Proceedings of the 26th International Conference on Offshore Mechanics and Arctic Engineering
OMAE2007
June 10-15, 2007, San Diego, California, USA

OMAE2007-29151

CFD HIGH L/D RISER MODELING STUDY

Yiannis Constantinides
Chevron Energy Technology Company

Owen H. Oakley, Jr.
Chevron Energy Technology Company

Samuel Holmes
Red Wing Engineering, Inc.

ABSTRACT

Fully three dimensional fluid flow simulations are used with a simple structural model to simulate very long risers. This method overcomes many shortcomings of methodologies based on two dimensional flow simulations and can correctly include the effects of three dimensional structures such as strakes, buoyancy modules and catenary riser shapes. The method is benchmarked against laboratory and offshore experiments with model risers of length to diameter ratios up to 4,000. RMS values of vortex induced vibration motions are shown to be in good agreement with measurements. The resources needed to model ultra deep water drilling and production risers are estimated based on current computer technology.

INTRODUCTION

In a prior paper we described an analysis of the vortex induced vibration (VIV) of a very long riser using fully three dimensional fluid flow simulations with a simple structural model [1]. This paper reports a continuation of this work with an emphasis on improved accuracy and reliability. In particular we have improved meshing strategies and mesh motion controls to solve problems with greater accuracy while modeling riser lengths up to four thousand diameters. These methods are extensively benchmarked against laboratory experiments covering a range of riser configurations and current conditions. Finally, an important objective here is to bring this technology to the point where it can become a general design tool.

BACKGROUND

A fundamental problem in the simulation of long marine risers is the scale of the computational fluid dynamics problem itself. The risers may have lengths of thousands of meters so that using three dimensional CFD solutions seems impossible. A practical approach to riser VIV predictions that has been proposed is to combine a series of two dimensional fluid flow solutions along the riser axis. These “strips” are pieced together with a structural model of the riser to obtain a prediction of the fluid-structure response. This method reduces the large three dimensional CFD problem to a large number of small two dimensional problems. However, this “strip” method has some serious shortcomings. In particular, the flow around bluff bodies is inherently three dimensional so the two dimensional strip solutions can only be expected to give approximate answers. In addition, the riser may be at a steep angle of attack to the flow as is often the case with steel catenary risers (SCRs) creating strong axial flow components. Also, VIV suppression devices like helical strakes that have a very strong three dimensional wake cannot be modeled. Finally, the strip method requires that some kind of interpolation method be used to estimate the forces between the strips. There is no general rule available to make such interpolations.

In addition to the sheer scale of the CFD problem, several other difficult problems remain. Although the structural response of the riser usually does not involve nonlinear material response, it usually includes nonlinear geometric effects such as large displacements and rotations. The presence of these effects means that a nonlinear structural

Copyright © 2007 by ASME

model must eventually be incorporated in the solution of riser problems and also that large mesh motions must somehow be accommodated in the fluid flow solution. Finally, it cannot be expected that the flow environment remains constant over the entire length of the riser. Both flow speed and direction may change. Any general solution must be able to treat these effects. As discussed earlier, we use a simple linear structural model for the solutions described here. However, we do show solutions that include the effect of changes in current speed and direction with depth and discuss how nonlinear structural models can be made part of this method.

In the following sections we describe the general method of solution combining 3D fluid flow solutions with a structural model. The problem of creating an optimal mesh is discussed in which the objective is to combine acceptable solution accuracy with good solution economy. We then describe benchmark calculations comparing the predicted response of short riser sections with laboratory experiments. Finally, several simulations of long scale model risers are compared with the recent laboratory experiments on small scale risers described in [9] and recent field experiments on large scale model risers [15].

BENCHMARK CASES

NDP experiments

The first set of experimental data used for benchmark consisted of the L/D=1407 riser model tested by the Norwegian Deepwater Programme (NDP) and reported in [9]. The experiment was carefully designed and instrumented to provide quality benchmark data. It was performed in a controlled laboratory environment A 38m length and 0.027m diameter riser was towed horizontally in a model basin. The riser ends were towed with the same speed to simulate a uniform current profile and in a circular pattern with one end being fixed to simulate a linear shear profile. A variety of configurations were tested including bare and straked riser configurations at a subcritical Reynolds number (<100,000). The riser was instrumented with both accelerometers and strain gauges. For the present study the bare riser in uniform and linear shear flow was selected as the benchmark case.

