VI International Conference on Computational Methods in Marine Engineering
MARINE 2015
F. Salvatore, R. Broglia and R. Muscari (Eds)

NUMERICAL STUDY OF VIV OVER A FLEXIBLE RISER

MARINE 2015

C.LOTHODE†*, G.FONTAINE†, E. GUILMINEAU§, A. WANG‡, F. VERTALLIER‡, M. MINGUEZ‡, A. CINELLO‡ AND D. GROSS†.

† K-Epsilon
300, route des Crêtes, 06902 Sophia-Antipolis, France
e-mail: corentin@k-epsilon.com, web page: http://www.k-epsilon.com

§ LHEAA, Ecole Centrale de Nantes
1, rue de la Noë, BP 92101, 44321 Nantes Cedex 3

‡ CITEPH Sponsors: Subsea7, Saipem, Technip, Doris Engineering
45 rue Louis BLANC, Maison de la Mécanique, 92038 Paris la Défense Cedex

Key words: Computational Methods, Offshore Structures, Fluid-Structure Interaction

Abstract. This study is based on the simulation of the fluid-structure interaction on risers. We aim to quantify the structural response of these long flexible pipes, used for the extraction of offshore petroleum when they are subjected to marine currents. The occurring phenomenon is known as VIV (Vortex induced vibration). These problems are a relevant challenge for several offshore companies, which are associated with K-Epsilon in a Citeph project. The project's goal is to use the FSI simulation tool developed by K-Epsilon and initially used for the simulation of flexible membranes such as sails, to model these VIV phenomena.

The problem of VIV in the case of a riser is a strongly coupled problem, meaning that the added mass is not negligible compared to the mass of the structure. This can be challenging for most fluid-structure interaction software. A strongly coupled algorithm is presented [6].

First, numerical results of fluid around cylinders are presented and compared to experimental results ([1], [10]) with several turbulence models, and time step sizes. Then, Chaplin's benchmark is presented with experimental / numerical comparison [3].

1 INTRODUCTION

VIV phenomenon has received significant amounts of research in the past. A large range of industrial problem needs to better understand when and where this behavior occurs. Applications range from fatigue on risers for the offshore industry, to renewable energy production using an immersed cylinder. A lot of studies both experimental and numerical have been conducted and have shown the difficulty to predict this phenomenon. The fluid-structure coupling is an important factor, and the response of the system can be very different depending on the parameters considered.

Numerical simulation of such mechanisms is promising in the prediction of these phenomena, but the main difficulty remains in developing algorithms able to solve a strongly coupled fluid-structure problem. This study shows how a correctly coupled algorithm can predict accurately VIV on risers.

2 FLUID-STRUCTURE COUPLING

2.1 Fluid solver

The fluid solver used is ISIS-CFD, of the software FINE/Marine™. It is developed by the DSPM team of the laboratory LHEEA. This solver is based on the incompressible Reynolds-averaged Navier-Stokes equations (RANSE), formulated in a strongly conservative way. The solver utilizes a finite volume approach and is generalized to handle non-structured meshes composed of arbitrary polyhedrons. Several turbulence model are implemented in ISIS-CFD. In this study, the models used are the SST k- ω and SST DES models of Menter.

The velocity field is obtained from the momentum equations and the pressure is obtained from the incompressibility constraint. The velocity-pressure coupling is done through the SIMPLE algorithm. Variables are stored at the cell center. Volume and surface integrals are evaluated by second order schemes. A second order backward time scheme is used for unsteady computations. For each time step, an internal loop is used to solve the non-linearities.

The ALE formulation (arbitrary Lagrangian-Eulerian) of the equations allows taking into account mesh morphing that occurs in fluid-structure interaction. The mesh deformation is based on an explicit propagation of the deformation state from the body to the rest of the domain. It is followed by a smoothing step that enforces a correct cell quality. This procedure is very fast (explicit), robust (with smoothing), and parallelized [6].

