Wind and current loads on a pipelaying crane vessel

Y F Zan¹, R N Guo¹, X Bai²-⁴, Y Ma¹, L H Yuan¹ and F X Huang³

¹ College of Shipbuilding Engineering, Harbin Engineering University, Harbin, China
² School of Naval Architecture and Ocean Engineering, Jiangsu University of Science and Technology, Zhenjiang, China
³ Offshore Oil Engineering Co., Ltd., Tianjin, China
⁴ E-mail: baixu@just.edu.cn

Abstract. Wind and current loads have a significant effect on crane vessel and pipelaying operations. This study analyzes a pipelaying crane vessel performing deep-water operations, and uses a computational fluid dynamics code to calculate the wind and current load and moment coefficients of the vessel during its operations. The simulation conditions are set to be identical to those of wind tunnel tests, and the crane vessel hull is divided into upper and lower parts, with the upper part influenced by the wind loads and the lower part influenced by the current loads. In this work, 3D hull models were constructed, and k-ε turbulence models were solved by the Reynolds-Averaged Navier–Stokes Equations. The wind and current loads on the hull model are obtained for 50 wind and current angle combinations. The simulation results are found to be in good agreement (within 10%) with the wind tunnel test results for most headings, verifying the feasibility of the numerical simulation method. Finally, the characteristics of the load coefficients are generalized.

Keywords: computational fluid dynamics (CFD); current loads; pipelaying crane vessel; wind loads

1. Introduction

When designing pipelaying crane vessels, designers must consider the wind and current loads because of their considerable effects on the vessels’ navigation, handling, and operational performance. During lifting and pipelaying operations, cranes and superstructures are influenced by wind loads, whereas the hull and other underwater parts of the vessel are influenced by current loads. Therefore, accurately assessing the wind and current loads can enhance the safety and efficiency of pipelaying crane vessel operations, which are crucial for engineering applications. The wind and current loads and their coefficients are calculated by performing wind tunnel tests or by directly using empirical formulas. However, wind tunnel tests are costly and unfeasible for engineering projects with small budgets. The empirical formula estimation method also has several limitations, as the estimation formulas are based on certain ship types and are not universally applicable. Using such formulas leads to low estimation accuracy. Thus, a new and feasible calculation method using computational fluid dynamics (CFD) is introduced.

Isherwood [1] performed wind tunnel tests on various models of commercial ship hulls, and analyzed the wind loads on the superstructures of different ship types to obtain their wind pressure characteristics and wind load coefficient equations. These equations were subsequently generalized to produce the Isherwood regression equation, which can be applied to various ship types but produces more accurate
results when calculating the wind loads experienced by ships with large bodies than for those with small bodies. This method is applicable to various ship configurations and for all angles of the wind relative to the bow. Gould [2] studied the effects of offshore wind on hull superstructures and presented a numerical equation for calculating wind load coefficients. The wind speed is expressed using a logarithmic equation and the complexities of offshore wind speed are described in terms of the wind gradient. In wind tunnel tests, the bows were divided into sub-blocks, and the lateral wind pressure is calculated to determine the overall wind pressure. The results can then be used to calculate load coefficients for all ship types. Andersen [3] conducted wind tunnel tests on 9000+ twenty-foot equivalent unit-class container ships, expressed the wind loads as two dimensionless coefficients, and noted that the container arrangements on the deck had a considerable influence on wind resistance. There has been increased application of Computational fluid dynamics (CFD), specifically Computational Wind Engineering, in wind engineering over the past 5 decades [4-6]. Ignazio et al. [7] used CFD numerical simulations to calculate the wind loads on hulls in different turbulence models and with various model mesh sizes, and compared the results with ship data obtained from wind tunnel tests. They discovered that CFD numerical simulations were practical and sufficiently precise for engineering projects. Jia et al. [8] built a three-dimensional model of a tanker and, using appropriate boundary conditions and hydrodynamic equations, calculated the vertical, horizontal, and sway wind load coefficients of hulls under different leeway angles. They then compared the wind load coefficients with those obtained from the Isherwood equation. Koop et al. [9] conducted CFD simulations on spherical and membrane-type liquefied natural gas carriers, shuttle cruise ships, and floating production storage and offloading vessels, and performed a comparative analysis against the results obtained in wind tunnel tests. Their analysis indicated that the CFD method was more precise than wind tunnel tests in calculating wind loads on floating marine structures in open waters. Janssen et al. [10] employed the CFD method to analyze the effectiveness of numerical simulations for predicting large container ship-related data and the effect of simplifications to the geometric modeling processes on the precision of solutions. Sayda et al. [11] calculated the wind load coefficients of container ships using CFD and compared them with those obtained through empirical formulas (i.e., current wind load calculation standards). Sulaiman [12] analyzed the wind loads on a drillship and compared the cases with and without a drilling tower. The results obtained using CFD were found to be similar to those collected in BMT’s boundary layer wind tunnel. In addition, valuable insights concerning the physical properties of wind loads on drillship superstructures were identified. Using CFD, the current loads on many offshore wind structures have been successfully analyzed and determined [13-15]. Koop and Berezontski [16, 17] applied CFD in the calculation of the current loads on a model and full scale semi-submersible wind structure, indicating the uncertainty that could result from the scale effect. It is important that CFD results are obtained before performing model tests in order to get an understanding of the range of forces expected and potential experimental errors.

