ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2016

NUMERICAL SIMULATION OF OBLIQUE TOWING TEST

Juan Sebastian Cardona
University of Liège
Faculty of Applied Sciences – ANAST, Liège
Place du 20 Août, 7 Bât.A1, 4000 Liège, Belgium
E-mail: jcardonadiaz@gmail.com

Florin Păcuraru
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domneasca Street, 800008, Romania,
E-mail: florin.pacuraru@ugal.ro

ABSTRACT

During the design of a ship, manoeuvrability performance should be measured in order to ensure course keeping and turning ability. In order to find out these characteristics, the numerical simulation of the Oblique Towing Test (OTT) has been developed, and the forces which act on the ship hull have been found numerically. Results are compared with experimental test carried out in INSEAN 2340 and in the DTMB 5512 ship model.

Keywords: INSEAN 2340, Oblique Towing Test (OTT), resistance curve, validation.

1. INTRODUCTION

During design of a ship, hydrodynamics characteristics and manoeuvrability performance must be studied in detail. At this stage of the design, hydrodynamic resistance should be known for a range of speeds including the design speed [1]. Besides, manoeuvrability performance of the specific ship should be measured or numerically determined in order to ensure course keeping and turning ability. All of them depend of the geometry of the vessel.

The towing tank test has been the most reliable tool for identifying the resistance of the ship by an experimental test using one scale model of the ship. On the other hand, Oblique Towing Test (OTT) must be done in a straight path (using towing tank) with the model in a drift angle with the fluid flow. By means of this test, it is possible to find out the damping coefficients depending on the translational velocities [2]. However, these two experimental tests are very expensive because they require the equipment and infrastructure necessary to carry them out [3].

Computational methods such as Boundary Element Method (BEM) [4] or Finite Volume Method (FVM) have been developed in the recent decades and they have been widely used in ship design for different purposes including ship resistance, propulsion, manoeuvring and seakeeping performance prediction. The use of computational method reduces the cost of hull models and time of experiment, allowing for a deeper investigation of the flow around the vessel, which is not easy to identify by experimental test visualizations. For ship manoeuvring, hydrodynamics derivatives could be obtained by CFD or Towing Tank Test, and then are usually used as input data for calculation of the turning circle test (turning ability), zig-zag maneuver (turning ability and counter maneuvering) and other maneuverability test required by the standards.

In the present study, the computational test is carried out using Computational Fluid Dynamics techniques based on FVM method to simulate the flow around the INSEAN

2340 scaled model and finally to calculate forces and moments that act on ship hull which experiences different drift angles, considering the viscous and wave resistance. Therefore, the Navier-Stokes equations for the incompressible fluid are solved considering free surface effects.

The force resistance for a wide range of speeds of the vessel is computed with a 13,7 % maximum of approximation error compared with experimental test of the model developed by IIHR [1].

2. THEORETICAL BASES

The 6DOF Navier-Stokes equations are solved in the computational domain around the ship using the software FINE™/MARINE from NUMECA. The turbulence model used for this computational algorithm is the Reynolds-averaged Navier-Stokes (RANS). Thus, fluid flow around the vessel can be expressed as in terms of velocity and pressure, [5]:

$$\rho\left(\frac{\partial u_i}{\partial t}+\frac{\partial\left(u_i u_j+u'_i u'_j\right)}{\partial x_j}\right)= \frac{\partial p}{\partial x_i}+\rho g+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right) \tag{1}$$

Incompressible viscous flow including gravity forces is expressed in the equation (1). However, the free surface interface between water and air is solved using the Volume of Fluid (VOF) method. This algorithm calculates a volume fraction $0 \le c \le 1$ for all the cells of the fluid domain, regarding the fluid portion in the cell. Thus, it is possible to calculate an equivalent value of density and viscosity for each cell in the fluid domain using the equations.

$$\begin{aligned}\rho &= \rho_{air}c+\rho_{water}(1+c)\\ \mu &= \mu_{air}c+\mu_{water}(1+c)\end{aligned} \tag{2}$$

The turbulent effect of eddy viscosity is solved using the turbulence model $\kappa-\varepsilon$. Thus, the eddy viscosity equation is calculated in terms of the turbulence kinetic energy per unit mass κ and the rate of the dissipation of the turbulence kinetic energy per unit mass ε.

After numerical calculation of the hydrodynamic forces, exerted on the ship hull in the X and Y direction, dimensionless coefficients of forces X´, Y´ and yaw moment N´ are calculated using the following equations [4]:

$$\begin{aligned}X' &= \frac{F_x}{0.5*\rho^2*T*L_{pp}}\\ Y' &= \frac{F_y}{0.5*\rho*u^2*T*L_{pp}}\\ N' &= \frac{N_z}{0.5*\rho*u^2*T*L^2{}_{pp}}\end{aligned} \tag{3}$$

3. GEOMETRY MODEL, COMPUTATIONAL FLUID DOMAIN AND SPACE DISCRETIZATION

In the present work, the INSEAN 2340 ship (see Fig.1) is used as a case study due to the good quantity of towing tank test data available for validation. Blue arrows are the local coordinate system in X, Y and Z direction located in the center of gravity of the ship.