DeepStar experiments

The second set of benchmarks consists of a vertically towed riser having an L/D of 4137, conducted as part of the DeepStar JIP [15]. A 147m length and 0.0356m diameter riser was suspended from a boat with an 800lb weight at the bottom end. The riser was towed in the Gulf Stream offshore Miami in sheared directional currents. The Reynolds number for this case was also subcritical. From this dataset the 40% straked riser configuration was selected.

APPROACH

All of the solutions shown here were produced using the AcuSolve™ finite element Navier-Stokes solver. AcuSolve is based on the Galerkin/Least-Squares formulation and supports a variety of element types. AcuSolve uses a fully coupled pressure/velocity iterative solver plus a generalized alpha method as a semi-discrete time stepping algorithm. AcuSolve is second order accurate in space and time for static solutions. Turbulence is modeled using either the Spalart-Allmaras Reynolds averaged Navier-Stokes turbulence model or the related Spalart detached eddy simulation model [14]. In either case, we used wall functions to describe the flow at the wall in all CFD simulations. This was done to economize on mesh size, but also because most risers have relatively rough walls. Wall functions reduce mesh size by providing an integrated relationship between the wall and the logarithmic region of the boundary layer. A more detailed discussion of the turbulence model and the wall treatment is given in [1].

The problem of riser motion is solved using a simple linear vibration analysis so that the displacements from the riser mean position are assumed small. The displacement is characterized by normal modes of vibration found using an eigenvalue analysis. The displacements of the riser are then a linear sum of the modal amplitudes times the corresponding eigenvectors. In the problems solved here, the riser axis is oriented in the z-direction and we assume the eigenmodes to be sinusoidal so the eigenvectors have the form:

$$S_i^n(z) = \sin\left(\frac{n\pi z}{L}\right) \qquad \textbf{(1)}$$

where the S_i^n is the eigenvector associated with the nth mode and the i index indicates the x or y directions in this case, L is the riser length, z is the distance along the riser axis and n is the mode number. It should be noted that the risers modeled here are tension dominated in that the bending stiffness of the riser is negligible compared to the stiffness due to tension. Also, the use of a sine shape in [1] implies that the riser tension is constant along the riser length.

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

$$\left[m_i^n\right]\{\ddot{\xi}_i^n\} + \left[K_i^n\right]\{\xi_i^n\} = f_i^n \qquad \textbf{(2)}$$

Where ξ_i^n are the modal amplitudes, with m_i^n and K_i^n being the associated mass and stiffness of each mode. Here we chose not to model the material damping of the riser structure because it is unknown. The presumed amount of damping is less than 1% and did not affect the solution in testing here.

Copyright © 2007 by ASME

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

$$f_i^n = \int_A T_i(x,y,z) \cdot S_i^n(z) dA \qquad \textbf{(3)}$$

The resulting f_i^n are then used with Equation [2] to find the model amplitudes (ξ_i^n) and then the displacements for the next time step using the trapezoidal rule to integrate. Note that no iterations between the fluid and structure are made within the time step. The resulting scheme is stable as long as the density of the riser is equal to or greater than the fluid density. A different integration scheme must be used if the density of the riser is less than that of the fluid, water. Also, because the motion of the riser is calculated from the surface tractions from the fluid solution at the end of the previous time step, the solid and fluid calculations are essentially staggered by ½ of a time step.

The mesh motion required to accommodate the changes in riser geometry were accomplished by explicitly controlling the motion of each node in the mesh. Furthermore, the motion of the boundaries of the mesh is controlled to accommodate large riser displacements. Thus the mesh is bowed as part of the simulation as the mean position of the riser changes in order to avoid contact between the riser and the mesh outer boundaries. This allows the mesh to be kept close to the riser so that variations in current speed and direction and be economically incorporated into the solution.

Finally, 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 time step used is varied from 0.01s to 0.005s so about 100 time steps are used in a vortex shedding cycle in a typical calculation.

Mesh Design

The approach taken here assumes that the fluid flow over the riser must be described using a three dimensional flow solution. No attempt is made to compromise the solution using quasi-three dimensional methods or other simplifications. Obtaining an economical solution is then simply finding the minimum number of degrees of freedom in the flow solution that captures the salient fluid flow effects and predicts the loads on the riser with sufficient accuracy. This problem has three parts: the resolution of the boundary layer, the resolution of the near wake and the resolution of the far field. A variety of mesh solutions have been used here in an attempt to find the most economical mesh, but all of these have the same general appearance as the mesh shown in Figure 1. Note that this cross section of the mesh has a wide wake region with distinct zones that are finer near the riser and coarser in the far wake. Large elements are used near the boundaries. This mesh has two to three times the density (number of nodes) as used in our earlier study [1]. The actual relative problem size depends on the configuration and the relative number of straked and bare lengths of riser. In general, from 4,000 to 4,500 nodes are required for every diameter of riser length. Thus a riser with and L/D of 5,000 would require a problem size of about 20M nodes.

