Real-time simulation of a COGAG naval ship propulsion system

Article *in* Proceedings of the Institution of Mechanical Engineers Part M Journal of Engineering for the Maritime Environment · February 2009

DOI: 10.1243/14750902JEME121

4 authors:

Marco Altosole
Università degli Studi di Genova

Giovanni Benvenuto
Università degli Studi di Genova

Massimo Figari
Università degli Studi di Genova

Ugo Campora
Università degli Studi di Genova

International Review of Mechanical Engineering (I.RE.M.E.), Vol. 7, n. 7
November 2013

Integration of Numerical Modeling and Simulation Techniques for the Analysis of Towing Operations of Cargo Ships

M. Altosole[1], D. Boote[2], S. Brizzolara[3], M. Viviani[4]

Abstract – *The paper is focused on some benefits gained by the integration of CFD results into the simulation process over time, in order to develop a whole numerical model representing the dynamics of a large merchant ship during towing/escorting operations in severe weather conditions. In particular, the shown application is referred to the simulation of a container ship, towed by two tugboats in a typical Mediterranean harbor. The developed simulator consists of a set of ordinary differential equations, mathematical relationships and numerical tables which represent the ship and the propulsion system dynamics, while CFD method is used in order to model the wind forces, eventually influenced by the presence of some harbor breakwaters, acting on the considered ship. The proposed simulator can be used as a powerful tool to plan safer towing arrangements and procedures, or to analyze in post-processing possible human errors in the case of unfortunate marine accidents.*

***Keywords**: CFD, Ship Dynamics, Time Domain Simulation, Towing Operations, Wind Action*

Nomenclature

c = flow speed at rudder;
f = fuel flow;
h = height above the sea level;
k_q = propeller torque coefficient;
k_t = propeller thrust coefficient;
n = shaft speed;
n_c = shaft speed setpoint;
r = ship rotational speed;
u = ship speed in surge direction;
v = ship speed in sway direction;
z = vertical axis for the wind profile definition;
x_G = longitudinal position of centre of gravity;
A_L = ship superstructure longitudinal area exposed to wind;
A_R = rudder area;
A_T = ship superstructure transversal area exposed to wind;
C_D = drag coefficient of the rudder;
C_L = lift coefficient of the rudder;
C_N = wind yawing moment coefficient;
C_X = longitudinal wind force coefficient;
C_Y = transversal wind force coefficient;
D = propeller diameter;
F = exponent for the wind profile definition;
I_{zz} = ship inertia moment about z-axis;
J_p = polar moment of inertia;
K_I = integral gain;
K_P = proportional gain;
L_{oa} = overall ship length;
M_E = engine torque;
M_{PROP} = propeller torque;
N = moments acting on the ship about z-axis;
N_W = wind moment about z-axis;
V_W = wind speed;
X = forces acting on the ship in x-axis direction;
X_{PROP} = propeller thrust;
X_{RUD} = rudder force in x-axis direction;
X_W = wind force in x-axis direction;
Y = forces acting on the ship in y-axis direction;
Y_{RUD} = rudder force in y-axis direction;
Y_W = wind force in y-axis direction;
ε = wind direction acting on the ship;
η_m = mechanical efficiency;
η_r = relative rotative efficiency of the propeller;
ρ = sea water density;
ρ_A = air density;
Δ = ship mass;

I. Introduction

Numerical modeling by CFD method and time domain simulation can save time and money on major engineering applications, because these techniques allow designers to predict results of interest before installing any equipment, eliminating costly on-site trials and error situations.

Since several years the application of simulation techniques to marine propulsion dynamics represents a research field of Genoa University.

Marine simulations can be used for a variety of purposes, such as ship performance analysis, machinery

Manuscript received and revised July 2013, accepted October 2013

performance analysis and propulsion control systems development [1]-[4].

In the following, the integration between CFD modeling and time domain simulation is described with regard to the development of a marine simulator, able to represent the behavior of a large merchant ship, towed by two tugboats in restricted space and in correspondence to different weather conditions.

The behavior of a ship under towing operation does not only depend on ship characteristics but it is heavily influenced by a series of boundary conditions which should be carefully considered from time to time. This is not so simple when critical conditions occur and the safety and integrity of the ship is entrusted mainly to the captain experience and sensitivity. Sometimes this is not enough and the underestimation of the real situation can compromise seriously the safety of the ship.

As a matter of fact, the evaluation of all playing variables and of their consequences on the ship response could be very complex and it requires specific calculation tools and large computational time. For this reason, simulation results can be successfully employed to investigate particular towing issues, as the proper use of the number and power of tugboats to be employed.

In the simulation procedure all the parameters influencing the ship response must be introduced and carefully quantified by proper mathematical algorithms.

For what the towed ship is concerned, the most important parameters consist of the shape and the amount of windage and submerged area, propulsion system performances (main engine and propeller, bow and aft thrusters) and the rudder hydrodynamic characteristics.

