Deepwater Offshore Specialty Symposium
Harbin, China, January 16 - 19, 2009
DOSS-71

VIV Analysis of Pipelines under Complex Span Conditions

James Wang, F. Steven Wang, Gang Duan, and Paul Jukes

J P Kenny, Inc., 15115 Park Row, Suite 300
Houston, Texas, 77084 USA

Abstract

Spans occur when a pipeline is laid on a rough undulating seabed or when upheaval buckling occurs due to constrained thermal expansion. This not only results in static and dynamic loads on the flowline at the span section, but also generates Vortex Induced Vibration (VIV), which can lead to fatigue issues. The phenomenon, if not predicted and controlled properly, will negatively affect pipeline integrity, leading to expensive remediation and intervention works.

Span analysis can be complicated by: long span lengths, a large number of spans caused by a rough seabed and multi-span interactions. In addition, the complexity can be more onerous and challenging when soil uncertainty, concrete degradation and unknown residual lay tension are considered in the analysis.

This paper describes the latest developments and a 'state-of-the-art' finite element analysis program that has been developed to simulate the span response of a flowline under complex boundary and loading conditions. Both VIV and direct wave loading are captured in the analysis and the results are sequentially used for the Ultimate Limit State (ULS) check and fatigue life calculation.

KEYWORDS: Boundary Condition (BC); DNV (Det Norske Veritas); FLS (Fatigue Limit State); FM (Force Model); KP (Kilometer Post); Mode Shape; Natural Frequency; RM (Response Model); VIV (Vortex-Induced Vibration); ULS (Ultimate Limit State); and Unit Stress.

Nomenclature

C_A	Added mass coefficient
C'_A	Equivalent added mass coefficient
D	Pipe OD including coating
EI_{conc}	Bending stiffness of concrete coating
EI_{steel}	Bending stiffness of steel pipe
SCF	Stress Concentration Factor
V	Pipe volume
Wc	Content weight
W_p	Pipeline weight
$W_p^{'}$	Equivalent pipeline weight
ρ_p	Pipe density
ρ'_p	Equivalent pipe density
ρ_{water}	Water density

Introduction

It is important to adopt an appropriate methodology to identify potential damage that could occur with the existence of a pipeline span. The integrity of the pipeline can then be evaluated with confidence in order to make a decision for the future service of the pipeline.

Free spanning pipeline analysis can be very challenging due to soil complexity, multimode vibrations and a high number of spans that can either be very long or interacting. The span evaluation is compliant with the design principles in DNV-RP-F105 [5] in this study. Based on the DNV code, the study of a free spanning pipeline includes both response and force models. The response model is based on a Vortex Induced Vibration (VIV) amplitude response where the VIV is caused by vortex shedding across the pipeline. There are two types of VIV to consider: in-line and cross-flow oscillation, which occur with lateral and vertical vibration, respectively. The pipe may also experience fatigue damage and local over-utilization due to direct waves, typically in shallow water. The influencing factors in VIV and direct wave loading assessments are:

- Pipe size, weight, and geometry;

- Additional weight such as content, insulation, and concrete coating if applicable;
- Current and wave parameters;
- Static and dynamic seabed soil stiffness;
- Span shoulder geometry;
- Residual lay tension; and;
- Operational conditions such as temperature and internal pressure.

Based on the input data above, VIV and direct wave loading analyses, including natural frequency calculations, are normally performed using a Finite-Element (FE) method for the following reasons:

- Complex seabed topology;
- Difficult identification of boundary conditions; and,
- Existence of span interactions.

Many studies have been preformed on spanning pipeline [1-4]. DNV-RP-F105 [5] provides a high level guideline for the span analysis; however, many key items are not documented in detail. This paper provides details regarding these key items in the procedure of the FEA modeling, fatigue calculation and ULS assessments, which becomes a practical methodology in pipeline span analysis.

FEA Modeling

In most cases, Finite Element Analysis (FEA) is necessary to accurately calculate natural frequencies, unit stresses and mode shapes - parameters required for fatigue life and ULS calculations. FEA modeling is thus a key to the success of span assessment.