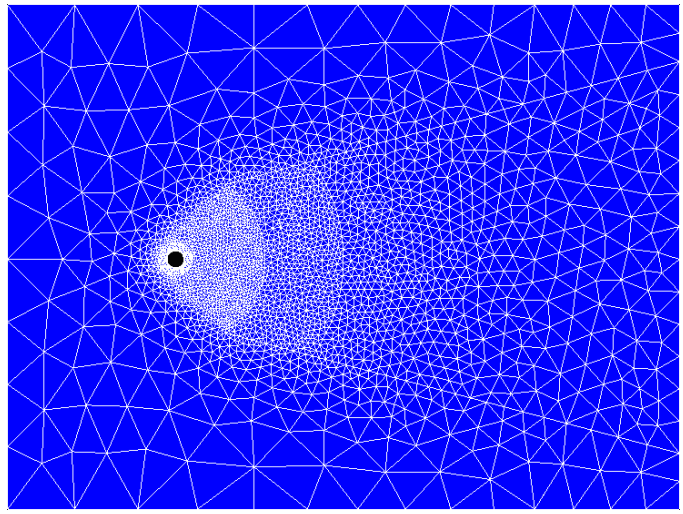

Figure 1 – Bare cylinder mesh.

Finally, Figure 2 gives some idea of the relative scales in the riser response problem. The figure shows an overall view of a towed riser model with an L/D of 4,137. This riser model will be discussed later in the paper in the discussion of the DeepStar field test experiments. The figure shows inlet and outlet surface of the mesh in green. The riser itself is not visible. Note that the mesh displacements are many times the length of the lateral boundaries of the mesh.

The specifications of the tow meshes constructed are listed in Table 1. It is important to note that we believe we used the minimum possible number of nodes to simulate the flow around the riser while still obtaining an acceptable engineering solution. This isn't meant to suggest that better solutions can't be obtained with a more refined mesh, but is rather intended to find the lower limits of problem size.

Mesh	Type	L(m)	L/D	Nodes	Elements
1	Bare	38	1407	3.1 M	10.9 M tet/prism
2	Strake (40%)	147	4137	10.0 M	32 M tet/prism

Table 1 – Summary of computational meshes used.

Copyright © 2007 by ASME

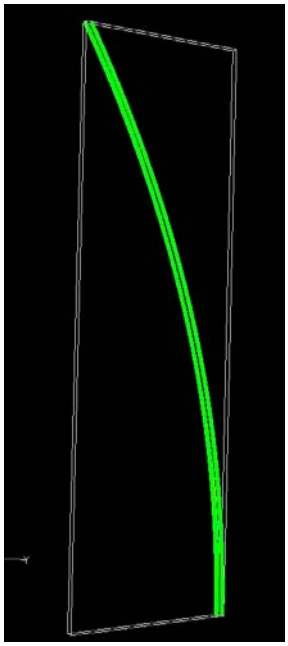

Figure 2 – Overall view of mesh used to simulate the towed L/D=4137 riser model. Note that green surfaces are the inlet and outlet surfaces of the CFD mesh. Also, the original mesh is straight so that the bowing of the riser and mesh is part of the simulation.

SIMULATION OF A MEDIUM L/D BARE RISER

Response prediction

The first set of simulations to validate the CFD model consisted of a medium aspect ratio riser case (L/D=1407). These include simulations of a bare riser in uniform and linear shear flow with zero current speed at one end. The basic features were modeled including riser properties and current speed. The towing rig's tensioning pendulum system was modeled as providing a constant tension. The experimental data consisted of several tow speeds ranging from 0.3 to 2.4 m/s. Although the data were not dense enough to capture potential reduction in response shifting from one mode to another, they were sufficient to identify the success of the model.

The results of the simulations were compared with the reported mean and maximum of the RMS (root mean square) value of the response along the riser. Since the instrumentation used was not dense enough to capture all the local maxima, a modal reconstruction method was used to derive the maximum response in the experiment, from the limited strain gauges and accelerometers. The maximum RMS is thus a derived and not an actual measured motion, inheriting the uncertainties of the method used to derive it. The mean value of the RMS along the riser was also used, since it is a more appropriate measure of the average response along the riser rather than a local maximum.

