NUMERICAL PREDICTION OF FLUIDELASTIC INSTABILITY THRESHOLD IN TUBE BUNDLES

Huvelin, F.
EDF R&D, Chatou, France
Longatte, E.
EDF R&D, Chatou, France
Baj, F.
CEA, Saclay, France
Souli, M.
LML, Lille, France

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 behavior generating possible instabilities and leading to possible short term failures through high magnitude 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 and code coupling methods, numerical simulation of flow-induced vibrations is now practicable for industrial purposes. The present paper is devoted to the numerical simulation of tube bundle vibrations in the presence of one-phase cross-flows. Results of computations are used to provide a numerical estimate of the critical flow velocity for the threshold of fluid-elastic instability in tube bundles without experimental investigation.

The methodology consists in simulating in the same time thermo-hydraulics and mechanics problems by using a mesh motion formulation for the fluid computation and a partitioned code coupling procedure for data transfer between fluid and structure solvers. A specific attention is paid to energy conservation at the fluid structure moving interface. The simulation of turbulence in the presence of moving boundaries and the presence of three-dimensional effects induced by the flow are particularly investigated (Huvelin et al., 2007). The present work results from high performance calculations performed on supercomputers and involving parallel computations. The good agreement between numerical and available experimental and analytical solutions is pointed out in the article.

This work is a first step in the validation of a computational process for the full numerical prediction of tube bundle vibrations induced by flows in the presence of turbulence and in the presence of fluidelastic effects responsible for complex coupling phenomenon between fluid and structure motions.

1. INTRODUCTION

Heat exchangers are vital components of power generation systems, which present many issues that are not understood in details such as vibrations of tube bundles under cross flows with possible instability development. This phenomenon is very complex. It is generally admitted that it is starting with a damping decrease of the system. In order to predict this instability, predictive methods have been developed in which the critical velocity of instability departure depends on parameters like structure damping in the presence of flow (Granger et al. 1993, Pettigrew et al. 1991).

In order to get a better understanding of the instability departure phenomenon, it may be interesting to consider coupled simulations of Computational Fluid Dynamics (CFD) and Computational Structure Dynamics (CSD) as proposed by Longatte et al. (2003) and Huvelin et al. (2007) for tube bundle applications. Two ways are possible : developing a devoted numerical tool or creating a coupling procedure between existing codes.

In this paper, the partitioned coupling algorithms initially proposed by Piperno et al. (2001) and Fahrat et al. (1997) are used. In the framework of classical fluid structure problems, small displacement and structure linear deformation are often investigared. However, the procedure involved in the present work is also convenient for large structure displacement and non linear deformation. The large displacement of the tubes is captured through efficient remeshing techniques based on Arbitrary Lagrange Euler (ALE) formulation in the CFD calculation.

To understand the mechanism of the tube instability development, one first considers a rigid body motion. This instability is related to the damping effect induced by the fluid. Small time steps are required for an accurate computation of the damping coefficient which is a relevant factor in fluid-elastic instability phenomena.

The present work is devoted to the simulation of the instability departure of a tube under cross-flow in an in-line square tube bundle. In the first part, the coupling method with existing codes is presented. In the second part, tube vibration simulations are proposed in order to check the ability of the tool to catch the threshold of instability departure.

2. COMPUTATIONAL PROCESS

The methodology is based on a partitioned procedure : the coupling procedure between the fluid and the structure problems is based on existing codes. The structure solver relies on a Lagrangian formulation, while the fluid solver is solved with an Eulerian formulation. In such coupling, the only link between fluid and structure codes is the boundary conditions at the fluid-structure interface. In order to simulate the coupled problem, the purpose is to ensure both the fluid adhesion to the structure and the stress tensor continuity at the interface. This implies that :

- The interface has to be tracked by the fluid and the structure solvers over the time.
- The fluid and structure codes are staggered in time.

The first item is the consequence of the mathematical description of the fluid and structure problems, while the second is due to the fact that the codes cannot be solved simultaneously.

For the first item, the interface tracking is solved by using an ALE method in the fluid solver (Longatte et al. 2003, Huvelin et al. 2007, 2008). When such a formulation is used, an arbitrary referentiel domain is introduced in which the Navier-Stokes equations are expressed.