This study analyzes a pipelaying crane vessel performing deep-water pipelaying and lifting operations, and uses numerical simulations to calculate the wind and current load and torque coefficients of this vessel during its operations. The simulation conditions are set to be identical to those of wind tunnel tests, and the pipelaying crane vessel hull is divided into upper and lower parts, with the upper part influenced by wind loads and the lower part influenced by current loads. The below-water portion of the model comprises a stacked mold, whereas the part above water comprises a superstructure and freeboard deck. The results obtained through numerical simulations are found to be markedly similar to those obtained through wind tunnel tests, which verifies the feasibility of the numerical simulation method.

2. Mathematical model

2.1. Basic theory
This study adopts a three-dimensional mathematical model built using the Reynolds-averaged Navier–Stokes equations with the open-source CFD code OpenFoam. By solving the $k$-$\varepsilon$ turbulence model, an unsteady incompressible current can be simulated. The continuity equation is as follows [18, 19]:

$$\nabla \cdot \mathbf{v} = 0$$  \hspace{1cm} (1)

and the corresponding momentum equation can be expressed as:

$$\frac{\partial \rho \mathbf{v}}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{v} + \rho \mathbf{v}) + \sigma_v \nabla \cdot \mathbf{v} + \sum \left\{ \nabla \mathbf{v} - \nabla \mathbf{v}^T \right\}$$  \hspace{1cm} (2)

where $\rho$ is the fluid density; $\mathbf{v} = (u, v, w)$ is the velocity vector of the fluid; $\mathbf{v} = \partial / \partial x \partial / \partial y \partial / \partial z$ is the divergence operator; $x = (x, y, z)$ is the position vector in the Cartesian coordinate system; $p$ is the dynamic water pressure; $g$ is gravitational acceleration; $\mu$ is the dynamic viscosity coefficient; $\tau$ is the Reynolds stress tensor; and $C_{\varepsilon}$ is the surface tension coefficient (assigned a value of 0.074 kg/s² in the calculations). $\kappa$ and $\gamma$ represent the curvature and fluid index function, respectively. We can write:

$$\tau = \frac{2}{\omega} \mu_S - \frac{2}{3} kl$$  \hspace{1cm} (3)

$$S = (\nabla v + (\nabla v)^T) / 2$$  \hspace{1cm} (4)

where $\mu$ is the eddy viscosity coefficient; $S$ is the strain-rate tensor; and $l$ is the Kronecker delta. The turbulent dynamic energy is $k = 0.5 \rho \cdot V^2$, and the turbulent kinetic energy dissipation rate ($\varepsilon$) is as follows:

$$\varepsilon = C_{\mu}^{0.75} \frac{k^{1.5}}{l}$$  \hspace{1cm} (5)

where $C_{\mu}^{0.75}$ is a dimensionless coefficient ($C_{\mu}^{0.75} = 0.09$ in this study) and $l$ is the length scale of the turbulence.

