MODEL- AND FULL-SCALE RANS SIMULATIONS FOR A DRIFT TANKER

(DOI No: 10.3940/rina.ijme.2017.a2.419)

J Yao, Key Laboratory of High Performance Ship Technology (Wuhan University of Technology), Ministry of Education; School of Transportation, Wuhan University of Technology, China

SUMMARY

The flow around a full-scale (FS) ship can be simulated by means of Reynolds-Averaged Naiver-Stokes (RANS) method, which provides a way to obtain more knowledge about scale effects on ship hydrodynamics. In this work, the viscous flow around a static drift tanker in full scale is simulated by using the RANS solver based on the open source platform OpenFOAM. The $k-\omega$ SST model is employed to approximate the eddy viscosity. To reduce computational time, wall function approach is applied for the FS simulation. The flow around the ship in model scale is simulated as well, but without using any wall function, i.e., using Low-Reynolds number mode. In order to verify the computations, detailed studies on the computational grid including investigation of the sensitivity of computed forces to y^+ (dimensionless distance of first grid point to wall) and grid dependency study are carried out. The computed forces are compared with available measured data. The scale effects are analysed and discussed by comparisons.

1. INTRODUCTION

Generally speaking, in the initial stage of ship design, a reliable way to access its hydrodynamic performance is to conduct model test in a tank. However, as we know due to scale effects the measured data must be further processed to get its real performance in full scale, i.e., the data need to be corrected to FS ship. Generally, for resistance we can use an empirical formula to correct ship friction coefficient, e.g. the one from 1957 International Towing Tank Conference (ITTC). That is actually for a flat plate. Of course, for a more real situation the roughness allowance should be considered as well, as proposed in the 19th ITTC (1990). However, for more complex problems, e.g. static drift for manoeuver, until now recommendations about the corrections of scale effect have been rarely reported in exiting publications. Therefore, to gain an insight into scale effects for such cases is of great significance to engineering application.

In past decades, Computational Fluid Dynamics (CFD) methods, especially the one based on solving RANS equations, made great progresses in ship hydrodynamics. It has been demonstrated in numerous publications that the simulations of ship motions by using RANS-based approaches can reach very high accuracy compared with experimental data. Theoretically, RANS method can be applied for the study of scale effect, but there remains challenges due to that full-scale (FS) RANS simulations usually require a lot of computational cells leading to extremely high computational time and it is very difficult to validate the computations because of a lack of experiment data. Thus, the existing studies on scale effects in ship hydrodynamics focus more on simple problems, e.g. resistance. For example, Ju and Patel (1994) improved a RANS method to predict the resistance components and nominal wake for a tanker in full and model scale. Eca and Hoekstra (2000) used eddy-viscosity turbulence models to predict scale effects on ship stern flows. More recently, Castro et al. (2011) mainly studied the resistance problem for a free surface container ship by RANS simulations. In all these researches, the scale effects on ship resistance and the stern flow were discussed and analysed. It was from their studies concluded that the predicted ship boundary layer at full scale was much thinner than that at model scale.

As stated above, rare publications have reported about scale effects on the flow around a drift ship. In this study, a tanker is taken as an example, and the turbulent flows around the bare hull in full and model scale are simulated by using the steady RANS solver in OpenFOAM. In the simulations, the $k-\omega$ SST model (Menter et al., 2003) is used to approximate eddy viscosity. For the FS simulation, wall function approach is applied to reduce computational time, while for the model-scale (MS) simulation no wall function is used. To verify the simulations investigation of the sensitivity of computed forces to y^+ and grid dependency study are firstly carried out. The predicted friction components of resistance (at zero drift angle) from both FS and MS simulations are validated by comparisons with the ones by 1957 ITTC correction formula. The computed results are compared with available measured data. The scale effects are analysed and discussed by comparisons of computed hydrodynamic forces and turbulence characteristics.

2. GENERAL CONSIDERATIONS

When using near-wall turbulence models in RANS method, a high grid resolution is required in the near wall region, since the first grid point near the wall has to be in sub-layer. As recommended by Wilcox (1993), y^+ should be less than 1 or at least less than 2. For FS simulations, such y^+ requirement causes very huge aspect ratios of computational cells. This will produce significantly large errors in flux calculations and may lead to numerical divergence. Theoretically, to overcome the problem, we can reduce cell size until a satisfying y^+ is achieved when generating grid, but this will lead to a surprising number of computational cells for ship geometries, e.g. several hundred million cells. At least until now the extremely high grid density remains unaffordable.

