Hydrodynamic Forces on Subsea Pipes due to Orbital Wave Effects

Lisa King

Jeremy Leggoe[1], Liang Cheng[2]
[1]School of Mechanical and Chemical Engineering
[2]School of Civil and Resource Engineering

CEED Client: Woodside Energy Ltd

Abstract

Pipeline stability design relies on an accurate prediction of the hydrodynamic forces induced by wave and current motion. The motion of wind generated waves is generally orbital, and the orbit paths flatten to an ellipsoid with depth. This leads to the assumption in deepwater that the vertical components of the wave motion tend to zero near the seabed. Due to this simplification wave motion is often modelled as rectilinear for the purposes of analysing on-bottom pipeline stability. This simplification predicts that the magnitudes of the hydrodynamic forces are equal on the forward and reverse half wave cycle. In computational fluid dynamics (CFD) modelling results have shown the drag force exerted on an on-bottom pipeline was 7% higher than expected on the forward half wave cycle, and 5% lower on the reverse half-wave cycle when waves were modelled as orbital rather than rectilinear. This paper describes CFD modelling to be conducted to investigate this phenomenon and verify the validity of this result.

1. Introduction

Pipeline stability is an integral part of the offshore oil and gas industry and in particular is a major challenge for pipeline operators in the Australian North West Shelf (NWS). Due to the severity of the environmental conditions in the NWS, pipeline stabilisation can be a significant project cost, contributing up to 30% of the capital expenditure (Zeitoun et al. 2009). Overdesign of the pipeline weight coat or use of secondary stabilisation methods are financially costly, and under design can compromise the pipelines production ability and pose an extreme safety risk if the pipe fails (Zeitoun et al. 2008). To ensure safety and efficiency in design it is critical that stability analyses are as accurate as possible, and this requires a comprehensive understanding of the hydrodynamic forces that will act upon the pipeline.

Hydrodynamic loads are defined as "flow induced loads caused by the relative motion between the pipe and the surrounding water" (DetNorskeVeritas 2008). The loads are made up of drag, lift and inertia forces. Drag and lift forces are related to water particle velocity (U), while inertia is a product of water particle acceleration (a). The motion of wind generated waves is generally orbital, with wave theories such as Airy Wave theory describing how the orbit paths are circular near the surface and flatten to ellipsoidal with depth. This forms the basis of the simplification that the vertical component of the wave velocity and acceleration tend to zero near the seabed. Traditionally the simplest approach to pipeline stability, illustrated in figure 1, has been to employ the Morison Equations and alter the submerged

weight (W_S) of the pipeline to ensure the lateral resistance of the system is great enough to resist the hydrodynamic forces acting on the pipe (Zeitoun et al. 2008).

$F_D(t) = drag\ force = {}^1/_2\, \rho D C_D U(t)|U(t)|$
$F_I(t) = inertial\ force = {}^{\pi D}/_4\, \rho C_M a(t)$
$F_L(t) = lift\ force = {}^1/_2\, \rho D C_L U(t)^2$
$N(t) = normal\ force = W_s - F_L(t)$
$R_L(t) = lateral\ resistance = r_L \times N(t)$
$D = pipe\ diameter$
$r_L = coefficient\ of\ lateral\ resistance$
$C_D, C_M, C_L = drag, inertia\ and\ lift\ coefficients$

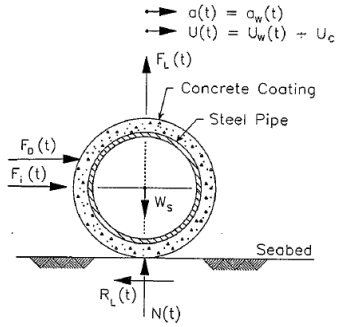

Figure 1 Traditional Stability Design Method (Allen et al. 1989).

