Simulation of Riser VIV Using Fully Three Dimensional CFD Simulations

Samuel Holmes
ACUSIM Software, Inc.

Owen H. Oakley, Jr.
Chevron Energy Technology Company

Yiannis Constantinides
Chevron Energy Technology Company

ABSTRACT

Fully three dimensional computational fluid dynamics (CFD) solutions are combined with structural models of a tensioned riser to predict riser vortex induced motion. The use of three dimensional CFD solutions overcomes many of the shortcomings of combining a series of strip or two dimensional simulations to calculate the fluid forces on the riser. Three dimensional vortex structures are treated correctly and straked risers and variations in angle of attack can be studied directly. The proposed method uses finite element methods that are tolerant of sparse meshes and high element aspect ratios. This allows economical solutions of large fluid domains while retaining the important features of the large fluid vortex structures which drive risers. Long risers can be treated with readily available computers and examples of simulations of riser with L/D over 1400 are given and compared with previously published experimental data. These examples are used to illustrate several points regarding the effects of the treatment of the riser structure as well as the efficacy of rotating frame or pinned riser experiments used to simulate sheared currents. The method can also be extended to sheared currents whose heading varies with depth.

INTRODUCTION

Predicting the vortex induced vibration (VIV) of marine risers (pipes) combines several difficult engineering analysis issues. The flow around the riser is characterized by vortex patterns that are naturally three dimensional and complex. The riser geometry and ocean current conditions are varied. Although the simplest case is for uniform flow in a direction normal to the riser axis, the flow angle of attack and direction may vary along the riser length. Furthermore, the riser structural motions are large enough that they are fully coupled to the flow problem so that riser motion will affect the fluid flow and vice versa. At the present time, most practical methods to study riser VIV combine a structural model (often linear) for the riser with a heuristic treatment of the forces from the fluid. Solutions have been built on both time domain and frequency domain analyses [References 1-3]. Perhaps the most difficult problem is the scale of the computational fluid dynamics problem itself. Risers of interest are very long so that a direct attack on the riser VIV problem using three dimensional CFD solutions seems impossible. One practical approach to riser VIV predictions has been proposed in which the fluid flow solution is obtained on a series of two dimensional planes along the riser axis [e.g. reference 3]. These “strips” are pieced together with a structural model of the riser to obtain a prediction of the fluid-structure response. This method is appealing because it reduces a large three dimensional CFD problem to a large number of smaller two dimensional problems. Furthermore, the method has had some success in comparison with experiments such as that reported in References [4 to 7]. However, the strip method has some serious shortcomings. First, the flow around bluff bodies is inherently three dimensional so that although the 2D solutions may approximate the flow under some conditions, they may not be good approximations in general. This is especially true in the modeling of strakes which produce significant velocities along the riser axis near the riser surface. A similar problem occurs when modeling steel catenary risers (SCRs) which are inclined to the flow over much of their length. Finally, the use of the strip method implies that some kind of interpolation method is assumed to estimate the forces between the strips. There is no general rule on how to make such interpolations.

In the approach taken here we attempt to use fully three dimensional CFD simulations coupled with relatively simple
structural models to predict riser VIV. This approach bypasses the main objections to the strip method but introduces new difficulties associated with the size of the numerical solution. A number of compromises are made in order to keep the size of the problem tractable. In particular, the fluid mesh is made very coarse. Finally, several simulations of long scale model risers are compared with the recent experiments on small scale risers described by Trim et al. [8].