2.2 Structure solver

The structure solver is ARA and is developed by the company K-Epsilon. It was initially developed as part of a project simulating a sailing boat. The initial purpose of the software was to solve the structure related to sailing boats (sails, mast and rig) with fluid-structure interaction, solving the fluid flow with a panel code. From an early stage, the solver integrated tools to solve strongly coupled problems that occurs frequently in sailing and therefore the notion of added mass and fluid coupling is strongly integrated into the structure solver.

ARA utilizes a finite element method formulation to resolve the structure. At each time step, a dynamic equilibrium is solved between elements generating internal forces and external forces due to the fluid, gravity, and other externally applied loads. Element behaviors are computed through functions that take as inputs position, velocity and acceleration of each of its nodes. Elements can contain internal variables to account for behavior such as plastic or viscous deformation. It computes forces in each of its nodes as well as their derivatives. It gives the mass matrix $[M]=\partial F/(\partial x\quad)$, damping matrix $[D]=\partial F/(\partial x\quad)$, and stiffness matrix $[K]=\partial F/\partial x$. These functions can be composed of different kinds of finite elements, as well as penalization method elements such as contact or sliding elements. The fluid-structure interface is also considered using a special kind of element.

The time scheme used is the Newmark-Bossak scheme. This scheme is used because of a favorable compromise between filtering high frequencies and accurately accounting for the low frequencies. In the case of highly non-linear cases, this scheme avoids creating energy.

2.3 Fluid-structure coupling

The coupling algorithm used in this study is the quasi-monolithic method ([6]). This method is based on an implicit coupling adapted for partitioned coupling (using two different solvers for solving the fluid part and the structure part), while conserving a monolithic behavior (convergence and stability). To achieve this, the structure solver is called in the non-linear loops of the fluid solver. The fluid solver algorithm is then preserved. The structure equilibrium is sought at each non-linear iteration of the fluid. An interface element is added to the structure. This element is obtained from the Jacobian matrix of the interface. In the case of an exact Jacobian matrix, the method is similar to the monolithic method. Here, to have a faster solution, the matrix is simplified to a simple diagonal matrix.

The use of a Jacobian matrix is not mandatory for implicit coupling, even for strongly coupled problems. Nonetheless, without this matrix the coupling would need a very low value of under-relaxation, implying a large number of non-linear iterations to reach convergence. With this method, the difference in computation time between a fluid computation and a fluid-structure computation has a ratio falling between 1 and 2 depending on the structural model size and the strength of fluid-structure coupling.

3 RESULTS

3.1 Fixed cylinder

The hydrodynamic of flows that can generate VIV around risers of circular section can be characterized by the Reynolds number (Re), defined as following: Re = (U * D) / ν. Where U is the flow velocity, D is the diameter of the cylinder, and ν the kinematic viscosity of the fluid (m^2/s).

Above a Re~45, alternating vortices are emitted into the wake of the cylinder: this phenomenon is called vortex shedding. The frequency of emission (Fst) is related to other dimension of the problem through the Strouhal number (St): St = (Fst * D) / U.

This phenomenon of vortex shedding is key in the appearance of VIV, and therefore a good knowledge of the Strouhal number is fundamental to VIV prediction.

Before studying the fluid-structure interaction over the whole riser, fluid validation was performed to choose the correct mesh discretization, time step size, and turbulence modeling. The characteristics of the computation are:

- Volumetric mass density: ρ=998.4 kg/m^3
- Dynamic viscosity: μ=0.00104362 Pa.s
- Cylinder diameter: D=1 m

A first time step law of dt1=D/35*U was used for RANS simulation. The same time step law was used for DES simulation as well as a second time step law dt2=D/50*U (see **Table 1**).

Table 1: Flow velocity and time step for each Reynolds number

Reynolds	U (m/s)	dt1= D/35*U (s)	dt2 = D/50*U (s)
43000	0.044948	0.635661	0.444950
71000	0.074216	0.384978	0.269484

In the case of a DES computation, a first velocity field is developed through a RANS computation and is restarted as a DES computation (**Figure 1**).

