VIII International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2019
E. Oñate, M. Papadrakakis and B. Schrefler (Eds)

DEVELOPMENT OF A CFD-BASED SCREENING TOOL FOR VIV PREDICTION

RAMIREZ V. Catherine[a,b], **GROSS David**[a], **PONCIN Jules**[a], **ROUX Yann**[a], **ASSIER Ludovic**[c], **VISONNEAU Michel**[b] **and LEROYER Alban**[b]

[a] K-Epsilon - 1300 route des Crêtes, 06560 Valbonne, France e-mail: catherine@k-epsilon.com, david@k-epsilon.com, jules@k-epsilon.com, yann@k-epsilon.com - web page: http://www.k-epsilon.com/

[b] Centrale Nantes. LHEEA (UMR-CNRS 6598), 1 rue de la Noë, 44321 Nantes Cedex 3, France e-mail: catherine.ramirez@ec-nantes.fr, michel.visonneau@ec-nantes.fr, alban.leroyer@ec-nantes.fr - web page: https://lheea.ec-nantes.fr/

[c] TOTAL E&P, Avenue Larribau, Pau Cedex, France e-mail: ludovic.assier@total.com - web page: https://www.ep.total.com

Key words: VIV, Slice method, FSI, RANS CFD, linear FEM, strong coupling

Abstract. This paper presents an efficient screening tool, based on the 2D strip method and the CFD-based approach. The proposed method allows the study of the dynamics of the vortex-induced vibration (VIV) phenomenon with almost the same accuracy as a fully 3D Fluid-Structure interaction (FSI) model, but with a reduction in computational times by a factor of nearly 20 compared to the 3D approach. Such a large reduction makes the use of CFD-based analysis feasible on a workstation within a reasonable time frame and opens the possibility of CFD approaches in a subsea pipeline design process.

1 INTRODUCTION

When a body of circular section is subjected to flow, a vortex shedding behavior develops which can induce an oscillatory displacement of the structure that will depend on the fluctuation of the efforts. This induced displacement, known as vortex-induced vibration (VIV), can excite the natural frequencies of the structure and make the system resonate, causing a premature fatigue failure.

To evaluate the risks of fatigue, engineers use tools developed and sufficiently validated to predict the fatigue of simple structures. Nevertheless, these simplified analysis tools are not adapted to the prediction of VIV of more complex oscillating structures, such as spools and subsea jumpers which can exhibit complex, multi-modal responses. Consequently, there is an industrial need for more accurate analysis methods.

Significant improvements can be achieved using three-dimensional fluid-structure interaction (FSI) simulations, based on computational fluid dynamics (CFD). However, detailed meshes, long simulation times and high performance computing are required. Knowing that industrial design processes demand the evaluation of various geometries and different structural designs of a large number of spools, a 3D FSI simulation that takes about two days for a single spool computation, is prohibitively slow for practical design use.

It is within this context, and after confirming the lack of precision of existing design analysis methods and classification society rules, that a collaboration between Total E&P and K-Epsilon was created. The main objective consists in developing a screening tool to estimate spool vibration phenomena within affordable industrial times, with moderate computational resources. Here the stated goal is to assess VIV behavior of a spool in a few hours on a high end work station with an accuracy sufficient for design analysis.

2 STATE OF THE ART

This section focuses on previous researches and results of different methods used to model the VIV phenomena. For a detailed description of the physics of VIV, the reader can refer to the review of Le Cunff [1], or for a deeper insight, to the references given by Willden [3].

The work of Le Cunff [1] gives a global overview of models that are used to predict the lifespan of cylindrical structures subjected to a flow. The presented methods, developed at the French Petrol Institute IFP, range from a simple modal analysis approach to coupled fluid-structure computations with resolution of the Navier-Stokes equations. Three of the methods are discussed below.

The modal approach presented in [1] performs a modal analysis to determine and select the potentially excited modes of the structure. Later, the modal amplitude is calculated and the lifting force of the cylinder is obtained from a large database of empirical data. Finally, a fatigue analysis is carried out to know the fatigue lifespan of the structure and to locate the most likely failure points. Although this method is fast, it is limited to studies of cross-flow displacements (CF). In addition, codes based on modal approaches use empirical coefficients which can lead to different results depending on the software for the same case, due to the large degree of scatter observed in experimental VIV data.

