Proceedings of the ASME 2011 30th International Conference on Ocean, Offshore and Arctic Engineering
OMAE2011
June 19-24, 2011, Rotterdam, The Netherlands

VIV Prediction of a Truss Spar Pull-Tube Array Using CFD

by **Yiannis Constantinides**, Chevron Energy Technology Company; Houston, TX, USA
Samuel Holmes, Red Wing Engineering, Inc.; Mountain View, CA, USA
Weiwei Yu, Chevron Energy Technology Company; Houston, TX, USA

Abstract

Complex flows through cylinder arrays, such as the case of pull-tubes located in the truss section of a truss spar, are very difficult to describe and analyze. It is especially difficult to predict and correct Vortex Induced Vibration (VIV) response using traditional tools that were developed to analyze single cylinder rather than arrays of cylinders. Computational Fluid Dynamics (CFD) offers the designer the ability to properly analyze these complex problems and increase the reliability of his design. In this study, a full scale truss spar with pull-tubes is modeled using CFD coupled with an FEA structural model of the pull-tubes for a fluid-structure interaction (FSI) computation. The VIV response of the pull-tubes is predicted and analyzed for different current headings and speeds.

Introduction

This paper addresses the problem of predicting vibration and hence fatigue damage in the tube arrays present underneath many offshore platforms. The potential damage arises from vortex induced vibration in the tube array which is difficult to predict because of the geometric complexity and irregularity of the array. Furthermore, the current conditions can vary over time and current speed and direction can vary with depth. The later condition is called a "sheared" current. Because of the many parameters one cannot expect to predict VIV in the array using existing experimental data for individual tubes or groups of tubes. This paper is an extension of a prior study [Reference 1] in which we examine the flow and VIV around the pull tubes in the upper bay of a truss spar. This earlier work showed the importance of wake effects on pull tube response.

In the current problem, 20 straked pull tubes ranging in diameter from 16 to 25 inches are in close proximity underneath a truss spar. The general arrangement is shown in Figure 1. The pull tubes originate about 10m above the water line and pass though the flooded centerwell in the platform (not shown) and extend down to below the soft tank. Each pull tube is about 170m long and is supported at several locations along its length.

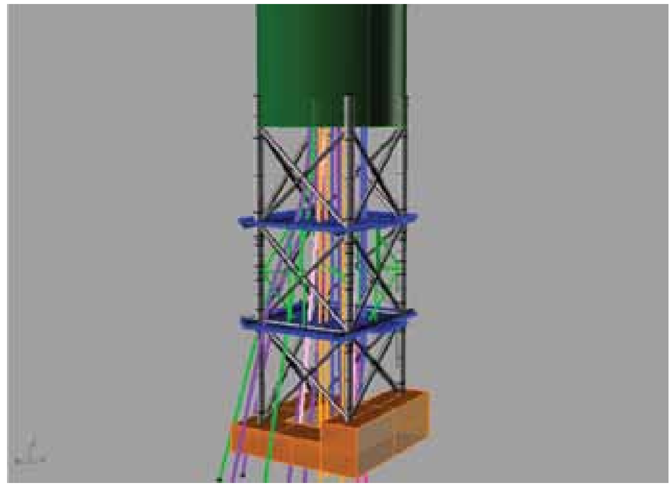


Figure 1 - Underside of truss spar showing truss and pull tubes heave plates, soft tank (bottom) and hard tank (top).

In order to predict the pull tube motion it is necessary to model the entire structure of each pull tube and its supports as well as the fluid flow along the pull tube. Although the pull tubes are assumed independent structurally, wake effects in the fluid in the array can result in complex interactions in the flow. With this in mind, the fluid volume simulated here includes the volume around and under the hard tank and well below the soft tank.