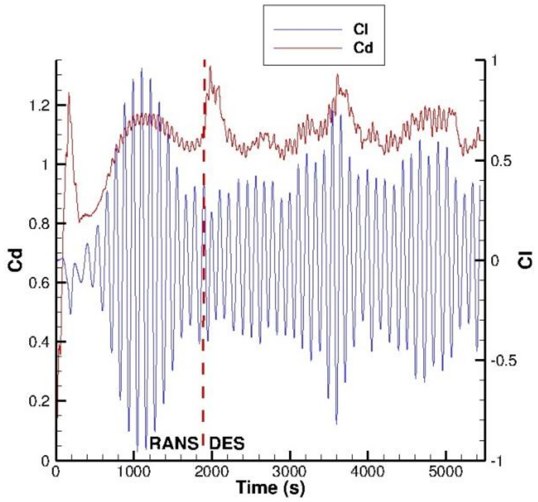

Figure 1: Drag and lift coefficients evolution for RANS/DES dt2, Re=43k

The vortex shedding process is the basis of the VIV mechanism. The vortex shedding generates lift fluctuations along the cylinder. This then leads to unsteady fluctuations in the drag forces which would otherwise be a steady loading on the cylinder. In the case of risers, these force variations can lead to a lock-in behavior where the shedding frequency changes to the natural frequency of the structure. We compare the Strouhal number and different RMS coefficients of lift and drag obtained against experimental values obtained in literature (see **Table 2**).

Table 2: Flow velocity and time step for each Reynolds number

			Re=43000			
	RANS	DES dt1	DES dt2	Exp [10]	Exp [8]	Exp [9]
Strouhal	0.192	0.191	0.202	0.19	0.192	0.189
Mean Cd	1.081	0.942	1.117	1.30	-	-
RMS Cd'	0.017	0.024	0.056	0.16	-	-
RMS Cl'	0.293	0.092	0.345	0.40	0.470	0.4-0.5

				Re=71000			
	RANS	DES dt1	DES dt2	Exp [10]	Exp [1]	Exp [8]	Exp [9]
Strouhal	0.206	0.215	0.202	-	-	0.188	0.187
Mean Cd	1.060	1.018	1.103	-	1.24	-	-
RMS Cd'	0.050	0.031	0.049	-	-	-	-
RMS Cl'	0.649	0.209	0.409	0.51	-	0.50	0.45-0.57

The Strouhal number is correctly obtained in the case of Re=43000, and a bit overestimated in the case of Re=71000. Both models (RANS and DES) show similar behavior in predicting the Strouhal number.

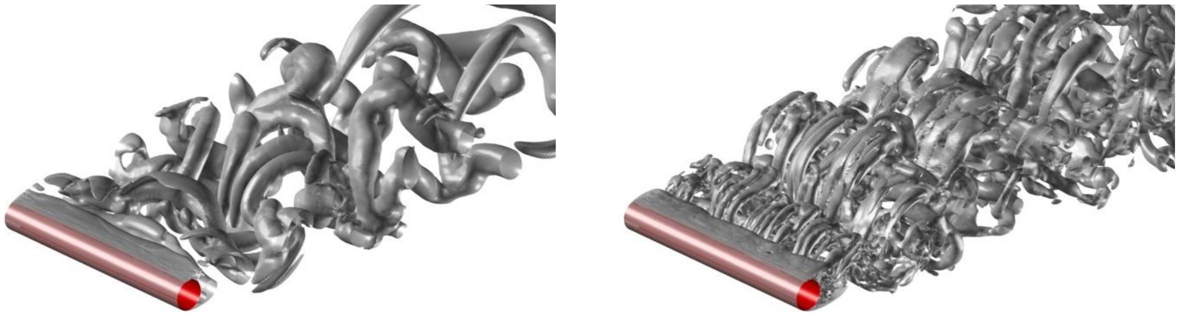

Figure 2: Vortex shedding, RANS (left), DES dt2 (right), Reynolds: 43k

Analysis of the results and visualization such as presented in **Figure 2** show significant differences in the forces and in the vortices structures. Forces are often under-predicted in numerical simulations, and there is a need to use models such as DES (with a very small time step) to approach experimental results. The result of such practice is very costly in terms of computing power and is not yet compatible with industrial means.