Looking for more accurate results, the fluid model is improved using a wake oscillator, based on the Van der Pol oscillator equation. This second approach searches to model the fluctuating nature of a turbulent flow by considering the oscillatory release of vortices, but giving no direct indication on the fluid flow. This model is more complete than the modal methods, and is fast enough to be used during the design phase.

Finally, the results of a previous strip method approach developed at the IFP are discussed. The work of Etienne [4], fully couples a two-dimensional FSI method, using the stream function-vorticity formulation of the Navier Stokes equations, with a finite element method (FEM) approach. Although no indication is given regarding the computation time scale of any of the above methods, the validation of the strip method, with experiments, shows that this method is both qualitatively and quantitatively able to capture the VIV phenomena.

Among others researchers using the strip method, there are also the contributions of Willden [3] and Duan [2]. These two works present approaches based on a loosely coupling between a set of two-dimensional flow calculations with a three-dimensional structural dynamics model.

Establishing a benchmark, the work of Chaplin [6] presents an experimental study carried out on a vertical riser exposed to a uniform stepped current ($Re \approx 4500$) with a set of different top tensions. In the tests, the lower 45% of the riser is subjected to the current, with the upper section in calm water. The riser is free to move in the in-line and cross-flow directions. Both the bottom and top ends are fixed using universal joints that fully constrain the displacement of the riser ends, but allow arbitrary rotation. Tension is applied to the upper end of the riser with a spring tensioning system. The experimental setup used in this work is shown in figure 1.

A parallel work of Chaplin [7], presented a comparison between the laboratory measurements and and a series of different numerical models. Paraphrasing the report, it is concluded that, in general, empirical codes were more successful at predicting cross-flow displacements and curvatures than codes based on CFD. In-line curvatures, potentially the cause of the largest fatigue damage, could not be computed by any of the empirically based codes, and in general, those based on CFD were also in very poor agreement with the measurements.

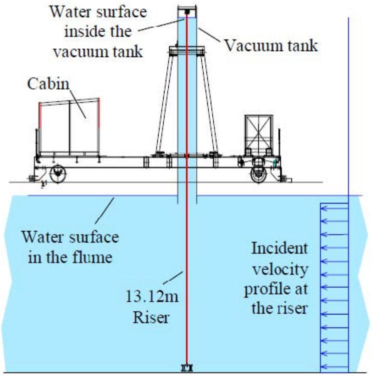


Figure 1: Experimental setup of the riser. Source: [6]

3 THE STRIP METHOD

In the strip method approach, a series of two-dimensional computational planes are placed along a structure perpendicularly to the spanwise axis. Each slice or plane is solved with the same method, but is individually solved and there is no transfer of information between the planes. The motions of the series of two-dimensional flow calculations is linked through the resolution of a three-dimensional structure.

Inherent to the strip method is that the three-dimensional effects in the fluid are neglected since only the local two-dimensional dynamics is resolved. Although three-dimensional eddies may develop in the wake of a cylinder ($Re > 180$), as it is mentioned by Willden [3], the VIV lock-in effect actually increases the coherence of the flow along the cylinder and makes the local flux essentially two-dimensional. Here, the lock-in effect refers to the synchronization of the vortex shedding and structural oscillatory frequencies such that, vortex shedding occurs at the natural frequency of the structure.

As mentioned also in [3], in order to capture the excitation and damping effect of the flow, a minimum of three slices are required per half-wavelength of vibration, therefore, the number of slices of simulation can be determined by considering either the highest vibration mode that is likely to be excited or the highest mode of interest.

4 PROPOSED NUMERICAL METHOD

Through the commercial coupling tool **K-FSI**, the strip method proposed in this work fully couples a velocity-pressure formulation of the two-dimensional, Navier Stokes equations with a three-dimensional, non-linear FEM model of the structure.

The **K-FSI** tool consists of the coupling between the structural code **K-Struct** and the fluid solver **ISIS-CFD**, included in the **FINE™/Marine** suite. **ISIS-CFD** is based on the finite volume method and solves the incompressible, unsteady Reynolds-averaged Navier-Stokes equations (URANS). **FINE™/Marine** is coupled in a segregated manner to **K-Struct**, the non-linear, unsteady finite element method structure solver, which forms the core of the **K-FSI** fluid-structure interaction suite.