Solution Method

In an earlier work [1] we studied the response of the pull tubes by modeling the fluid flow in the uppermost bay in a truss spar. In this earlier study we assumed that the pull tubes are fixed at the bottom of the hard tank and at the top of the uppermost heave plate and we modeled only the structure and fluid between these two elevations. We also did not examine the effect of sheared currents. These assumptions limited problem size but of course the assumption of perfectly fixed supports for the pull tubes is not completely satisfying or correct. In the current study we treat the pull tube supports more accurately and model the full length of the pull tube. We also calculate the structural properties of the pull tubes (the eigenvalue problem) using finite element methods accurately modeling pull tube geometry and supports, etc.

All of the solutions were obtained using the commercial CFD code AcuSolve™. AcuSolve is a finite element based code which is second order accurate in space and time and supports a variety of turbulence models. In this study, turbulence effects were modeled using Spalart's detached eddy simulation (DES) which is described in References 2 and 3.

For economy, wall functions are used in the boundary layer so that the flow is not resolved down to the wall. Here, the CFD code is used to solve both the fluid flow (Navier-Stokes equations) and the structural response equations. However, the structural model is reduced to a simple linear vibration model so this task requires little computational effort.

Structural model - The solution of the pull tube structural response is obtained using the flexible body option in AcuSolve™. In this method, the problem of riser motion is solved by finding the eigenvalues and eigenvectors associated with the pull tube alone and thus characterizing the pull tube motion as a simple linear vibrating structure. Thus the pull tube motion is described by a series of eigenvectors which are translated into mesh motions. In this case, we found the eigenvalues and eigenvectors for each pull tube prior to starting the FSI problem using the finite element code Abaqus™ using beam elements to describe each tube. About 500 elements are used along the pull tube length with the beam node locations on the pull tube centerline. Using the Abaqus output at the beam nodes the displacements on the pull tube surface and hence the displacements associated with each mode in the AcuSolve solution (S_i^n) are found though interpolation along the pull tube length.

With this approach, the motion of the riser is assumed to be a linear summation of the various modes. The response of each of *n* modes is found by solving the equation:

$$\left[m_i^n\right]\{\ddot{\xi}_n\}+\left[K_i^n\right]\{\xi_n\}=f_n \qquad [1]$$

In each time step, the surface tractions on the riser are projected onto the eigenvectors to find the values f_i :

$$f_i=\int_A T(x,y,z)S_i^n(z)dA \qquad [2]$$

The resulting f_i are then used with Equation [1] to find the displacements for the next time step using the trapezoidal rule to integrate. Note that AcuSolve iterates between the fluid and structure solutions within the time step to improve accuracy. The mesh motion required to accommodate the changes in riser geometry were accomplished by using an Arbitrary Langrangian-Eulerian (ALE) mesh movement strategy.

In the case studied here, each pull tube is supported at about 10 positions along its length. Using the finite element structures code in a separate calculation we found the mode shapes and associated mass for a range of modes in each pull tube. Figure 2 shows example shapes for a typical pull tube. The eigenvalue analysis typically predicted that the lowest modal frequency of vibration for any tube in the array is on the order of 0.7 Hz. Fundamental vortex shedding frequencies for the ocean current conditions modeled thus far are from 0.5Hz to 0.7 Hz maximum, so we expect that the lowest modes of response of the tubes will be the most important. Most of the FSI simulations are thus run using a structural model comprised of the lowestthsix modes of response. The frequency associated with the 6 tubes. mode varies from 1.8Hz to 2.6Hz for all the pull tubes.