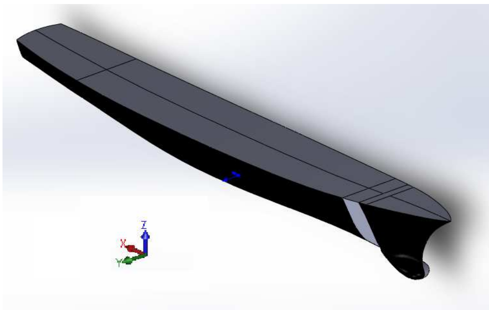

Fig.1 The 3D model of the bare hull of the INSEAN 2340 ship.

The INSEAN 2340 ship in a 1/24.8 scale is simulated during the study in order to determine the forces and moments that act on ship hull and also to investigate the flow behaviour for different drift angle condition. And experimental data of Oblique Towing Test is available in [2] for the DTMB 5415 model, which is an identical geosym of the INSEAN 2340. Main dimensions of the ship are listed in Table 1.

Table 1. Main dimension of the scaled INSEAN 2340 ship

No.	Dimension	Unit	Value
1	Scale	---	1/24.82
2	LPP	[m]	5.72
3	Draft	[m]	0.248
4	Beam	[m]	0.76
5	Wetted area	[m^2]	4.786
6	Displacement	[t]	0.549

Two computational domains have been used for this study, the first computational domain, used to measure the resistance of the ship takes advance of the symmetry of the flow around the bare hull. Thus, the symmetry in the XZ plane allows considering just half the fluid domain. The second computational domain, used for the different drift angle (Fig.2), should contain both sides of the ship because the drift angle produces non-symmetric wake behind the hull that has to be captured.

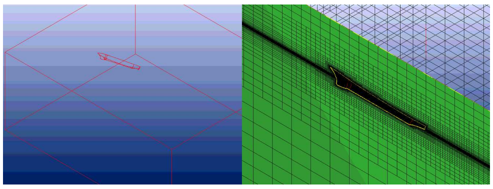

Fig.2 Computational domain of a ship with a drift angle (left hand) and Mesh surface of the symmetry plane for the domain without drift angle (right hand).

The dimension of the computational domain is very important for avoid violation of boundary conditions that can affect the accuracy of the results. Thus the fluid domain is extended 3 times the LPP downstream, 1 time upstream and 2 times the beam direction at each side.

A Cartesian mono-block unstructured grid of about 2 million cells has been generated to cover the entire computational domain along the bare hull. The grid topology is a H-H type. The minimum spacing (initial spacing normal to the body surfaces) is calculated based on a value y^+ of 5. The mesh dependence analysis (Fig.3) of the case with an intermediate speed of the ship 2.097m/s is presented.

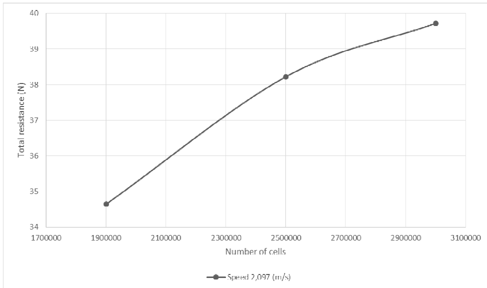

Fig.3 The Mesh independence analysis for an intermediate speed of the model.

The special refinements of the mesh are done on the bow surfaces of the hull, where high gradients are expected, and on the initial free surface level (Fig.2 right hand) in order to capture the air-water interface with better accuracy.

4. RESULTS AND DISCUSSIONS

The curve of ship resistance is built using 6 different speeds of the model, including maximum speed and design speed (from 0,373 to 3,37 m/s). The CFD results of force resistance in X direction, F_x, which is total ship resistance, are compared with experimental data of the same model (Fig.4).

Table 2 shows the Force in X direction calculated by numerical simulations at six different speeds and the experimental results taken from [1]. Maximum approximation error between numerical and experimental results is found in the intermediate points with a maximum value of 13,7%.

