VERIFICATION AND VALIDATION OF CALCULATIONS OF THE VISCOUS FLOW AROUND KVLCC2M IN OBLIQUE MOTION

S.L. Toxopeus

Maritime Research Institute, Netherlands

ABSTRACT

Viscous-flow calculations have been conducted for the KVLCC2M hull form in oblique motion. Several different drift angles were considered in this study. For one drift angle, a detailed grid-dependency study was carried out in order to obtain the uncertainty in the results. It is observed that in order to arrive at reliable results, special attention must be paid to the gridding of the computational domain. To validate the results, a comparison with integral as well as field variables available from measurements is made. The paper addresses the methods used and a detailed discussion about the accuracy of the results is presented. Very encouraging results are obtained, but the relatively high level of uncertainty in the evaluation of the pressure components requires further attention.

INTRODUCTION

Before results of viscous-flow calculations can be used practically in design studies, the uncertainty and accuracy of the results for similar cases should be known. Otherwise, conclusions based on erroneous results might be drawn, resulting in sub-optimal designs. Therefore, demonstration of the capabilities of viscous-flow solvers for a wide range of ship types is required.

During the Tokyo CFD Workshop 2005, participants were invited to conduct calculations for a full-block tanker hull form, the KVLCC2M, in steady manoeuvring motion. As part of the work for this workshop, an extensive series of viscous-flow calculations has been conducted for the KVLCC2M hull form in oblique motion. Drift angles ranging up to 12° were considered in this study. For the 12° drift angle case, a detailed grid-dependency study was conducted in order to obtain the uncertainty in the results. Additionally, the results have been compared to experimental data from NMRI for validation.

PARTICULARS OF THE SHIP AND TEST CONDITIONS

The hull form under consideration is the KVLCC2M. The particulars of this hull form are presented below, taken from the website of the Tokyo CFD Workshop 2005 (www.nmri.go.jp/cfd/cfdws05):

<table>
<thead>
<tr>
<th>Designation</th>
<th>Model scale (1:64.4)</th>
<th>Full scale</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length $L_{pp}$</td>
<td>4.97 m</td>
<td>320.0 m</td>
</tr>
<tr>
<td>Beam $B$</td>
<td>0.9008 m</td>
<td>58.0 m</td>
</tr>
<tr>
<td>Draught $T$</td>
<td>0.3231 m</td>
<td>20.9 m</td>
</tr>
<tr>
<td>Block coefficient $C_{B}$</td>
<td>0.81</td>
<td>0.81</td>
</tr>
<tr>
<td>Wetted area $S$</td>
<td>6.58919 m$^2$</td>
<td>27316 m$^2$</td>
</tr>
</tbody>
</table>

The measurements were carried out with the model restrained from moving in any direction relative to the carriage. Bilge keels, rudder and propeller were not present during the model tests and were therefore not modelled in the calculations.

The calculations were conducted with an undisturbed water surface, i.e. neglecting the generation of waves. The Reynolds number in the calculations was $3.945 \times 10^6$, corresponding to a full scale ship speed of 15.5 knots.

NUMERICAL PROCEDURES

Coordinate system

The origin of the right-handed system of axes used in this study is located at the intersection of the waterplane, midship and centre-plane, with $x$ directed aft, $y$ to starboard and $z$ vertically upward. Note that all coordinates given in this paper are made non-dimensional with $L_{pp}$ unless otherwise specified. All velocities are made non-dimensional with the
undisturbed velocity $U$. The forces and moments presented in this paper are given relative to the origin of the coordinate axes, but in a right-handed system with the longitudinal force directed forward positive and the transverse force positive when directed to starboard.

A positive drift angle $\beta$ corresponds to the flow coming from port side.

**Flow solver**

The calculations presented in this paper were performed with the MARIN in-house flow solver PARNASSOS, see Hoekstra and Eça [1] and Hoekstra [2]. This solver is based on a finite-difference discretisation of the Reynolds-averaged continuity and momentum equations with fully-collocated variables and discretisation. The equations are solved with a coupled procedure, retaining the continuity equation in its original form. The governing equations are integrated down to the wall, i.e. no wall-functions are used.

In PARNASSOS several eddy-viscosity turbulence models are available. In a numerical calculation of the flow around a ship, the turbulence model selection is not only based on the quality of the predictions, but also on the numerical robustness and the ability to converge the solution, i.e. reduce the iterative error to the desired value.