In particular, for the present work, each fluid computation slice uses the same mesh, however, each plane can be placed at different angles of incidence. The developed tool transfers information between the different 2D slices and the 3D structure through ports called sockets, using the ZeroMQ distributed message library. The use of sockets allows the communication of multiple fluid calculations, which are executed in parallel and independently, with a single structural calculation. Each fluid computation may in turn also be computed locally in parallel using the message passing interface (MPI) protocol.

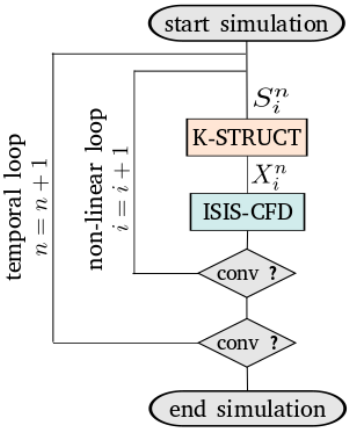


Figure 2: Coupling algorithm

The following algorithm is executed: several 2D CFD calculations are run independently on different processors. Once each of the fluid domains is resolved, the resultant fluid efforts are sent to the structural solver at each non-linear iteration. The structural solver then uses the information obtained from each one of the fluid computations to load the structure and computes the displacements. Finally, it communicates the new displacements to each of the fluid computations. Each fluid calculation communicates independently with the structure solver using a particular socket.

5 SIMULATION SETUP

The numerical model of this paper follows the experimental setup presented in the work of Chaplin [6], and shown in Figure 1. Two FSI models are simulated, a fully 3D fluid simulation and the proposed strip method. The structural and fluid configuration, common to both models, is presented below. Information particular to each model is detailed later on.

The structural mesh consists of 180 Euler-Bernoulli beam elements. The top and bottom ends of the riser are fully constrained in translation and free in rotation to model the universal joints at each end. A constant tension of 405 N is applied at the top of the riser. The structural properties presented in Table 1 have been used for model. Both the effect of gravity and buoyancy are represented in the structure model.

For both fluid approaches, the meshes used are non-structured, hexahedral/quadrilateral meshes. A no-slip boundary condition is used on the riser surface. The meshes were designed with a target y^+ value of $y^+ = 0.5$. Refinement boxes are used to capture the near and far wake. The $k - \omega$ (SST-Menter) model is used to model the turbulence, the water density is considered as $\rho_f = 999.207 \text{ kg} \cdot \text{m}^3$, and the dynamic viscosity is defined as $\mu_f = 0.001002 \text{ Pa} \cdot \text{s}$.

Table 1: Structural parameters used to model the riser

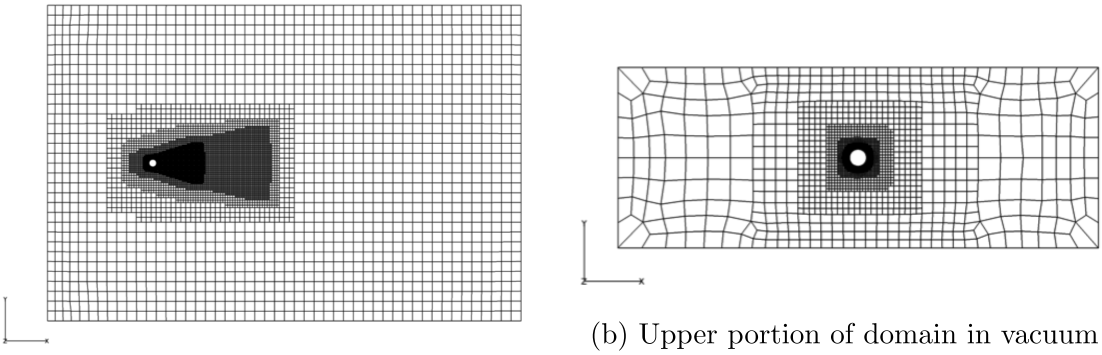
L (m)	**Diameter** (m)	**L/D**	**EA** (kN)	**EI** ($\text{N} \cdot \text{m}^2$)	**GJ** ($\text{N} \cdot \text{m}^2$)	**Top tension** (N)	**Linear mass** ($\text{kg} \cdot \text{m}^{-1}$)
13.12	0.028	469	5880	29.880	80.660	405	1.848