General

The pipeline is modeled using 2-node pipe elements as they are simple and normally provide enough accuracy. The element size used in the model can be 1OD as a start based on DNV-RP-F105 [5]. The DNV code also states that short elements may be required, especially for short span/higher order modes.

The FE model can be single pipe with or without concrete coating, depending on project requirements. "Concrete Modeling" is discussed in the later section.

FE modeling of the span analysis is divided into two phases: static and dynamic (modal). In the static phase the sag deflection under the operating conditions, after the pipeline is laid on the seabed, is determined. In this phase, soil-pipe interaction is modeled using node-to-surface contact. In the dynamic phase, the natural frequencies and corresponding mode shapes are resolved and springs are used to model the interaction between soil and pipe. The dynamic phase is a linearised procedure that indicates linear effects, and any nonlinearity such as plasticity and friction are ignored in the dynamic phase even if these effects have been included in the static contact model. Therefore, only spring elements can be used to model dynamic soil stiffness.

Model Length

Based on DNV-RP-F105 [5], the boundary condition applied at the ends of the flowline section model should represent the continuity of the pipeline. Therefore, sufficient lengths of the pipeline at both sides of the span should be included, if possible, to account for the effects of the side spans. The length of the FEA model depends on the number of critical spans, span interactions, and span isolation of the pipeline region. The length also depends on model boundary conditions, computation time and result accuracy.

There are several methods to define model length. One of them is described in the following:

- Use a separate FEA model on a smooth seabed with a maximum possible span length;
- Use the relevant pipe geometry and soil conditions;
- Fix both ends;
- Identify the virtual anchor spacing – ideal model length;
- Identify the final model length using a comparison study.

As an example, a case was conducted where a 4,000-meter long pipe was laid on a flat seabed with a 50-meter span in the middle. The effective axial forces along the pipeline were obtained, shown in Figure 1. The virtual anchor spacing was determined to be 2,000 meters. This indicates that the pipe will be fully constrained by the soil-seabed friction, rather than the end constraints, if the model size is larger than 2,000 meters.

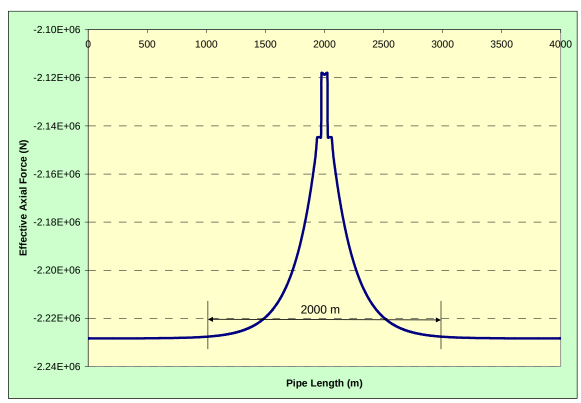

Figure 1: Effective Axial Forces

A comparison study was then performed on an actual seabed to demonstrate that an even shorter model length is sufficient to account for the effect of side spans. The study was conducted using a 400-meter model and the determined 2,000-meter model, respectively. The extracted natural frequency and unit stress indicates that the difference between the two models is less than 0.3%. Their deformed pipeline shapes and ULS results are also identical to each other. This proves that a shorter model length of 400 meters is sufficient for the analysis in this case. Certainly, depending on the project, the final model length will vary.

Fluid Mass Consideration

The fluid content is not normally modelled using elements in ABAQUS. To capture the impact from the content on the natural frequency and unit stress, the options below can be used:

Option 1: Equivalent Pipe Density

$\rho_p' = \rho_p + M_c/V$

Where,

ρ_p' Equivalent pipe density

ρ_p Pipe density

M_c Content mass

V: Pipe volume

Option 2: Equivalent Pipe Weight plus Equivalent Added Mass Coefficient

$W_p' = W_p + W_c$

$C_A' = C_A + \text{Content Mass}/(0.25 \cdot \pi \cdot D^2 \rho_{water})$

Where,

W_p' Equivalent pipe weight

W_p Pipe weight

W_c Content weight

C_A': Equivalent added mass coefficient

C_A: Added mass coefficient

D: Pipe OD including coating

ρ_{water}: Water Density

Concrete Modeling

For single pipes with concrete coating, the concrete can be modeled as an outer pipe relative to the steel pipe. The inner steel pipe and outer concrete pipe can be unbonded with a relative axial displacement slippage. The inner steel pipe and outer concrete pipe can also be bonded, sharing the same nodes. For both unbonded and bonded models, concrete degradation can be taken into account by using reduced concrete stiffness. In addition, a pure single pipe model can be used with an equivalent weight and stiffness for both pipe and concrete. The pure single pipe model does not capture the slippage phenomenon.

Modeling field joints in the FE span analysis may not be necessary, as the joints are too short to impact the overall deformed pipeline curvature. However, the stress concentration due to the discontinuity at field joints should be considered. The Stress Concentration Factor (SCF), depends on pipe and concrete geometry, and concrete degradations, and normally increases the bending stresses which affect the ULS and fatigue results. The SCF can be calculated using the proposed equation below:

$$SCF = 1 + \left(\frac{EI_{conc}}{EI_{steel}} \right)^{0.75}$$

Where,

EI_{conc}: Bending stiffness of concrete coating;

EI_{steel}: Bending stiffness of steel pipe.

It should be noted that this equation is similar but different from the equation defined in Section 6.2.5 of DNV-RP-F105 [5] for an analytical calculation (approximate response quantities). In the DNV equation, the full young's modulus of concrete is used, and Kc is constant accounting for the deformations/slippage in the corrosion coating and the cracking of the concrete coating, and Kc is 0.25 for PP/PE coating. In the equation defined above, either full or degraded concrete stiffness is used without Kc (i.e. Kc = 1). The SCF results using the equation above apply to both the response model and the force model when the fatigue life and ULS are calculated. The equation is easy to use and accurate once calibrated.

The calibration of the equation is performed using a separate FE model. In the model, the effect of pure bending imposed at both ends of a single 50-meter long pipe with a concrete coating is studied. The model uses pipe elements for both the pipe and concrete coating. There are two sub-models: the base model and the field joint model. The base model has a concrete coating over the entire pipe, and the field joint model has a concrete gap located in the pipe middle to represent the field joint, as shown in Figure 2 and Figure 3. The SCF can then be obtained as the ratio of bending stresses extracted from two FE models.

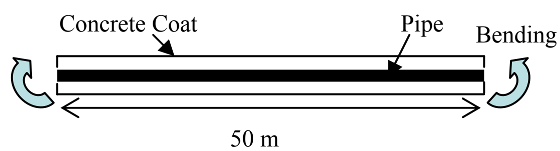

Figure 2: Base Model without Field Joint

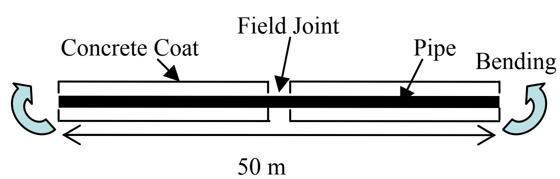

Figure 3: Model with Field Joint

An example is provided using data from Table 1 and the calibration results are presented in Table 2. The results show that the SCF's obtained using the equation and the separate FEA model are in close agreement. The equation method is slightly more conservative.

Pipe OD (mm)	Pipe Thickness (mm)	Pipe Density (kg/m^3)	Pipe Elastic Modulus (GPa)
609.6	15.875	7,850	207
Poisson's Ratio	Concrete Thickness (mm)	Concrete Density (kg/m^3)	Concrete Elastic Modulus (Gpa)
0.3	50	3,300	31.3