3.2 Simulation of a riser with fluid-structure interaction: the Chaplin experiment

The study previously conducted on a fixed cylinder has shown that highly refined computation (in time and space) were necessary to describe cylinders similar to a riser. Though, RANS simulation has shown that vortex shedding could be correctly described in term of frequency, even if the resulting effort was a bit off. VIV is located in a lock-in zone, where the structure oscillates around its natural frequency and not the Strouhal frequency. Hence, the hypothesis was made that VIV could be reasonably described as long as vortex shedding was correctly predicted.

The current study of a riser takes the setup of Chaplin [3] . The riser has an aspect ratio of L/D=468, a bending moment of 29.9Nm², and diameter D=0.028m. The computation of interest here is C1 of [3], with a velocity field of U=0.16m/s (Re=4500). The configuration is presented in **Figure 1**. One of the difficulties is that large deformations in the vacuum tube can cause cell crushing because of the proximity between the riser and the wall.

Laminar (no turbulence model active) and RANS SST k-ω models were studied and compared. A RANS turbulence modeling approach is necessary for high Reynolds number for which a laminar approach is not sufficient to take into account turbulent vortical behavior.

The domain of the fluid computation is composed of two parts. The upper part is of the dimension of the vacuum tube, and the lower part is where the riser is subject to a flow velocity field.

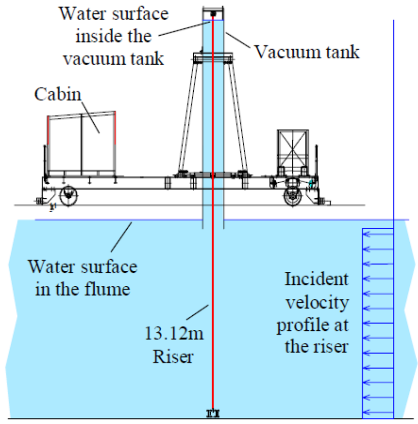

Figure 3: Configuration of the experiment

The mesh is composed of unstructured hexahedra with hanging nodes. To ensure a mesh of a reasonable size, cells were stretch in the z-direction while preserving a good refinement in x and y, comparable to the previous study conducted for fixed cylinders. The wake is refined in order to capture the vortices. The mesh in the boundary layer was refined to a y+ of 0.5.

The structure of the riser is modeled by beam elements. The boundary conditions are the following: free to rotate (except around z), and fixed in translation for the bottom and the top of the riser, and an imposed tension at the top. Several configurations of elements were tested and the number of elements chosen in this study is 99 beam elements for 100 nodes (about 600 degrees of freedom). The following properties were used to describe the beam elements: ES=5880000 N, EIx=EIy=29.9 Nm², GJ=80.66 Nm², and linear mass=1.85 kg/m.

The riser's deformations results from the combined action of lift and drag forces. Two aspects are of interest: the mean value of the in-line deformation (in the direction of the flow), and the fluctuations of both in-line and cross-flow deformations. The deformations are non-dimensionalized by the diameter and the length of the riser.

Figure 4 shows the deformation of the two modeling approaches compared to the Chaplin experiment. **Figure 5** shows the corresponding curvatures as these results are meaningful for the study of fatigue. It is interesting to see that in both cases the correct mode is captured for both in-line and cross-flow. The RANS approach tends to over predict deformations, but reproduces correctly the curvatures and the overall shape of the deformation. This is especially true in the cross-flow direction where it correctly captures the inflection in the lower deformation.