5.1 Fully 3D fluid model

To ensure a reasonable mesh size, the cells are stretched along the Z axis, whereas an isotropic discretization is kept in the X and Y directions, to correctly capture the cylinder and the near wake. In order to match the experimental set-up, the fluid domain is confined for $Z > 5.904$ m to represent the vacuum tank containing the upper 55 % of the riser. A view of the lower and upper sections of the mesh is given in Figures 3a and 3b, respectively. Boundary conditions are presented in Table 2. Riser motions are captured using mesh deformation in an arbitrary Lagrangian Eulerian (ALE) approach.

Table 2: Boundary conditions imposed on the fully 3D fluid domain

Type		Faces	Riser height
Outlet	Constant pressure	X_{max}	$Z < 5.904$ m
Inlet	Constant velocity ($0.16\ \mathrm{m \cdot s^{-1}}$)	X_{min}, Y_{min}, Y_{max}	$Z < 5.904$ m
Wall	Slip condition	X_{min}, X_{max}, Y_{min}, Y_{max}, Z_{max}	$Z > 5.904$ m
Wall	Slip condition	Z_{min}, Z_{max}	$Z < 5.904$ m



(a) Lower portion of domain exposed to current

(b) Upper portion of domain in vacuum tank

Figure 3: Fully 3D fluid domain

5.2 Strip method (2D Fluid calculations - 3D structure)

To assess the proposed strip method, two calculations have been executed with a total of 20 and 40 slices, respectively. Half of the slices are positioned at $Z < 5.904$ m, and the other half at $Z > 5.904$ m. An identical 2D mesh is used, for each and every one of the slices along the riser, even for $Z > 5.904$ m.

This is done under the consideration that 3D interactions are neglected by the strip model, and that no confinement effect is expected, given the distance from the riser to the walls, according to the experimental set-up. The 2D fluid mesh is shown in Figure 4. Unlike, the 3D simulations, the riser motions may be treated using rigid body translations of the fluid domain and hence, the cost of mesh deformation is avoided.

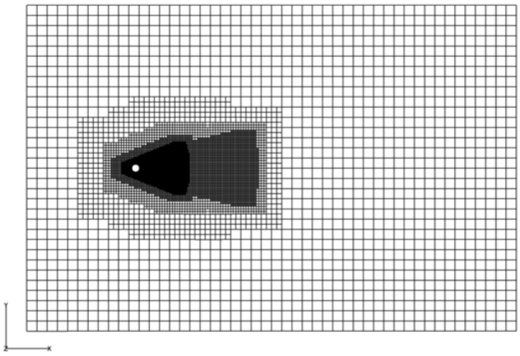


Figure 4: Numerical fluid 2D domain

To improve the capture of the vortex shedding, a slightly larger refinement box is used in comparison to the fully 3D model. The imposed boundary conditions are presented in Table 3.

Table 3: Boundary conditions imposed on each 2D fluid domain

	Type	Faces	Position of plane
Outlet	Constant pressure	X_{max}	all
Inlet	Null velocity ($0\ \mathrm{m \cdot s}$)	X_{min}, Y_{min}, Y_{max}	$Z > 5.904$ m
Inlet	Constant velocity ($0.16\ \mathrm{m \cdot s^{-1}}$)	X_{min}, Y_{min}, Y_{max}	$Z < 5.904$ m
Wall	Mirror condition	Z_{min}, Z_{max}	all

6 NUMERICAL RESULTS

Taking as the reference the experimental results of Chaplin [6], a comparison of the two approaches is done for the in-line (IL) and cross-flow (CF) direction displacements.

A summary of the in-line displacement results is presented in Table 4. The mean IL displacement and the displacement envelopes are shown in Figures 5 and 6. Note, that the mean displacement has been filtered out of the IL response in Figure 6a to make the differences in IL oscillations more apparent.

Both, the fully 3D approach and the strip method, underestimate the maximum of the mean displacement for the in-line direction, however the 3D method provides a better representation of the IL oscillation amplitudes. The vertical position of the maximum amplitude is well predicted with both methods. See Figure 5.