The Morison equations calculate hydrodynamic forces using the horizontal components of the fluid velocity and acceleration, as well as constant drag, inertia and lift force coefficients. This approach is well suited to steady flow, yet has been found to underestimate the forces in oscillatory wave conditions. A comprehensive research program, conducted by the Pipeline Stability Design Project (PIPESTAB), highlighted a significant phenomenon called the 'wake effect'. The 'wake effect' refers to the impact the wake created by the previous half wave cycle has on the hydrodynamic forces exerted when it reverses back over the pipe in the following half wave cycle. The wake effect is largely responsible for the inaccuracy of Morison's equations. The study concluded that the drag and lift coefficients must reflect the time variation of the forces, as well as the modification of the local velocity in the pipeline vicinity, by accounting for the wake being swept back and forth over the pipe (Lammert, Hale & Jacobsen 1989).

The simplification of wave motion to rectilinear at depth has also resulted in wave motion in tank testing for on-bottom pipeline stability traditionally being generated as rectilinear. There are two principal tank testing methods used; the first method involves placing a stationary structure in a flow field, and the second is the carriage technique which involves dragging a suspended structure through a stationary fluid in such a way as to simulate the desired relative motion between the fluid and the structure (Bryndum, Jacobsen & Brand 1983). These two methods generally only model the oscillation of the horizontal components of the hydrodynamic forces and assume the vertical components are negligible.

As a result of these conventions stability analysis methods assume the magnitude of the hydrodynamic forces exerted by waves is equal on the forward and reverse half cycle. In CFD research conducted by Mengmeng Xu, 2010, an interesting anomaly was highlighted. Modelling returned results that showed the drag force exerted on an on-bottom pipeline was 7% higher than expected on the forward half wave cycle, and 5% lower on the reverse half-wave cycle when waves were modelled as orbital rather than rectilinear. This asymmetry contradicts these traditional methods, and this project seeks to examine the validity of conventional experimental approaches used to model hydrodynamic forces on subsea pipelines. This asymmetry is particularly important when a 'Generalised lateral stability method' is employed for design, as a degree of lateral movement is allowed. In this quasi-stable pipeline design method, wave loading asymmetry can result in net displacement in a particular direction, as well as net sediment transfer in this direction, which could affect the pipeline performance and integrity. These net movements could be missed with traditional tank testing models, and the purpose of this project is to investigate whether this asymmetry exists, and if so when it is significant and must be allowed for in design.

2. Model Formulation

The modelling of hydrodynamic forces due to orbital wave motion has been conducted in this project through the use of CFD software ANSYS Fluent. ANSYS Fluent is a finite element analysis platform which solves the Reynolds Averaged Navier-Stokes (RANS) equations.

2.1 Computational Domain and Boundary Conditions

The computational domain consists of a circular pipeline sitting on the bottom of the domain, the dimensions of which were developed from the results of a parametric analysis. The length and height of the domain were varied over a range of 100-300m and 15-50m respectively, and the velocity profiles in the vicinity of the pipeline were compared to establish when the results became independent of the geometry. The upstream length of the domain must be sufficient to allow the flow from the inlet to fully develop before it reaches the pipeline, and the downstream length must be sufficient to fully capture the upstream flow and wake characteristics that flow back over the pipeline during the reverse half wave cycle. Due to the nature of the modelling setup blockage exists where the flow crosses the pipeline, so the domain height must be adequate to ensure this blockage isn't significant. From the results of the parametric analysis it was found that the minimum upstream and downstream length is 50 pipe diameters (D), and the minimum height is optimally 50D. Note that for simulations in a water depth of less than 50D a smaller height can be used; however the blockage effect will become more significant.

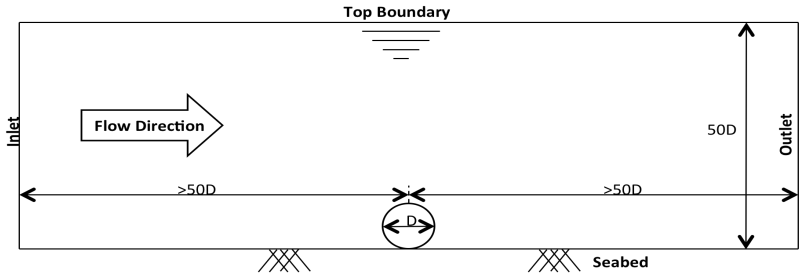

Figure 2 Schematic diagram showing the geometry, boundary conditions and the computational domain.

