RANS Simulation of the Viscous Flow around Hull of LNG Ship in Confined Water

Mechanical Engineering Faculty
Universiti Teknologi Malaysia (UTM)
Skudai, Johor Bahru, 81310, MALAYSIA
*Adi@fkm.utm.my

Abstract: This research work describes the hydrodynamic effects of lateral and vertical forces on Liquefied Natural Gas (LNG) ship hull which are produced due to the effect of the restricted water by using Computational Fluid Dynamics (CFD) techniques. Liquefied Natural Gas (LNG) tanker often travels through shallow waters with visible banks for its operations. Manoeuvring of vessel in restricted water is relatively difficult and exposed to higher risk level. Travelling the ship near the bank may result in generating hydrodynamic loads such as sway force and yaw moment on ship hull besides changing velocity and pressure contours around the hull and the generation of asymmetric flow around the ship, hence may lead to potentially dangerous situations. In this research work the behaviour of a LNG carrier in confined water where there is a restriction in water depth and in waterway width has been studied experimentally and numerically. Influence of hydrodynamic forces and moments on the hull ship and wake pattern behind ship hull has been investigated using CFD. The numerical results have been validated by comparison with model testing in Marine Technology Center (MTC) of University Technology Malaysia (UTM).

Key-Words: Bank effect, Shallow water, Streamlines, LNG carrier, CFD

1 Introduction

Nowadays, focusing on the ship navigation in restricted waters such as canals, ports and channels with increasing the size and dimensions of LNG ships, container ships, but then, confined waters for navigation aren’t increased at the same level. As a result, ships are influenced by lateral and vertical hydrodynamic forces which produced by canal wall and bed. Hydrodynamic effect of restricted water (bank- shallow) on the ship manoeuvring is called “ship-bank effect”.

Hydrodynamic ship behaviours in restricted water are significant influenced by banks and shallow water while she navigates in confined water. LNG Tanker “Tenaga Kelas Satu” owned by Malaysian International Shipping corporation berhad (MISC Berhad) was grounded at Lake Charles in Jun. 2004 and Starboard bow below waterline touched the bank channel after failing to steer back since the ship turn to starboard. On the other hand, the same incident occurred for another LNG tanker “Tenaga Kelas Empat” owned by MISC Berhad at Suez Canal in Nov. 2007. [1]

Influence of ship-water velocity ratio, depth of water, propeller rotation rate, bank geometry, rudder angle and ship hull types on hydrodynamic forces of model tests conducted, also all coefficients and parameters of motion formulation determined by experimental tests and extent each items affected by confined channel width and finite depth. [2]

Ship hydrodynamics characteristics are significant affected by vertical and slope walls of restricted waterways. There are noteworthy hydrodynamic consequences in experimental tests with model tanker to determine the yaw moment and sway force on ship manoeuvring. [3]

Channels with varying width are causing asymmetric flow field around ship’s hull and vertical and horizontal hydrodynamic forces in presence of shallow water and close bank to the ship. Several characteristics of asymmetric forces on ship’s hull due to the bank and shallow water effect are caused by varying width of channel. Results of simulations are shown that it is noticeably large to the possible dangers of grounding and collision for ship traveling in varying width of channel. [4]
Changing the geometry of bank has significant effect on ship’s hull hydrodynamic forces. Several kinds of bank such as vertical bank, submerged bank, surface piercing bank with varying in horizontal situation between ship and bank, slope, water depth above the flooded bank was conducted in experimental tests to determine the yaw moment, surge and sway forces on LNG tanker’s hull. [5,6 and 7]

LNG carrier often travels through restricted waters with visible banks to load and unload its hazardous cargo. When ship deviates from the center of channel, the ship unavoidably goes close the bank, and results in non-symmetric flow around the ship hull. As a consequence of Bernoulli’s theorem, pressure difference is generated between port and starboard sides of vessel. A low pressure region generated due to the accelerated flow between the vessel and bank and manifest into a suction forces at stern of vessel attracting the vessel to bank, while a cushioning effect is induced at the bow which will result in a yaw moment swinging the bow away from bank. Such situations affect manoeuvring and course keeping which can lead to potentially dangerous situations, especially for LNG tanker.