The one-equation model proposed by Menter [3] is the most commonly used turbulence model in PARNASSOS. This model leads to a remarkably robust method and allows convergence of the solution to machine accuracy in many cases. The Spalart correction to account for the effects of stream-wise vorticity, described in Dacles-Mariani et al. [4], is adopted in the turbulence model. No attempts have been made to add special features for modelling transition. So the basis turbulence model acts as the transition model as well.

**Procedure for uncertainty estimation**

The uncertainty, $U_\delta$, of any integral or local flow quantity $\phi$ is estimated with a procedure based on a least squares root version from Eça and Hoekstra [8] of the Grid Convergence Index (GCI), proposed by Roache [9]. Two basic error estimators are involved in the present procedure for uncertainty estimation: the extrapolation to grid cell size zero performed with Richardson extrapolation, $\delta_{RE}$; and the maximum difference between the data points available, $\Delta M$.

Ignoring the round-off and iterative errors, the error estimation $\delta_{RE}$ obtained by Richardson extrapolation is defined as: $\delta_{RE} = \phi_i - \phi_0 = \alpha h_i^p$. In this definition, $\phi_i$ is the numerical solution of any local or integral scalar quantity on a given grid $i$, $\phi_0$ is the estimated exact solution, $\alpha$ is a constant, $h_i$ identifies the representative relative grid cell size (or relative step size) and $p$ is the observed order of accuracy. The relative step size $h_i$ is calculated using $(n_{i+1}/n_i)$, with $n_{i+1}$ the number of nodes in stream-wise direction for the finest grid, and $n_i$ the number of nodes in stream-wise direction for grid $i$. The typical relative step size of 1 refers therefore to the finest grid.

Based on experience with several variants of uncertainty estimation procedures and on the outcome of the Workshop on CFD Uncertainty Analysis, see Eça and Hoekstra [10], the following options were adopted in the present calculations:

- Determine the observed order of accuracy, $p$, from the available data.
- For $0.95 \leq p < 2.05$, $U_\delta$ is estimated with the GCI and the standard deviation $U_\sigma$ of the fit:
  
  $U_\delta = 1.25 \delta_{RE} + U_\sigma$.

- For $0 < p < 0.95$, the same error estimate is made but is then compared with the value of $\Delta M$ multiplied by a factor of safety of 1.25, so that $U_\phi$ is obtained from:
  
  $U_\phi = \min(1.25 \delta_{RE} + U_{fit}, 1.25 \Delta M)$.

- For $p \geq 2.05$, a new error estimate $\delta_{RE}^*$ is calculated in the least squares root sense with $p=2$. The uncertainty then follows from:
  
  $U_\delta = \max(1.25 \delta_{RE}^* + U_{fit}, 1.25 \Delta M)$.

- If monotonic convergence is not observed, $U_\delta = 3 \Delta M$.

Based on the uncertainty analysis, it is assumed that the numerical solution $\phi_0$ for zero step size will be bound with 95% confidence by the interval: $\phi_i - U_\delta < \phi_0 < \phi_i + U_\delta$.

**Computational domain and grid topology**

Several grid topologies have been used for the calculation of the flow around the KVLCC2M double model [5]. The results presented in this paper were all obtained on structured grids with H-O topology with extra grid clustering close to the bow and propeller plane.

For the zero-drift case, a single-block calculation was conducted while for the non-zero drift case the
The domain was decomposed into effectively two blocks. The six boundaries of the computational domain are the following: the inlet boundary is a transverse plane located upstream of the forward perpendicular; the outlet boundary is a transverse plane downstream of the aft perpendicular; the external boundary is a circular or elliptical cylinder; the remaining boundaries are the ship surface, the symmetry plane of the ship or coinciding block boundaries and the undisturbed water surface.

The flow around the hull at non-zero drift angles has no port-starboard symmetry and the computational domain must be extended to cover the port side as well. Furthermore, a larger domain is required in order to incorporate the drift angle. On each side of the domain the grid consists of an inner block and an outer block, see Figure 1. The inner block is the same for all yaw angles and the outer block can deform to allow for the drift angle of the ship. Therefore grids for various drift angles can be made efficiently. Use is made of an in-house grid generator, see Eça et al. [6].

