Longatte, E., Baj, F., Magnaud, J.P., 2009. Simulation de l'interaction fluide – structure dans un faisceau de tubes nombre de Reynolds lev, Congrès Francais de Mécanique, Marseille, France.

Barbut, G., Marcel, T., Braza, M., Harran, G., Longatte, E., Baj, F., Magnaud, J.P., Hoarau, Y., 2010. Simulation of fluid structure interaction in a tube array under cross flow at high Reynolds number. In: ETMM8, Marseille, France.

Barbut, G., Braza, M., Harran, G., Hoarau, Y., Vos, J., Longatte, E., Pasutto, T., 2007. Prdiction de l'coulement turbulent dans un faisceau de tubes en configuration statique et en mouvement d'oscillation force. Congrès Francais de Mécanique, Grenoble, France.

Benek, J.A., Steger, J.L., Dougherty, F.C., Buning, P.G., 1986. Chimera: a grid-embedding technique. Technical Report, AEDC-TR-85-64, Arnold Engineering Development Center.

Benhamadouche, S., Laurence, D., 2003. LES, coarse LES and transient RANS comparisons on flow across a tube bundle. Int. J. Heat Fluid Flow 24, 470–479.

Bergeaud, V., Tajchman, M., 2007. Application of the Salome software architecture to nuclear reactor research. In: SCS Spring Simulation Multiconference on High Performance Computing Symposium, Norfolk, USA.

Berkooz, G., Holmes, P.J., Lumley, J.L., 1993. The proper orthogonal decomposition in the analysis of turbulent flows. Annu. Rev. Fluid Mech. 25, 539–575.

Blevins, R.D., 1990. Flow-induced Vibrations, 2nd edition.

Bonet, J., Peraire, J., 1991. An Alternating Digital Tree algorithm for 3D geometric searching and intersection problems. Int. J. Numer. Methods Eng. 31 (1), 1–17.

Bouhadji, A., Bourdet, S., Braza, M., Hoarau, Y., Rodes, P., Tzabiras, G., 2002. Turbulence modeling of unsteady flows with a pronounced periodic character. In: Haase, W., Selmin, V., Winzell, B. (Sci Eds.), Notes on Numerical Fluid Mechanics and Multidisciplinary Design, Progress in Computational Flow-Structure Interaction, vol. 81. Springer.

Bourguet, R., Braza, M., Dervieux, A., 2007a. Reduced-order modeling for unsteady transonic flows around an airfoil. Phys. Fluids 19, 111704.

Bourguet, R., Braza, M., Dervieux, A., 2011. Reduced-order modeling of transonic flows around an airfoil submitted to small deformations. J. Comput. Phys. 230, 159–184.

Bourguet, R., Braza, Harran, G., El Akoury, R., 2008. Anisotropic Organised Eddy Simulation for the prediction of non-equilibrium turbulent flows around bodies. J. Fluids Struct. 24 (8), 1240–1251.

Bourguet, R., Braza, Perrin, R., Harran, G., 2007b. Anisotropic eddy viscosity concept for strongly detached unsteady flows. AIAA J. 45 (5), 1145–1149.

Bourguet, R., Braza, M., Sevrain, A., Bouhadji, A., 2009. Capturing transition features around a wing by reduced-order modeling based on compressible Navier–Stokes equations. Phys. Fluids 21 (9), 111701–111701.

Braza, M., Bourguet, R., Ouvrard, H., Perrin, R., 2010. Turbulence modeling for strongly detached unsteady high-Re flows, based on refined PIV experiments. In: CAIMS-SCMAI, Canadian Symposium on Fluid Dynamics.

Braza, M., Perrin, R., Hoarau, Y., 2006. Turbulence properties in the cylinder wake at high Reynolds number. J. Fluids Struct. 22, 757–771.

Braza, M., 2000. The direct numerical simulation at the service of turbulence modeling methodologies for unsteady aerodynamic flows. In: ECCOMAS, Barcelona, Spain.