Through these simulations, we could reproduce the behavior of the riser in the case of an academic case with a low Reynolds number. The amplitude of the deformation and number of modes was correctly reproduced. This work has as a goal to put in place methodologies to obtain similar results with realistic efforts in term of computation power and real time.

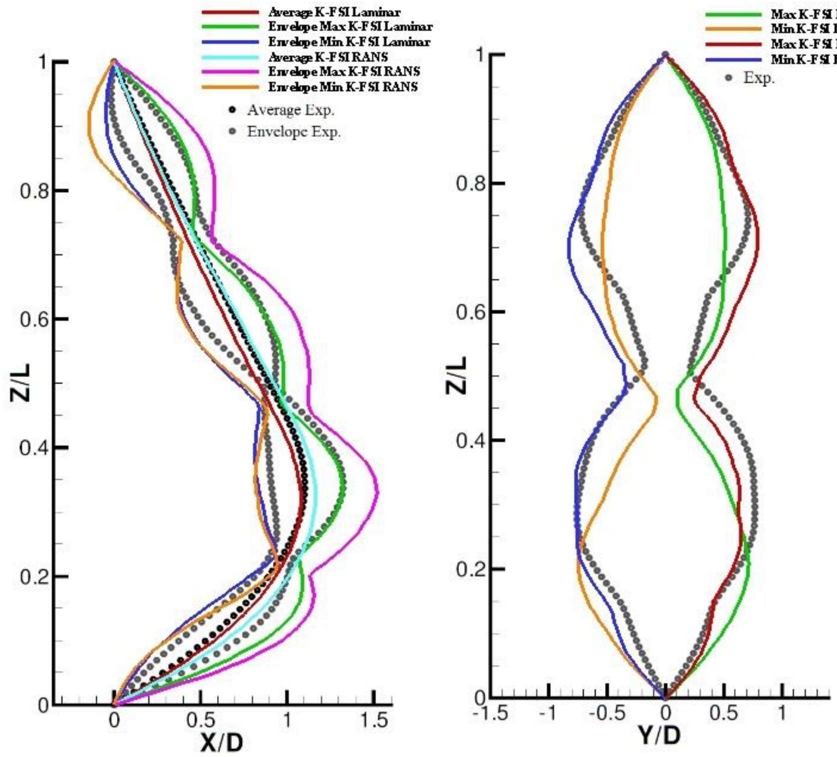

Figure 4: In-line and cross-flow deformation of the riser

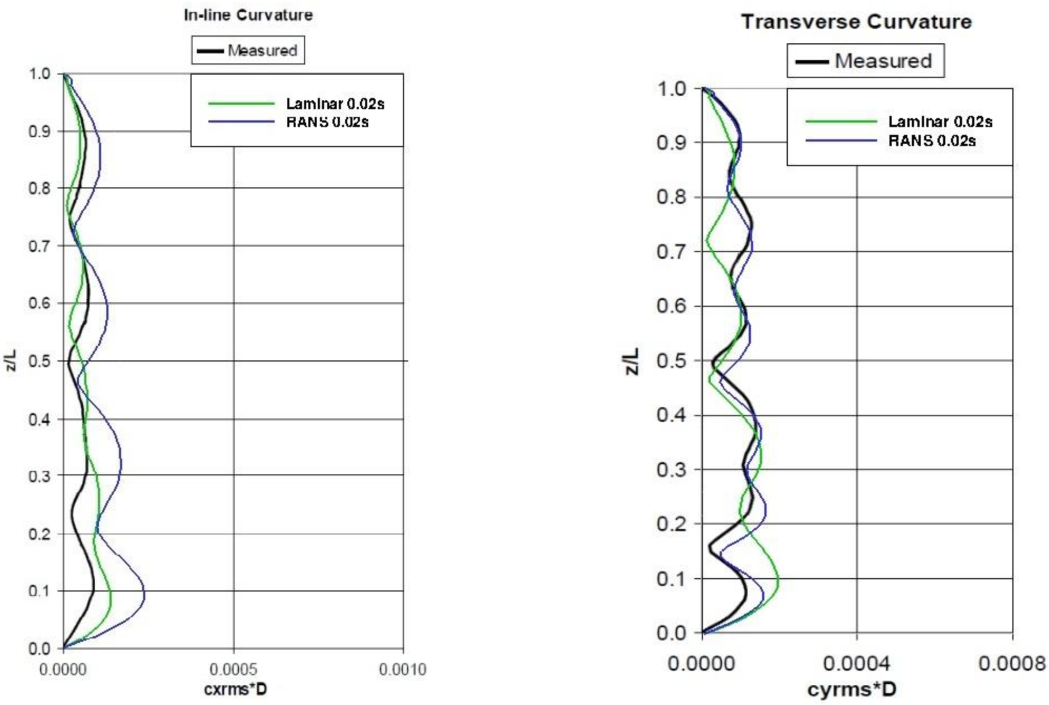

Figure 5: Curvature of the riser (in-line and cross-flow)