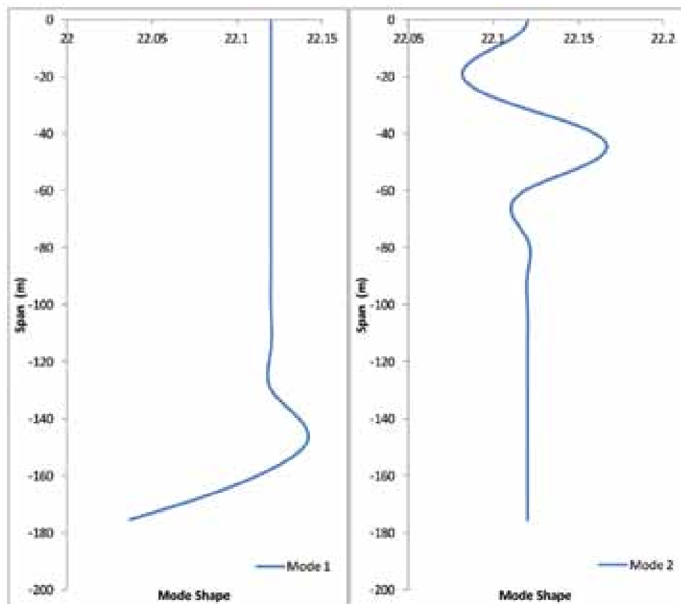


Figure 2. Example of pull tube with associated mode shapes corresponding to 0.7Hz for mode1 and 1.5Hz for mode 2.

Pull Tube R1 – Here we use pull tube R1 to illustrate the method and results for all the pull tubes. ABAQUS V6.9-1 [4] is used to obtain the natural frequency and mode shape for this pull tube. The origin of the coordinate system used is at the center of the spar at the mean water line. The initial configuration of the pull-tube lies on the ABAQUS X-Z plane. For pull tube R1, The pull-tube assumed heading towards 180° counter-clockwise from ABAQUS positive X-axis. The pull tube is 0.244m in diameter and 175m long. For modal analysis, two-node linear beam elements with a hybrid formulation is applied to model the pull tube using 500 elements along the pull-tube.

The pull tube is supported by decks and other supports at nine elevations. These supports are modeled in the FEA using guide nodes defined at contact points between pull-tube and the supports. The support from decks is modeled by defining spring elements connecting the guide nodes and the corresponding pull-tube node. The spring stiffness of the supports is on the order of 10^8 N/m in both X and Y directions so that the supports are stiff in terms of displacements. As designed, the vertical stiffness and three rotational stiffnesses at the supports are small and are assumed to be zero here. In order to correct for the effects of any installed changes in the pull tube geometry, a static solution with gravity, buoyancy and riser lateral loads is found first and then the frequency analysis is performed to extract the first 6 modes.

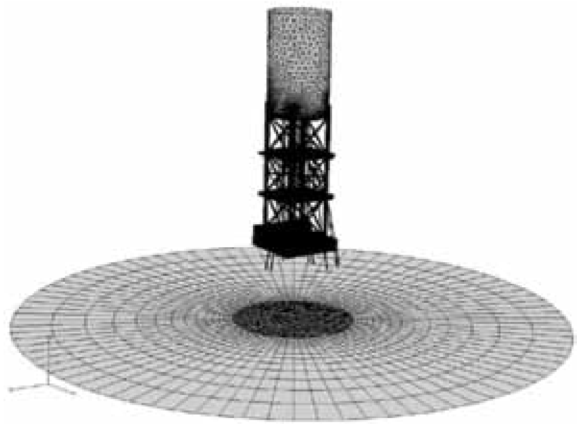
Fluid model – The overall fluids mesh is a circular cylinder as shown in Figure 3a. The purpose of this shape is to facilitate changes in current heading which can be changed with a simple command without any mesh changes. The mesh models a section of the ocean 364m in diameter by 200m deep. The mesh is made up primarily of tetrahedral elements but the mesh in the boundary layer on the spar platform is made of wedge elements. Mesh density is kept high in the region of the truss so that vortex shedding is modeled regardless of current direction. Also, the outer region of the mesh, far from the platform is made of wedge elements. These elements are slender and aligned horizontally to facilitate the propagation of sheared currents while keeping the problem size small. Table 1 gives the nodes and elements counts for the mesh.