The use of wall functions avoids the above limitations of near-wall turbulence models and considerably decreases computational time. The wall function approach was firstly proposed by Launder and Spalding (1974) and a few more advanced versions were developed later, e.g. the one proposed by Shih et al. (2003). The advantage of wall function approach is that grids with larger y^+ are allowed, however using wall functions has its own requirement, i.e., the first grid point must be in log-layer, usually $y^+ > 30$. Many studies reported in literatures have validated the wall function approach and generally good results were obtained by it. Anyway, the advantage makes it possible to simulate FS ship flows by RANS method.

One issue when using wall function approach is that an appropriate y^+ need to be firstly found out. If the wall function approach is used in both FS and MS simulations, we have to generate different grids and find appropriate y^+ for both respectively. If so, it decreases the comparability of the computed results more or less because of the different y^+ and using different grids. In this work, another strategy is adopted to achieve the study, i.e., a same grid but at different scale is used for FS and MS simulations, however, for the FS simulation wall function approach is adopted. Since according to many pre-computations, if both simulations are based on a same grid (at different scale), when the mean value of y^+ is in range of log-layer for the FS simulation, the mean y^+ is exactly around 1 for the MS simulation. This means y^+ in both full and model scale satisfy their own application condition respectively.

The tanker considered here is named KVLCC2, a very large crude carrier designed by Maritime & Ocean Engineering Research Institute (MOERI), Korea. No FS ship was built and it was a widely-used benchmark hull for comparison purpose on ship CFD Workshops, e.g. the Workshops of Tokyo (Hino, 2005) and Gothenburg (Larsson et al., 2010). Due to the low speed, the deformation of free surface together with the related effects such as sinkage and trim is not considered in present simulations. Moreover, the study by Castro et al. (2011) showed that scale effects almost did not have impact on ship wave. So that neglecting the free surface elevation in this study is expected to have negligible influences on the numerical accuracy and main conclusions.

The RANS solver in OpenFOAM is based on a finite volume method. The related issues including governing equations, turbulence model, boundary condition, and so on, have been already described in previous publication by the author (Yao, 2015), in which the solver has been validated for MS simulations, showing a promising accuracy.

3. SHIP DATA, GRID AND BOUNDARY CONDITIONS

KVLCC2 is the second variant of the MOERI tanker with more U-shaped stern. The main particulars of the tank in full and model scale are listed in Table 1, where L_{pp} is the length between perpendiculars, B is breadth, T is draught, C_B is block coefficient F_r is Froude number and Re is the Reynolds number. As well know, usually when Re reaches around 10^5, the flow becomes fully turbulent.

Table 1 Particulars of the tanker in full- and model scale, scale ratio 1:110

Particulars	FS	MS
L_{pp} $[m]$	320	2.9091
B $[m]$	58	0.5273
T $[m]$	20.8	0.1891
C_B	0.8098	0.8098
Approach speed, F_r	7.96 $[m/s]$, 0.142	0.76 $[m/s]$, 0.142
Re	2.55×10^9	2.21×10^6

A ship-fixed Cartesian coordinate system *o-xyz* is defined to describe ship motion, where the origin *o* is located at the intersection of mid-ship sections and undisturbed free surface, *x*-axis towards bow, *y*-axis towards starboards and *z*-axis vertical downwards. The computational domain is limited by a box. The boundary of the box in front of the ship extends to $1.5L_{pp}$ from *o*. The boundary behind the ship extends to $2L_{pp}$. The side boundaries and bottom boundary extend to $1.5L_{pp}$ and L_{pp} respectively.

An unstructured grid is generated for the simulations by using the software Gambit. Figure 1 shows a half unstructured grid in a single block at starboard side of the KVLCC2 and enlarged view at stern. The grid consists of triangular prisms in the attached layers and tetrahedrons in the remaining space. It can be generated in a simple way: first generate surface grid on hull surface using triangular elements; next extrude the hull surface mesh to form a zone of mesh layers wrapping around the hull; finally fill the rest of the domain fully with tetrahedrons. The tetrahedron size is controlled by an increasing ratio (here 1.2) with which the edge length of tetrahedron increases with the distance from the outmost mesh layer.

This study involves four types of boundary conditions: wall, inlet, outlet and symmetry. Because of the assumption of double-body flow, the free surface is seen as a symmetry plane. A symmetry boundary condition is imposed on it. On the wall (hull) the no-slip condition is imposed. For boundaries in far field, the boundaries located on windward side relative to the main flow are set as inlets, while if the boundaries locate on leeward side relative to the main flow, they are treated as outlets.