NOMENCLATURE

\( a = \) motion amplitude [m]
\( A = \) area (m²)
\( D = \) riser diameter [m]
\( K^R_i = \) riser stiffness matrix
\( L = \) riser length (m)
\( m^R_i = \) riser mass matrix
\( n = \) mode number
\( Re = \frac{UD}{ν} = \) Reynolds number
\( S^i_n = \) eigenvector
\( t = \) time [s]
\( T_i = \) Surface traction
\( U = \) current velocity; maximum for sheared profile [m/s]
\( x, y, z = \) coordinates (m)
\( \tilde{S}_i = \) Eigenvector of ith mode

THE CENTRAL PROBLEM OF 3D SIMULATION

The driving motivation for methods such as the strip method for riser motion prediction is to make the simulation tractable, i.e. to reduce the computational effort to a manageable level. Even with current advances in hardware and software, fully three dimensional solutions of the flow around a long riser is not possible using currently accepted standards for mesh resolution and CFD methods. The problems are simply too large by an order of magnitude or more. The approach proposed here is to perform these simulations using much sparser meshes than used in current practice. For example, here we want to simulate the flow around cylinders at Reynolds numbers on the order of 30,000 (based on diameter). A brief survey of past and present literature reveals many solutions of this type. For example, in 1993, Kato et al. [9] used a mesh with about 20,000 nodes per diameter of length for flow over a cylinder at Re=10,000. Using LES they show good results for drag and also the acoustic signature predicted at a distance in terms of frequency and amplitude. The dominant noise terms are the shedding frequency and its first two harmonics or precisely the terms that we might expect to determine the motion of a free rigid cylinder. Because the noise at a distance is calculated from the time derivative of the integrated surface tractions on the cylinder (using Lighthill’s analogy [10]) one expects that the prediction of the spatially averaged total tractions on the cylinder is accurately determined. Today, a calculation similar to that in [9] might use one million nodes or more per diameter of length of the cylinder. For example, Kim and Mohan [11] present results at sub-critical Reynolds numbers (1.4e5) using a grid with 3.4 million cells in a finite volume solution using Large Eddy Simulation (LES).

The approach proposed here takes advantage of past experience that shows that many of the features of flow past a cylinder can be captured on relatively coarse grids. These include flow gradients in the axial direction, such as the correlation length for vortex shedding and other features. In order to predict the motion of flexible risers it is only necessary to approximate the integrated surface tractions fairly accurately as a function of axial position and it is not necessary to predict the details of pressure fluctuations around the riser surface. In what follows we show that the overall surface tractions can be adequately predicted.

A final comment is needed here regarding the effects of using coarse meshes to predict unsteady turbulent flows. We propose to use detached eddy simulation (DES) or unsteady Reynolds averaged Navier-Stokes (URANS) to model turbulence. In either case, we hope to resolve the large eddies that have the biggest effect on the loads on the riser. Recognizing that the resolved scales may not include all of those that cannot be considered isotropic, it needs to be shown that the effect of unresolved scales can be neglected or that they are sufficiently accounted for in the turbulence model. This problem is not completely resolved in this paper for all flow conditions of interest but is the object of ongoing study.

THE EXPERIMENTS

The experiments described by Trim et al. [8] were performed as part of the Norwegian Deepwater Programme (NDP). The experiments were performed in a tow tank that provided well controlled flow conditions. In all cases, a model riser made of 27 mm fiberglass pipe was towed in a tank. Testing was also done with various straked configurations by adding sleeves to the riser. Here we will only consider the straked riser cases where the strakes extend over 62% of the riser length. In all the experiments the riser is pinned at the ends and either towed by the two ends to simulate uniform current or towed in a circle by one end to simulate sheared current conditions as illustrated in Figure 1. Note that the sheared current configuration is an approximation of a vertical straight riser in a sheared current. In particular, the riser sits in a rotating reference frame which might have some effect on response so that results may differ from those of a vertical riser in a sheared current. We will examine the effect of this experimental method of producing a sheared current later in this paper in a single example and show that it is unlikely that the method causes problems with the data used here.

Instrumentation in the experiments consisted of strain gages and accelerometers along the length of the riser. Trim et al. [8] use a model analysis method combining data from both the acceleration histories and the strain gage data to obtain the displacements which we use for comparisons here.
Figure 1. Tow tank configurations to simulate uniform currents (top) and sheared currents (bottom).

NUMERICAL METHOD

All of the solutions shown here were produced using the AcuSolve™ finite element CFD 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.