Figure 3 and Figure 4 compare the predicted and experimental response for uniform and linear shear current profiles. Overall, the mean RMS response along the riser is well predicted, within the uncertainties and error bounds of the experimental and computational model. Deviations from the maximum values can be attributed to the length of the simulation since the response was not stationary in either simulation or experiment, not achieving lock-in at low velocities and the accuracy of the CFD method. This is also discussed further in the response characterization section. CFD predicts a low amplitude response at the very low speeds and low modes, and the higher response at higher speeds. It also predicts a higher response for the uniform current cases than the linear shear cases, similar to the experiments. Overall the success at the higher speeds is important since these are the critical cases in riser design.

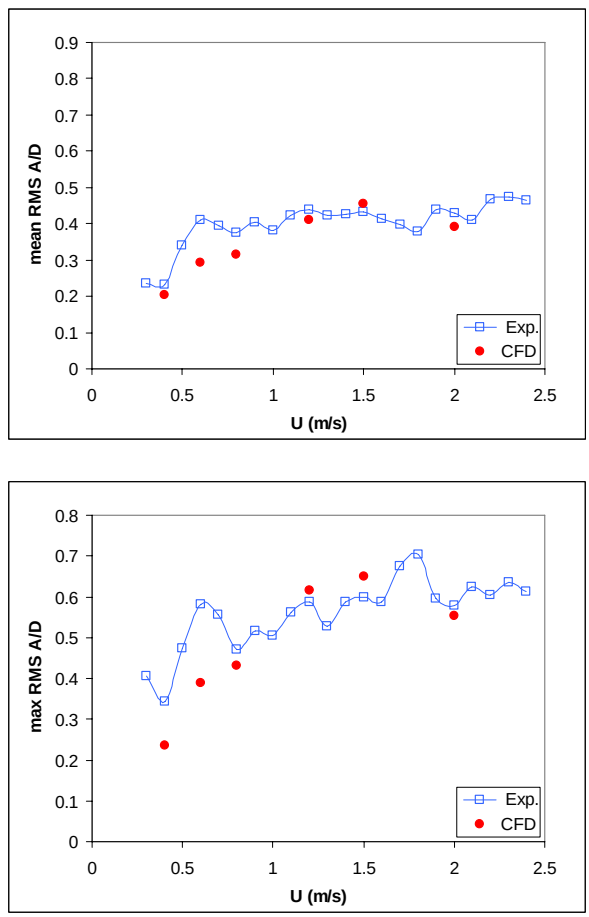

Figure 3 – Temporal mean and max RMS response along the riser for linear shear flow (L/D=1407).

Copyright © 2007 by ASME

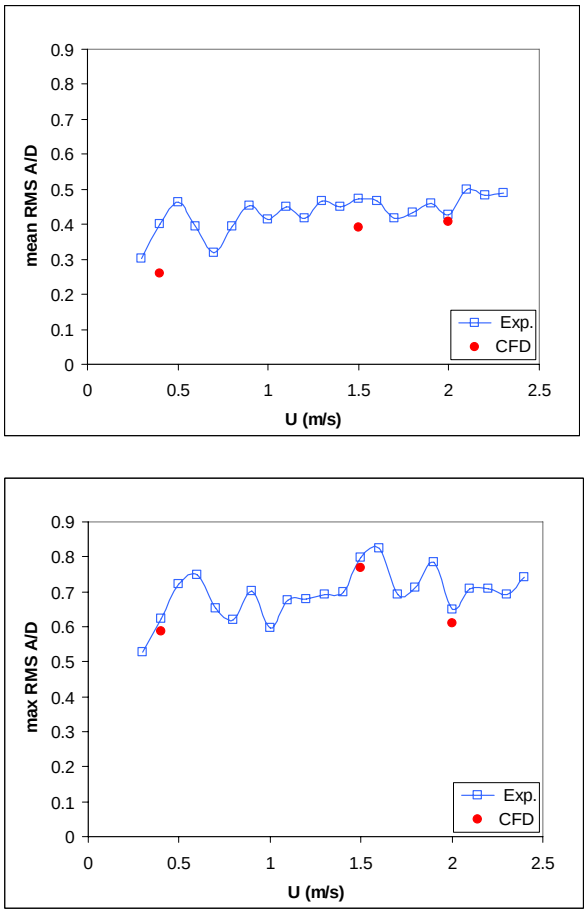

Figure 4 – Temporal mean and max RMS response uniform flow (L/D=1407).