Table 1: Pipe and Concrete Coat Data

Cases	Field Joint Length	Bending Stress		SCF		Diff (%)
		w/ Field Joint (Pa)	w/o Field Joint (Pa)	from FEA	from Equa -tion	
Full Con-crete	0.1m	1.59 E+ 08	9.64 E+ 07	1.65	1.73	4.7
	1m	1.59 E+ 08		1.65	1.73	4.7
70% Con-crete	0.1m	1.59 E+ 08	1.09 E+ 08	1.46	1.56	6.9
	1m	1.59 E+ 08		1.46	1.56	6.9

Table 2: SCF Calibration Results

Comparison between the FEA and Analytical Methods for a Span with Fixed Condition

This section is a comparison study of the FEA model for single pipe with concrete coating versus the empirical SCF approach – analytical method defined the in DNV-RP-F105 [5]. To demonstrate the difference between the two methods, an example model of 50-meter fixed-fixed single pipe with a concrete coating was used, shown in Figure 4, using the input data from Table 1.

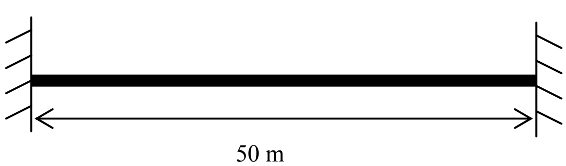

Figure 4: Pipeline Model for FEA versus Analytical Method

Single Pipe without Concrete

Without concrete, the results in Table 3 show a difference between the two methods of no more than 0.12% for the natural frequencies, and 5.20% for the unit diameter stresses. This indicates that the analytical method based on DNV code is quite accurate for the case with fixed boundary condition, and without concrete coating.

	Direction	FEA	Analytical	Diff (%)
Natural Frequency	Cross-Flow	0.870	0.870	0.00
	In-Line	0.869	0.870	0.12
Unit Stress	Cross-Flow	401	423	-5.20
	In-Line	401	423	-5.20

Table 3: Natural Frequency (Hz) and Unit Stress (MPa) Comparison for Single Pipe without Concrete

Single Pipe with Concrete

The natural frequency results are presented in Table 4 and Table 5 for the in-line and cross-flow, respectively. The results show that the FEA result with 95% concrete stiffness degradation matches the analytical result. The unit diameter stress results are also compared, as shown in Table 6 and Table 7, and the result show that the FEA unit diameter stress with 95% concrete stiffness degradation matches the analytical result, too. It should be noted that the match of the 95% degradation of concrete stiffness is only for the pipe case defined in Table 1. If the pipe and concrete are different, the match point between the FEA and analytical methods may differ. This indicates that the analytical method using the empirical constant defined in the DNV code does account for concrete degradation, but the represented degradation amount varies from case to case.

Concrete Degra-dation	FEA	Analytical	Diff (%)
0%	0.88	0.745	-15.34
40%	0.808	0.726	-10.15
80%	0.73	0.705	-3.42
95%	0.698	0.692	-0.86

Table 4: Natural Frequency (Hz) Comparison – In-Line

Concrete Degra-dation	FEA	Analytical	Diff (%)
0%	0.886	0.746	-15.80
40%	0.816	0.728	-10.78
80%	0.74	0.703	-5.00
95%	0.711	0.694	-2.39

Table 5: Natural Frequency (Hz) Comparison – Cross-Flow

Con-crete Degra-dation	FEA		Analytical		Diff (%) Dia-meter Unit Stress
	SCF	Dia-meter Unit Stress	SCF	Dia-meter Unit Stress	
0%	1.728	807	1.182	582	-27.88
40%	1.496	699	1.124	553	-20.85
80%	1.220	571	1.055	519	-9.10
95%	1.077	504	1.019	501	-0.62

Table 6: Unit Diameter Stress (MPa) Comparison – In-Line

Con-crete Degra-dation	FEA		Analytical		Diff (%) Dia-meter Unit Stress
	SCF	Dia-meter Unit Stress	SCF	Dia-meter Unit Stress	
0%	1.728	810	1.182	582	-28.19
40%	1.496	703	1.124	553	-21.35
80%	1.220	576	1.055	519	-9.87
95%	1.077	510	1.019	501	-1.67

Table 7: Unit Diameter Stress (MPa) Comparison – Cross-Flow

Comparison between the FEA and the Analytical methods for a Pipe Span at Actual Seabed

DNV RP-F105 and DNV's fatigue analysis software, FATFree [6] define four boundary conditions: "pinned-pinned", "pinned-fixed", "fixed-fixed" and "single span on seabed". Among them, "pinned-pinned" and "pinned-fixed" may be used as an initial assessment for interacting spans. However, it has been noted that the interacting spans cannot be accurately analyzed using the analytical method.