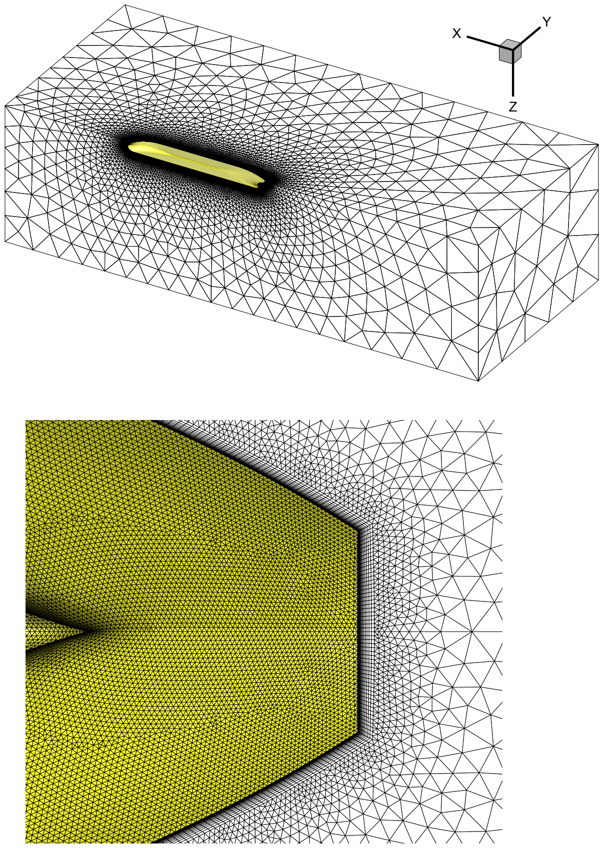


Figure 1. A half unstructured grid at starboard side of KVLCC2 (left) and enlarged view at stern (right)

For the FS simulation, the near-wall treatment of wall function approach, i.e., High-Reynolds number mode, is activated, while for the MS simulation the Low-Reynolds number mode (without using wall functions) is applied. More details on this point can be found in previous publication.

4. CHOICE OF y^+ AND GRID DEPENDENCE STUDY

4.1 SENSITIVITY INVESTIGATION OF y^+

In order to investigate the sensitivity of y^+ to the computed results, four grids are generated by systematically decreasing the spacing of the first grid point to hull surface. The information of the grids is summarized in Table 2. The four grids have the same surface mesh on hull surface. Starting from grid1 to grid4, the spacing of the first grid point to hull surface reduces by half. The increasing ratio of thickness between adjacent layers is 1.2 for each grid. The number of mesh layers varies from 10 to 20 from grid1 to grid4.

Table 2. Information of the grids for sensitivity investigation of y^+

grid	NH (1)	NC (2)	NL (3)	$\Delta y/L_{pp}$ (4)	δ/L_{pp} (5)
grid1	30828	1128854	10	1.375×10^{-4}	0.00357
grid2	30828	1214222	13	0.687×10^{-4}	0.00333
grid3	30828	1334892	17	0.344×10^{-4}	0.00361
grid4	30828	1473010	20	0.172×10^{-4}	0.00321

(1) NH is the number of surface elements on the hull surface.
(2) NC is the number of volume elements in the whole domain.
(3) NL is the number of mesh layers attached to the hull surface.
(4) Δy is the spacing of first grid point to the hull surface.
(5) δ is the total thickness of mesh layers.

The static drift motion at $\beta = 16°$ is simulated by using the four grids in model and full scale, where β is drift angle. The computed results including coefficients of longitudinal force X', side force Y' and yaw moment N' are plotted in Figure 2. The forces and moment have been made non-dimensional by

$$Force' = \frac{Force}{0.5\rho u_0^2 L_{pp} T}, \qquad Moment' = \frac{Moment}{0.5\rho u_0^2 L_{pp}^2 T}.$$

where ρ is water density and u_0 is ship speed. The corresponding computed y^+ are showed in Table 3, including maximum, minimum and mean values.

It is seen from Table 3 that with decreasing Δy by half y^+_{mean} almost reduces by half as well for both FS and MS simulations, and the maximum and minimum values also decrease. For the FS simulations, all Δy are larger than 30, satisfying the condition that the first grid point is located in log-layer. For the MS simulations, whereas, only the y^+ based on the grid3 and grid4 meet the condition, i.e. the first grid point is located in sub-layer, usually $y^+ < 5$.