Buffoni, M., Camarri, S., Iollo, A., Salvetti, M.V., 2006. Low-dimensional modeling of a confined three-dimensional wake flow. J. Fluid Mech. 569, 141–150.
Bui-Thanh, T., Damodaran, M., Willcox, K., 2003. Proper Orthogonal Decomposition Extensions for Parametric Applications in Transonic Aerodynamics. AIAA Paper, 4213.
Caillaud, S., 1999. Excitation force et contrle actif pour la mesure des forces fluide-lastiques. PhD Thesis.
Chandesris, M., 2007. Modélisation des écoulements turbulents dans les milieux poreux et à l'interface avec un milieu libre. PhD report.
Chen, 1987. Flow-induced Vibration of Circular Cylindrical Structures. Hemisphere Publishing Corporation.
Chesshire, G., Henshaw, W.D., 1990. Composite overlapping meshes for the solution of partial differential equations. J. Comput. Phys. 90 (1), 1–64.
Connors, H.J., 1970. Fluid-elastic vibration of tube arrays excited by cross-flow. Flow-Induced Vibration in heat exchanger, Chicago, N.Y.: ASME, pp. 42–46.
Couplet, M., Basdevant, C., Sagaut, P., 2005. Calibrated reduced-order POD–Galerkin system for fluid flow modeling. J. Comput. Phys. 207, 192.
De Langre, E., 2001. Computational modeling in fluid-structure interactions. Houille Blanche (1), 34–38.
Deloze, T., Hoarau, Y., Dusek, J., 2010. Chimera method applied to the simulation of a freely falling cylinder in a channel. Eur. J. Comput. Mech. 19/5–7, 575–590.
Deloze, T., Hoarau, Y., Dusek, J., 2012. Transition scenario of a sphere freely falling in a vertical tube. J. Fluid Mech. 711, 40–60.
Dervieux, A., Braza, M., Dussauge, J.P., 1998. Computation and Comparison of Efficient Turbulence Models for Aeronautics, Notes on Numerical Fluid Mechanics, 65, Publisher Vieweg.
Dougherty, F.C., Kuan, J., 1989. Transonic store separation using a three-dimensional Chimera grid scheme. AIAA Paper, AIAA-89-0637.
Dowell, E., Hall, K., 2001. Modeling of fluid–structure interaction. Annu. Rev. Fluid Mech. 33, 445–490.
Farhat, C., Lesoinne, M., 1997. Improved Staggered Algorithms for the Serial and Parallel Solution of Three-Dimensional Nonlinear Transient Aeroelastic Problems. Center for Aerospace Structures, 97-11, University of Colorado, Boulder, Colorado, AIAA Journal.
Glolub, G.H., Ruiz, D., Touhami, A., 2007. A hybrid approach combining Chbyshev filter and conjugate gradient for solving linear systems with multiple right-hand sides. SIAM J. Matrix Anal. Appl. Soc. Ind. Appl. Math. 29 (3), 774–795.
Gottlieb, D., Orszag, S.A., 1977. Numerical Analysis of Spectral Methods: Theory and Applications. SIAM-CMBS, Philadelphia.
Gotz, J., Iglberger, K., Feichtinger, C., Donath, S., Rde, U., 2010. Coupling multi-body dynamics and computational fluid dynamics on 8192 processor cores. Parallel Comput.
Graham, W.R., Peraire, J., Tang, K.Y., 1999. Optimal control of vortex shedding using low-order models. Int. J. Numer. Meth. Eng. 44, 973–990.
Granger, S., Padoussis, M.P., 1996. An improvement to the quasi-steady model with application to cross-flow induced vibration of tube arrays. J. Fluid Mech. 320, 163–184.
Haase, W., Braza, M., Revell, A., 2009. Sci Eds. DESider – A European Effort on Hybrid RANS-LES Modeling, Notes on Numerical Fluid Mechanics and Multidisciplinary Design, 103, Springer, ISBN 978-3-540-92772-3.
Hachem, E., Rivaux, B., Kloczko, T., Digonnet, H., Coupez, T., 2010. Stabilized finite element method for incompressible flows with high Reynolds number. J. Comput. Phys. 229 (23), 8643–8665.
Hover, F.S., Miller, S.N., Triantafyllou, M.S., 1997. Vortex-induced vibration of marine cables: experiments using force feedback. J. Fluids Struct. 11, 307–326.
Huvelin, F., 2008. Couplage de codes en interaction fluide structure et application aux instabilités fluide-élastiques. PhD Thesis.
Jus, Y., 2011. Modélisation et simulation numérique des vibrations induites par écoulements autour d'obstacles cylindriques seuls ou en en réseaux, PhD Thesis UPMC.