The solution of the riser structural response was obtained using the P_FSI or “practical fluid-structure interaction” body option in AcuSolve. The problem of riser motion is solved by finding the eigenvalues and eigenvectors associated with the riser alone and thus characterizing the riser motion as a simple linear vibrating structure. The displacements of the riser are then a linear sum of the modal amplitudes times the corresponding eigenvectors. In this case, we assumed the eigenmodes to be sinusoidal so the eigenvectors have the form:

\[ S_i^n(z) = \sin\left(\frac{n \pi z}{L}\right) \]  

where the \( S_i^n \) are the eigenvectors in the x and y directions associated with the nth mode, \( L \) is the riser length, \( z \) is the distance along the riser axis and \( n \) is the mode number. For the examples given here we chose to represent the riser motion with either 20 or 30 modes each for the inline and cross flow motions.

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

\[ \left[ m_i^n \right] \ddot{\xi}_n + \left[ K_i^n \right] \dot{\xi}_n = f_n \]  

where \( \dot{\xi}_n \) is the mode amplitude and the \( m_i^n \) and \( K_i^n \) are 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 although it can be easily included in the analysis if desired. The presumed amount of damping is less than 1% and did not affect the solution in testing here.

In each time step, the surface tractions on the riser are projected onto the eigenvectors to find the values \( f_n \):

\[ f_n = \int_A T_i(x,y,z) \cdot S_i^n(z) dA \]  

The resulting \( f_n \) are then used with Equation [2] to find the model amplitudes (\( \dot{\xi}_n \)) 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 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 \( \frac{1}{2} \) of a time step.

The mesh motion required to accommodate the changes in riser geometry were accomplished by explicitly controlled mesh motion rather than using an arbitrary Langrangian-Eulerian (ALE) strategy. This is more economical and also allows the overall mesh shape to be changed to accommodate the bowing of the riser during the simulation. In every other respect, the solution is equivalent to that obtained using ALE and the fluid and structural motions are fully coupled. As in ALE methods, the flow variables are calculated at the new nodal locations after the mesh is moved using the finite element shape functions. Figure 2 shows an overview of the mesh used for a bare riser. The mesh is bowed as part of the simulation as the mean position of the riser changes in order to limit the distortion of the mesh in the vicinity of the riser that would occur if all of the deformation was accommodated within fixed outer boundaries.

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 0.005 s so about 200 time steps are used in a vortex shedding cycle in a typical calculation.
The objective of the mesh design for the riser simulation is to provide adequate flow resolution using the minimum number of nodes. The approach used here is to provide increased mesh resolution in the wake and near the cylinder and lower resolution in other areas. Figure 3 shows a cross section of a typical mesh using this scheme. This mesh of a bare riser has about 2,500 nodes in this plane or about the resolution used in the simulations described in [9]. The mesh uses only 38 nodes around the circumference of the cylinder. In the problems solved here we wanted to use a total of about 1.6 million nodes in order to keep solution times short on available computer hardware. In order to make the mesh more economical, the mesh is stretched along the riser axis after it is generated creating elongated elements. AcuSolve is very tolerant of the resultant degradation in element quality so this change in the mesh does not affect the solver behavior but does reduce the resolution in the axial direction. Thus, at any given location, the mesh can better resolve smaller eddies that are aligned with the riser than those with other alignments.

A similar mesh for straked cylinders was also developed corresponding to the riser model with 62% strake coverage described in [8]. The presence of the strakes means that care must be taken to provide adequate resolution in the axial direction and the mesh cannot be stretched very much. Thus the total number of nodes per unit length increases by a factor of 4 over the bare riser model. Figure 4 illustrates the mesh strategy used for the straked cylinder. The mesh uses 3.9 M nodes and 7.2 M tet and prism elements.