Table 3. Summary of y^+ obtained from the y^+ sensitivity investigation

grid	FS			MS		
	y^+_{min}	y^+_{max}	y^+_{mean}	y^+_{min}	y^+_{max}	y^+_{mean}
grid1	712.74	7708.63	4241.15	0.30	11.72	5.97
grid2	239.38	3357.95	2111.09	0.10	7.07	2.92
grid3	90.88	1751.01	1050.71	0.064	3.66	1.43
grid4	35.48	913.28	525.89	0.029	1.79	0.72

Meanwhile we observe from Figure 2 that the computed results obtained from FS and MS simulations have large differences, especially for X'. When decreasing y^+ the change of X', as well as Y' and N', becomes smaller gradually, but Y' and N' based on the FS simulations show a bigger change. This means the FS simulation is more sensitive to y^+ than the MS simulation. Actually, less than 0.8% difference is found between the results based on grid3 and grid4 for FS and MS simulations, in particular for the latter the side force, as well as the yaw moment, is nearly unchanged.

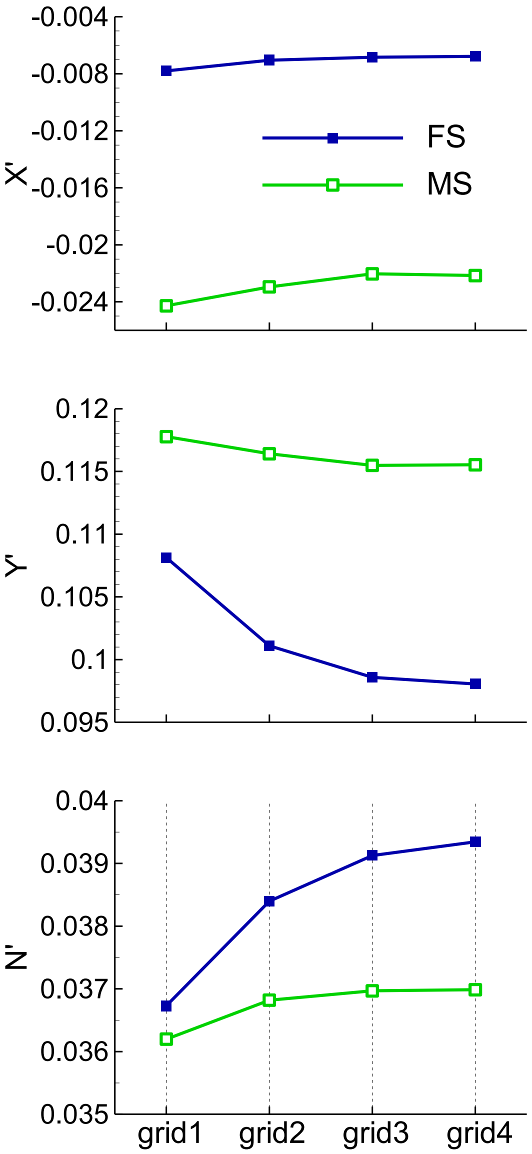
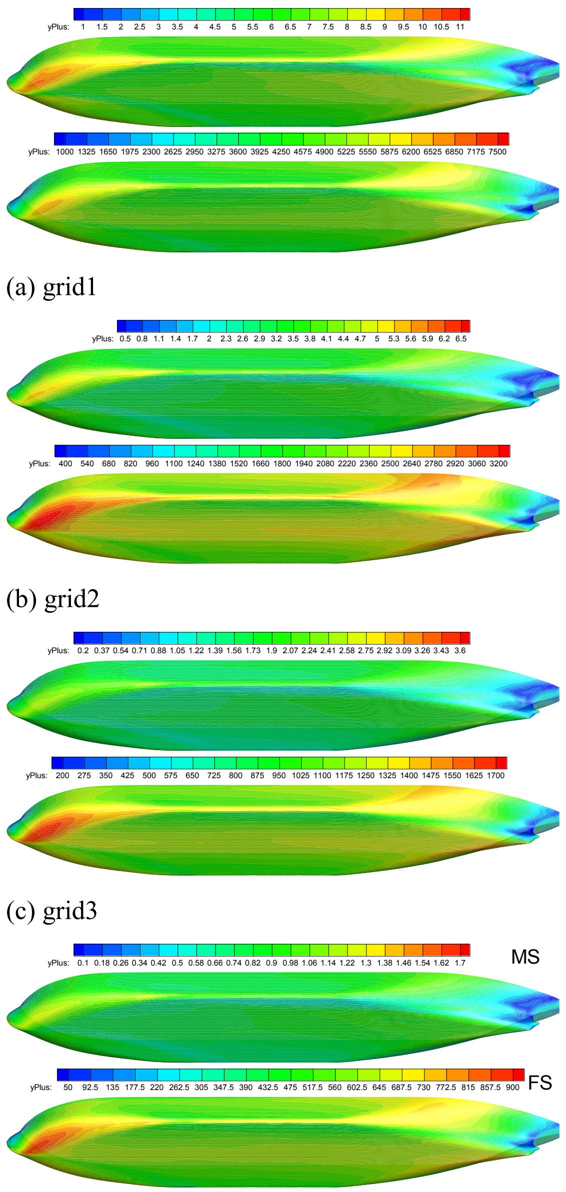