Nodes	18.7M
Tet elements	70.0M
BL Wedges	12.6M
Outer sea wedges	278k

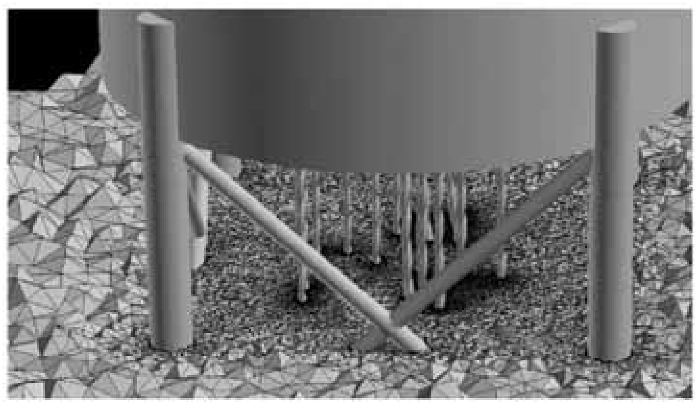
Table 1. Node and element count for mesh.

The boundary layer on the pull tubes and truss elements is made up of wedge elements. The first element thickness in the critical wedge elements is 4mm which gave a range of y+ from 10 to 50 on these surfaces for most problems. No attempt was made to model the boundary layer on the hard tank or the soft tank in reduce problem size.

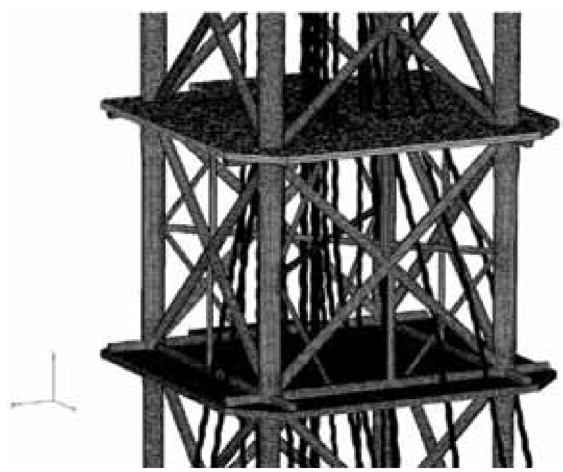
The time step for the calculations is constant and is chosen to resolve the fluid flow as accurately as possible. For the meshes used here, the typical time step used is 0.05s so about 60 time steps are used in a vortex shedding cycle. Run time for most problems was from 24 hours to 48 hours using 188 cores on a 48 node cluster with infiniband connectivity.



(a)



(b)



(c)

Figure 3. Fluids mesh.

Figure 4 shows velocity vectors on a cut plant in the truss area under a typical flow condition. The velocity vectors are colored with velocity magnitude. This picture illustrates the close proximity of the pull tubes and the potential for strong wake effects for downstream tubes. Also note that the each vector is associated with a single element so the vectors give and indication of mesh density. Note that the mesh around the pull tubes is isotropic so that the wakes of the tubes can be captured regardless of current direction.