Strain prediction

The overall mean and max RMS motion are not definitive indices of the success as the response profile over the riser. In this case we compare the predicted strain along the riser with the experimental values reported in [13]. Stress or strain is directly related to fatigue damage and it is a better quantity to compare from a fatigue prediction prospective. It is often more difficult to predict strain and acceleration since both motion and frequency have to be well predicted. In addition the contribution of high harmonics to the strain is significant and the computational model must be able to predict them in order not to underestimate strain. For the cases of 2.0 m/s linear shear and 2.0 m/s uniform flow the microstrain along the riser is plotted against the experiment Figure 5. The strain is in good agreement for both cases. In the case of linear shear a strong end-effect is present at the top in the form of a standing wave. If the simulation was carried out for more cycles, this end effect would have reduced in amplitude since it is non-stationary.

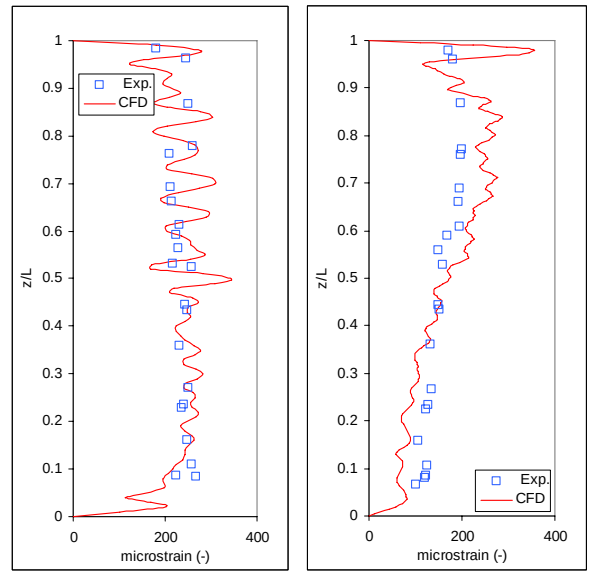

Figure 5 – Crossflow strain comparison for 2.0 m/s uniform flow (left) and 2.0 m/s shear flow (right).

Frequency content and high harmonics

We compared the predicted and measured reduced frequencies ($f = f_{obs} D / U$) based on the tow velocity. This gives 0.140 for uniform flow and 0.128 for sheared flow compared with 0.142 and 0.127 in the experiment for the 2 m/s case. Another important contributor to the strain and fatigue, especially for high mode long risers, is the high frequency content due to high harmonics [15]. The acceleration spectrum in Figure 6 reveals that the current method can capture these important effects. The amplitude is close to what is expected for a low mode, low modal density configuration like this. Strain spectra also contained similar amount of energy in the 3rd harmonic. To our knowledge this is the first numerical simulation of a flexible cylinder that high harmonics have been predicted

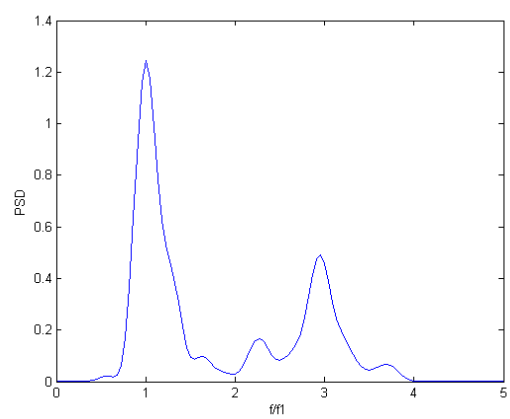

Figure 6 – Crossflow acceleration spectrum for uniform flow 1.5 m/s at z/L=0.9 showing the presence of the 3rd harmonic. Frequency is normalized to the primary Strouhal frequency

Copyright © 2007 by ASME

SIMULATION OF A HIGH L/D PARTIALLY STRAKED RISER

A simulation of a high L/D riser was performed to demonstrate that this method can be used to model deepwater risers with today's capabilities. A number of simplifications were made to the actual problem, dictated by hardware limitations and complexity of the experiment, making this a demonstration exercise rather than a benchmark. The results were compared against a model scale experiment described by Vandiver et al. in [15]. In the experiment, the instrumented riser was towed in the open sea. Because of background currents in the sea, this simulates a long riser in a current that varies with depth. The model riser has a length of 147m and an L/D of 4137. The upper 60% of the riser is bare and the bottom 40% is straked. A large mass with a pinned joint was used to provide a boundary condition at the bottom of the riser. During the tow experiment the top part of the riser has a significant tilt angle and the bottom part is approximately vertical. Thus the riser approximates a typical 5000-6000 ft SCR (steel catenary riser) experiencing an in-plane current.