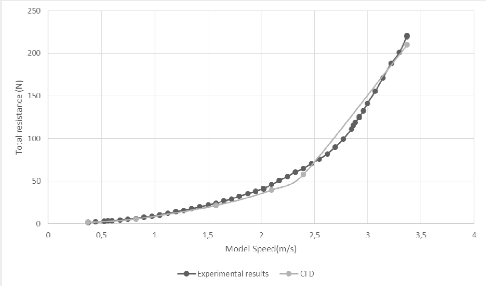

Fig.4 Resistance curve of the INSEAN 2340 ship model

Table 2. Comparison of forces on X direction

Speed	Froude Number	Force Experiment	Force CFD	Error
(m/s)	**-**	**(N)**	**(N)**	**(%)**
0,373	0.05	1,7	1,8	5,9
0,823	0.11	6,4	5,8	9,1
1,574	0.21	24,2	21,6	10,7
2,097	0.28	46,0	39,7	13,7
2,395	0.32	65,1	57,8	11,2
3,371	0.45	219,5	210,0	4,3

OTT or Pure drift test is done by computational simulations with a constant inflow speed, corresponding to a Froude number of 0.28, at six different drift angles β. Due to the asymmetry of the flow, the vessel experiences hydrodynamic forces along the longitudinal (X-axis) and transverse (Y-axis) axes. But, if the longitudinal center of gravity of the vessel is different than the longitudinal center of forces, the vessel can also experience a yaw moment N about the vertical axis Z. Table 3 shows the Forces in X and Y direction calculated by numerical simulations at the discrete speeds. Dimensionless coefficients of forces X´, Y´ and yaw moment N´ are calculated for the drift angle 0°, 4°, 8°, 12°, 16° and 20° using the set of Equations 3.

Table 3. Forces and moments calculated for six drift angle

---	0°	4°	8°	12°	16°	20°
Fx (N)	39,70	44,85	63,51	99,50	153,85	239,96
Fy (N)	0,00	50,97	121,38	191,26	294,90	400,31
Nz (Nm)	0,00	200,59	415,85	654,25	955,35	1210,80

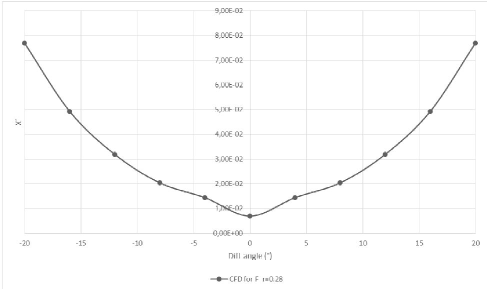

Fig.5 CFD results of the force coefficient X´ in the INSEAN 2340 ship model

The CFD results are compared with experimental data taken from [3]. The results of the CFD simulations, as dimensionless coefficients of forces in Y´ and yaw moment N´, show good agreement with experimental data. Maximum approximation error between numerical and experimental results are found in the drift angle of 20° with a value of 13%. As is shown in Fig.6 and Fig.7, the error increases as soon as the drift angle increases probably due to drift angle, which generates high wave elevation and complex flow behaviour.

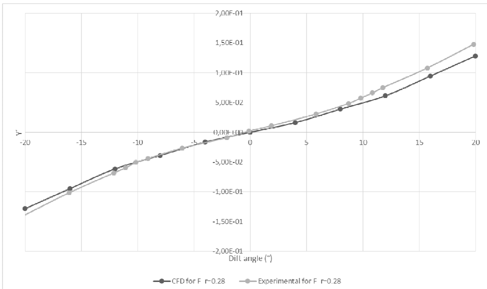

Fig.6 CFD results of the force coefficient Y´ in the INSEAN 2340 ship model compare DTMB 5512 ship model [3].

In the case of the dimensionless yaw moment N´ satisfactory agreement with experimental data is also found. Maximum approximation error between numerical and experimental results are found in the drift angle of 20° with a value of 23%.

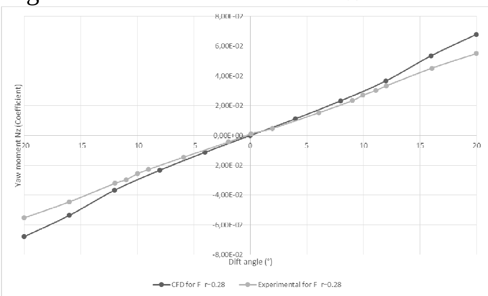

Fig.7 CFD results of the yaw moment coefficient N´ in the INSEAN 2340 ship model compare DTMB 5512 ship model [3].

Ship model with a drift angle of 16° [2] are compared with the computational results obtained in the INSEAN 2340 in this study. Wake, pattern and wave elevation (Fig.5) are in accordance; especially in the aft part increasing towards the starboard where wave elevation contours seems to have the same pattern.

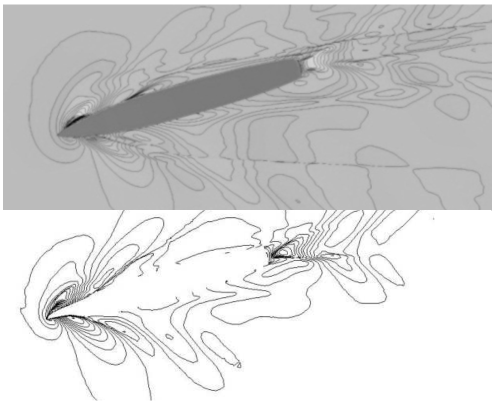

Fig.8 CFD results of the wave elevation in the INSEAN 2340 ship model compare DTMB 5512 ship model [2].