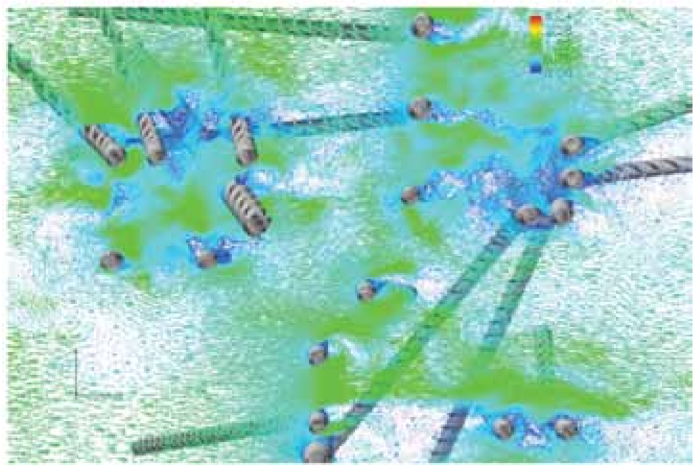
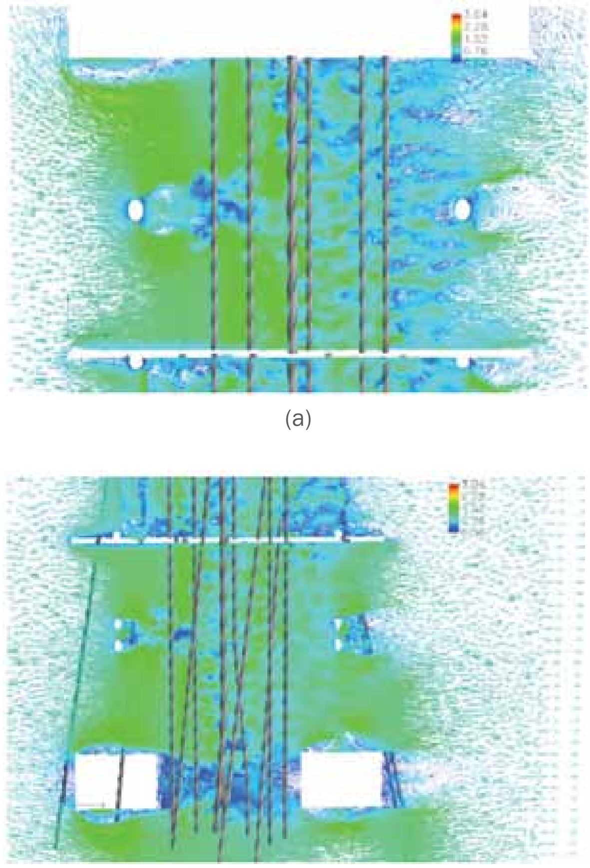


Figure 4. Velocity vectors around pull tubes with contours of velocity magnitude.

CFD Results

Flow visualization

The computed flow velocities were visualized in an effort to understand the flow through the riser array. Figure 5 shows the vectors of a vertical slice along the centerline colored by the velocity magnitude. The flow accelerates as it flows under the spar resulting in a 30% increase in mean velocity. The X truss member cross section generates a strong wake at different locations in the bay. The horizontal members that support the heave plates, shown in the cross-section, also generate strong wakes. The velocity slice shown here has considerable wake influence and unsteady flow, especially at locations behind vertical members such the truss members and pull tubes. The bottom ballast tank, identified by two rectangles, has a typical bluff body flow with complex separation and recirculation zones.



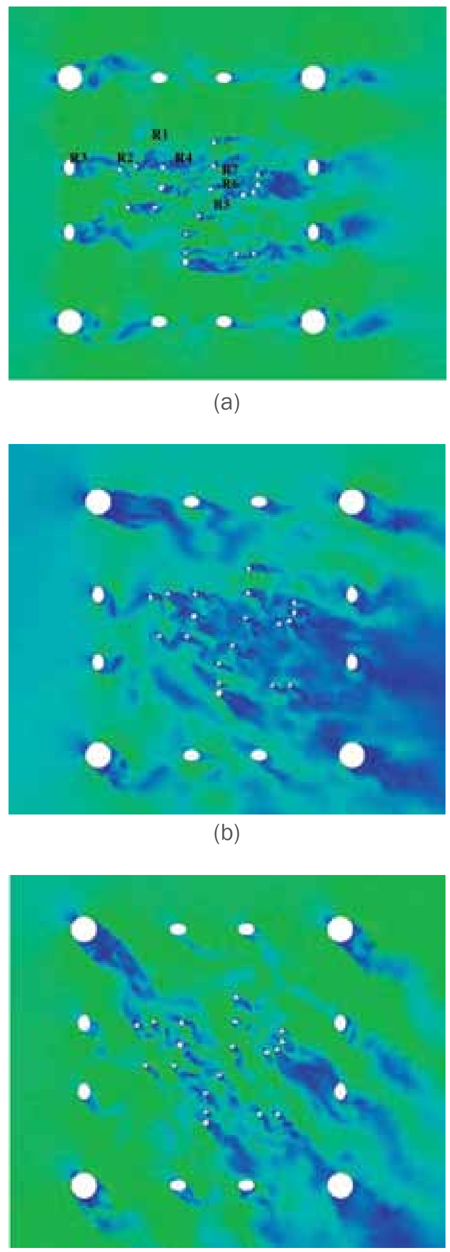
(a)

(b)

Figure 5. Velocity vectors colored by magnitude of vertical slice at the centerline of the domain.