Other meshes developed for this study include a shorter version of the bare cylinder mesh which models a riser length of 5.4 m (53 K nodes) and a dense version of this mesh (3.01 M nodes) which is used to explore the effect of mesh refinement; see Table 1. Short straked riser sections corresponding to one riser pitch length are also modeled using coarse and fine meshes to examine mesh sensitivity.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Type</th>
<th>L(m)</th>
<th>L/D</th>
<th>Nodes</th>
<th>Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Bare</td>
<td>38</td>
<td>1407</td>
<td>1.7 M</td>
<td>7.2 M tet/prism</td>
</tr>
<tr>
<td>2</td>
<td>Strake (75%)</td>
<td>38</td>
<td>1151</td>
<td>3.87 M</td>
<td>32 M tet</td>
</tr>
</tbody>
</table>

In the experiments, 75% of the riser is covered with strakes with a 13% loss due to gaps. The analysis mesh does not model the gaps.
Finally, it is important to note that here we believe we used the minimum possible number of nodes to simulate the flow around the NDP 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.

SIMULATION OF LONG RISERS

A series of eighteen calculations were performed as direct simulations of particular NDP experiments. These include simulations of bare risers in uniform and sheared flow and straked risers in uniform flow. Although not as extensive as the NDP experiments themselves, these simulations are sufficient to give a good picture of the successes and failures of the proposed method. The results of the simulations were compared with experiments by comparing dominant predicted and observed modes and frequencies, root mean square (RMS) amplitude of vibration along the length of the riser and maximum RMS amplitude from all points on the riser. In general, the dominant mode number and frequency are well predicted so the most telling comparisons are based on amplitude of vibration. Figures 5 and 6 compare predicted and measured amplitudes of vibration for a bare riser at uniform current speeds of 0.4 m/s and 0.8 m/s. In Figure 5 and Figure 6, the dashed curves are the experimentally derived values based on the analysis method described in [8]. Note that in general, these values tend to be higher than amplitudes based only on accelerometer data. Figure 5 compares RMS displacements in the in-line and cross-flow directions at a current speed of 0.4 m/s. Mode 3 is dominant in both experiment and analysis and the maximum amplitudes are about the same, but occur in different parts of the riser. Note the asymmetry in the experimental data that gives some indication of the complexity of the VIV process. Also, the indicated mode number in the in-line direction from the simulation is higher than that reconstructed from the experimental data.

Figure 6 makes a similar comparison for a current speed of 0.8 m/s. Again the dominant mode number is well predicted by the analysis. Although only cross flow data is available here, the analysis and experiment appear to be in reasonable agreement. However, the dominate mode shape (mode 5) shows more clearly in the CFD solution. Note that the agreement between analysis and experiment is not always as good as indicated here. This will be shown later when the displacements are compared over a wide range of flow conditions.

A total of 10 runs were completed for the bare riser in uniform flow at current speeds from 0.3 m/s to 2.0 m/s. The data show the expected increase in in-line (IL) and cross-flow (CF) mode number with current speed and the CF values as summarized in Figure 7. The predictions seem to agree fairly well with the experimental data at low values of current speed, but overestimate the mode number at high values of current speed. At this time it is not clear whether this difference in mode number at higher current speed is due to the treatment of the riser structure or to a deterioration of the CFD solution at higher Reynolds numbers.
We also compared both maximum and average rms displacements along the riser with the data given in [8]. Both measures produce similar results when compared to the experimental data. The maximum observed values are presented in Figure 8. Here the CFD simulation tends to underestimate the experimental results at low current speeds and over predict the displacements at intermediate speeds.

We also made similar comparisons for the case of sheared currents. In these simulations we first simulated a sheared current by placing a uniformly sheared current normal to the riser at the inlet to the mesh. Simulations were completed at 0.4 m/s, 0.8 m/s, and 1.5 m/s maximum current speed.

Figure 9 shows the predicted RMS displacements as a function of position at a current speed of 1.5 m/s. Also shown are the observed maximum and mean RMS displacements from the experiments. Here the CFD solution seems to agree less well with the experimental data, and this is also true for the other current speeds tested. Figure 10 compares the three available simulations with the experimental measurements on the basis of maximum observed RMS displacement. The predicted displacements are much lower for two of the three cases.