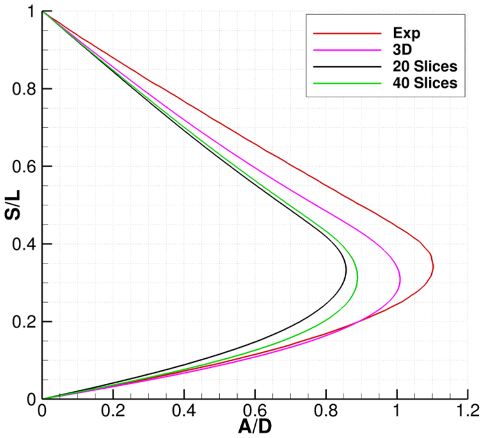
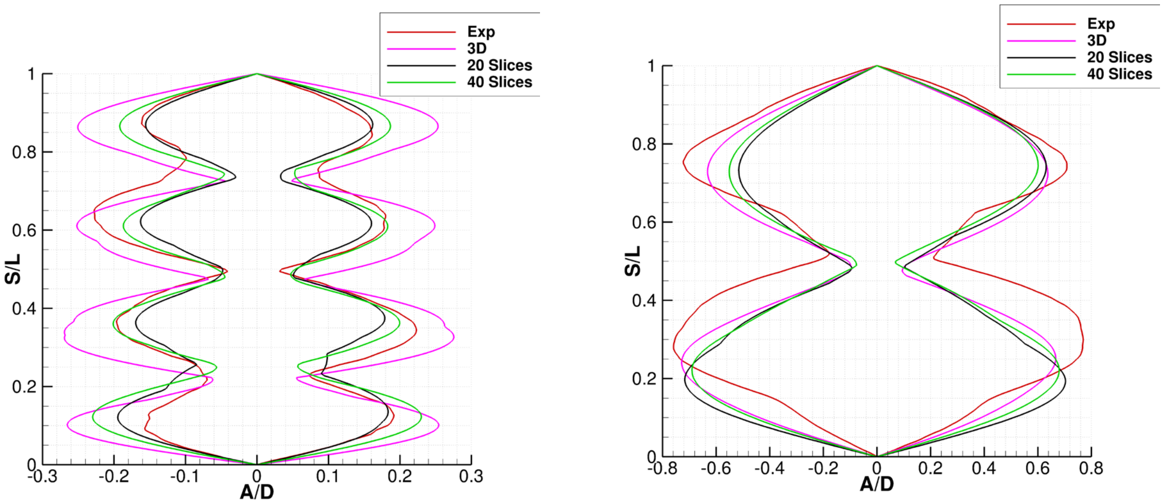


Figure 5: Mean (IL) displacement(A/D)

Regarding the amplitudes of the in-line oscillations, the fully 3D model overestimates the envelopes but properly predicts the position of the anti-nodes and nodes. Regardless of the number of slices, a better match with the experiment is observed for the slice method. The envelopes of the cross-flow displacement are underestimated by both methods, but are slightly weaker for the strip method. The position of the nodes and anti-nodes is correct regardless of the method, (see Figure 6a and 6b).

Table 4: Comparison of the mean in-line direction results

Method	Maximum displacement (D)	Z-Position (m)	Z-Position / L
Experiment	1.102	4.422	33.7%
3D	1.009	4.048	30.9%
20 Slices	0.857	4.373	33.3%
40 Slices	0.889	4.155	31.7%



(a) (IL) envelopes, no mean disp. (A/D)

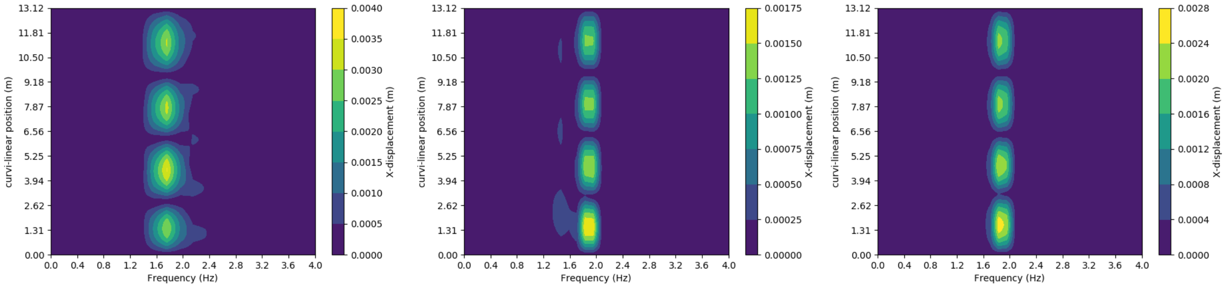
(b) (CF) envelopes (A/D)