The horizontal cross-section shown in Figure 6 is located near the top of the uppermost bay and shows contours of the total velocity magnitude at 3 current headings: 0, 24 and 45 degrees. The free stream velocity is uniform with a magnitude of 1.5 m/s. The upstream truss members, either vertical or X shape, influence the flow causing speedup to the fluid between them. The flow pattern and influence of the wake to the risers is different in the three headings. For example, R1 is influenced significantly by the wake of the top left truss member at 24 degrees heading (Figure 6b). Pull tubes R2, R3 are influenced at a 45 degree heading (Figure 6c) by truss member that is directly upstream. Looking at the interaction between the pull tubes, it is obvious that the large eddies in their wakes affect any pull tube located directly downstream. This dynamic wake can cause buffeting and dynamic forces on the downstream tubes that will lead to increased response. As a result, a downstream pull-tube with strakes can vibrate even though the strakes would ordinarily mitigate VIV effectively.

Other interaction effects are shown in Figure 6a for a heading of 0 degrees. Here R3 is shown to influence R2 and R4. The cluster of R5, R6 and R7 is also influenced by upstream pull tubes as well by each other's wake as they are located in close proximity. The study of the 3 headings suggests that the response of individual pull tubes will be different for each heading as the wake and flow structure will change significantly. No pull tube was found to operate independent of any wake effect from other structures.



(a)

(b)

(c)

Figure 6. Flow field (velocity magnitude) at 0, 24 and 45 degree heading at the top bay.

VIV response evaluation

A number of simulations were performed allowing the pull tubes to freely vibrate in the presence of a uniform current. The current speeds were varied from 1 m/s to 2.0 m/s to represent typical extreme current events in the Gulf of Mexico. Because the pull tubes extend to a depth of about 160m these current speeds are close to extreme loop current events.

Returning to tube R1 as an example, we see that the dynamic response of R1 along the length, shown in Figure 7, is similar to the shape of mode 1 shown in Figure 2. Thus R1 response is dominated by mode 1 in the range of velocities of interest. Looking at the response of pull tubes R1 through R7 in Figure 8 for 0 degree heading, each pull tube has a different maximum amplitude of response (along its length). The response amplitude varies from 0.02 to 0.12 standard deviations (std) of normalized response amplitude A/D. The response difference can be explained by the location of the pull tubes with respect to other tubes and truss members with the upstream cluster R1- R3 having less VIV than the downstream cluster R4-R7.

We also found that differences in structural properties between the pull tubes is another important factor in determining response in this example because the various pull tubes have different natural frequencies and hence lock-in velocities. Finally, we compared the simulation results with predicted VIV amplitudes for bare and straked cylinders under similar flow conditions using simple formulas for VIV based on single cylinder data. In this comparison the simple models predicted greater VIV amplitudes for bare tubes but lower amplitudes for straked tubes. It would seem that estimates based on single tube models without interaction effects are not useful for predicted VIV this problem.

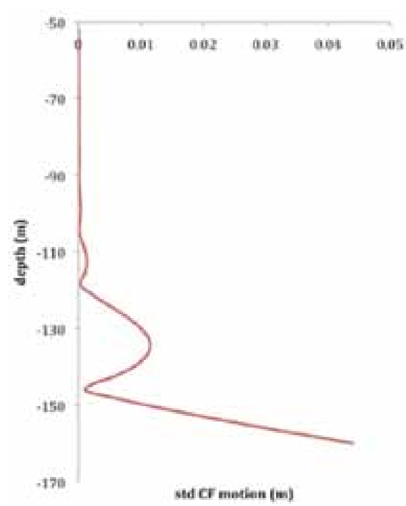


Figure 7. Dynamic response of R1 along span.