To demonstrate the differences between the analytical and FEA methods, an interacting span with an actual seabed shown in Figure 5, is used. The natural frequency, unit stress and fatigue life are calculated using both two methods, and the results are presented in Table 8. The results show that fatigue life is overestimated with the analytical approach using the "single span on the seabed", whereas the fatigue life is underestimated with the analytical approach using the "pinned-fixed". It can also be derived that the fatigue life will be further underestimated if the "pinned-pinned" analytical approach is used. A more realistic result for the interacting span is achieved by the FEA method since the natural frequency and unit stress can be accurately calculated using FE modeling.

Items	FEA	Analytical Method (DNV RP-F105 /DNV FATFree)			
		Single Span on Seabed	Diff to FEA (%)	Pined - Fixed	Diff to FEA (%)
Natural Fre-quency –In-line, Hz	1.061	1.114	5.0	0.964	-9.1
Natural Fre-quency –Cross-Flow, Hz	1.192	1.145	-3.9	0.964	-19.1
Unit Stress - IL, MPa	442	908	105.4	1362	208.1
Unit Stress - CF, MPa	502	947	88.6	1362	171.3
Fatigue Life, Year	146	185	26.7	6.23	-95.7

Table 8: Result Comparison using Actual Seabed

Fatigue Analysis and ULS Check

The fatigue analysis is conducted for each span corresponding to the applicable changes in concrete coating thickness, seabed topography, water depth, span gap, and environmental data. In the fatigue analysis, static load, VIV load (In-line and Crossflow) and direct wave loads (In-line if in shallow water) are considered in the calculation. In the ULS check, static load, VIV load (In-line and Crossflow) and the direct wave loads (In-line) are considered in the calculation. The span gap is calculated as the average value over the central third of the span based on suggestions from DNV-RP-F105 [5], [7]. During the fatigue and ULS assessment the key items: worst condition identification, wave and current data directionality, direct wave load consideration, and result sensitivity are discussed below.

Worst Conditions

The span analysis should consider the soil stiffness variation and concrete degradation. The following tolerance can be used:

- The soil static and dynamic stiffness: nominal - 30%;
- Concrete condition – Young's modulus: nominal -30% and nominal -100%.

In theory, higher dynamic soil stiffness will result in a higher natural frequency thus enhancing fatigue life, and higher concrete degradation should impair the fatigue life. However, when the residual tension is unknown for an existing pipeline, it is determined when the FEA pipe profile matches the survey profile. Under this condition, a higher tension may be required to have profile match if the concrete stiffness is decreased. As a result, zero concrete stiffness may not be the worst case for an existing pipeline.

An example using the data from Table 1 and Figure 5 for a new pipeline is presented to demonstrate the influences of soil stiffness and concrete condition - the fatigue results are presented in Table 9.

The results indicate that the static soil stiffness does not influence the fatigue life and lower dynamic soil stiffness decreases the fatigue life. It can also be concluded that the concrete stiffness degradation has a huge impact to the fatigue life.