The $k$-$\varepsilon$ equation is written as:

$$\frac{\partial k}{\partial t} + \nabla (k \mathbf{v}) = \nabla (\frac{\nu_t}{\sigma_k} \nabla k) + 2 \frac{\nu_t}{\rho} \left| \nabla \mathbf{v} \right|^2 - \varepsilon$$  \hspace{1cm} (6)

$$\frac{\partial \varepsilon}{\partial t} + \nabla (\varepsilon \mathbf{v}) = \nabla (\frac{\nu_t}{\sigma_\varepsilon} \nabla \varepsilon) + 2C_{\varepsilon} \nu_t \left| \nabla \mathbf{v} \right|^2 \frac{\varepsilon}{k} - C_{2\varepsilon} \frac{\varepsilon^2}{k}$$  \hspace{1cm} (7)

where $\nu_t$ is the turbulent kinetic viscosity coefficient, and the empirical coefficients are $C_{\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_k = 1.30$, and $\sigma_\varepsilon = 1.00$.

2.2. Sea-level wind field model

It is important to analyze the atmospheric wind profile when studying the wind loads that impact on pipe-laying crane vessel superstructures. In most studies, the exponential wind profile is used to show exponential changes in the average speed of wind along the surface of the sea with changes in relative altitude. The equation below is used to determine the average wind speed [20]:

$$\frac{V(h)}{V(10)} = \left( \frac{h}{10} \right) ^\alpha$$  \hspace{1cm} (8)

where $h$ is the height above the surface of sea water (m) and $\alpha$ is an exponent representing the velocity profile. For velocity profiles over the ocean, $\alpha$ is usually between 0.11 and 0.14. $V(h)$ is the average speed of wind at height $h$, and $V(10)$ is average speed of wind at a height of 10 m.

The wind field model is constructed using the aforementioned exponential wind profile at a ratio of 1:120. The data and curve are displayed in figure 1. For the wind tunnel tests, pipelaying crane vessels in water were selected as the study objects, and the wind and current loads on the hulls (expressed as
wind and current load coefficients) were measured at 15° intervals in the approach stream angle. The longitudinal force, lateral force, and sway torque were measured, allowing the wind-load-related coefficients of vessel parts subjected to wind speeds above water and the current-load-related coefficients of vessel parts subjected to current speeds below the water to be obtained.

Figure 1. Atmospheric boundary layer wind profile in wind field simulations.

2.3. Calculating wind and current load coefficients

The effects of winds and currents on the operations of a pipe-laying crane vessel occur in three directions, causing the vessel to sway, yaw and surge. The effects have three degrees of freedom in the average force and moment. The equations below illustrate how the longitudinal (X) and lateral (Y) forces, and the moment around the z-axis N are made:

\[
\begin{align*}
X &= \frac{1}{2} C_x \rho_a V^2 L^2 \\
Y &= \frac{1}{2} C_y \rho_a V^2 L^2 \\
N &= \frac{1}{2} C_n \rho_a V^2 LT
\end{align*}
\]  

(9)

where \( C_x \), \( C_y \), and \( C_n \) are the force and torque coefficients in their respective directions; \( L \) is the pipelaying crane vessel length; \( \rho_a \) is the fluid density; \( V \) is the fluid velocity; and \( T \) is the crane vessel hull water depth. When calculating pipelaying crane vessel current loads, the seawater density was set to \( 1.04 \times 10^3 \text{kg/m}^3 \). In contrast, when calculating the wind load on hull superstructures, the air density was set to 1.29 kg/m³. By converting the above equations, the following wind and current load coefficient equations are obtained:

\[
\begin{align*}
C_x &= \frac{2X}{\rho_a V^2 L^2} \\
C_y &= \frac{2Y}{\rho_a V^2 L^2} \\
C_n &= \frac{2N}{\rho_a V^2 LT}
\end{align*}
\]  