Figure 6: Displacement envelopes

6.1 Spectral analysis

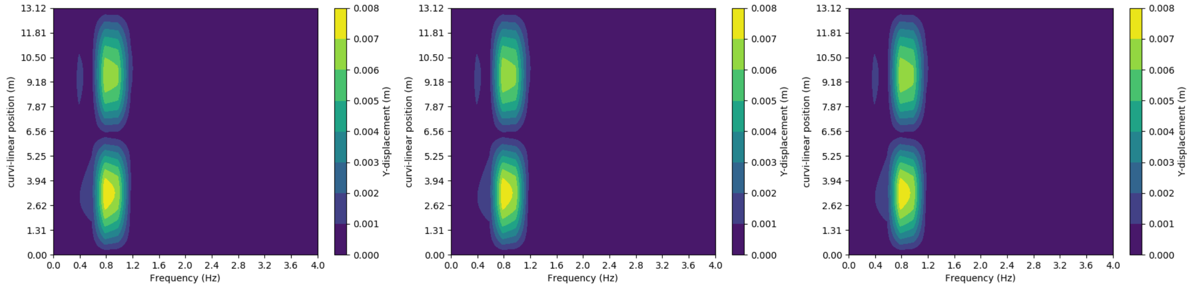
Examining the frequency content by spectral analysis, the dominant mode for the in-line (IL) direction of both methods is the 4^{th} mode. Although the corresponding frequency of the 4^{th} mode is similar, the range of frequency for the 3D method (1.4 to 2.1 Hz) is a bit wider than that of the strip method (1.7 to 2.1 Hz). The anti-node showing the maximum amplitude corresponds to the second lowest anti-node, according to the 3D method, and to the bottom most, for the strip method for both, 20 and 40 slices.

Both methods indicate that the dominant mode for the cross-flow (CF) direction corresponds to the 2^{nd} mode. As with the in-line spectral analysis, the range of the dominant mode is somewhat larger for the 3D method (0.3 Hz for the strip method, and 0.5 Hz for the 3D method). Both methods point to the bottom anti-node as the one with maximum displacement.



(a) Fully 3D (b) 20 slices (c) 40 slices

Figure 7: Spectral analysis, (IL) displacement



(a) Fully 3D (b) 20 slices (c) 40 slices

Figure 8: Spectral analysis, (CF) displacement

6.2 Simulation time

Computational time has been measured for both methods and, as shown in Table 5, a gain has been observed with the strip method. Considering the total of the CPU hours, and compared with the full 3D method, the strip method using 20 slices is 18.6 times faster, whereas the one using 40 slices is 10.4 times faster than the 3D computation. To give an example, 9 hours are required to get a result comparable to a 3D simulation using a desktop computer, with 20 cores and 20 slices.

Table 5: Comparison of the computational cost

	3D	20 slices	40 slices
Simulation time for the same number of time steps (hours)	51	9	16
Number of processors	64	20	20
Total CPU hours	3262	176	313

6.3 Comparison with others tools

This section presents a comparison between the proposed strip method developed within the **K-FSI** suite and the results of the tools assessed in the work of [7]. These tools are: USP, Norsk Hydro, Orcaflex vortex tracking, VIVANA, VIVIC, VICoMo, ABAVIV, and SHEAR7. The empirical and wake oscillator tools VIVANA, SHEAR7, VICoMo and ABAVIV were not developed to determine the in-line deformation of a riser.

6.3.1 In-line (IL)

All of the tools underestimate the in-line deformation, but the position of the maximum in-line deformation is globally well captured. In general, the methods developed in the present work stand out from the other ones. In particular the mean in-line displacement and the in-line envelopes are more accurately captured. Only the two **K-FSI** approaches and Orcaflex predicts nodes and anti-nodes, though only the two **K-FSI** results presented here succeed to determine the right number of anti-nodes and the proper dominant mode. In contrast, Orcaflex incorrectly predicted a fifth mode response.