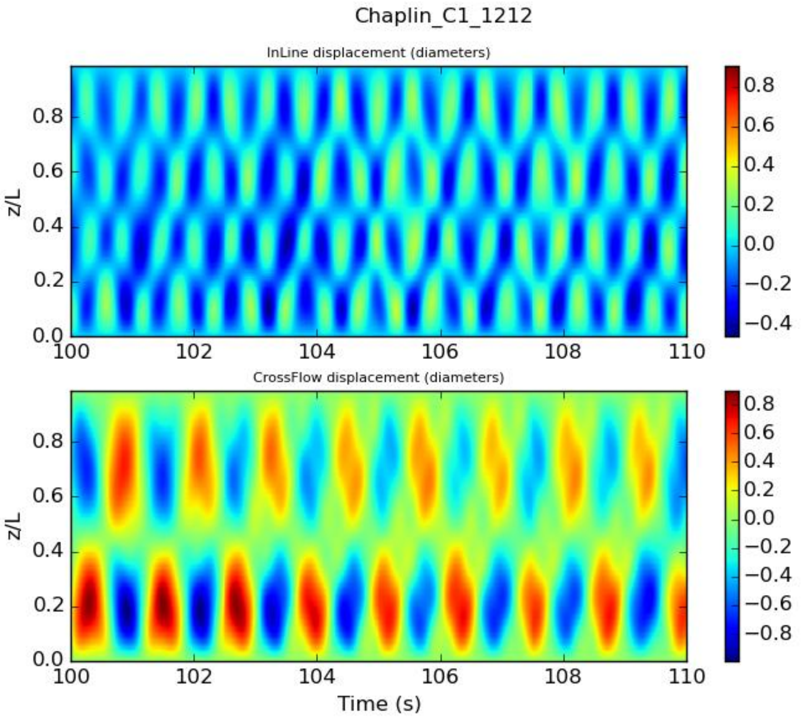

Figure 6: Inline and crossflow displacement (in diameter) relative to time

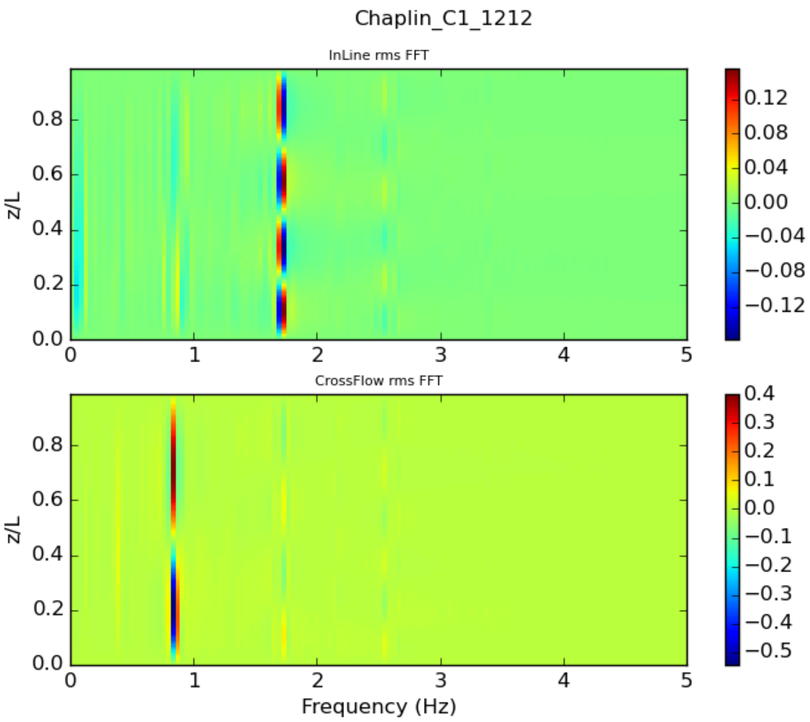

Figure 7: Inline and crossflow displacement spectrum

4 CONCLUSION

This study allowed to study separately the problems related to the flow behavior around a cylinder and those related to the fluid-structure interaction. Some difficulties were encountered during the study of fixed cylinders to predict accurately the forces. However it was observed that it is possible to reduce the accuracy of the fluid flow problem without deteriorating the structural response in the case of fluid-structure interaction. The next step of this work would be to confirm this tendency in the cases where the turbulence has a bigger impact.

REFERENCES

[1] Cantwell, B. and Coles, D. An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder. *Journal of fluid Mechanics* (1983), **136**:321-374.

[2] Laboratory measurements of vortex-induced vibrations of a vertical tension riser in a stepped current. *Proceedings of the 8th International Confernce on Flow-Induced Vibration, FIV2004*, Paris.

[3] Chaplin, J.R., Bearman, P.W., Huera Huarte, F.J. and Pattenden, R.J. Blind predictions of laboratory measurements of vortex-induced vibrations of a tension riser. *Journal of Fluids and Structures* (2005), 21(1):3–24.