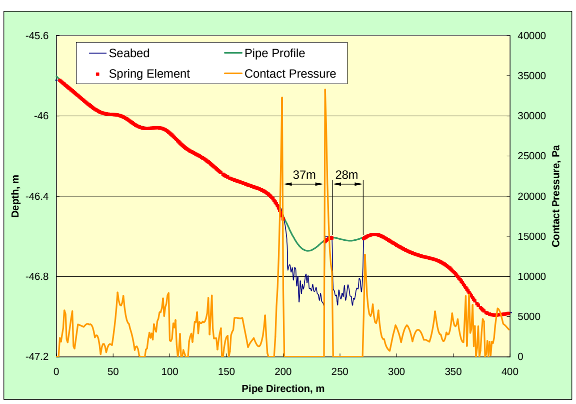

Figure 5: Seabed and Pipe Profiles

Tolerance (%)	Concrete Stiffness	Static Soil Stiffness	Dynamic Soil Stiffness
0 (Base Case)	146	146	146
-30	110	146	127
-100	23	N/A	N/A

Table 9: Fatigue Life Results at Concrete Degradation and Soil Stiffness Variation

Wave and Current Data Directionality

During the fatigue calculation, the wave and current magnitude and direction are required [8]. However, the direction information may not always be available. Therefore, the conservative assumption of direction combination of wave and current is adopted.

If the directions of both currents and waves are not available, perpendicular assumption can be made in the analysis. However, if the direction of only one phenomenon is available, users must define the same directionality for both phenomena.. This is due to how the current and wave statistics are intended. They are independent, but in order to depict most accurately, the probability density function should be joint, i.e. there should be a three dimensional matrix of probabilities associated with current velocity (Uc), wave period (Tp) and wave height (Hs) for each direction. Therefore, the probability density functions are interpreted as simultaneous, i.e. the wave data and current data are assumed to act in the same direction at all times, as such wave and current are assumed locked to each others direction. If users have more directions for one of the phenomena, wave for example, it is appropriate to impose current in all directions for which wave is active. It may not be conservative to assume that the direction of the other is perpendicular to the pipe.

Direct Wave Load

Direct wave loading may not be necessary in the analysis primarily depending on water depth, wave data, pipe size, etc. Normally, the direct wave is influential in shallow water. A sensitivity study was conducted as an example to verify the impact of water depth. Using various pipe sizes, the fatigue life due to direct wave loading at various water depths are calculated and presented in Figure 6. The results show that fatigue life increases exponentially and below a certain water level, the influence from the direct wave loading is negligible.

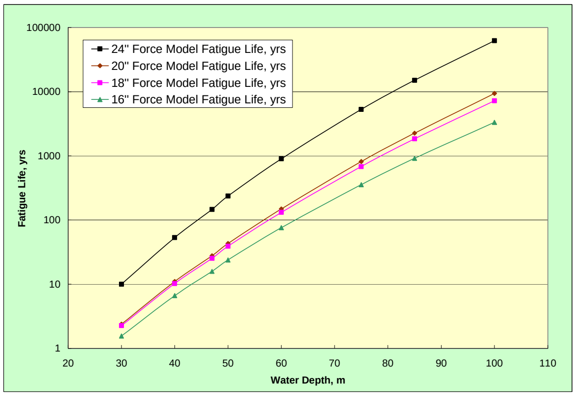

Figure 6: Fatigue Life due to Direct Wave Load

For the ULS calculation due to direct wave loading, the loading may be considered as acting on one or two interacting span segments at a time depending on the loading direction, shown in Figure 7 and Figure 8. The ULS check for both scenarios is required and either of them can be considered a worst case depending on the shoulder length, lateral friction and direct wave load magnitude.