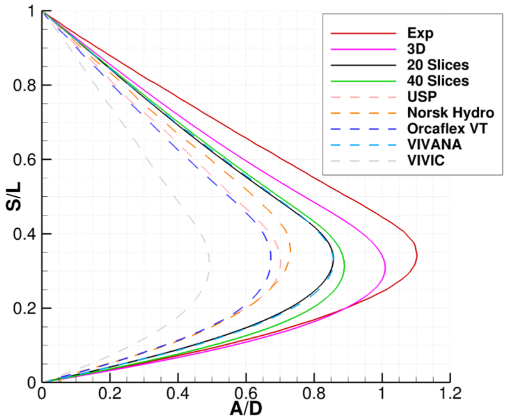
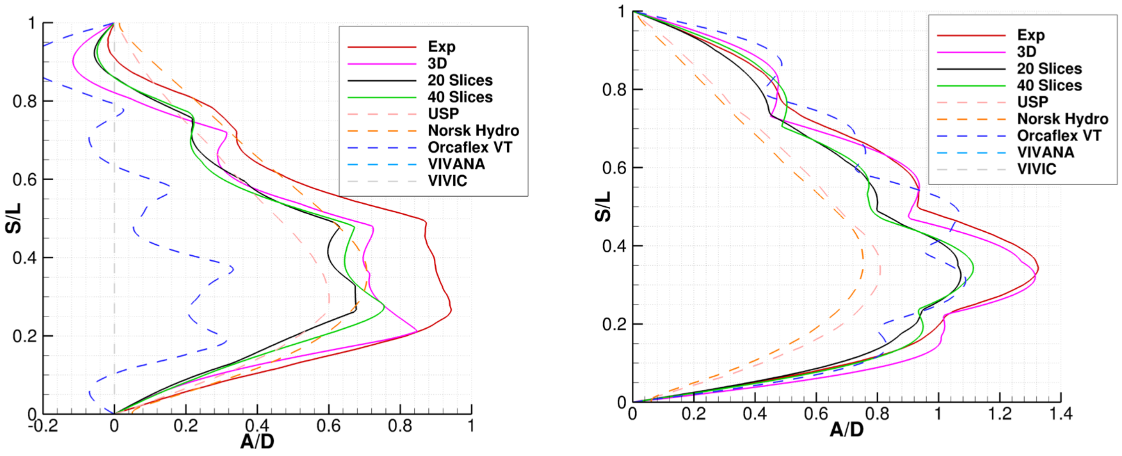