![Figure 1: Inner and outer blocks (coarsened) at 12° drift angle.](image)

The inner block is generated with a number of cells similar to the grids as used by Eça and Hoekstra [5] for the zero-drift case. Based on early calculations by Toxopeus [7] grid clustering at the propeller plane and the bow of the ship was applied to resolve the gradients of the flow at these locations more accurately.

To incorporate the drift angle of the ship, the inner block is rotated around the vertical z-axis over the desired yaw angle. Then the outer block is generated around the inner block. The cell stretching used in the inner block is automatically applied to the outer block as well. It was decided to have matching interfaces between the blocks so that the inner and outer blocks could be merged. The size of the outer blocks is chosen such that the rotated inner block can smoothly be incorporated in the outer grids. This means that increasing drift angles will result in wider domains. The size of the domain is based on the use of a solver for potential flow to calculate the velocities in the inflow and external planes. Before starting the calculations, the separate blocks are merged into one block for the port side of the ship and another block for the starboard side of the ship.

For each grid, the variation in the number of grid nodes in the stream-wise, normal and girth-wise (nξ, nη and nζ) directions is presented in Table 2, which includes also the maximum y+ value for the cells adjacent to the hull, designated y2+. That was obtained during the calculations. Note that also a calculation with zero drift angle was conducted in order to be able to determine the relation between the drift angle and integral or local variables consistently.

<table>
<thead>
<tr>
<th>β</th>
<th>nc</th>
<th>nξ</th>
<th>nη</th>
<th>nζ</th>
<th>nodes</th>
<th>y2+</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>449</td>
<td>81</td>
<td>45</td>
<td></td>
<td>1.6×10⁶</td>
<td>0.32</td>
</tr>
<tr>
<td>3°</td>
<td>449</td>
<td>95</td>
<td>2×45</td>
<td></td>
<td>3.8×10⁶</td>
<td>0.40</td>
</tr>
<tr>
<td>6°</td>
<td>449</td>
<td>95</td>
<td>2×45</td>
<td></td>
<td>3.8×10⁶</td>
<td>0.55</td>
</tr>
<tr>
<td>9°</td>
<td>449</td>
<td>95</td>
<td>2×45</td>
<td></td>
<td>3.8×10⁶</td>
<td>0.69</td>
</tr>
<tr>
<td>12°</td>
<td>449</td>
<td>95</td>
<td>2×45</td>
<td></td>
<td>3.8×10⁶</td>
<td>0.80</td>
</tr>
</tbody>
</table>

Table 3 presents the sizes of the computational domains for the drift case calculations. For increasing drift angles, the computational domain size is increased in order to be able to incorporate the inner block in the outer deforming mesh.

<table>
<thead>
<tr>
<th>β</th>
<th>inlet Lpp</th>
<th>outlet Lpp</th>
<th>width Lpp</th>
<th>depth Lpp</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>-0.73</td>
<td>0.92</td>
<td>2×0.18</td>
<td>0.18</td>
</tr>
<tr>
<td>3°</td>
<td>-0.74</td>
<td>0.93</td>
<td>2×0.42</td>
<td>0.36</td>
</tr>
<tr>
<td>6°</td>
<td>-0.75</td>
<td>0.94</td>
<td>2×0.49</td>
<td>0.36</td>
</tr>
<tr>
<td>9°</td>
<td>-0.76</td>
<td>0.95</td>
<td>2×0.55</td>
<td>0.36</td>
</tr>
<tr>
<td>12°</td>
<td>-0.76</td>
<td>0.95</td>
<td>2×0.61</td>
<td>0.38</td>
</tr>
</tbody>
</table>

For a drift angle of 12°, a series of geometrically similar grids has been generated in order to investigate the discretisation error. The grid coarsening has been conducted in all three directions. For some of the grids however, the distance of the first node to the hull surface has been maintained in order to capture the velocity gradients in the boundary layer. Table 4 presents the number of nodes and y2+ values for these grids.
Table 4: Properties of grids for uncertainty analysis.