Figure 2. Computed results obtained from the sensitivity investigation of y^+

Figure 3 shows the y^+ distributions on the hull surface obtained by using the four grids in model and full scale. We notice firstly that the distribution patterns of y^+ based on the four grids are very similar to each other for the MS simulations or FS simulations. y^+ changes rapidly near the blunt parts of the hull, such as bow, stern and bilge, where flow velocity varies rapidly as well. Whereas y^+ changes mildly in the region of ship bottom and ship sides with small curvature.

The y^+ distributions based on FS and MS simulations display similar features, e.g. the maximum and minimum values occur near the bow or stern. The range of small y^+ (here in blue) near the stern based on the FS simulations is obviously smaller than that based on the MS simulations. There also exist differences between the y^+ distributions on ship bottom. It seems that the y^+ in model scale shows a slower change on ship bottom.



(a) grid1

(b) grid2

(c) grid3

(d) grid4

Figure 3. y^+ distributions obtained by using the four grids in model and full scale

4.2 GRID DEPENDENCY STUDY

According to above investigation, the grid4 is selected to be the coarse grid and the initial grid to generate a medium and fine grid for the grid dependency study. The medium grid is generated by increasing the size of the triangle element of hull surface mesh of grid4 with the ratio $\sqrt{3}$. The fine grid is generated based on the medium grid in the same way. Other settings for generating the medium and fine grids remain unchanged. The grid details are listed in Table 4.

Table 4. Information of the grids for grid dependency study

grid	NH	NC	NL	$\Delta y/L_{pp}$	δ/L_{pp}
Coarse / grid4	30828	1473010	20	0.172×10^{-4}	0.00321
medium	108952	4052114	20	0.172×10^{-4}	0.00321
fine	340464	11715340	20	0.172×10^{-4}	0.00321

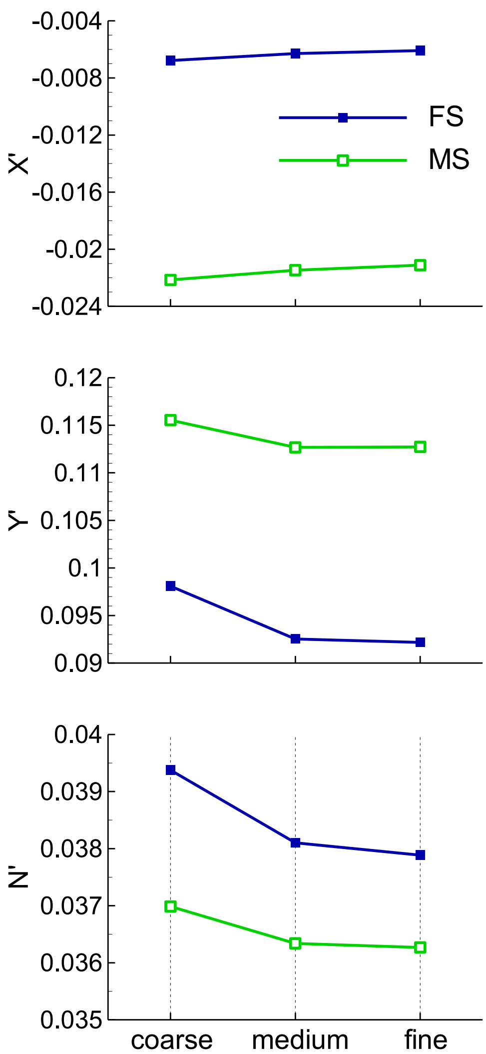


Figure 4. Computed results obtained from the grid dependency study

The difference among the computed y^+ distributions based on the three grids is small, due to the identical spacing of first grid point to hull surface. Figure 4 shows the computed X', Y' and N' obtained by using the coarse, medium and fine grids. We see from the figure that when refining the grid the change of X', Y' or N' becomes smaller, especially of Y' and N'. The difference between X', Y' or N' based on the medium and fine grid is less than 1%. However, the FS simulation shows more sensitivity to grid density than the MS simulation in general.

Based on the above sensitivity study of grid, we can conclude that when $\Delta y/L_{pp} = 0.172 \times 10^{-4}$ both the FS simulations and MS simulations satisfy their own using condition of near-wall treatment respectively, i.e. High-Reynolds and Low-Reynolds number mode, and when decreasing Δy the computed results show the feature of convergence and when the spacing reaches $0.172 \times 10^{-4} L_{pp}$ the results change very little. Although using the medium grid can obtain good results according to the grid dependency study, the fine grid is still used in the following computations due to the advantage that high grid density can obtain more detailed flow characteristics. On the other hand, the computational time using the fine grid is acceptable. Here a fine-grid-based computation requires around 1.5 days for the convergent solution using 4 processors on a small workstation.

