



OTC 18348

## Application of CFD for Vortex-Induced Vibration Analysis of Marine Risers in Projects

M. Dixon and D. Charlesworth, DeepSea Engineering & Management

Copyright 2006, Offshore Technology Conference

This paper was prepared for presentation at the 2006 Offshore Technology Conference held in Houston, Texas, U.S.A., 1–4 May 2006.

This paper was selected for presentation by an OTC Program Committee following review of information contained in an abstract submitted by the author(s). Contents of the paper, as presented, have not been reviewed by the Offshore Technology Conference and are subject to correction by the author(s). The material, as presented, does not necessarily reflect any position of the Offshore Technology Conference, its officers, or members. Papers presented at OTC are subject to publication review by Sponsor Society Committees of the Offshore Technology Conference. Electronic reproduction, distribution, or storage of any part of this paper for commercial purposes without the written consent of the Offshore Technology Conference is prohibited. Permission to reproduce in print is restricted to an abstract of not more than 300 words; illustrations may not be copied. The abstract must contain conspicuous acknowledgment of where and by whom the paper was presented. Write Librarian, OTC, P.O. Box 833836, Richardson, TX 75083-3836, U.S.A., fax 01-972-952-9435.

### Abstract

The application of CFD (Computational Fluid Dynamics) to the issue of vortex induced vibration (VIV) of marine risers is a rapidly developing area of engineering which offers the potential to improve design capability whilst reducing associated testing costs. However, the tools and processes to achieve this are still being developed, tested and validated by a number of groups within the offshore industry and as yet only sporadic project application has been made of CFD for VIV analysis of marine risers.

This paper discusses the key issues associated with the application of CFD to projects followed by a case study illustrating a suitable area of application given the current level of maturity of the numerical tools and the timeframes required to perform the work. The first part of the paper includes a review of the available turbulence models comparing their benefits and drawbacks as well as the important issue of validation and benchmarking with recommendation on acceptable levels of accuracy given the application objectives and requirements.

The case study is based on a deepwater drilling riser and the assessment of understanding the VIV performance differences obtained when using staggered bare and buoyant joints along the riser. It is a generally accepted view that staggering bare joints improves riser VIV response however current analysis techniques do not provide the capability to evaluate the staggering and no hard evidence exists (e.g. test data) in support of a staggering philosophy. Through the use of CFD the influence of the non-cylindrical bare joints is captured in the analysis process permitting at least a qualitative evaluation of particular staggered configurations. This application and the stated objectives of the work are used to highlight the main points of the preceding discussion.

### Introduction

The last years have seen an increasing interest in the use of Computational Fluid Dynamics for the assessment of the Vortex-Induced-Vibration (VIV) behaviour of marine risers. This is still an on-going process where CFD methods are undergoing rigorous testing to prove their applicability in marine riser design. CFD is widely used in other industries (automotive, aerospace, energy) but investment has been made to achieve sufficiently good accuracy in the models for the various industryspecific flow regimes encountered. The offshore/subsea industry is now starting to do the same in realising the potential for CFD application to VIV, process simulation, splash zone loading etc. Currently, project application of CFD is mainly in complimentary role. Deepsea uses CFD for several types of projects such as workover riser VIV, strake and fairing design/analysis, as well as riser tower VIV analysis and drilling riser configuration development.

This paper aims to outline the procedure used for the development of the configuration of a drilling riser using CFD analysis for the assessment of the hydrodynamic behaviour of the drilling riser components which is being used as input to standard industry-accepted tools for frequency domain VIV analysis (Shear7).

### Key CFD Modelling Aspects

There are several levels of complexity, resource expense and accuracy with respect to CFD modelling. This paper focuses on the use of the RANS (Reynolds Averaged Navier Stokes) approach which is a good combination of efficiency and accuracy required for use in commercial work. The DES approach combines features of RANS formulations and LES methods. It was developed in an attempt to improve the predictive capabilities of turbulence models in highly separated flows and it is based on the idea of covering the boundary layer using a RANS approach and switching to a LES mode in detached regions. The DES method is at least an order of magnitude more computationally intensive than RANS methods and two-dimensional or axisymmetric calculations are not feasible.

LES (Large Eddy Simulation) solves the instantaneous Navier Stokes equations full instantaneous Navier-Stokes equations up to a prescribed “large scale” and uses “subgrid scale” models for the small-scale motion using appropriate filters to remove very fine time and length scales. LES