One concern with the sheared current runs is the effect of the experimental method used to produce a sheared current. In the experiments, the model riser is towed in a circle in a still tank producing a sheared approach velocity. Thus the fluid motions take place in a rotating reference frame when viewed from a fixed inertial reference frame. The fluid motions produced by the riser motion will be affected when viewed from this reference frame vis a’ vis a reference frame fixed to the riser.

For example, fluid motions along the riser axis as might occur in the wake are subjected to Coriolis forces. Because this might explain some of the differences between analysis and experiment, we also completed a simulation in which we...
modeled the riser and fluid motions in a rotating reference frame. This was accomplished by putting in the correct components of velocity at the mesh inlet and adding the Coriolis and centripal effects as body forces in the flow. This experiment did not produce a significant change in the RMS displacements. Although this may be satisfying to the experimentalist, it does not provide an explanation for the low values of predicted displacement in the sheared current simulations.

Further tests also showed that increasing the number of modes in the simulations had little effect. At this time the differences in predicted and observed in sheared flow might be attributed to either the simplified structural model or to errors in the CFD solution caused by the use of a coarse mesh.

Further tests also showed that increasing the number of modes in the simulations had little effect. At this time the differences in predicted and observed in sheared flow might be attributed to either the simplified structural model or to errors in the CFD solution caused by the use of a coarse mesh.

STRAKED RISERS

Two simulations where completed with the straked riser mesh corresponding to cases of uniform flow over a riser with 62% strakes as defined in Reference [8]. In these test, the strakes were concentrated at one end of the riser rather than distributed evenly along the riser. Thus the maximum displacements were measured at the unstraked section of the riser. The straked riser geometry is based on the information provided in the reference. The two cases analyzed were for flow velocities of 0.4 and 0.8 m/s. The presence of the strakes tended to disrupt the vortex shedding and reduce the motions on the straked end of the riser while the bare end produced somewhat higher amplitudes. Figure 11 shows a visualization of the flow with pressure contours on the riser and pressure iso-surfaces in the flow. The later surround the main vortex cores in the flow.

Figure 12 compares maximum RMS displacements with the data over a limited range of currents. The results show the expected trend of increasing displacement with current but general conclusions cannot be drawn due to the limited amount of data. Future work will continue this comparison to higher values of current speed.

SUMMARY & CONCLUSIONS

Two dimensional CFD simulations are often combined with a structural model of the riser to provide an economical solution for riser response. This “strip theory” approach has several limitations as the actual flow around the riser is strongly three dimensional and some geometries such as strakes are obviously not handled correctly. In this paper we propose an alternative approach based on fully three dimensional CFD simulations.
This meshing technique has the ability to model risers at any angle of attack, straked risers and other geometries and should be able to include three dimensional flow effects. The compromise required in the near term is that very coarse meshes must be used in the simulations. The meshes used here also use some asymmetric elements and thus sacrifice resolution in the riser axial direction. Even so, the problems used about 1.7 M nodes for bare risers and 3.8 M nodes for straked risers.

The resulting CFD solution is combined with a simple riser structural model (modal analysis) that is linear and allows only motion in the horizontal plane although further extensions are readily available. Riser and mesh motion are coupled in the solution in each time step but without iterations between the solid and fluid. The resulting solutions are compared with available laboratory data on a 38 m long riser model based on RMS displacements and mode number for most but not all cases.

The experiments and analyses differ in a number of noteworthy cases. The analysis under predicts the motion of bare risers at low current speeds and over predicts the motion at intermediate current speeds. This may be due to the lack of mesh resolution or other compromises in the present model and suggests that further work is needed. However, increased mesh resolution and other changes are well within the capabilities of current computers. Finally, it should be remembered that the current calculations were compared with riser experiments at about 1/10 scale and that the comparison is expected to be more difficult at full scale Reynolds numbers.

REFERENCES