5. VALIDATION AND DISCUSSION

5.1 RESISTANCE

The case of zero drift angle (i.e. resistance) is firstly validated since the computed friction resistance in full scale, as well as in model scale, can be compared with the one by the 1957 ITTC formula $C_F = 0.075/(\log_{10}\mathrm{Re} - 2)^2$, where C_F is friction coefficient and Re is Reynolds number. Here for the FS simulation Re is 2.55×10^9 and for the MS simulation Re is 2.21×10^6 as given in Table 1. As stated in previous section, the fine grid is used, however because of symmetric flow for resistance case here only a half computational domain is considered to reduce computational time.

Table 5 summarizes the results for resistance in full and model scale, where C_T and C_P are coefficients of total resistance and pressure respectively. The total resistance and its components are made non-dimensional by ship length, ship speed and wetted surface area of the ship hull in still water for comparison purpose. The measured C_T is from National Maritime Research Institute (NMRI) of Japan and published on the Workshop on Verification and Validation of Ship Manoeuvring Simulation Methods (SIMMAN, 2014). The C_F in full and model scale are computed by 1957 ITTC formula and $C_P = C_T - C_F$.

Table 5. Comparison of resistance with available data

Coefficients	MS			FS		
	CFD	Exp.	Err. (%)	CFD	Exp.	Err. (%)
$C_T \times 10^3$	5.14	5.25	-2.10	2.40	--	--
$C_F \times 10^3$	3.90	3.97*	-1.76	1.29	1.37*	-5.84
$C_P \times 10^3$	1.24	1.28	-3.13	1.11	--	--

*Computed by 1957 ITTC formula.

It is shown that the MS computation underestimates C_T, but only around 2.1% error is found. The underestimation may be due to the effects of free surface which is seen as a rigid plane in this study. The predicted C_F in model scale is quite close to the friction coefficient estimated by 1957 ITTC formula which corresponds to a flat plate of ship length. An underestimation is observed for C_P in model scale as well and the error is around 3.13%. The FS computation also underestimates C_F, but the error is a little bit large, here around 5.84%. There is an around 10% difference between the computed C_P in full and model scale, while the MS C_F is around three times larger than the FS C_F caused by scale effects. On the whole, the computed MS resistance agrees quite well with the measured data, and it can be expected that the FS computation is of certain accuracy since the friction coefficient is close to the estimated one.

5.2 STATIC DRIFT

The static drift motions at drift angle $\beta = 4°, 8°, 16°$ are simulated by using the fine grid in model and full scale. The computed X', Y' and N' are presented in Figure 5 including the measured data in model scale offered by NMRI on SIMMAN 2014. The comparison shows that the results in model scale are quite promising since the maximum error between the computed and measured results is less than 6%. The MS X' are in complete agreement with the measured X'. For the MS Y' or N', the error becomes larger when drift angle increases. The error is around 5.5% for N' at $\beta = 16°$.

The results based on the MS simulations have significant differences with that based on the FS simulations. The MS X' is around 2.2 times of the FS X' at $\beta = 0°$ and it increases to 3.5 times at $\beta = 16°$. This means the influence of scale effects on X' becomes larger at a larger drift angle, which can be explained by that when the ship performs a larger amplitude motion the turbulence around it becomes more complicated, e.g. occurrence of vortex, resulting in a larger impact on X'. However, this characteristic is not reflected by Y' and N'. All computed MS Y' are larger than the corresponding FS Y', and the differences are around 27.14%, 23.48% and 22.56% at $\beta = 4°, 8°, 16°$ respectively. Whereas, for N'all MS values are smaller than the corresponding FS values, and the differences are around 12.4%, 8.26% and 4.22% at $\beta = 4°, 8°, 16°$ respectively. Above analyses indicate that scale effects impact much on the hydrodynamic forces. In particular, scale effects lead to a more than 20% difference for Y', which means the v-related hydrodynamic derivatives in full and model scale, such as Y'_v, will have the same level difference, where v is lateral velocity of ship. As we know Y'_v is a main hydrodynamic derivative to determine ship course-keeping and turning abilities, so that scale effects will have much influence on ship manoeuvrability.