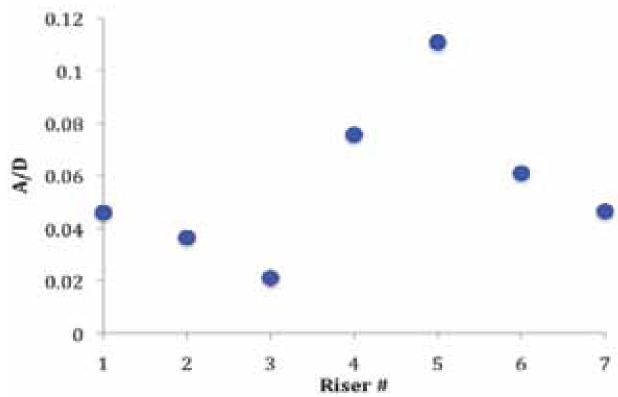


Figure 8. Crossflow std crossflow motion of selected pull- tubes at 0 degree heading.

The variation of the VIV response with current speed for 0 degree heading is shown in Figure 9. For the tubes R1 and R5 the cross-flow response increases with the velocity and reaches a maximum at a value of 0.25 for R5 and 0.1 for R1 at the maximum velocity. With resonance frequencies in the range of 0.5Hz only mode 1 is excited with a natural frequency of 0.7 Hz for R1. The frequency content of the hydrodynamic pressure appears to be broad band due to the unsteady incoming flow caused by the wake of upstream tubes.

The response is affected by the current heading which results in changes in the upstream geometry for any particular pull tube. Figure 10 shows the response of R1 in 0, 24 and 45 degrees of 1.5 m/s current. From Figure 6a we see that for a 0 degree heading R1 does not sit directly in the wake of any large upstream members. While at 24 degrees the vertical truss member is directly in front of R1. The strong wake from the truss results in higher VIV amplitudes for R1 at this heading.

Figure 11 compares the time history of displacement for R1 and R5 for a 1.5 m/s current at 0 degree heading. Because R5 is downstream a number of pull tubes, its response is significantly higher than that of R1. Also the time history is complex as might be expected in the wake of a straked cylinder.

Overall, it is a big challenge to understand the response of pull tubes physics since the flow is complex and there are multiple sources of wake excitation (truss members, other tubes) that will affect an individual tube. CFD can model all these effects and provide results that will reduce the number of assumptions one will need and refine the design. The only way, based on our understanding, to accomplish this design with existing empirical tools is to build in sufficient conservatism in the design. Even then it is impossible to determine the proper inflow conditions and strake efficiencies without extensive experimental data including tube bundle experiments.

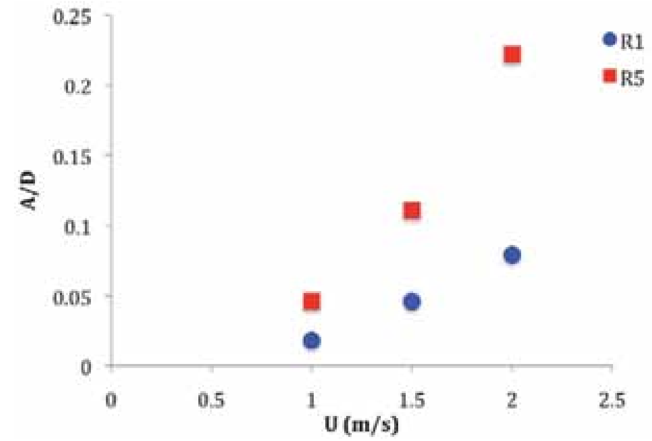


Figure 9. Crossflow std motion at different current speeds for 0 degree heading.