<table>
<thead>
<tr>
<th>id</th>
<th>β</th>
<th>n₀</th>
<th>n₁</th>
<th>n₂</th>
<th>h₀</th>
<th>nodes</th>
<th>y²</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12°</td>
<td>449</td>
<td>95</td>
<td>2×45</td>
<td>1.00</td>
<td>3.8×10²</td>
<td>0.80</td>
</tr>
<tr>
<td>2</td>
<td>12°</td>
<td>409</td>
<td>87</td>
<td>2×41</td>
<td>1.10</td>
<td>2.9×10²</td>
<td>0.75</td>
</tr>
<tr>
<td>3</td>
<td>12°</td>
<td>361</td>
<td>81</td>
<td>2×37</td>
<td>1.24</td>
<td>2.2×10²</td>
<td>0.62</td>
</tr>
<tr>
<td>4</td>
<td>12°</td>
<td>329</td>
<td>74</td>
<td>2×33</td>
<td>1.37</td>
<td>1.6×10²</td>
<td>0.71</td>
</tr>
<tr>
<td>5</td>
<td>12°</td>
<td>297</td>
<td>65</td>
<td>2×29</td>
<td>1.51</td>
<td>1.1×10²</td>
<td>0.94</td>
</tr>
<tr>
<td>6*</td>
<td>12°</td>
<td>249</td>
<td>65</td>
<td>2×25</td>
<td>1.81</td>
<td>8.1×10¹</td>
<td>0.78</td>
</tr>
<tr>
<td>7**</td>
<td>12°</td>
<td>225</td>
<td>48</td>
<td>2×23</td>
<td>2.00</td>
<td>5.0×10¹</td>
<td>1.15</td>
</tr>
<tr>
<td>8***</td>
<td>12°</td>
<td>177</td>
<td>41</td>
<td>2×19</td>
<td>2.55</td>
<td>2.8×10¹</td>
<td>1.25</td>
</tr>
<tr>
<td>9***</td>
<td>12°</td>
<td>145</td>
<td>33</td>
<td>2×15</td>
<td>3.11</td>
<td>1.4×10¹</td>
<td>1.73</td>
</tr>
<tr>
<td>10**</td>
<td>12°</td>
<td>121</td>
<td>33</td>
<td>2×13</td>
<td>3.73</td>
<td>1.0×10¹</td>
<td>1.32</td>
</tr>
<tr>
<td>11**</td>
<td>12°</td>
<td>113</td>
<td>24</td>
<td>2×12</td>
<td>4.00</td>
<td>6.5×10¹</td>
<td>2.17</td>
</tr>
</tbody>
</table>

For grid 5, it was unfortunately not possible to converge the results until the required convergence criterion was reached. Therefore the results for this grid are dropped from further analysis.

**Boundary conditions**

At the ship surface the no-slip condition is applied directly and the normal pressure derivative is assumed to be zero. The undamped eddy viscosity, the variable in Menter's one-equation model, vanishes at a no-slip wall.

Symmetry conditions are applied at the undisturbed water surface and on the ship symmetry plane (for the zero-drift condition). At the inlet boundary, the velocity profiles are obtained from a potential flow solution, which also determines the tangential velocity components and the pressure at the external boundary. At the outlet boundary, stream-wise diffusion is neglected and the stream-wise pressure derivative is set equal to zero.

For the drift cases, the lift generated by the hull form is modelled in the potential flow solution by applying a vortex sheet on the symmetry plane of the ship. At the stern of the ship, the Kutta condition (the flow leaves the trailing edge smoothly) is applied, which allows the solution of the unknown vortex strengths on the sheet. Since the only purpose of the potential flow solution is to set the boundary conditions for the viscous flow solution at the inlet and external boundaries, vortex shedding from the bilges of the ship is omitted.

**RESULTS AND DISCUSSION**

**Numerical Convergence**

In the calculations a reduction of the maximum difference in pressure between consecutive iterations to 5×10⁻⁵ was adopted as the convergence criterion. It is assumed that this is sufficiently small compared to the discretisation error and therefore the iteration error is ignored in the uncertainty analysis.

For all cases, the adopted convergence criterion results in a reduction of the difference in the (total) force and moment components between consecutive iterations of well below 1×10⁻⁵.

**Computing times**

All computations except for 6° drift angle have been conducted on a PC using a single Pentium 4 processor with 2.4 GHz clock cycle frequency and 1 GB of internal memory. The calculations for 6° drift angle have been conducted on a SGI supercomputer and the computation times for this computer have not been recorded.

The computing times required for the calculations are presented in Table 5.

Table 5: Computing times.