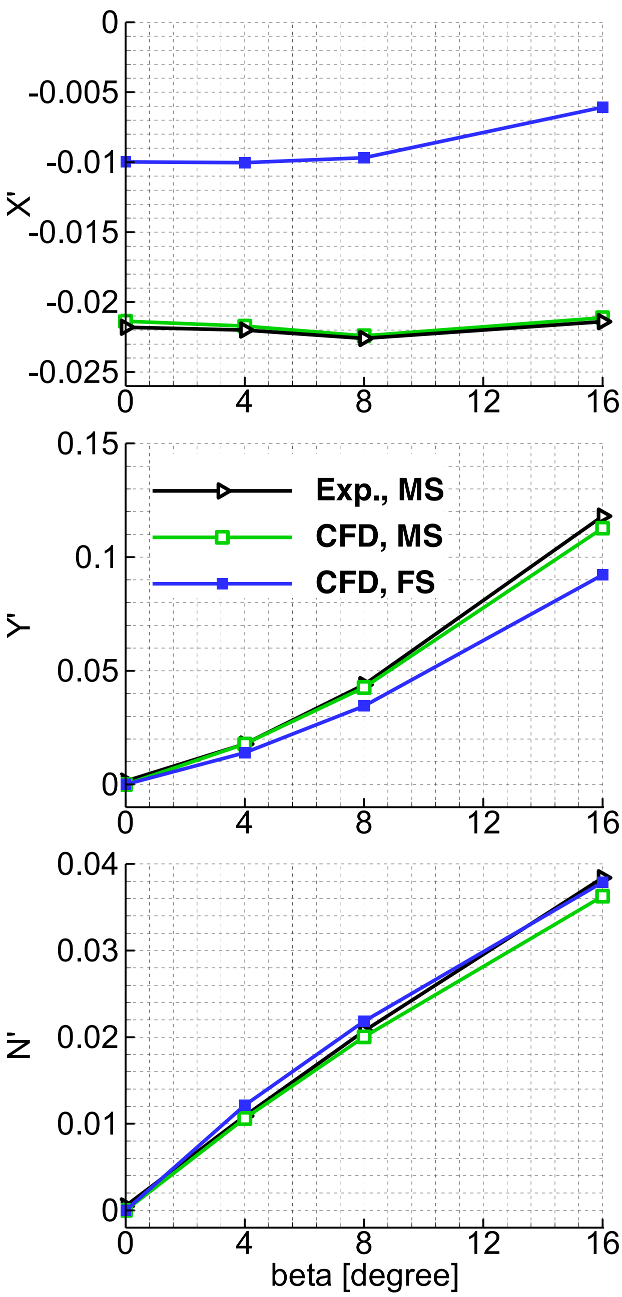


Figure 5. Comparison of X', Y' and N' with meaured data

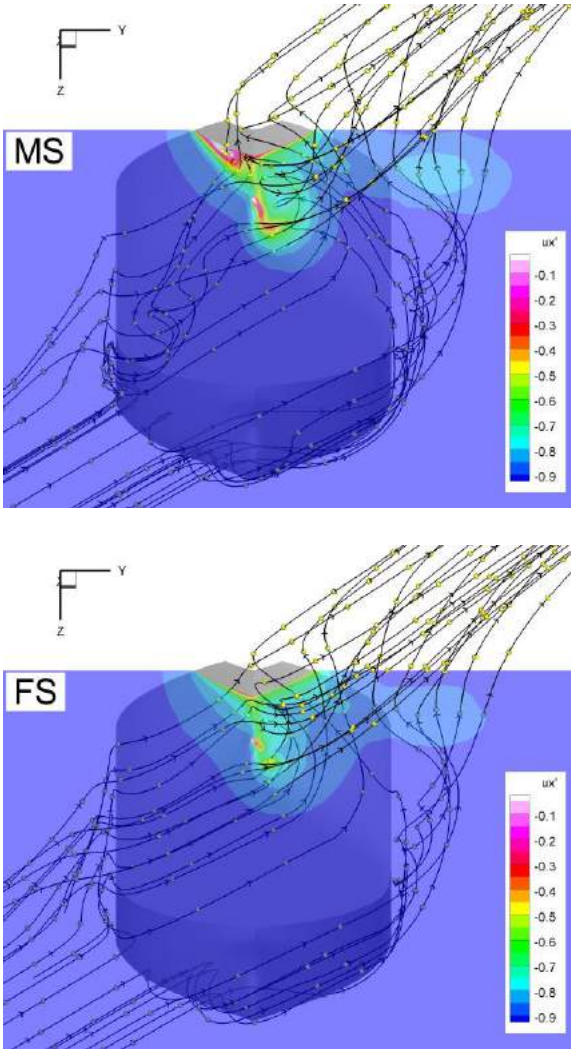


Figure 6. Contours of axial velocity at $x/L_{pp} = -0.48$ and stream lines for the case $\beta = 16°$