(10)
3. Model building

When performing wind tunnel tests on pipelaying crane vessels, wind-load-based numerical simulations are used for crane vessel parts above the water and current-load-based numerical simulations are employed for crane vessel parts below the water. For the CFD calculations, hull models are constructed using CATIA. The main dimensions of the HYSY201 hull are given in Table 1. The data are output in STL format to enable subsequent simulation model meshing. The overall crane vessel model is illustrated in Figure 2.

| Parameter                  | Data (m) |
|----------------------------|----------|
| Overall length             | 204.65   |
| Length between perpendiculars | 185.00   |
| Breadth                    | 39.20    |
| Depth                      | 14.00    |
| Mean draft                 | 8.313    |

Figure 2. HYSY201 hull model.

During simulations on the pipelaying crane vessel model, wind tunnel test-related parameters are used. The hull is divided into an upper and lower part at a water depth of 8.313 m. Hull parts under the water are made using a stack mold, and the model undergoes STL formatting. Figures 3 and 4 illustrate the wind load on the hull above the water and the current load on the hull below the water, respectively.

Figure 3. Effect of wind load on the hull above the water.  
Figure 4. Effect of current load on the hull below the water.

4. Comparative analysis of the calculation results

4.1. Wind load coefficients

Hull parts above the water surface were affected by wind loads. Therefore, simulations were done between 0° and 360°, at intervals of 15°. The results obtained were compared with the results from wind tunnel tests. In Figure 5-7, the $X$-axis, $Y$-axis current load coefficients, and current load moment coefficients on the $Z$-axis are shown. It is important to note that the effect of the value of $C_y$ will be
greater than $C_x$ on the overall wind load on the vessel because the projected side area is many times
greater than the projected front area. The detailed $C_x$ and $C_y$ closely resemble the wind tunnel data. For
the lateral force coefficient $C_y$, there is an agreement between the results obtained for 120° and 215°,
while some slight variations are observed for wind angles from 45° to 120°.

The longitudinal force causes resistance when the wind direction $\psi < 90^\circ$ or $\psi > 270^\circ$ and pushes
the ship forward when $90^\circ < \psi < 270^\circ$. The increases in force are not monotonous. The maximum wind
resistance occurs near 30° and 330°, and the maximum push is achieved near 150° and 210°. The lateral force is markedly strong in a wide range of approximately 30° either side of $\psi = 90^\circ$ and 270°. The yaw
torque changes exhibit an approximately S-shaped distribution when the wind direction $\psi = 0–180^\circ$.

**Figure 5.** The x-axis wind load coefficient.  
**Figure 6.** The y-axis wind load coefficient.  
**Figure 7.** The z-axis wind load torque coefficient.

### 4.2. Current load coefficient

For hull parts below the water surface were affected by current loads. Simulations were done between
0° and 360°, at intervals of 15°. The results obtained were compared with the results from wind tunnel
tests. Figures 8 and 10 show that the results of the CFD study and wind tunnel tests are in agreement.
Figure 9 shows the variation trend for the transverse force coefficient as captured by CFD. At angles of
90° and 180°, there is a slight underestimation of the lateral force coefficient.

The longitudinal force causes resistance when the wind direction $\psi < 120^\circ$ or $\psi > 240^\circ$ and pushes
the ship forward when $120^\circ < \psi < 240^\circ$. Under the current load effect, local peak values of the negative
longitudinal force occur when $\psi$ is around 30° and 330°, and local peak values of the push appear for $\psi$
near 165° and 195°. The lateral force is markedly strong in a wide range of approximately 30° either
side of $\psi = 90^\circ$ and $270^\circ$. The yaw torque changes from positive to negative at the $90^\circ$ mark, and the current load effects change accordingly.

![Figure 8](image1.png)  ![Figure 9](image2.png)  ![Figure 10](image3.png)

**Figure 8.** The $x$-axis current load coefficient.  **Figure 9.** The $y$-axis current load coefficient.  **Figure 10.** The $z$-axis current load coefficient.