Figure 9: Comparison mean displacement (IL) direction of the different tools



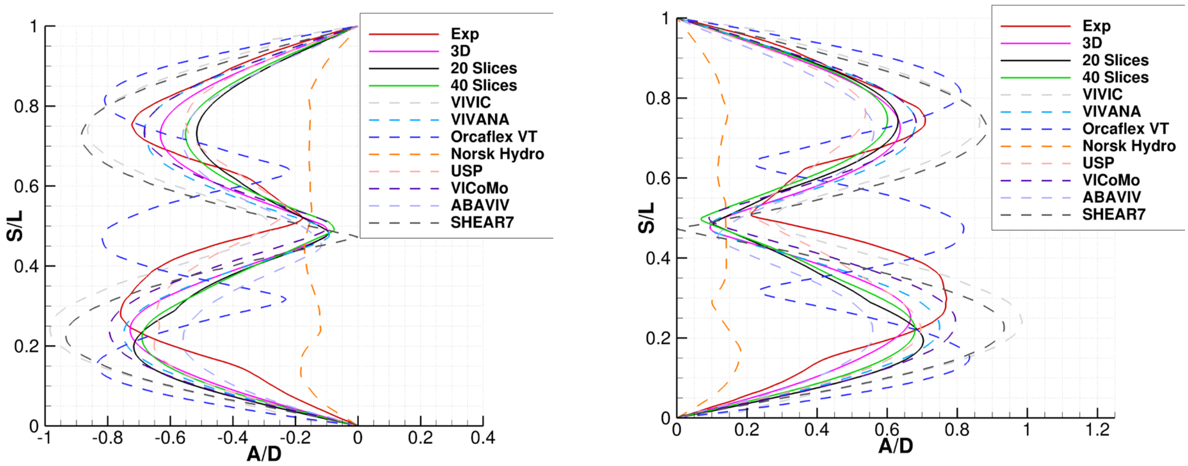
(a) Minimum displacement envelopes

(b) Maximum displacement envelopes

Figure 10: Comparison envelopes (IL) direction of the different tools

6.3.2 Cross-flow (CF)

The majority of the softwares gives the right mode for the cross-flow envelopes with reasonably correct amplitudes. Nevertheless, some are not suited to predict the CF displacement (Orcaflex : wrong mode, Norks Hydro : wrong mode and wrong amplitude).



(a) Minimum displacement envelopes

(b) Maximum displacement envelopes

Figure 11: Comparison of the envelopes for the different tools, (CF) direction

7 CONCLUSIONS AND FUTURE WORK

The results of the present study are compared with benchmark presented in Chaplin [6] and lead to the following conclusions:

- It emerges from all these simulations that the strip method offers results comparable to those of the 3D method while offering a significant time saving.
- In general, the strip method leads to a good agreement with the experimental data, and increasing the number of slices has been shown to yield a gain in precision though at the cost of greater computation time.
- Compared with some of the more recognized tools used in the oil industry, the **K-FSI** tool presented here yielded the closest results to the experiments.
- Both the fully 3D FSI and the strip approach developed were found to be reliable, robust and precise methods.
- The proposed approach makes it possible to perform an FSI simulation on a riser in a few hours, on a desktop machine dating from 2013.

Given the fact that past simulations proved to be incapable of predicting the deformations and frequencies associated with the VIV on a riser, the obtained results are very encouraging. Considering the possibility of making a specific solver for the VIV analysis, and the capacity of new processors, a drastic reduction in computational time can be expected. The developed strip method could be extended to handle more complicated geometry configurations such as spools and tandem riser configurations where the wake of one cylinder influences the behavior of the other. Examination of the case of the experimentally measured spool of [8] is the focus of ongoing work.

REFERENCES

[1] Le Cunff, C., Biolley, F., Fontaine, E., Etienne, S., & Facchinetti, M. L. (2002). Vortex-induced vibrations of risers: theoretical, numerical and experimental investigation. Oil & Gas Science and Technology, 57(1), 59-69.

[2] Duan, M., Fu, B., & Wan, D. (2016). The effect of top tension on VIV model analysis of a vertical flexible riser. In Proceedings of the second conference of global Chinese scholars on hydrodynamics (pp. 11-14).

[3] Willden, R. H., & Graham, J. M. R. (2004). Multi-modal vortex-induced vibrations of a vertical riser pipe subject to a uniform current profile. European Journal of Mechanics-B/Fluids, 23(1), 209-218.

[4] Etienne, S., Biolley, F., Fontaine, E., Le Cunff, C., & Heurtier, J. M. (2001, January). Numerical simulations of vortex-induced vibrations of slender flexible offshore structures. In The Eleventh International Offshore and Polar Engineering Conference. International Society of Offshore and Polar Engineers.

[5] Norberg, C. (2001). Flow around a circular cylinder: aspects of fluctuating lift. Journal of Fluids and Structures, 15(3-4), 459-469.

[6] Chaplin, J. R., Bearman, P. W., Huarte, F. H., & Pattenden, R. J. (2005). Laboratory measurements of vortex-induced vibrations of a vertical tension riser in a stepped current. Journal of Fluids and Structures, 21(1), 3-24.

[7] Chaplin, J. R., Bearman, P. W., Cheng, Y., Fontaine, E., Graham, J. M. R., Herfjord, K., & Meneghini, J. R. (2005). Blind predictions of laboratory measurements of vortex-induced vibrations of a tension riser. Journal of Fluids and Structures, 21(1), 25-40.

[8] Gross, David and Roux, Yann and Rousse, Benjamin and Pétrié, François and Assier, Ludovic and Minguez, Matthieu (2018). Experimental and Numerical Study of Vortex Induced Vibrations on a Spool Model. ASME 2018 37th International Conference on Ocean, Offshore and Arctic Engineering, V002T08A029–V002T08A029.