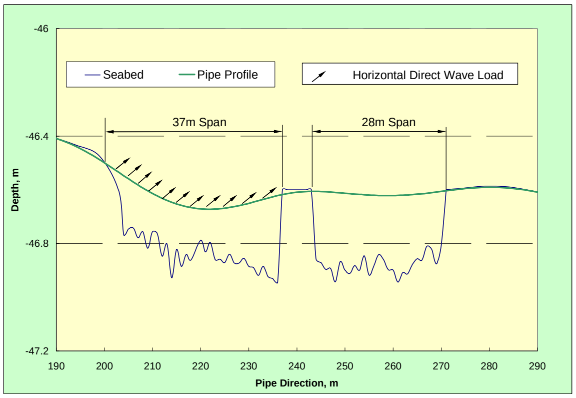

Figure 7: Wave Load applied to One Segment

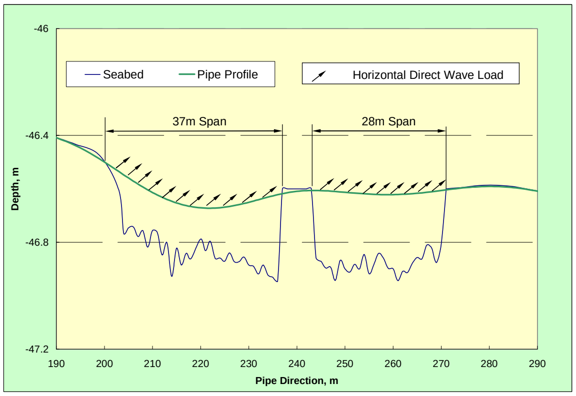

Figure 8: Wave Load applied to Two Segments

Result Sensitivity

Sensitivity due to Metocean Data

In the analysis, it is important to verify the influences from the tolerance or data range of the metocean data to identify the worst impact. An example using the following metocean data is presented and fatigue results are shown in Figure 9

- Base case: Nominal current and nominal wave data;
- Current – 0.1 m/s: Nominal current – 0.1 meter/second;
- Current + 0.1 m/s: Nominal current + 0.1 meter/second;
- Wave Period + 1s: Nominal wave period + 1 second;
- Wave Height + 0.5m: Nominal wave height + 0.5 meters.

The results show the impact of tolerance or variation of metocean data, which should be considered in the analysis.

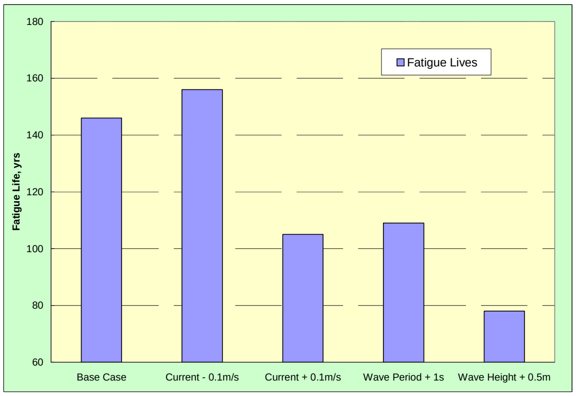

Figure 9: Fatigue Life Sensitivity with Metocean Data

Bending Moment Sensitivity

Bending moment in the ULS calculation due to direct wave loading can be very sensitive for interacting spans, especially for spans with a short shoulder in between. The sensitivity depends on the loading magnitude as well as the lateral friction factor. Refer to example shown in Figure 10. When the wave force is initially applied to two span segments, the lateral friction at the shoulder holds the pipe from moving laterally. During this stage, the bending moment due to wave load behaves linearly with the direct wave force. However, the moment greatly increases with the force when the pipe at the shoulder starts to move laterally. From this point, the friction can no longer hold the pipe and the moment becomes very sensitive to the applied force.

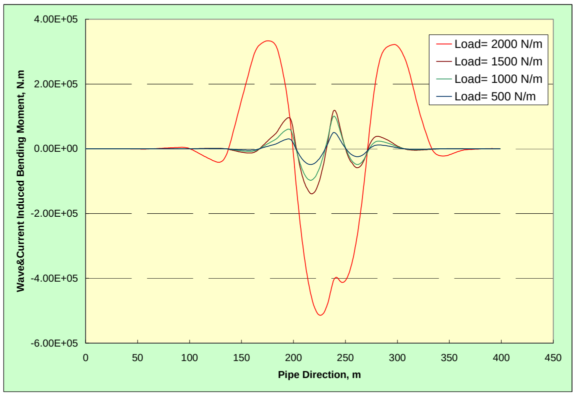

Figure 10: Bending Moments with Various Environment Loads

This indicates that the ULS calculation should be performed for a range of friction factors as well as a range of direct wave loading forces due to the tolerance of current and wave data. This allows for capture of the highest bending moment in all possible scenarios thus identification of the maximum ULS.