For the second item, the time advancement is addressed by using a coupling scheme. Due to the fact that the fluid and the structure cannot be solved simultaneously at each time step, one of the two boundary conditions has to be predicted in order to allow the computation of the current time step. Usually, the procedure can be split into four steps : (1) predicting the structure displacement at the fluid-structure interface, (2) updating the fluid mesh motion (by means of an ALE formulation), (3) solving the fluid problem and computing the fluid forces acting on the structure and (4) solving the mechanical problem. A good prediction of the fluid-structure interface is required in order to ensure the energy conservation at the interface. Many coupling schemes with explicit (Farhat et al. 1997) or implicit (Abouri et al. 2005) procedures have been studied.

An improved serial staggered procedure (Piperno et al. 2001) has been chosen for this computation. It satisfies both velocity and displacement continuity conditions at the interface without violating the geometric conservation law with a first order approximation. The following relations are ensured at the interface for prediction of displacements and forces :

$$u_f^{n+1/2} = u_s^n + \frac{\Delta t}{2}\dot{u}_s^{n+1} \quad (1)$$

$$F_s^{n+1} = 2F_f^{n+1/2} - F_s^n \quad (2)$$

Δt designates the time step of the coupled computation. In the present work, the time step of fluid and structure calculations is the same one. u_s^n and u_f^n designate respectively the velocity of the fluid structure interface provided by the structure solver and transmitted to the fluid solver. F_f^n and F_s^n designate respectively the loading applied on the fluid structure interface provided by the fluid solver and transmitted to the structure solver.

The computation procedure of a fluid structure coupled problem with a partitioned procedure can be described as depicted in Figure 1.

3. CASE DESCRIPTION

The configuration involves a square in-line tube bundle containing ten half tubes and fourteen full tubes as shown in Figure 2. The geometric parameters are described in Table 1. Only one tube is moving in the cross-flow direction and it is assumed rigid. It has only one vibration mode in the lift direction. The properties of the fluid are those of water in ambient temperature. They are reported in Table 2.

A flow inlet velocity is imposed upstream of the fluid domain, while a free outlet is imposed downstream. For boundary conditions at the tubes and at the two sides of the domain, wall conditions are imposed in order to ensure fluid adhesion.

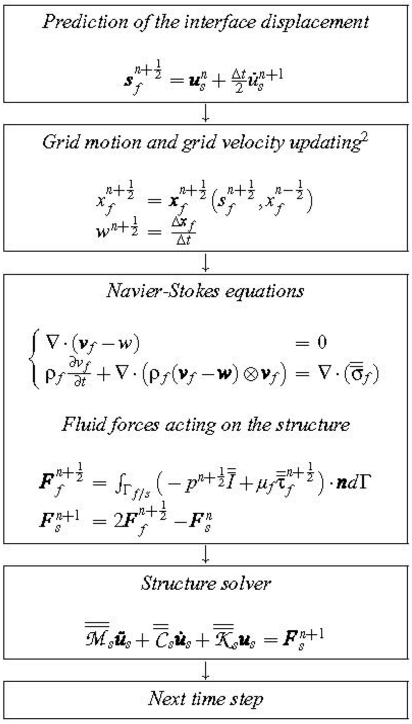

Figure 1 : *Computational procedure for simulation of a fluid structure coupled problem.*

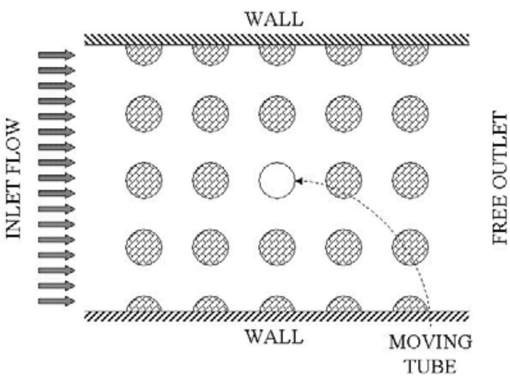

Figure 2 : Case configuration.

Three series of numerical simulations are presented below. The first one with a fluid at rest, the second one with a laminar flow and the third one with a turbulent flow across the tube bundle. The mechanical parameters corresponding to the three series are reported in Tables 3 and 4.

The goal is to catch the threshold of instability departure of the internal moving tube. In presence of flow, the properties of the structure and the inlet fluid velocity are chosen in order to get values of reduced velocities between *1* and *5*. An initial displacement is given to the structure at the first time step of the fluid solver. In the present calculations, the ratio between the initial displacement of the tube and its diameter equals to 5.10^{-4}. This is a very small value which places this analysis in a linear framework from a mechanical point of view.

A numerical mesh and space convergence has been performed in order to catch the expected flow and structure properties (Huvelin et al. 2007, 2008). The mesh involved in the work presented below involves about *150000* elements (Figure 3).