However the test is expensive and takes long time, Model test remain the primary source of data for marine hydrodynamic test. using computational fluid dynamics (CFD) techniques in expecting the hydrodynamic forces influence on vessel is gradually increasing in the maritime community, this include further study of bank induced hydrodynamic forces in proximity of bank in complicated situations. CFD also applied in investigation of bank induced hydrodynamic forces on hull of a ship in confined waterway.

The results presented generally confirmed the feasibility of CFD technique to study on bank effect. Application of CFD in hydrodynamic analysis is particularly interesting since they can be used for a wide range of different application. Although these methods are still not wholly reliable, their use as a design tool will become widely accepted and used in the near future. [8].

Present study focus on examination of ship-bank interaction effects on flow pattern around the hull and nominal wake pattern in propeller plane of LNG tanker in restricted water using CFD simulation. Commercial CFD package, ANSYS-CFX, which using 3D Reynolds-averaged Navier-Stokes (RANS) solver based on finite volume method (FVM) is applied in this study.

### 2 Modeling

The ship model studied in this research work for numerical simulations and experiments is a Tenaga Class LNG tanker, at scale of 1/80. Table 1 and Fig. 1 are shown the main characteristics and the body lines of ship.

<table>
<thead>
<tr>
<th>Table 1: Main characteristics of LNG model</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Full scale</strong></td>
</tr>
<tr>
<td>L_{bp} (m)</td>
</tr>
<tr>
<td>Beam (m)</td>
</tr>
<tr>
<td>Draft (m)</td>
</tr>
<tr>
<td>Block coefficient, C_b</td>
</tr>
<tr>
<td>Wetted surface area (m^2)</td>
</tr>
</tbody>
</table>

Same Froude number in model and full scale ensures the model and full-scale ships exhibit similar behaviour. In general, the Froude number, \( F_n \), is defined as:

\[
F_n = \frac{V}{\sqrt{gL_{wl}}}
\]

Where \( V \) is the velocity of the ship, \( g \) is the acceleration due to gravity, and \( L_{wl} \) is the length of the ship at waterline level.

### 3 Numerical Simulations

A commercial RANS code Ansys CFX has been used in this study for simulating the flow around the LNG hull at the different speeds. CFX code utilizes the finite volume method for simulation. Figs. 2, 3, 4 and table 2 show the arrangements and mesh information for the LNG carrier in restricted and open water, respectively.
In Cartesian tensorial form the general Reynolds Average Navier-Stockes formulation for continuity is written as,

\[- \frac{\partial p}{\partial t} + \sum_{i} \left( \frac{\partial (\rho u_i)}{\partial x_i} \right) = 0 \]  \tag{2}

Momentum formulation is become as follows:

\[- \frac{\partial p}{\partial t} + \sum_{i} \left( \frac{\partial (\rho u_i u_j)}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \delta_{ij} \frac{\partial u_l}{\partial x_l} \right) \right) + \frac{\partial}{\partial x_j} \left(- \rho u_i u_j' \right) + f_{bi} \]  \tag{3}

In the above equation \( u_i \) is \( i^{th} \) Cartesian component of total velocity vector, \( \mu \) is molecular viscosity, \( (-\rho u_i u_j' \right) \) is Reynolds stress, \( \delta_{ij} \) is Kronecker delta and \( p \) is static pressure. The Reynolds stress should be demonstrated to near the governing equations by suitable turbulent model. For solution the RANS equation and turbulence velocity time scale, it is used by Boussinesq’s eddy-viscosity supposition and two transport equations. The body force is expressed by \( f_{bi} \).

For determination the 3D viscous incompressible flow around the ship’s hull is used the ANSYS-CFX14.0 code. The parallel version of CFX concurrently calculates the flow formulations using numerous cores of computers.