The riser was modeled with a mesh of 9.9M nodes which was near the limit of the 32 CPU computer used. The riser motion simulation was restricted to 26 modes in each of the x and y directions due to computer memory limitations so the lowest modes were omitted as they were not thought to make a significant contribution to the riser motions. Thus the cross flow motions included modes 5 to 30 and the in-line motions included modes 18 to 43. The dominant cross flow mode in the simulation was found to be mode 27. A unidirectional steady current was modeled not accounting for the directional unsteady current along the riser in the experiment. Tension variations due to the boat motion were not modeled.

The experimental data include strain measurements along the riser. Due to a twist of the fiber optic strain gauges along the riser, the exact orientation of the sensors was unknown.. In addition, the measurement includes tensile strain contributions that are not reported by CFD. Thus, the comparison with the experiment was meant to show trends rather than quantitative results, also baring in mind the modeling simplifications. That said, we compared transverse bending strain along the riser predicted from the CFD simulation with the experimental data in Figure 7. As shown in the figure, the two plots are qualitatively similar but show some differences. In particular, the experiment shows an increase in RMS strain near the juncture between the bare and straked portions of the riser. This was not reflected in the simulation results and may be caused by the omission of higher modes in the model Also, the simulation results clearly showed the effect of the dominate mode 27. It is thought that the use of more and higher modes in the analysis should improve the results. The overall predicted RMS strains was similar to those observed in the experiments. The stresses at the ends of the riser, where the critical design areas are located, are conservatively predicted. We also note that accurate predictions of high derivatives in physics, like strain, are very difficult to accomplish. Motion response values of 0.6 RMS A/D in the bare region and 0.1 in the strake region were predicted and found to be in good agreement with similar experiments.

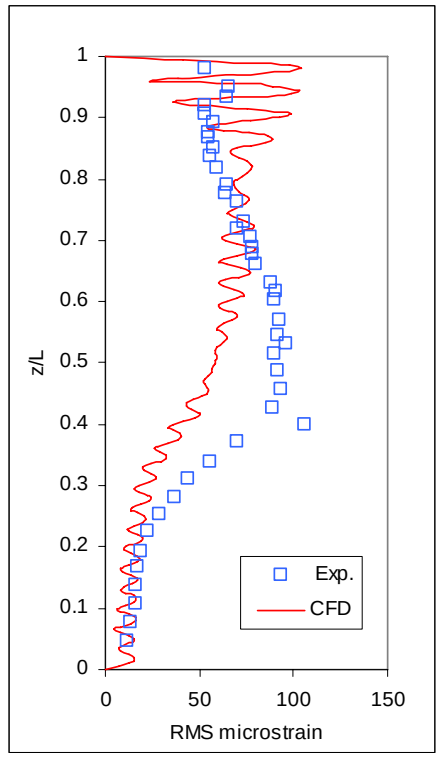

Figure 7 – Strain comparison for an L/D=4137 riser with 40% strake coverage in the Gulf Stream current. The orientation of the experimental measurements is unknown due to a twist in the fiberoptic fiber. Note that z = 0 is at the riser bottom.

RESPONSE CHARACTERIZATION

Dynamic response

The crossflow response for the 1407 L/D was analyzed in an effort to understand the complex cable dynamics. For the low mode cases in uniform flow a steady standing wave pattern was observed as expected, see Figure 8. At higher current speeds, the primary mode number increases and also the number of modes participating. The standing wave pattern is still present at times but mostly dominated by traveling waves for the cases examined. At the ends of the riser standing waves dominate due to the reflection boundary condition of the structure.

The high mode shear flow cases shown in Figure 9 reveal a strong traveling wave pattern with standing waves at the ends of the riser, due to the end conditions. The excitation is caused near the top of the riser at the high current region and

Copyright © 2007 by ASME

the wave propagates to both ends. This region is also referred to as power-in and was observed to shift in time. In the cases examined, the power-in region was located in the high current region between 0.8 and 0.95 z/L. The shift to areas of higher and lower velocity may be caused by the inline motion of the riser that drifts back and forth due to the slow change in drag force, or due to the mode or frequency shift.