The choice of the time step is depending on a fluid criterion and on a structure criterion imposed by the vibration frequency of the tube. In order to catch with accuracy the frequency and the damping in water of the tube, it is necessary to have enough iterations per period.

Tube bundle length	0.07	m
Pitch to diameter ratio	1.44	-

Table 1: Geometric parameters.

Density	1000	$kg.m^{-3}$
Dynamic viscosity	1.10^{-3}	$kg.m^{-1}.s^{-1}$

Table 2: Fluid physical properties.

Natural frequency	2.5	Hz
Natural damping	0.0437	%
Mass	0.298	kg
Diameter	0.01	m

Table 3: Structure physical properties used for the configurations in laminar flow.

Natural frequency	14.3	Hz
Natural damping	0.25	%
Mass	0.298	kg
Diameter	0.01215	m

Table 4: Structure physical properties used for the configurations in fluid at rest and in turbulent flow.

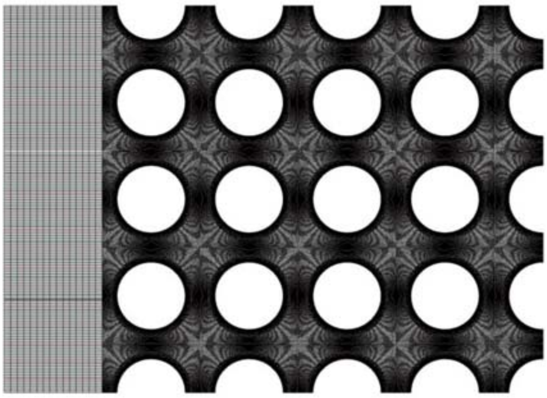

Figure 3 : Example of fluid domain mesh.

4. NUMERICAL RESULTS

4.1 Simulations in the presence of fluid at rest

A first simulation in quiescent fluid is presented in order to show the ability of the tool to forecast added mass and viscous damping with a good accuracy. The results are compared to available analytical results providing approximations of vibration frequency and damping of the tube moving in fluid at rest. The results obtained with different time steps provide the same frequency and damping. Results are in good agreement with expected solutions.

Parameters in fluid at rest	Numerical	Analytical	
Frequency	11.56	11.55	Hz
Damping	1.13	1.14	%

Table 5: Results of fluid structure coupled computation in fluid at rest with structure parameters of Table 4 (Huvelin et al. 2007, 2008).

4.2 Simulations in the presence of cross flow

The goal of this simulation is to catch the threshold of instability departure of the structure under cross-flows. For each inlet velocity to be considered in this case, the flow remains laminar and two-dimensional. A flow field resulting from a two-dimensional simulation is provided in Figure 4. Several inlet velocity values are considered. Two examples of simulations are illustrated on Figures 5 and 6 with flow velocities located below and above the tube instability threshold. The different behaviors of the structure are reproduced : stable and instable. The evolution of the tube vibration frequency and damping for several reduced velocity values is given in Figures 6 and 7. It is shown that the instability threshold is reproduced : the damping of the tube under cross flow falls to zero.

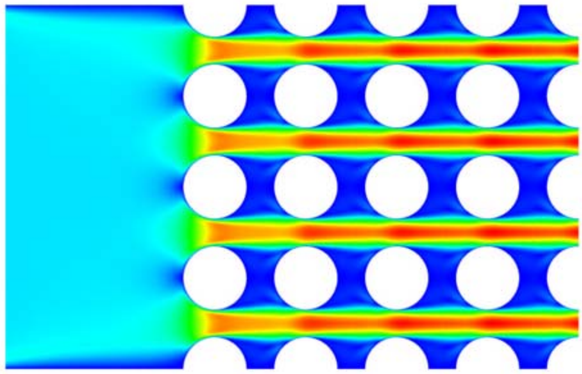

Figure 4 : Example of fluid flow field for an inlet velocity of 0.015 $m.s^{-1}$ with structure parameters of Table 3.