The domain boundaries are depicted in Figure 2. The domain inlet and the top of domain are set as velocity inlets, with the inlet specified by a user defined function (UDF) developed for this project. The bottom of the domain represents the seabed and as such is defined as a stationary no-slip wall with the roughness of coarse sand. Similarly the interface between the pipe and the sea-water is a no-slip wall with the roughness of concrete. Lastly the domain outlet is a pressure outlet. The mesh width decreases towards the centre of the domain and the mesh height decreases towards the bottom of the domain, ensuring the mesh is most dense around the pipeline. The mesh around the pipeline is contoured to ensure that there is minimum skewness or distortion of elements in the area of interest. The mesh density was increased until the results of the simulations converged. Approximately 130,000 elements are necessary for mesh independence for a 200m x 50m domain.

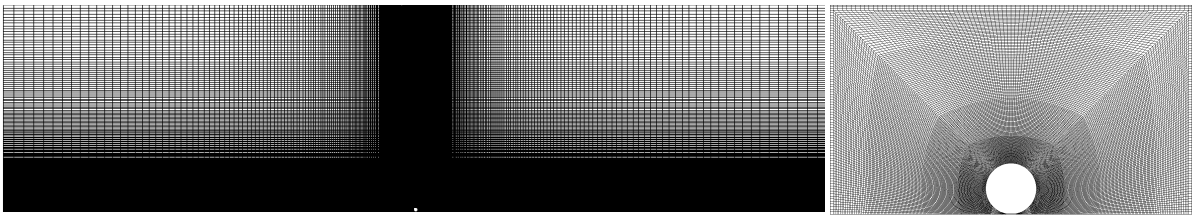

Figure 3 Mesh grids in computational domain (left). Mesh grids around the pipe (right).

2.2 Selection of a Turbulence Model

The RANS equations require a turbulence closure model and the selection of a turbulence model is a compromise between accuracy and computational efficiency. ANSYS Fluent offers a wide range of turbulence models from the Spalart-Allmaras one equation model to far more complex second moment models. Mirroring the experimental set-up of Akoz and Kirkgoz (2009), simulations were run with the RNG k-ε, k-ω and the SST k-ω turbulence models to determine which most accurately predicted the experimental PIV results. The two-equation eddy-viscosity models are the most widely used in industry (Hanjalic 2004).

The k-ε model in-particular is considered to be the backbone of many industrial CFD packages, and is known for being numerically robust and computationally efficient while having over three decades of use and validation behind it (Hanjalic 2004). The model works by solving two transport equations for the turbulent kinetic energy (k) and the rate of turbulent energy dissipation (ε). The downside of the model is its' insensitivity to adverse pressure gradients which makes it unreliable in regions such as the boundary layer. The RNG (renormalisation group) k-ε model provides analytical formula for the turbulent Prandtl number and accounts for low-Reynolds number effects. This makes it more accurate and suitable for a larger range of flows than the standard model (ANSYS 2011).

The k-ω model solves a second transport equation for the specific dissipation rate (ω) as opposed to ε. The advantage of this model is the seeming lack of modification required in the near wall region compared to k-ε, however the model is sensitive to the free-stream values of k and ω outside of the shear layer (ANSYS 2011). To overcome this sensitivity a model which combined the k-ε model in the free-stream region with the k-ω model in the near wall region was developed. The model is known as the Shear Stress Transport (SST) k-ω model and it combines the transport equations from both models through a blending function (ANSYS 2011).

The results of the three simulations were compared against the experimental results by plotting the velocity profiles in the near pipe region. Upstream of the pipeline all models have a reasonable degree of agreement with the experimental results, however downstream of the pipeline the k-ε model is significantly less accurate which is likely to be due to its weakness in dealing with the flow separation that occurs at the pipeline. The k-ω and SST k-ω models return very similar results, however the SST k-ω has been selected for this project due to its superiority in the near wall region.

