OTC 20217

Advanced Pipeline Riser Pull-In Analysis for a Fixed Offshore Production Platform

Weiyong Qiu, SPE, S.-H. Mark Chang, Xuejie Liu, and Jim King, Technip

This paper was prepared for presentation at the 2009 Offshore Technology Conference held in Houston, Texas, USA, 4–7 May 2009.

Abstract

This paper describes an advanced finite element analysis (FEA) methodology to analyze pipeline riser pull-in through an existing pull tube supported by a fixed offshore production platform. The fixed steel platform stands in 1,350 ft of water in the Gulf of Mexico. The challenges come from: (1) multiple bends along the pull tube; (2) 40 supports along the pull tube; (3) total riser length of more than 1,500 ft (a world record); (4) contacts between riser and pull tube, between pull-head and pull tube, between pull cable and pull tube, and between riser and seabed; (5) friction forces due to contacts; and (6) elastic-plastic bending of the riser due to the small bend radius along the pull tube. To incorporate these complexities of the riser and pull tube system in the FEA model, the general FEA software ABAQUS was selected. The detailed ABAQUS model based upon the first-end installation approach includes riser, pull tube, back tension, seafloor and the fixed platform.

This paper also presents pull tube strength design criteria and pipeline riser strain design criteria. The maximum computed stress along the pull tube and the maximum computed total bending strain along the pipeline riser are utilized to confirm that both pull tube and riser meet the pull tube strength design criteria and riser bending strain criteria. The pull-in loads and associated support loadings can be applied to verify the capability of the crane or winch and the structural design of the platform.

Introduction

A pipeline riser was designed to be tied back to a fixed platform in the Gulf of Mexico where the water depth is approximately 1,350 ft. Two pull tubes, i.e., Pull Tube 1 and Pull Tube 2, were to be considered for tie-back applications in the project. The application of pull tubes either in floating production platforms, especially in Spar, or in fixed production platforms, is accepted practice in the Gulf of Mexico. Walker and Davies [Ref. 1] explained the mechanics of pipeline riser pull-in in detail. They identified three stages of pipeline riser pull-in which are:

1. The initially straight riser encounters the curved walls of the pull tube and has to under go bending to proceed further.
2. The riser progresses through the bend with its shape being held in conformation with the walls of the pull tube.
3. As the riser enters the straight portion of the pull tube, it has a residual curvature resulting from the plastic deformation it has undergone; this causes the pullhead eventually to contact the inner side of the pull tube.

The pull-in load is given in each stage in analytic format. Walker and Davies also provided test results for a multiple-bend pull tube to verify the analytical assessments of the total riser pull-in load.

This paper presents an advanced pipeline riser pull tube pull-in analysis using time domain methodology. The finite element program ABAQUS version 6.5 is used for the analysis. Pull Tubes 1 and 2 both undergo the riser pull-in analysis. The strength analyses are performed for API X65 riser pipe which is empty during pull-in. Stresses are calculated for pull tubes. Bending strains are calculated for the pipeline riser. The pull-in loads and reactions of pull tube supports are also obtained by FEA analysis. Therefore, the advanced FEA approach can provide much more information than the analytical approach.

This paper also presents pull tube strength design criteria and pipeline riser strain design criteria. The maximum computed stress along the pull tube and the maximum computed total bending strain along the pipeline riser are utilized to confirm that both pull tube and pipeline riser meet the pull tube strength design criteria and pipeline riser bending strain criteria. The pull-in loads and associated support loadings can be applied to verify the capability of the crane or winch and the structural design of the platform.