<table>
<thead>
<tr>
<th>id</th>
<th>β</th>
<th>nodes</th>
<th>iterations</th>
<th>CPU time</th>
<th>u(n₀n₁)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>12°</td>
<td>1.6×10⁶</td>
<td>569</td>
<td>33178</td>
<td>3.6</td>
</tr>
<tr>
<td>3°</td>
<td>3.8×10⁶</td>
<td>549</td>
<td>130393</td>
<td>6.2</td>
<td></td>
</tr>
<tr>
<td>6°</td>
<td>3.8×10⁷</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>9°</td>
<td>3.8×10⁷</td>
<td>1175</td>
<td>266985</td>
<td>5.9</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>12°</td>
<td>3.8×10⁶</td>
<td>1885</td>
<td>343179</td>
<td>4.7</td>
</tr>
<tr>
<td>2</td>
<td>12°</td>
<td>2.9×10⁶</td>
<td>1251</td>
<td>151967</td>
<td>4.2</td>
</tr>
<tr>
<td>3</td>
<td>12°</td>
<td>2.2×10⁶</td>
<td>958</td>
<td>98848</td>
<td>4.8</td>
</tr>
<tr>
<td>4</td>
<td>12°</td>
<td>1.6×10⁶</td>
<td>1403</td>
<td>85910</td>
<td>3.8</td>
</tr>
<tr>
<td>5</td>
<td>12°</td>
<td>8.1×10⁵</td>
<td>794</td>
<td>38242</td>
<td>6.0</td>
</tr>
<tr>
<td>6</td>
<td>12°</td>
<td>5.0×10⁵</td>
<td>644</td>
<td>10543</td>
<td>3.3</td>
</tr>
<tr>
<td>7</td>
<td>12°</td>
<td>2.8×10⁵</td>
<td>360</td>
<td>2238</td>
<td>2.2</td>
</tr>
<tr>
<td>8</td>
<td>12°</td>
<td>1.4×10⁵</td>
<td>329</td>
<td>1061</td>
<td>2.2</td>
</tr>
<tr>
<td>9</td>
<td>12°</td>
<td>1.0×10⁵</td>
<td>330</td>
<td>758</td>
<td>2.2</td>
</tr>
<tr>
<td>10</td>
<td>12°</td>
<td>6.5×10⁴</td>
<td>363</td>
<td>613</td>
<td>2.6</td>
</tr>
</tbody>
</table>

**Uncertainty analysis**

In this and following sections, the forces and moments presented are made non-dimensional using respectively ½ρU²LₚₚT and ½ρU²Lₚₚ²T, in accordance with specifications for the Tokyo CFD Workshop 2005. CX is the longitudinal force, CY the transverse force, CZ the vertical force, CK the heeling moment, CM the pitching moment and CN
the yawing moment with respect to the origin of the
xyz coordinate system, which is located at station 10.
For a drift angle of 12°, the predicted values of the
friction (index f) and pressure (index p) components
as well as the total force and moment coefficients are
presented in Table 6 with the estimated uncertainties.
Based on an analysis of the results for each grid, it
was decided to use the 6, 7 or 8 finest grids for the
uncertainty analysis. The number of grids \( n_g \) used
depended on the scatter in the results for the coarsest
grids.

Table 6: Uncertainty analysis, \( \beta = 12^\circ \).