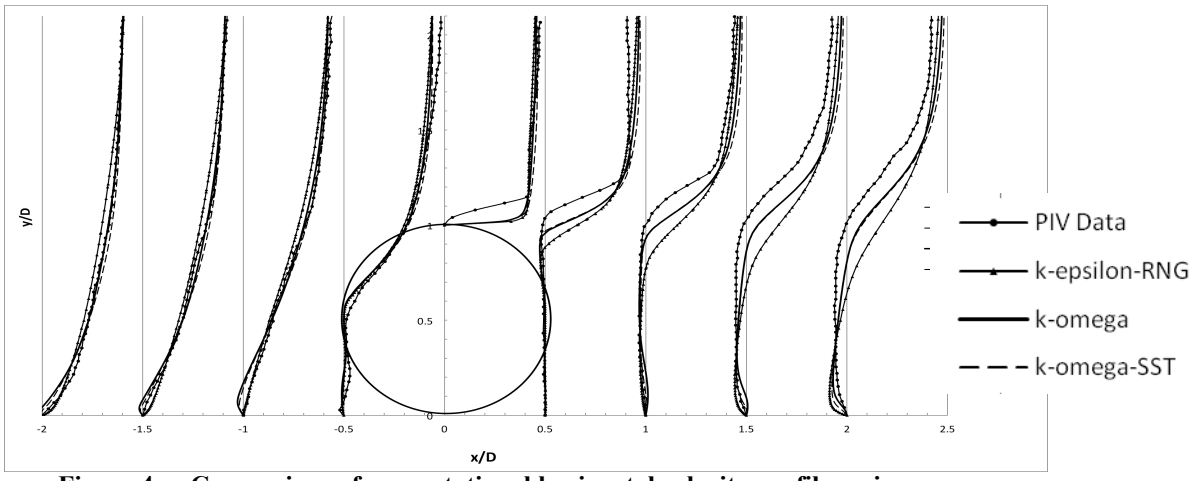

Figure 4 Comparison of computational horizontal velocity profiles using RNG k-ε, k-ω and SST k-ω turbulence models with experimental PIV profiles (Akoz & Kirkgoz 2009) for Re_D =5000

Comparison of the wake streamlines produced by the SST k-ω model with the experimental data shows a good replication of the wake size and form; this further verifies the use of the SST k-ω turbulence model.

(a)

(b)

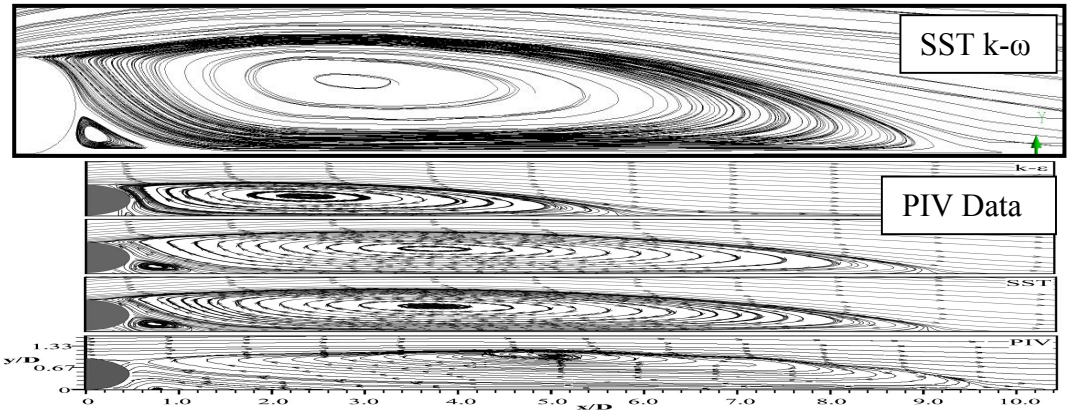

Figure 5 Velocity streamlines of wake using SST k-ω turbulence models and experimental data for flow with Re_D =5000 (a) Results from FLUENT (b) Experimental PIV results (Akoz & Kirkgoz 2009).