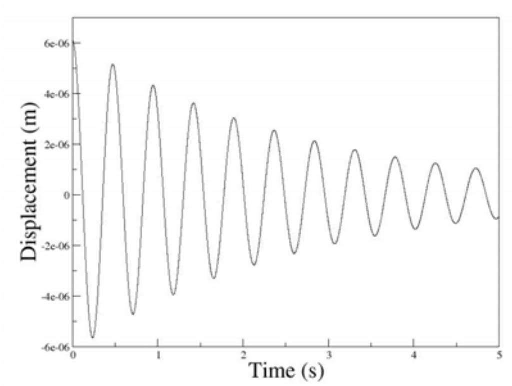

Figure 5 : *Example of tube displacement time history for an inlet velocity of 0.015 $m.s^{-1}$ with structure parameters of Table 3.*

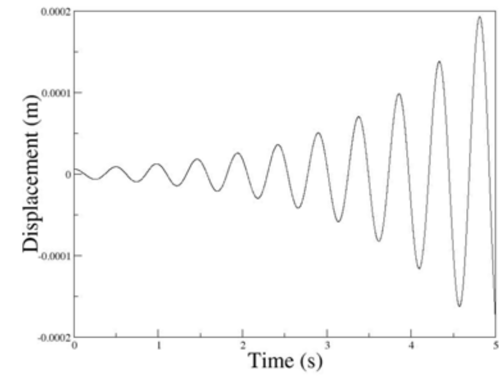

Figure 6 : Example of tube displacement time history for an inlet velocity of 0.3 $m.s^{-1}$ with structure parameters of Table 3.

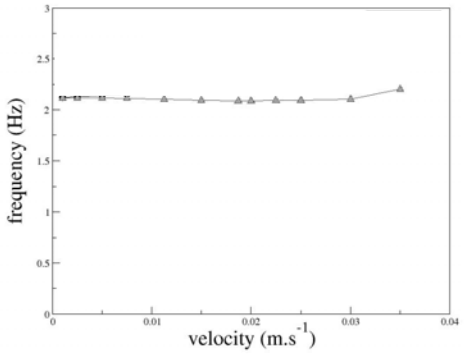

Figure 7 : Tube vibration frequency in terms of flow inlet velocity with structure parameters of Table 3 (Huvelin 2008).

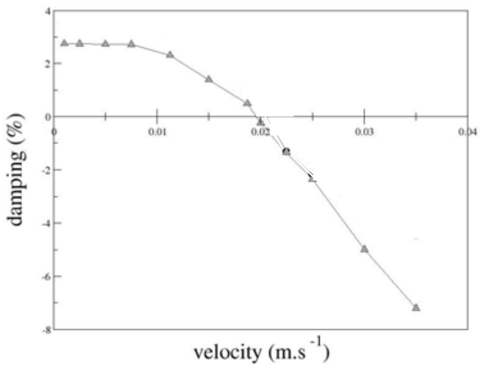

Figure 8 : Tube vibration damping in terms of flow inlet velocity with structure parameters of Table 3 (Huvelin 2008).

It would be interesting to compare these results to experimental data. However in this configuration, with laminar flows, for low Reynolds numbers, very limited number of experimental data are available. That is the reason why other computations have also been performed for higher Reynolds numbers. Numerical results are presented below. For high Reynolds numbers, experimental data are available which is very helpful for the validation of numerical results. However, turbulence patterns of flows have to be taken into account and three-dimensional calculations are required. Hence, the numerical identification of instability threshold requires high performance computations and it is CPU time consuming.

4.3 Simulations in the presence of turbulence

In what follows, one compares the results of fluid structure coupled calculations performed in 2D and in 3D. The purpose is to show that for high Reynolds numbers, the instability threshold identification required a 3D computation since an accurate estimation of flow fields and of near-wall fluid forces is required.

Figure 9 features a 2D flow field at moderate Reynolds number. An illustration of three-dimensional effects induced by the flow is given in Figure 10. 3D effects are significant. As a results, fluid forces and tube motion deduced from 2D and 3D computations are not the same. As shown in Figure 11, a 2D calculation tends to overestimate the near-wall fluid forces acting on the tube because no energy dissipation is taken into account in the tube length direction. That is the reason why the tube displacement deduced from a 2D calculation is not relevant for the identification of the tube instability threshold.

5. CONCLUSION

The goal of this paper is to find the threshold of instability departure for a structure under flow. 2D and 3D flows are considered. The computation involves a code coupling procedure based on a time explicit integration. An improved serial staggered procedure has been chosen which is the most appropriate formulation from CPU and accuracy point of view.

A first simulation with a quiescent fluid shows that the code ensures the prediction of tube frequency and damping in the presence of fluid. It is necessary to use a small time step in order to obtain values close to the analytical expected solutions.

Then, a computation with laminar flow shows that the code can forecast instability departure. It will be useful to use an higher initial displacement in order to make fluid-elastic forces non-linear and to check the behavior of the damping.

Finally, a computation with 3D turbulent flows is discussed and possible 3D effects of flow on near-wall fluid forces are investigated. In such case, 3D computations are required to capture tube instability development.