Vibration Modes

The VIV influence from higher vibration modes varies from case to case. A sensitivity study is performed to identify the impact from each mode, as shown in Figure 11. The results show the fatigue life decreases from around 15 years to 13 years when the first two modes are considered, compared to the case when only the first mode is considered. The results also show that the third and fourth modes have negligible impacts on the fatigue results. Therefore, using the first mode only in the fatigue calculation is not conservative.

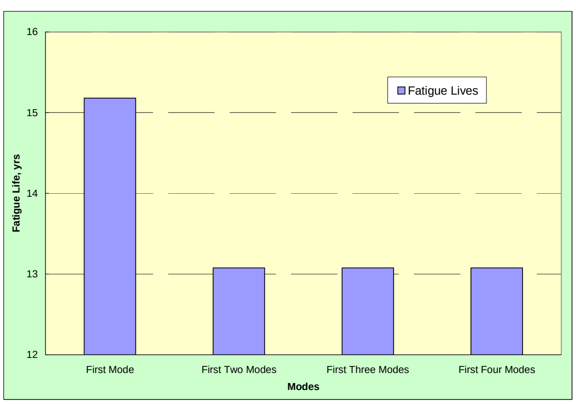

Figure 11: Fatigue Life due to Mode Consideration

Slugging Condition

The slugging condition, if occurs, should be analyzed. Under this condition, the pipe may sag deeply due to increased content weight. If the seabed used is trusted, no seabed modification is required and the sagging pipe may contact the seabed at mid-span. If the seabed is questionable in the span range, it is conservative to modify the seabed such that the sagging pipe makes no seabed contact in that span area. The results from two scenarios may be completely different. Considering the previous example for the slugging condition, the ULS results are 0.76 and 0.84, for the case with and without the middle touching, respectively.

Summary and Conclusions

This paper presents a practical methodology for analyzing free span pipelines. The methodology with details and examples, highlights key factors in FE modeling and fatigue and ULS calculations during the analysis. The methodology has been used on real projects in various scenarios, yielding the following main conclusions:

- Advanced numerical FE tools can adequately simulate the span of pipelines in static and dynamic phases with a good understanding of the DNV RP-105 [5]. In FE modeling, special care is taken in determining element size, model length, fluid mass consideration, concrete induced SCF (Stress Concentration Factor) at field joints, etc.;
- Identification of worst condition is required with variation of concrete degradation, and soil stiffness.
- Special care is to be exercised as well for consideration of wave/current directionality, the influence from the direct wave loading, and metocean magnitude tolerances;
- In the ULS check, it should be noted that the bending moment is very sensitive to the lateral friction, especially for the interacting span with a very narrow shoulder in between;
- The assessment of the slugging condition is needed - slugging may increase or decrease the ULS results.

It is believed that this methodology can be used as a starting point for projects with complicated spans.

Acknowledgements

The author would like to thank all that have provided input into this work.

References

[1] Fyrileiv, O., Mork K., and Chezhian, M., (2006) “Ormen Lange Pipeline Project – Free Span Design in Deep Waters,” 1st IOPF.

[2] Fyrileiv, O., Mork K., and Chezhian, M., (2002) “Structural Response of Pipeline Free Spans based on Beam Theory,” Proceedings of OMAE2002, Osla, Norway.

[3] Nielsen, F. G. “A suggested procedure for estimating Vortex Induced Vibrations for long free spanning pipelines”, Norsk Hydro Report 2002a.

[4] Nielsen, F. G., Kvarme, S.O. and Soreide, T., “VIV response of long free spanning pipelines”, 21st International conference on offshore mechanics and arctic engineering, (OMAE), Oslo, Norway, 2000b.

[5] DNV (2006), "Free Spanning Pipeline," DNV-RP-F105.

[6] DNV (2003), "FATFree User Manual," 2003-0511.

[7] DNV (2000), "Submarine Pipeline System," DNV OS-F101.

[8] DNV (2000), "Environmental Conditions and Environmental Loads", No. 30.5.

Copyright ©2008 The Deepwater Offshore Science Society. All rights reserved.