5. Conclusion
This study has analyzed the effects of wind and current loads on crane vessels using CFD and three-dimensional crane vessel hull models built using CATIA. Subsequently, sea-level wind fields were simulated and numerical wind tunnels were constructed using OpenFoam. Preprocessing was employed to establish initial $k$-$\varepsilon$ turbulence model conditions, and calculations were performed to obtain longitudinal, lateral, and yawing wind and current load coefficients under various leeway angles. The results obtained through CFD were found to be similar to those obtained from the wind tunnel tests. In addition, the method proposed in this study enhances the prediction accuracy of wind loads on crane vessels and vessels sharing similar physical characteristics.

Acknowledgments
This project was supported by the National Natural Science Foundation of China (grant numbers 51809067, 51879125), Science and Technology Project of China National Offshore Oil Corporation (grant number CNOOC-KJ 135 GJJS 07 GC 2020-02), the High-tech Ship Scientific Research Project of Industry and Information Technology of PRC for Research on Key Technologies of Integrated Disassembly Equipment for Super Large Offshore Oilfield Facilities.
Acknowledgments for urban physics: Importance, scales, and opportunities concerning modeling of near-field pollutant dispersion in the built environment. Barcarolo D, Andrillon Y, Jacquin E and Ledoux A 2016 Evaluation of wind loads on FPSO topsides using a numerical wind tunnel. Koo A and Bereznitski A 2011 Model tests by wind tunnel of a FLNG Hull. In: OMAE2012 pp 731-42

viola I M 2009 Downwind sail aerodynamics: A CFD investigation with high grid resolution. Ocean Eng. 36 974-84

Jia B Z, Wu L Z, Ji R, Wang X L and Han X S 2018 Prediction and comparison of wind loads on VLCC based on CFD. Dalian Haishi Daxue Xuebao/J. Dalian Maritime Univ. 44 1-7

Koop A, Rossin B and Vaz G 2012 Predicting wind loads on typical offshore vessels using CFD. In: Proc. ASME 31th Inter. Conf. Ocean, Offshore Arctic Engin. (OMAE2012) pp 731-42

Janssen W D, Blocken B and van Wijhe H J 2017 CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications. J. Wind Engin. Industr. Aerodynamics 166 106-16

Saydam A Z and Taylan M 2018 Evaluation of wind loads on ships by CFD analysis. Ocean Eng. 158 54-63

Sulaiman Z 2019 Predicting wind loads on the topside of a drillship using CFD. In: Proc. Int. Offshore Polar Eng. Conf. pp 2779-86

Huang Z J, Kim J, Jang H and Slocum S T 2016 Current drag from tow and wind tunnel tests of a FLNG Hull. In: ASME 2016 35th Inter. Conf. Ocean, Offshore Arctic Engin. (American Society of Mechanical Engineers Digital Collection)

Koop A 2020 Using CFD to determine scale effects on current loads of offshore vessels in side-by-side configuration. Ocean Eng. 195 106707

Dong Q, Yang J and Guo X 2020 Measurements of current loads on side-by-side semi-submersibles in a wind tunnel. Ocean Eng. 201 107126

Koop A and Bereznitski A 2011 Model-scale and full-scale CFD calculations for current loads on semi-submersible. In: ASME 2011 30th Inter. Conf. Ocean, Offshore Arctic Engin. (American Society of Mechanical Engineers Digital Collection) pp 147-57

Koop A 2016 Determining side-by-side current loads using CFD and model tests. In: ASME 2016 35th Inter. Conf. Ocean, Offshore Arctic Engin. Vol 2 CFD and VIV (American Society of Mechanical Engineers Digital Collection) V002T08A051

Holzinger G 2015 OpenFOAM: A little user-manual CD-Laboratory-Particulate Flow Modelling (Linz: Johannes Kepler University)

Jasak H 2009 OpenFOAM: Open source CFD in research and industry. Inter. J. Naval Archit. Ocean Engin. 1 89-94

Barcarolo D, Andrillon Y, Jacquin E and Ledoux A 2016 Evaluation of wind loads on FPSO topsides using a numerical wind tunnel. In: ASME 2016 35th Inter. Conf. on Ocean, Offshore and Arctic Engin. Vol 2 CFD and VIV (American Society of Mechanical Engineers Digital Collection) V002T08A006