The next step will be to compare the results provided by the tool in the presence of turbulence and 3D flows and to compare them to available experiment data.

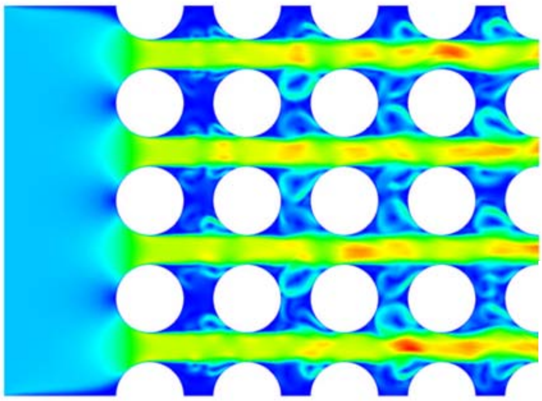

Figure 9 : Example of flow field for an inlet velocity of 0.075 m.s^{-1} with structure parameters of Table 4 (Huvelin 2008).

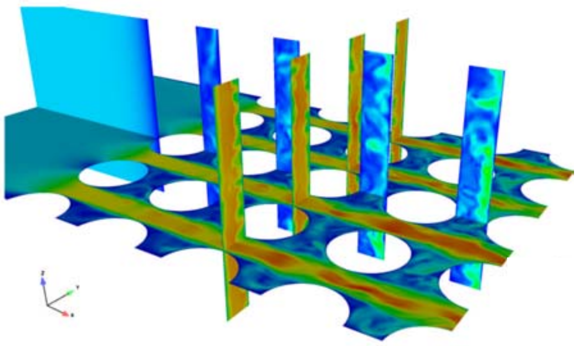

Figure 10 : Flow three-dimensional effects for an inlet velocity of 0.075 m.s^{-1} with structure parameters of Table 4 (Huvelin 2008)

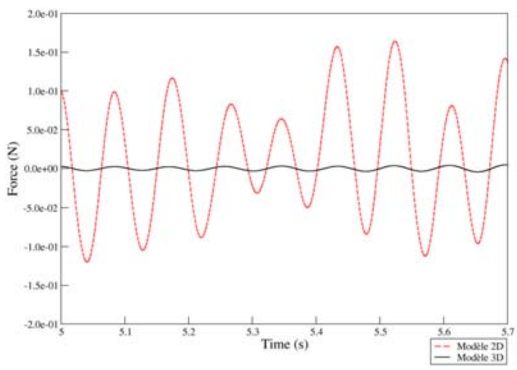

Figure 11 : Comparison of fluid forces acting on the moving tube estimated by two-dimensional and three-dimensional computations (Huvelin 2008).

6. REFERENCES

Abouri, D., Parry, A., Hamdouni, A., Longatte, E., 2005, A stable fluid-structure interaction algorithm: application to industrial problems. Journal of *Pressure Vessel Technology*, **128** : 1-8.

Huvelin, F., 2008, Fluid structure interaction code for simulation of fluid-elastic instability threshold in tube bundles. *PhD. Thesis. To be published*.

Huvelin, F., Morais, M., Baj, F., Magnaud, J.P., Longatte, E., Souli, M., 2007, Numerical simulation of tube bundle vibrations under cross flow. In *Pressure Vessels and Piping Division Conference*. San Antonio, Texas.

Farhat, C., Lesoinne, M., Stern, P., Lantéri, S., 1997, High performance solution of three-dimensional nonlinear aeroelastic problems via parallel partitioned algorithms : methodology and preliminary results. *Advances in Engineering Software*, **28**: 43-61.

Granger, S., Campiston, R., Lebret, J., 1993, Motion-dependent excitation mechanisms in a square in-line tube bundle subject to water cross-flow : an experimental modal analysis. *Journal of fluids and structures*, **7**: 521-550.

Longatte, E., Bendjeddou, Z., Souli, M., 2003, Methods for numerical study of tube bundle vibrations in cross flows, *Journal of Fluids and Structures*, **18**: 513-528.

Pettigrew, M.J., Taylor, C.E., 1991, Fluidelastic instability of heat exchanger tube bunldes : Review and design recommandations. *Journal of Pressure Vessel Technology*, **113**: 242-256.

Piperno, S., Farhat, C., 2001, Partitioned procedures for the transient solution of coupled aeroelastic problems - Part II : energy transfer analysis and three-dimensional applications. *Computer methods in applied mechanics and engineering*, **190**: 3147-3170.