Le Tallec, P., Gerbeau, J.F., Hauret, P., Vidrascu, M., 2005. Fluid structure interaction problems in large deformation. C.R. Mécanique 333, 910–922.
Lever, J.H., Weaver, D.S., 1986. On the stability of heat exchanger tube bundles. Part 1. Modified theoretical model. J. Sound Vib. 107, 375–392.
Liang, C., Papadakis, G., 2007. Large eddy simulation of cross flow through a staggered tube bundle at subcritical Reynolds number. J. Fluids Struct. 23, 1215–1230.
Lohner, R., Sharov, D., Luo, H., Ramamurti, R., 2001. Overlapping unstructured grids. AIAA Paper, AIAA-2001-0439, Reno, NV.
Longatte, E., Bendjeddou, Z., Souli, M., 2003a. Methods for numerical study of tube bundle vibrations in cross flows. J. Fluids Struct. 18, 513–528.
Longatte, E., Bendjeddou, Z., Souli, M., 2003b. Application of Lagrangian Eulerian formulations to flow-induced vibration problems. J. Press. Vessel Technol. 125 (4), 411–417.
Longatte, E., Verreman, V., Souli, M., 2009. Time marching for simulation of fluid structure interaction problems. J. Fluids Struct. 25 (1), 95–111.
Ma, X., Karniadakis, G., 2002a. A low-dimensional model for simulating three-dimensional cylinder flow. J. Fluid Mech. 458, 181.
Ma, X., Karniadakis, G., 2002b. A law-dimensional model for simulating 3D cylinder flow. J. Fluid Mech., 181–190.
Mavriplis, D., Nastase, C., Wang, L., Burgess, N., 2009. Progress in high-order discontinuous Galerkin methods for aerospace applications, Forty-seventh AIAA Aerospace Science Meeting. AIAA-2009-601, Orlando, FL.
Meakin, R.L., 1993. Moving body overset grid methods for complete aircraft tilt rotor simulations. AIAA Paper, AIAA-93-3350.
Menter, F.R., Kuntz, M., Bender, R., 2003. A scale-adaptive simulation model for turbulent flow predictions. AIAA Paper, AIAA 2003-0767, Reno.
Morand, H.J.P., Ohayon, R., 1995. Fluid Structure Interaction. Wiley Sons.
Moreno, M., De Langre, E., Le Quere, P., 2000. A Large Eddy Simulation of the turbulent forcing spectrum induced by axial flow on a rod. In: 7th Conference on Flow-Induced Vibrations, Lucerne.
Nakahashi, K., Togashi, F., Sharov, D., 2000. Intergrid-boundary definition method for overset unstructured grid approach. AIAA J. 38 (11), 2077–2084.
Noack, B., Afanasiev, K., Morzynski, M., Tadmor, G., Thiele, F., 2003. A hierarchy of low-dimensional models for the transient and post-transient cylinder wake. J. Fluid Mech. 497, 335.
Papadakis, G., 2008. A novel pressure–velocity formulation and solution method for fluid structure interaction problems. J. Comput. Phys. 227, 3383–3404.
Piperno, S., Farhat, C., 2001. Partitioned procedures for the transient solution of coupled aero-elastic problems. Part II. Energy transfer analysis and three-dimensional applications. Comput. Methods Appl. Mech. Eng. 190, 3147–3170.
Planchard, J., Conca, C., Thomas, B., Vanninathan, M., 1994. Problèmes mathématiques en couplage fluide structure, applications aux faisceaux tubulaires. Eyrolles.
Pomarede, M., Longatte, E., Sigrist, J.F., 2010. Numerical simulation of an elementary vortex-induced-vibration problem by using fully-coupled fluid solid system computation. Int. J. Multiphys. 4 (3), 273–291.
Price, S.J., Padoussis, M.P., 1984. An improved mathematical model for the stability of cylinder rows subject to cross-flow. J. Sound Vib. 77 (1), 19–37.
Ribes, A., 2008. Salome platform: focus on the component model. In: International Conference CBHPC, Karlsruhe, Germany.
Ribes, A., Caremoli, C., 2007. Salome platform component model for numerical simulation. In: Compsac, 31st Annual International Computer Software and Applications Conference, vol. 2, pp. 553–564.
Rogers, S.E., Suhs, N.E., Dietz, W.E., 2003. PEGASUS 5: an automated preprocessor for overset-grid computational fluid dynamics. AIAA J. 41 (6), 1037–1045.