The shear stress transport (SST) turbulence model had been used in this study, because it gave the best results in comparison with other turbulence models. The equations are shown as follows:

Equation of \( \kappa \):

\[ \frac{\partial}{\partial t} (\rho k) + \sum_{i} \left( \frac{\partial (\rho u_i k)}{\partial x_i} \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \]  \tag{4}

Equation of \( \omega \):

\[ \frac{\partial}{\partial t} (\rho \omega) + \sum_{i} \left( \frac{\partial (\rho u_i \omega)}{\partial x_i} \right) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \]  \tag{5}

Where \( G_k \) and \( G_\omega \) express the generation of turbulence kinetic energy due to mean velocity gradients and \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) express the active diffusivity of \( \kappa \) and \( \omega \). \( Y_k \) and \( Y_\omega \) represent the dissipation of \( \kappa \) and \( \omega \) due to turbulence. \( D_\omega \) expresses the cross-diffusion term, \( S_k \) and \( S_\omega \) are user-defined source terms.[9]

The computational settings for using the ANSYS-CFX is tabulated in table 3 as follows:

Table 3: computational settings

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computing</td>
<td>64-bit Desktop pc 6GB of RAM</td>
</tr>
<tr>
<td>Simulation type</td>
<td>Steady state</td>
</tr>
<tr>
<td>Mesh type</td>
<td>Unstructured hybrid(tetrahedral/prism)</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>k-\omega (Shear stress transport)</td>
</tr>
<tr>
<td>Wall modeling</td>
<td>Automatic wall function based on a law of the wall formulation</td>
</tr>
<tr>
<td>Advection scheme</td>
<td>CFX high resolution</td>
</tr>
</tbody>
</table>
5 Results and Discussion

A series of model experiments to measure the ship-bank interactive forces were conducted to validate the numerical calculations. The experiments were conducted in the towing tank at Marine Technology Centre of University Technology Malaysia (UTM) that shows in Fig. 5.

Fig. 5: ship model in testing situation to study the effect of bank and shallow water

Fig. 6 shows the measured hydrodynamic forces and moments while various ship speed, ship-bank distance, and water depth.

Fig. 6: Effect of ship speed on yaw moment at H/D =1.6

The vessel experienced a bow out moment, which becomes bigger as the ship-bank distance decreases. It is also noticed that for a given ship-bank distance, the bank effect increase with higher ship speed. In general, the bank effect reduces with the increase of ship-bank distance and the reduction of ship speed.

The streamlines on the LNG ship’s hull model surface predicted by CFX code has shown a good agreement with streamlines that generated by Paint Smear Test (PST) on the hull surface model test which conducted in MTC-UTM and as can be seen in Figs. 7 and 8.

Fig. 7: Paint Smear Test velocity streamlines

Fig. 8: CFD velocity streamlines

Fig. 9 shows that the streamline around the ship hull in open water is symmetric on both sides on the ship hull at the streamline around her hull are asymmetric near bank as shows in Fig. 10. In addition, the pressure contours are symmetric in case of deep water and asymmetric under the effect of bank and shallow water, as shown in Figs. 11 and 12. The high velocity under the hull bottom in shallow water, it might lead to hull grounding.

Fig. 9: streamlines around the ship in deep water

Fig. 10: streamlines around the ship in confined water
4 Conclusion

CFD simulations and experiments have been conducted for studying the bank effect on a LNG tanker in open and restricted water conditions. In according to the presented computational results based on RANS equations, the following conclusions are taken:

- The proximity of bank and shallow water has a significant effect on the hydrodynamic pressure and velocity contours around the ship hull.
- The hydrodynamic interactive force and moment increase with reducing of ship-bank distance. For a given ship bank distance, the bank effect is more pronounced with an increasing of ship speed.
- In according to high velocity under the bottom in shallow water, it may lead to ship hull grounding.

References:
[2] M., Fujino, Experimental studies on ship manoeuvrability in restricted waters, Part 1,