<table>
<thead>
<tr>
<th>Quantity</th>
<th>( n_g )</th>
<th>( \phi_0 )</th>
<th>( \phi_1 )</th>
<th>( U_\phi )</th>
<th>( p )</th>
</tr>
</thead>
<tbody>
<tr>
<td>CX</td>
<td>7</td>
<td>-</td>
<td>-1.78x10^{-2}</td>
<td>12.0%</td>
<td>(2)</td>
</tr>
<tr>
<td>CXf</td>
<td>6</td>
<td>-1.57x10^{-3}</td>
<td>-1.54x10^{-2}</td>
<td>2.1%</td>
<td>1.45</td>
</tr>
<tr>
<td>CXp</td>
<td>7</td>
<td>-</td>
<td>-2.32x10^{-2}</td>
<td>50.0%</td>
<td>(3)</td>
</tr>
<tr>
<td>CY</td>
<td>7</td>
<td>6.67x10^{-1}</td>
<td>6.43x10^{-1}</td>
<td>5.6%</td>
<td>1.13</td>
</tr>
<tr>
<td>CYf</td>
<td>6</td>
<td>1.84x10^{-1}</td>
<td>1.70x10^{-1}</td>
<td>13.4%</td>
<td>1.34</td>
</tr>
<tr>
<td>CYp</td>
<td>7</td>
<td>6.45x10^{-1}</td>
<td>6.26x10^{-1}</td>
<td>4.8%</td>
<td>1.25</td>
</tr>
<tr>
<td>CZ</td>
<td>7</td>
<td>3.41x10^{-2}</td>
<td>3.21x10^{-2}</td>
<td>4.7%</td>
<td>0.51</td>
</tr>
<tr>
<td>CZf</td>
<td>6</td>
<td>3.20x10^{-3}</td>
<td>3.20x10^{-3}</td>
<td>4.7%</td>
<td>0.52</td>
</tr>
<tr>
<td>CZp</td>
<td>7</td>
<td>3.00x10^{-2}</td>
<td>3.00x10^{-2}</td>
<td>4.7%</td>
<td>0.52</td>
</tr>
<tr>
<td>CK</td>
<td>7</td>
<td>-</td>
<td>-3.07x10^{-2}</td>
<td>10.3%</td>
<td>(2)</td>
</tr>
<tr>
<td>CKf</td>
<td>6</td>
<td>2.16x10^{-4}</td>
<td>1.74x10^{-4}</td>
<td>12.9%</td>
<td>0.44</td>
</tr>
<tr>
<td>CKp</td>
<td>8</td>
<td>-</td>
<td>-3.24x10^{-4}</td>
<td>19.7%</td>
<td>(1)</td>
</tr>
<tr>
<td>CM</td>
<td>7</td>
<td>-3.94x10^{-2}</td>
<td>-3.86x10^{-2}</td>
<td>6.1%</td>
<td>(2)</td>
</tr>
<tr>
<td>CMf</td>
<td>7</td>
<td>1.09x10^{-1}</td>
<td>1.08x10^{-1}</td>
<td>0.7%</td>
<td>1.71</td>
</tr>
<tr>
<td>CMP</td>
<td>7</td>
<td>-3.97x10^{-2}</td>
<td>-3.97x10^{-2}</td>
<td>5.9%</td>
<td>(2)</td>
</tr>
<tr>
<td>CN</td>
<td>7</td>
<td>-</td>
<td>2.53x10^{-2}</td>
<td>14.8%</td>
<td>(2)</td>
</tr>
<tr>
<td>CNf</td>
<td>7</td>
<td>-2.94x10^{-4}</td>
<td>39.3%</td>
<td>0.39</td>
<td></td>
</tr>
<tr>
<td>CNp</td>
<td>7</td>
<td>-</td>
<td>2.55x10^{-4}</td>
<td>14.2%</td>
<td>(2)</td>
</tr>
</tbody>
</table>

(1) Oscillatory convergence
(2) Monotonous divergence
(3) Oscillatory divergence

The absolute uncertainty in the pressure components
is larger than in the friction components. The
uncertainty in the longitudinal friction component is
about one-third of the uncertainty in the longitudinal
pressure component. For the other forces and
moments, the uncertainty in the friction component is
at least one order of magnitude smaller than the
uncertainty in the pressure component. Since most
integral forces and moments are dominated by the
pressure component, this results in relatively large
uncertainties in the overall forces and moments.

In Figure 2 the friction component of the longitudinal
force is graphically presented for the different grids.
Although scatter exists, the results appear to converge
for a relative step size below 2.5. However, due to
scatter in the pressure component, convergence is not
found for the overall longitudinal force coefficient.

In Figure 3, the convergence of the side force
coefficient with grid refinement is presented. It is
seen that upon grid refinement, the estimated value
for CY (indicated by cfd) comes closer to the
experimental value (indicated by exp). Considerable
scatter is visible on the data and therefore it is not
easy to establish whether data points are located in
the asymptotic range of convergence. This is typical
for this type of calculation, as already observed
previously by e.g. Eça et al. [10] and Hoekstra et al.

Looking at the yawing moment CN, the maximum
difference between the estimated values for all grids
is 5.1%. Because the difference between the
estimated values is relatively small and scatter on the
data is found, monotonous divergence is found and
extrapolation to zero step size could not be made.
This results in a relatively large uncertainty of 14.8%.
Figure 4 shows however that the results on the finest
grids are within 0.5% from the measured value. Even the result for the coarsest grid is within 3.1% from the measurement, which is very acceptable.

To further verify the results, the velocities along a horizontal cut located behind the propeller hub are compared for several different grid densities. This cut was located in a plane (designated the WAKE1 plane) perpendicular to the flow at a distance along the longitudinal axis of the ship of 0.48·LPP behind midship and at a vertical position of z=-0.05. Figure 5 presents the axial, transverse and vertical velocities obtained from the results of grids 1 through 4 and 6, together with the experimental results.