The temporal maximum crossflow response was also found to be around 1-1.7 A/D for the high mode cases, showing good correlation with rigid cylinder 2 DOF experiments. The traveling speed of the wave was 55 m/s compared with an estimated 52 m/s using an added mass coefficient of 1.

One of the implications of the shift in the power-in region and mode is that the location of high VIV motion will shift after time, resulting in a non stationary process. Short simulations will thus yield uncharacteristic results biased to one point when compared with experimental data that include a large number of cycles. To get more representative statistics, simulations must be run for as many cycles as the experiments or to compare short windows in both experimental and computational data that show similar response. The complexity and unsteadiness of the VIV response make benchmarking a significant challenge.

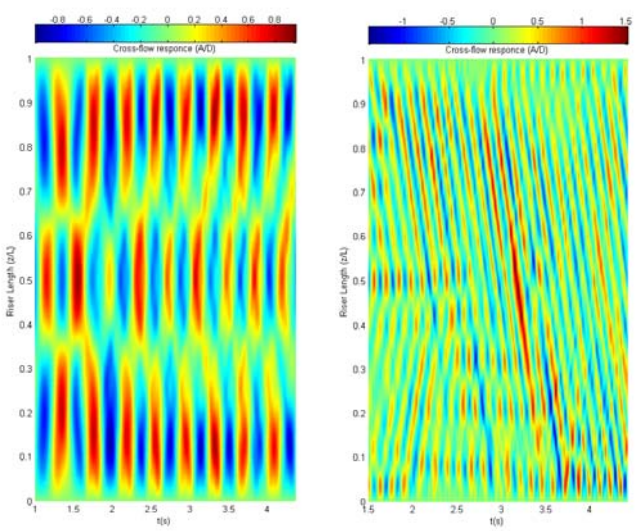

Figure 8 ***Uniform current crossflow response snapshot for 0.4m/s (left) and 1.5m/s (right) for the L/D= 1407 riser.***

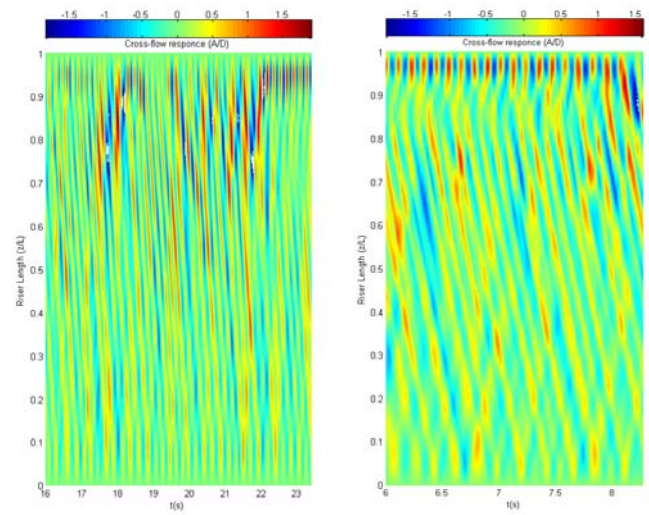

Figure 9 ***Response in shear currents, 1.2m/s left and 2.0m/s right for the L/D= 1407 riser.***

Flow visualization

A sample flow pattern is presented in Figure 10 for the linear shear case at 1.2 m/s. As the vibration moves down the riser vortices are shed in the wake. The vortices break down and split. Vortex shedding modes such as 2P and 2S have been identified in the isolated cases examined.

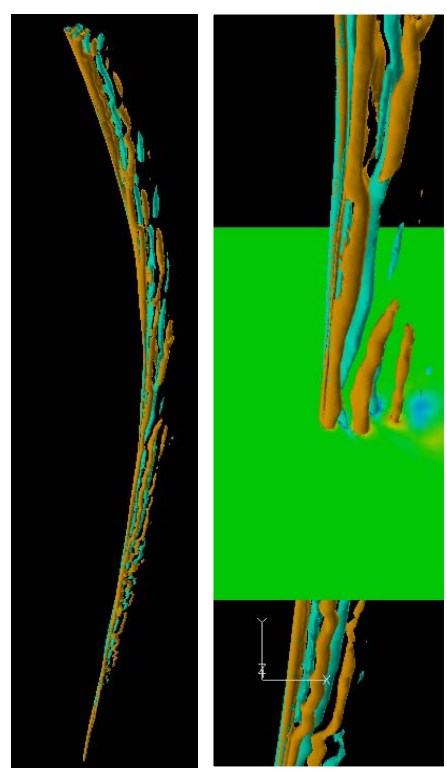

Figure 10 – Vorticity contours for shear flow 1.2 m/s.