Rollet-Miet, P., Laurence, D., Ferziger, J., 1999. LES and RANS of turbulent flow in tube bundles. Int. J. Heat Fluid Flow 10, 241–254.
Rowley, C., Colonius, T., Murray, R., 2004. Model reduction for compressible flows using POD and Galerkin projection. Physica D 189 (1–2), 115.
Schaefer, M., Heck, M., Yigit, S., 2006. An Implicit Method for the Numerical Simulation of Fluid Structure Interaction. Springer, Berlin, Heidelberg.
Sergent, E., 2002. Vers une mthodologie de couplage entre la simulation des grandes chelles de la turbulence et des modles statistiques. PhD Thesis.
Sherer, S.E., Scott, J.N., 2005. High-order compact finite-difference methods on general overset grids. J. Comput. Phys. 210 (2), 459–496.
Simonin, O., Barcouda, M., 1986. Measurements of fully developed turbulent flow across tube bundle. Third International Symposium on Applications of Laser Anemometry to Fluid Mechanics, Lisbon, pp. 21.5.1–21.5.5.
Sitaraman, J., Wissink, A., Potsdam, M., Sankaran, V., 2008. Parallel unsteady overset mesh methodology for a multisolver paradigm with adaptive Cartesian grids. Twenty-sixth Applied Aerodynamics Conference, AIAA-2008-7177, Honolulu, HI.
Spalart, P.R., 2000. Strategies for turbulence modeling and simulations. Int. J. Heat Fluid Flow 21, 252–263.
Spalart, P.R., Allmaras, S.R., 1994. A one-equation turbulence model for aerodynamic flows. La Recherche Arospatiale 1, 5–21.
Spalart, P.R., Jou, W.-H., Strelets, M., Allmaras, S.R., 1997. Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. In: Liu, C., Liu, Z. (Eds.), Advances in LES/DNS, First AFOSR International Conference on DNS/LES. Greyden Press, Louisiana Tech University.
Speziale, C.G., Sarkar, S., Gatski, T., 1991. Modeling the pressure-strain correlation of turbulence: an invariant dynamical systems approach. J. Fluid. Mech. 227.
Strelets, M., 2001. Detached eddy simulation of massively separated flows. AIAA paper, 2001-0879.
Tanaka, H., Takahara, S., 1981. Fluid elastic vibration of tube array in cross-flow. J. Sound Vib. 77, 19–37.
Theodorsen, T., 1935. General theory of aerodynamic instability and the mechanism of flutter. Tech. Rep., 496, NACA.
Thompson, J., Soni, B., Weatherill, N., 1999. Part II. Unstructured grids. In: HandBook of Grid Generation, Chapter 14-126-1. CRC Press, Boca Raton.
Travin, A., Shur, M., Strelets, M., Spalart, P., 2000. Detached-eddy simulation past a circular cylinder. J. Flow Turbul. Combust. 63 (1–4), 293–313.
Vigo, G., Dervieux, A., Mallet, M., Ravachol, M., Stoufflet, B.,1998. Extension of methods based on the proper orthogonal decomposition to the simulation of unsteady compressible Navier–Stokes flows. In: CFD 98, Proc. 4th ECCOMAS Conf. Wiley, pp. 648–653.
Vigo, G., 2000. The Extension of P.O.D. to Complex System with Non-Homogenous Boundary Conditions, Application to a Turbulent Pulsed Jet, INRIA RR, 3945.
Wang, Z.J., 2007. High-order methods for the Euler and Navier Stokes equations on unstructured grids. Prog. Aerosp. Sci. 43 (1–3), 1–41.
Weaver, D.S., 2008. Some thoughts on the elusive mechanism of fluidelastic instability in heat exchanger tube arrays. In: Zolotarev, I., Horacek, J. (Eds.), Flow-induced Vibration. , pp. 21–28.

Wolf, P., Staffelbach, G., Gicquel, L.Y.M., Poinsot, T., 2009. Massively parallel large eddy simulation of combustion in annular gas turbines. In: SciDAC, San Diego, USA.
Wylie, B., Geimer, M., Nicolai, M., Probst, M., 2007. Performance Analysis and Tuning of the XNS CFD Solver on BlueGene/L, Computer Science, 4757, Proceedings of the 14th EuroPVM/MPI Conference, Springer, pp. 107–116.
Yetisir, M., Weaver, D.S., 1993. An unsteady theory for fluid-elastic instability in an array of flexible tubes in cross-flow. J. Fluids Struct. 7, 751–766.
Yokokawa, M., Itakura, K, Uno, A., Ishihara, T., Kaneda, Y., 2002. 16.4-Tflops direct numerical simulation of turbulence by a Fourier spectral method on the earth simulator, SC2002, Baltimore, MD.