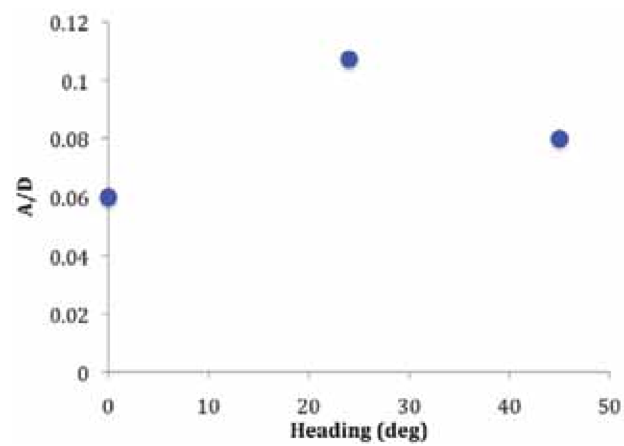


Figure 10. Combined std response magnitude of R1 as a function of heading.

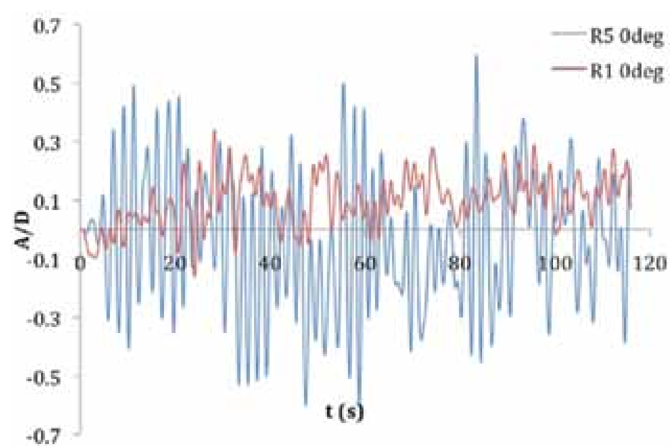


Figure 11. Crossflow timeseries comparison between R1 and R5 for 1.5 m/s.

Conclusions

CFD was successfully used to evaluate the response of a full scale truss spar pull tube array using exact design parameters. The truss spar and pull tubes were modeled with adequate detail including FEA structural models for the pull tubes. The model is easily handled by a medium size cluster and requires 24 to 48 hours to run.

Flow visualization revealed a complex flow in the tube array with increase flow velocity under the spar hard tank. The truss members as well as the pull tubes create complex wakes that interact and affect the downstream pull tubes causing vibrations due to wake buffeting. The pull tube response was found to vary from pull tubes to pull tube due to different properties and location. Current heading is identified as an important factor with the worse case for any particular pull tube occurring when it is downstream other tube and truss elements. It appears that CFD offers the best hope to predict these complex wake effects and predict VIV for this type of problem.

Nomenclature

a = motion amplitude, [m]
A = area, [m2]
Cd = drag coefficient based, [-]
Cl = lift coefficient, [-]
D = riser diameter, [m]
f_i = modal force, [N]
m_i = system mass, [kg]
n = modenumber[-]
K_i^n = system stiffness [N/m]
T = period of oscillation, [s]
Tn = system natural period of oscillation, [s]
T = Surface traction [Pa]
V = current velocity, [m/s]
Vrn = nominal reduced velocity, V Tn/D, [-]
e_i = unit vector [-]
ξ_i = modal amplitude [m]
std = standard deviation

References

[1] Constantinides Y., Oakley H. O. Jr. and Holmes, S. “Analysis of Turbulent Flows and VIV in Truss Spar Risers”, OMAE-92674 (2006).

[2] Holmes S., Oakley H. O. Jr. and Constantinides Y., “Simulation of Riser VIV using Fully Three Dimensional CFD Simulations,” OMAE-92124 (2006).

[3] Constantinides Y., Oakley H. O. Jr. “Numerical Prediction of Bare and Straked Cylinder VIV”, OMAE-92334 (2006).

[4] ABAQUS/Standard Users Manual, V. . Hibbitt, Karlsson and Sorensen, Pawtucket, RI 12860, Inc.