2.3 User Defined Input Function

A UDF function has been developed to define the velocity inlets for the domain, and it requires the specification of nine input parameters (wave height (H), wave length (L), wave period (T), water depth (h), reference current velocity ($U(z_r)$), reference height (z_r), seabed roughness height (z_0), turbulent inlet intensity (ININ) and turbulent scale factor (INSF)). The UDF utilises Airy Wave Theory to define the inlet wave profile, defining both the vertical and horizontal components of velocity with the following equations (Barltrop & Adams 1991):

$$u(z,x,t) = \frac{\pi H cosh\big(k(z+h)\big)}{T sinh(kh)} \cos\left(2\pi\left(\frac{x}{L} - \frac{t}{T}\right)\right)$$

$$v(z,x,t) = \frac{\pi H sinh\big(k(z+h)\big)}{T sinh(kh)} \sin\left(2\pi\left(\frac{x}{L} - \frac{t}{T}\right)\right)$$

There is also a small boundary layer included in the description of the wave profile. To define the current profile the logarithmic profile given in DNV-RP-F109 below was used:

$$U(z) = U(z_r)\frac{\ln(z+z_0) - \ln(z_0)}{\ln(z_r+z_0) - \ln(z_0)}$$

Lastly to prescribe the turbulent model input parameters the following relationships which were found to be adequate for verification were used (Akoz & Kirkgoz 2009):

$$k = \frac{3}{2}[(ININ)u_0]^2$$

$$\varepsilon = \frac{C_\mu k^{\frac{3}{2}}}{(INSF)L}; \qquad \omega = \frac{\varepsilon}{C_\mu k}$$

3. Results and Discussion

Preliminary simulations have returned very interesting results suggesting that an asymmetry is in fact present. There have not however been a sufficient number of simulations run yet and adequate verification performed to allow any reliable conclusions to be formed.

4. Conclusions and Future Work

A large test matrix has been developed to guide the remaining simulations that will be undertaken as part of this project scope. The test matrix covers a range of cases which will investigate the effects of pipe diameter, water depth, wave height, wave period and current velocity on the hydrodynamic forces exerted on subsea pipelines. Conclusions and recommendations will be formulated once the analysis of all cases has been completed. The scope of this project is limited to varying the parameters of regular wave series however in reality wave series are irregular; therefore further research beyond the scope of this project could include investigating the phenomena under irregular wave conditions. Future work can also include a 3D CFD simulation which would be particularly beneficial in accounting for the wake effect on hydrodynamic forces, and validating the 2D results.

6. References

Akoz, MS & Kirkgoz, MS 2009, 'Numerical and Experimental Analyses of the Flow Around a Horizontal Wall-Mounted Circular Cylinder', *Transactions of the Canadian Society for Mechanical Engineering*, vol. 33, no. 2, pp. 189-215.

Allen, DW, Lammert, WF, Hale, JR & Jacobsen, V 1989, 'OTC6055: Submarine Pipeline On-Bottom Stability: Recent AGA Research'. Available from: OnePetro.

ANSYS 2011, *ANSYS FLUENT 13.0 Theory Guide*, ANSYS Inc.

Barltrop, NDP & Adams, AJ 1991, *Dynamics of Fixed Marine Structures*, Third edn, Butterworth-Heinemann Ltd, London.

Bryndum, MB, Jacobsen, V & Brand, LP 1983, 'OTC4454: Hydrodynamic Forces From Wave and Current Loads on Marine Pipelines', in *Offshore Technology Conference*, Houston, Texas.

DetNorskeVeritas 2008, *Offshore Standard DNV-OS-F101: Submarine Pipeline Systems*, DNV, Norway.

Hanjalic, K 2004, *Closure Models for Incompressible Turbulent Flows*, Department of Applied Physics, Delft University of Technology, Netherlands.

Lammert, WF, Hale, JR & Jacobsen, V 1989, 'OTC6058: Dynamic Response of Submarine Pipelines Exposed to Combined Wave and Current Action'. Available from: OnePetro.

Xu, M 2010, CFD Modelling of Seabed Shear Stresses Around Subsea Pipelines, University of Western Australia.

Zeitoun, HO, Tornes, K, Cumming, G & Brankovic, M 2008, 'Pipeline Stability: State of the Art', *ASME Conference Proceedings*, vol. 2008, no. 48203, pp. 213-228.

Zeitoun, HO, Tornes, K, Cumming, G & Willcocks, J 2009, 'A Stability Design Rationale - A Review of Present Design Approaches', in *ASME Conference Proceedings*.