Figure 6 shows a comparison of flow features to check the scale effects. The MS and FS contours of the axial velocity at the propeller plane $x/L_{pp} = -0.48$ and stream lines for the case $\beta = 16°$ are compared in the figure. The axial velocity is made non-dimensional by free-stream flow velocity. The comparison of the contours shows that a thinner boundary layer is obvious in full scale and thus higher velocities are present at the propeller plane. The differences in boundary layers have an impact on the stern wake, which will affect the incoming velocities into propeller and consequently the propeller hydrodynamic performance. Two main differences are observed from the comparison of stream lines. The first one is that strong vortexes occur at model scale near the after part of port side. The second is that the stream lines at model scale starting from the bow pass the starboard side, then many of them merge into the stern wake with the stream lines starting from the after part of port side. The steam lines at full scale do not display these features. The MS flow structure is more complex.

6. CONCLUSIONS

Nowadays using CFD methods is an effective way to study scale effects on ship hydrodynamics. In this study, the flows around a drift tanker in model and full scale are simulated by means of RANS method based on OpenFOAM. In order to verify the computations, a detailed study on grid are firstly carried out. It includes two aspects: sensitivity of y^+ to the computed results and grid dependency study. A suitable grid is finally found for both the MS and FS simulations, i.e., Low-Reynolds number mode for MS simulation but High-Reynolds number mode for FS simulation.

The computed resistances in model and full scale, together with the components, are compared with available data. Good agreements are achieved. Especially the error of the friction resistance in full scale is acceptable (around 5.84%) compared with the values based on 1957 ITTC friction line. This demonstrates the computation is of high accuracy. Satisfactory agreements are also found for the static drift case through the comparison of the computed results with available measured data. The maximum error in model scale is around 5.5% for the considered cases.

This study has shown scale effects have large impacts on the hydrodynamic forces acting on the tanker and flow around it. The largest influence is found for the longitudinal force, around 3 times difference. The difference for side force is even more than 20% and will affect the tanker's manoeuvrability. The flow characteristic in full scale is quite distinguishing on the boundary layer and vortex structure. These differences will have an impact on the loads on the propeller and rudder. In further work, FS simulation will be performed for a full appended ship to obtain further knowledge of scale effects on hull-propeller-rudder interaction.

7. ACKNOWLEDGEMENTS

This work was supported by National Natural Science Foundation of China (Grant No. 51609188, 51609187, 51609186), Hubei Key Laboratory of Inland Shipping Technology (Grant No. NHHY2015002) and Fundamental Research Funds for the Central Universities (WUT: 2016IVB006, 2017IVB007).

8. REFERENCES

1. *19th ITTC report of the powering performance committee*, 1990. 19th International towing tanker conference, Madrid, Spain.
2. CASTRO, A. M., CARRICA, P. M. and STERN, F., 2011. *Full scale self-propulsion computations using propeller for the KRISO container ship KCS*, Computers & Fluids, 51, 35-47.

3. ECA, L. and HOEKSTRA, M., 2000. *Numerical prediction of scale effect in ship stern flows with eddy-viscosity turbulence models*, 23rd Symposium on Naval Hydrodynamics, 553-568.
4. HINO, T., 2005. Proceeding of CFD Workshop, Tokyo.
5. JU, S. and PATEL, V., 1994. *A numerical approach for predicting the total resistance and nominal wakes of full-scale tankers*, 19th Symposium on Naval Hydrodynamics, 371-387.
6. LARSSON, L., STERN, F. et al., 2010. Proceeding of CFD Workshop, Gothenburg.
7. LANUDER, B. E., and SPALDING, D. B., 1974. *The numerical computation of turbulent flow*, Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp.269-289.
8. MENTER, F. R., KUNTZ, M. et al., 2003. *Ten years of industrial experience with the SST turbulence model*, Turbulence, Heat and Mass Transfer 4.
9. SHIH, T. H. POVINELLI, L. A. and LIU, N. S., 2003. *Application of generalized wall function for complex turbulent flows*, Journal of Turbulence, 4, 1-16.
10. SIMMAN, 2014. *Workshop on verification and validation of ship manoeuvring simulation methods*, December, Copenhagen, Denmark.
11. WILCOX, D. C., 1993. *Turbulence modelling for CFD*, Second Edition, DCW Industries.
12. YAO, J. X., 2015. *On the propeller effect when predicting hydrodynamic forces for manoeuvring using RANS simulations of captive model tests*, Doctoral Dissertation of Technical University of Berlin, Germany.