In these graphs, it is seen that upon grid refinement the results in general come increasingly closer to the measurements. Additionally, the differences between two successive grids reduce upon grid refinement.

A further investigation is made of the convergence of the pressure on the hull at two different locations: x=-0.4 and x=0.4. For x=-0.4 (bow), see Figure 6, it is seen that only marginal differences exist between the different grids. Except for the location at which the vortex generated at the bow leaves the hull (approximately at y=0.012) no significant differences exist. This indicates that for the flow around the bow the discretisation is sufficiently dense for the grids selected for the comparison.

For the pressure at the hull surface at x=0.4, see Figure 7, the differences are more pronounced.
Especially at the position of the separation of the vortex generated at the stern (at y=0.012) differences between the successive grids are visible. Also more to the starboard (leeward) side, differences between the coarsest grid in the graph (grid 6, hi = 1.81) and the finer grids appear. The results from grid 6 fail to capture the strong gradients in the pressure distribution.

Finally, the change in the longitudinal distribution of the side force upon grid refinement is examined. Figure 8 shows that once again, the results on the finest grid approximate the experimental results best. At the bow and midship region (-0.6 < x < 0.2), the difference between the solutions at various grid resolutions is negligible but at the stern (x > 0.2), the coarser grids in the graph do not follow the experiments as well as the finer grids. Similar to Figure 3, this means that the side forces on the coarser grids are slightly underpredicted compared to the finer grids.

From the uncertainty analysis, it is concluded that a proper choice of the grid density may depend on the purpose of the calculations. A fine grid is required to arrive at accurate results. However, for comparative purposes a coarser grid solution still captures all relevant flow phenomena. In optimisation studies, a coarser grid might be a good option to compare different hull forms.

Even between the finest grids, some differences are still clearly visible, in integral variables as well as in local field quantities. Other studies for zero drift, see Eça and Hoekstra [5], indicate the possibility to obtain a lower uncertainty when the grid nodes in girth-wise direction are stretched to the water plane. This will be examined in further studies.

Validation of integral coefficients

Table 7 presents the results of the calculations for each drift angle $\beta$ as well as a comparison between the calculated variables and the measured ones. In the following sections, only the results found with the finest grid will be discussed for 12° drift angle.

<table>
<thead>
<tr>
<th>$\beta$</th>
<th>cfd</th>
<th>exp</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>CX</td>
<td>CY</td>
</tr>
<tr>
<td></td>
<td>$\times 10^2$</td>
<td>$\times 10^2$</td>
</tr>
<tr>
<td>$-3^\circ$</td>
<td>1.77</td>
<td>0.91</td>
</tr>
<tr>
<td>0°</td>
<td>1.74</td>
<td>0.00</td>
</tr>
<tr>
<td>3°</td>
<td>1.77</td>
<td>0.91</td>
</tr>
<tr>
<td>6°</td>
<td>1.79</td>
<td>2.26</td>
</tr>
<tr>
<td>9°</td>
<td>1.79</td>
<td>4.23</td>
</tr>
<tr>
<td>12°</td>
<td>1.78</td>
<td>6.43</td>
</tr>
</tbody>
</table>

Table 8: Error estimates

<table>
<thead>
<tr>
<th>$\beta$</th>
<th>$\varepsilon_{CX}$</th>
<th>$\varepsilon_{CY}$</th>
<th>$\varepsilon_{CN}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$-3^\circ$</td>
<td>-2%</td>
<td>-27%</td>
<td>8%</td>
</tr>
<tr>
<td>0°</td>
<td>1%</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>3°</td>
<td>1%</td>
<td>-28%</td>
<td>25%</td>
</tr>
<tr>
<td>6°</td>
<td>1%</td>
<td>-12%</td>
<td>5%</td>
</tr>
<tr>
<td>9°</td>
<td>4%</td>
<td>-7%</td>
<td>3%</td>
</tr>
<tr>
<td>12°</td>
<td>2%</td>
<td>-9%</td>
<td>-1%</td>
</tr>
</tbody>
</table>

Figure 8: Side force distribution, $\beta=12^\circ$ (bow to the left of the figure)

Figure 9: Integral values as a function of drift angle

Table 7: Integral values

Table 8: Error estimates
Except maybe for the results for 3° drift angle, the predictions obtained by the calculations are very promising. In almost all cases the prediction error $\varepsilon$ (defined by $\varepsilon = \phi_{\text{cfd}}/\phi_{\text{exp}} - 1$) is within 10% from the measurements. Noteworthy is the consistent underprediction of the transverse force, while both the longitudinal force and yawing moment are predicted quite accurately. Figure 9 presents the longitudinal and transverse forces and the yawing moment as a function of the drift angle. More results can be found in Eça et al. [12] and in the proceedings of the Tokyo CFD Workshop 2005.