The wave contours for four different drift angles have been plotted in Fig.9.

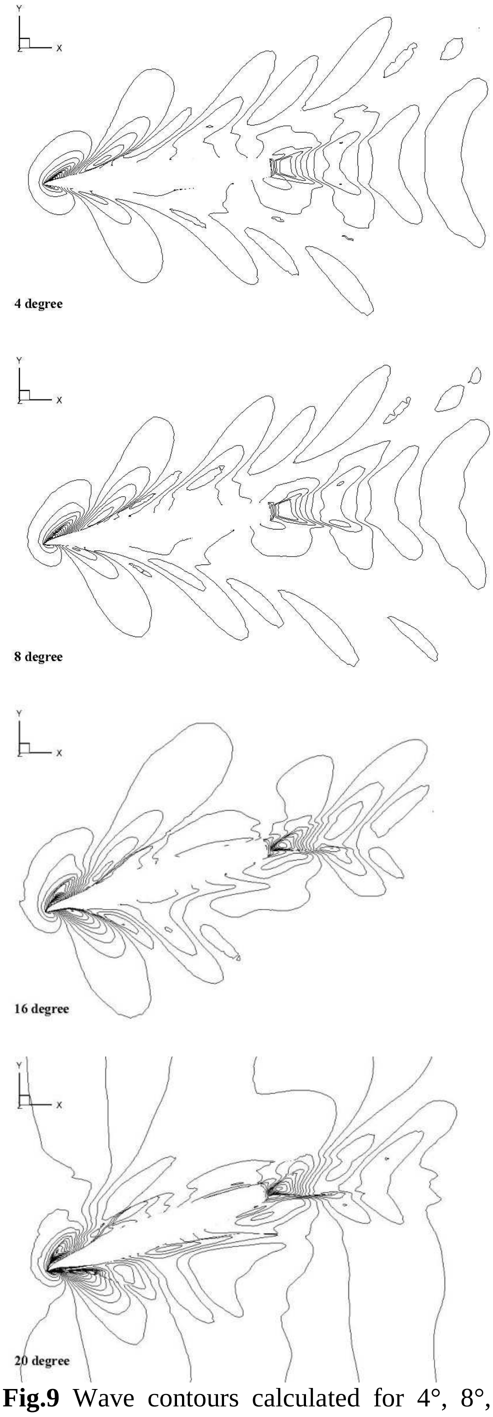

Fig.9 Wave contours calculated for 4°, 8°, 16° and 20° drift angles

5. CONCLUSIONS

The computational method used in this study solves the ship resistance and shows a good agreement with the measurement data, but the result can be further improved by using finer grids and smaller time step.

From the comparison of the dimensionless coefficient, computed and measured, differences from 13% to 23 % have been founded for large drift angles.

Additional computations for the simulation of the Planar Motion Mechanism (PMM) test could be considered for the future work

ACKNOWLEDGEMENTS

This study has been accomplished in the frame of the EMSHIP - European Masters Course in Advanced Ship and Offshore Design Ref. 159652-1-2009-1-BE-ERA MUNDUS-EMMC, at "Dunarea de Jos" University of Galati, Faculty of Naval Architecture.

REFERENCES

[1]. **Olivieri, A.**, et al., "*Towing tank experiments of resistance, sinkage and trim, boundary layer, wake, and free surface flow around a naval combatant INSEAN 2340 model*", Report no. IIHR-TR-421, IOWA UNIV., 2001.

[2]. **Hajivand, Ahmad, and Hossein Mousavizadegan S.,** "*Virtual simulation of manoeuvring captive tests for a surface vessel*", International Journal of Naval Architecture and Ocean Engineering 7.5 pp. 848-872, 2015.

[3]. **Yoon, H.S**., "*Phase-averaged stereo-PIV flow field and Force/moment/motion measurements for surface combatant in PMM manoeuvres*", PhD thesis, The University of Iowa, 2009.

[4]. **Hess, J.L.,** "*Panel methods in computational fluid dynamics*", Annual Review of Fluid Mechanics 22, pp. 255–274, 1990.

[5]. **Rawson, K.J., Tupper, E.C.,** "*Basic ship theory*", Butterworth & Heinemann Publishing House, Oxford, 2001.

[6]. **Wilson, R., Paterson, E. and Stern, F.,** "*Unsteady RANS CFD Method for Naval Combatants in Waves*", Proceedings of 22nd Symposium Naval Hydrodynamic, Washington, pp.532-549, 1998.

Paper received on December 28th, 2016