Proceedings of the Seventeenth (2007) International Offshore and Polar Engineering Conference
Lisbon, Portugal, July 1-6, 2007

ISBN 978-1-880653-68-5; ISBN 1-880653-68-0(Set); ISSN 1098-6189 (Set)

Riser VIV Analysis by a CFD Approach

Kevin Huang, Hamn-Ching Chen
Ocean Engineering Program
Department of Civil Engineering
Texas A&M University, College Station, Texas, USA

Chia-Rong Chen
Department of Mathematics
Texas A&M University, College Station, Texas, USA

ABSTRACT

In this paper, we present a CFD approach for riser vortex induced vibration (VIV) analysis in uniform current. The riser has a L/D of 1400, and is partitioned into 30 segments with uniform current specified on each segment. The drag and lift coefficients of each segment are then calculated using an unsteady Navier-Stokes numerical method on an overset (Chimera) grid system. These drag and lift coefficients are assembled to determine the riser displacements for each riser segment. The riser motion equation is expressed through riser modal shape superposition, and integrated in time domain by using Runge-Kutta scheme. At each time step the data grids for each riser segment are updated according to the riser's instantaneous displacements. The predicted riser VIV results are compared to the experimental data and previous publications. General agreements are observed. It is concluded that the present numerical method is valid and effective for riser VIV prediction.

KEY WORDS: riser, vortex induced vibration (VIV), modal response, Chimera, time domain, computational fluid dynamics.

INTRODUCTION

Riser vortex induced vibration has been an active research area over the past twenty years, majority of the work is focused in 2D or short risers. Recently experiments on long riser (L/D=1400) have been conducted at Marintek's Ocean Basin in Trondheim (Trim et al. 2005). Figure 1 shows the testing schematics plan view. The riser model has a mass ratio of 1.6, and length of 38m. It is towed through the wave basin to generate desired current conditions. The testing was performed under different the current conditions, i.e. uniform and sheared current. Some experimental data are published in Trim's paper (2005). This VIV experiment has generated considerable interest since it published some detail experimental results of riser VIV with large L/D. As the offshore oil and gas industry is heading toward ultra deepwater fields, to disclose and understand the characteristics of long riser VIV is of particular importance for field development and riser system design. Model testing is generally a favorable approach to provide design data and verifications. However, it does have its limitations too, such as facility availability and capacity limits, model scale limit, difficulty of current profile generation, cost concerns, etc. Under such condition, CFD provides a valuable compensation and alternative to model testing.

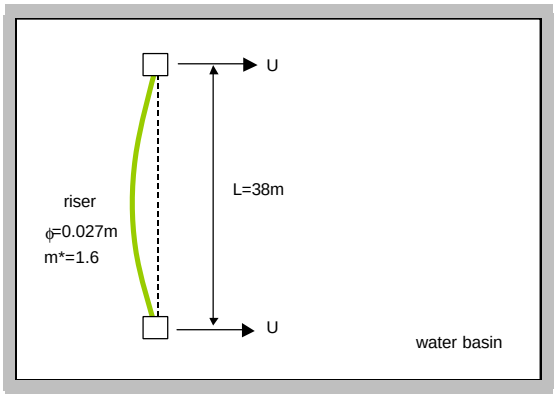

Fig. 1 Riser VIV Testing Plan View Schematics

Full 3D riser VIV time domain simulation has been a formidable area due to its computational effort. However, progress has been constantly made recently, with the help of ever increasing computational power and storage space. As of the moment this paper is published, a computational fluid domain with about 1 million elements can be practically handled by single processor PCs., and clusters have also been used for large-scale simulations with significantly more elements. Some published fully 3D CFD simulations of riser VIV (Holmes et al. 2006) have attempted to use elements number in the order of 10 million. They have used unstructured data grids, achieved reasonably well results, and demonstrated the possibility of calculating riser VIV using full 3D CFD approach.

In this paper we intended to further demonstrate that the long riser VIV could also be analyzed by using Chimera (overset grid) technique embedded CFD approach. This method has been previously validated and applied to different riser VIV studies (Pontaza, Chen & Chen, 2004, 2005a, 2005b; Pontaza, Chen & Reddy, 2005; Pontaza & Chen 2006). The Chimera technique is particularly well suited for the computational fluid dynamics (CFD) involving moving objects such as risers. A very fine data grid (body grid) is attached to the riser and on