**Longitudinal side force distribution**

To understand the manoeuvrability of ships and to be able to generate reliable generic mathematical manoeuvring models, the longitudinal distribution of the side force is of interest. Therefore, the predicted longitudinal distribution of the lateral force has been compared to the experimental values to determine the accuracy of the predictions, see Figure 10.

The comparison shows that although the side force according to Table 7 is systematically underpredicted, the predicted distributions are for both drift angles very close to the measurements. Apparently the physics of the force distribution are captured well by the calculations and therefore the accuracy of this prediction is judged to be good.

![Figure 10: Side force distribution (bow to the left of the figure)](image)

**Comparison of field quantities**

Experimental data are available at a transverse cut just behind the propeller hub (in the WAKE1 plane at $z=-0.05$). The graphs in Figure 11 provide a comparison between the experiments (markers) and calculations (lines). Except for a small region at the windward side ($-0.02<y<0$ for $\beta=12^\circ$) the calculations follow the measurements closely. Some discrepancies are seen in the prediction of the axial velocity between $0<y<0.02$, but the transverse and vertical velocities in this region are matching the experiments. The velocity profile in this area is influenced by flow separation from the propeller hub and by the vortex shed from the stern, see also Figure 7.

![Figure 11: Velocity in WAKE1 plane, $z=-0.05$](image)

For the present study, the experimental data of the wake field in the complete WAKE1 plane has kindly been made available by NMRI. Figure 12 presents a comparison of the axial velocity fields between the experiments (dotted lines) and the calculations (solid lines) for $0^\circ$, $6^\circ$ and $12^\circ$ drift angle. This figure shows that in most parts of the plane, the viscous-flow calculations correspond well with the experiments. Even for $12^\circ$ drift angle, the strength and position of the vortex generated at the starboard bow (its centre is located at $y=0.11, z=-0.03$) is quite accurately captured by the calculations.
In the port side area (windward), discrepancies are found for the 12° drift case, however. In the calculations, the contour lines are straightened while for the experiments, the contour lines retain their hook-shape. Also just behind the propeller hub for 0° drift angle, the hook-shape in the measurements appears more pronounced than in the calculations. This can be attributed to the turbulence modelling, as was also observed by Eça et al. [10]. A further study using a k-ω turbulence model is proposed.

Overall, it is concluded that the flow field at the propeller plane is quite accurately predicted.

![Figure 12: Axial velocity in WAKE1 plane, (solid lines: cfd, dotted lines: exp; top β=0°, middle β=6°, bottom β=12°)](image)

CONCLUSIONS

Simulations have been conducted of the viscous flow around the KVLCC2M hull form at several drift angles. For 12° drift angle, a grid convergence study was performed to study the uncertainty in the results. The finest grid used in this study contained 3.8 million points.

It is shown that with the finest grid, no significant changes in the flow field quantities occur, compared to the second finest grid. Some integral quantities however still vary upon grid refinement. It is shown that differences between solutions on increasingly finer meshes tend to decrease. But especially the pressure components of the forces appear sensitive to the grid refinement. It is concluded that possibly the number of grid nodes in girth-wise direction needs to be increased for improved accuracy.

Detailed comparisons with experimental data show that the main flow features are well predicted even when looking at discrete positions in the flow field. Qualitatively, promising results are obtained. For practical purposes however, the accuracy of the results should be improved. For the current calculations, the predicted yaw moment is close to the measurements but the side force is underpredicted. Reason for these discrepancies might be the neglect of the water surface deformation. Some aspects of the calculated flow fields can also be improved by choosing a different turbulence model. Both should be studied in future research.

ACKNOWLEDGEMENT

Part of the work conducted for this paper has been funded by the Commission of the European Communities for the Integrated Project VIRTUE. This project is part of the Sixth Research and Technological Development Framework Programme (Surface Transport Call).

REFERENCES