Copyright © 2007 by ASME

CONCLUSIONS & RECOMMENDATIONS

A full three dimensional CFD model was developed and coupled with a simple structural model to predict the complex fluid structure interaction of a riser in strong currents.

- The model was validated through a number of cases including different current profiles and strake coverage.
- Results were satisfactory and compared well with experiments to the degree set by the limitations, uncertainties and assumptions of the available data and CFD model.
- The presence of the 3rd harmonic was identified in the crossflow direction and had an important contribution in the high mode strain and acceleration predictions.
- For a more definitive benchmark, simulations have to be carried out for as many cycles as the experiment. Comparisons should be made using the same post processing and filtering techniques. Looking at shorter windows and comparing other statistics besides the RMS value is also important.
- Modeling a full scale deepwater risers is possible using a medium size computational cluster.

NOMENCLATURE

A = motion amplitude [m]

D = riser diameter [m]

L = riser lenght [m]

x, y, z = coordinates [m]

U = current velocity; maximum for sheared profile [m/s]

$Re = UD/\nu$ = Reynolds number [-]

t = time [s]

K_i^n = riser stiffness matrix

m_i^n = riser mass matrix

n = mode number

S_i^n = eigenvector

T_i = Surface traction

ξ_i = Eigenvector of ith mode

REFERENCES

[1] Holmes, S., Oakley O. H. and Y. Constantinides, "Simulation of Riser VIV Using Fully Three Dimensional CFD Simulations", OMAE2006-92124, 2006

[2] Constantinides Y, and Oakley O. H, "Numerical prediction of bare and straked cylinder VIV", OMAE2006-92334, 2006

[3] Larsen, C. M., Passano, E., Barrholm, G. S. and K. Koushan, "Non-Linear Time Domain Analysis of Vortex Induced Vibrations for Free Spanning Pipelines" OMAE 2004-51404, 2004

[4] Larsen, C.M., Koushan, K. and E. Passano, "Frequency and Time Domain Analysis of Vortex Induced Vibrations" OMAE 2002-28064, 2002

[5] Herfjord, K., Drange, S.O., Kvamsdal T. Assessment of vortex-induced vibrations on deepwater risers by considering fluid-structure interaction, J.OMAE, 121 1999

[6] Shulz, K. W. and T. S. Meling, "Multi-Strip Numerical Analysis for Flexible Riser Response," OMAE2004-51186, 2004

[7] Shulz, K. W. and T. S. Meling, "VIV Analysis of a Riser Subjected to Step and Multi-Directional Currents," OMAE2005-67144, 2005

[8] R. H. J. Willden, and J.M.R. Graham, "CFD Simulations of the Vortex-Induced Vibrations of Model Riser Pipes," OMAE2005-67197, 2005

[9] Trim, A. D., Braaten, H., Lie, H. and M. A. Tognarelli, "Experimental Investigation of Vortex-Induced Vibration of Long Marin Risers," J. Fluids and Structures, Sept. 2005

[10] Kato, C., Lida, A., Takano, Y., Fujita, H. and M. Takano, "Numerical Prediction of Aerodynamic Noise Radiated from Low Mach Number Turbulent Wake," AIAA 93-0145, 1993

[11] Lighthill, H. J., "On Sound Generated Aerodynamically I. General Theory," Proceedings of the Royal Society Series A, Vol. 211, pp 564-587, March 1952

[12] S. E. Kim and L. S Mohan, "Prediction of Unsteady Loading on a Circular Cylinder in High Reynolds number Flows using Large Eddy Simulation," OMAE2005- 67044, 2005

[13] Trim, A. D., et all, "Experimental Investigation of Vortex-Induced Vibration of Long Marin Risers," Flow Induced Vibration, Ecole Polytenchique, July, 2004

[14] Spalart, Deck, Shur, Squires, Streletes & Travin, 2005, "A New Version of Detached-Eddy Simulation, Resistant to Ambiguous Grid Densities", Theoretical & Computational Fluid Dynamics

[15] Vandiver J. K., Sweithenbank S., Jaiswal V. and H. Marcolo, "The Effectiveness of Helical Strakes in the Suppression of High-Mode-Number", OTC-18276, 2006

Copyright © 2007 by ASME