[4] Cinello, A., Pétrié, F., Rippol, T., Molin, B. and Damblans, G. Comparison between static, forced oscillation and free oscillation model tests for the assessment of galloping instability at high reynolds numbers. In ASME 2013 *32nd International Conference on Ocean, Offshore and Arctic Engineering* (2013), V04AT04A049–V04AT04A049. American Society of Mechanical Engineers.

[5] Damblans, G., Berhault, C., Le Cunff, C., Molin, B., Wiet, P., Cinello, A., Deglaire, T., and Legras, J.-C. Investigations on galloping of non-circular cross sections. In ASME 2013 *32nd International Conference on Ocean, Offshore and Arctic Engineering*, V007T08A023–V007T08A023. American Society of Mechanical Engineers.

[6] Durand, M., Interaction fluide structure souple et légère, application au voilier, Ph.D. Thesis, Ecole Centrale de Nantes, 2012.

[7] Huang, K., Chen, H.-C., and Chen, C.-R. Vertical riser VIV simulation in uniform current. *Journal of energy resources technology* (2010), 132(3).

[8] Norberg, C. Pressure forces on a Circular Cylinder in Cross Flow. *IUTAM Symposium on Bluff-Body Wakes, Dynamics and Instabilities* 1992, 1993.

[9] Norberg, C. Fluctuating lift on a circular cylinder: review and new measurements. *Journal of Fluids and Structures* (2003), **17**(1):57–96.

[10] Szepessy, S. and Bearman, P.W. Aspect ratio and end plate effects on vortex shedding from a circular cylinder. *Journal of Fluid Mechanics* (1992), **234**:191–217.

[11] Travin, A., Shur, M., Strelets, M., Spalart, P. Detached-Eddy Simulation Past a Circular Cylinder. *Flow, turbulence and combustion* (1999), **63**:293-313.