In the case of tug (or tugs), for each unit involved in the operations, the variables are the towing attachment point coordinates (of tugs and of the ship as well), the cable length and the towing force as a function of throttle settings and tugs conditions.

Finally, the boundary conditions should be considered: wind direction and intensity, the effect of breakwaters and buildings on wind speed distribution, direction and speed of sea current (if any) and the water depth in the operation area.

In this paper a quite complex application of a towing simulation is presented, related to a container ship in a typical Mediterranean harbor. The simulation includes all the previously considered parameters, in order to achieve a complete spectrum of all possible ship responses and to identify which of them can be considered sustainable or dangerous.

II. Simulation Model

A time domain numerical model to represent the berthing and towing/escorting operations of a large merchant ship, towed by two tugboats, has been designed. The non-linear time domain simulation runs in Matlab-Simulink® software environment, a widely used platform for the dynamic systems simulation. The simulator consists of a set of ordinary differential equations (ODEs), algebraic equations and numerical tables that represent the various elements of the propulsion system and maneuverability behavior of the ship. The following elements are modeled: the ship motions in three degrees of freedom (surge, sway and yaw), the main engine performance, the side thrusters, the rudder and the environmental actions (wind and current).

For each modeled element, numerical models with different level of accuracy have been developed, taking into account the general objective of a good balance between the reliability of the simulation results and the code performance.

By numerically solving the several differential equations, it is possible to obtain the ship trajectory during towing condition. The simulation inputs, given as time histories, are:

- the tugboats pulls (intensity and direction);
- the action of the ship side thrusters;
- the bridge lever position of the ship;
- the ship rudder angle;
- the wind and current actions (speed and direction).

The outputs are the time histories of the ship motions and of the propulsion machinery behavior. In order to have a better understanding of the ship manoeuvres, a 3D visualization in a virtual marine scenario is also available (developed by using Virtual Reality Toolbox of Matlab®).

II.1. Ship Dynamics Equations

The possible six motions of a ship are illustrated in Fig.1, where also the two kinds of reference systems are shown: one is moving with the ship (*oxyz*: body-fixed), while the other is an inertial reference system ($OX_0Y_0Z_0$: earth-fixed).

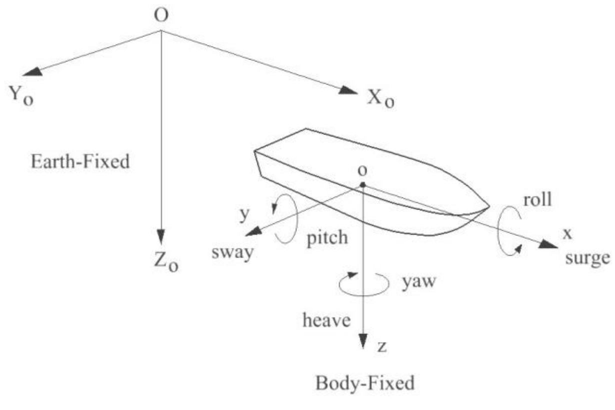

Fig. 1. Definitions of reference frames and ship motions (surge, sway, heave, roll, pitch and yaw)

The theoretical model adopted for the proposed application, takes into account only three degrees of freedom of the ship: surge, sway and yaw; so the inertial and not inertial coordinates systems can be reduced to the horizontal plane illustrated in Fig. 2, where the origin of the body-fixed system is coincident with the centre of gravity *G* of the vessel, characterized by the velocity vector *V*.

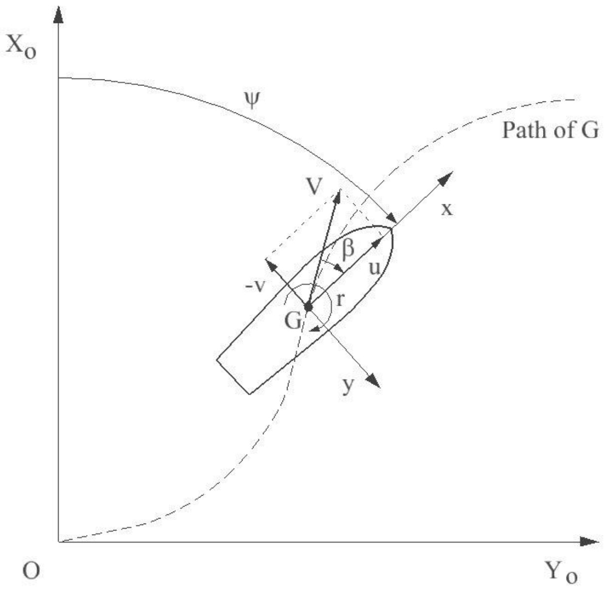

Fig. 2 Reference system used for ship equations

On the basis of Fig. 2, the traditional manoeuvrability equations (derived by the application of the Newton's law and written with reference to the body-fixed coordinates system) are:

$$\begin{aligned} X &= \Delta(\dot{u} - vr) \\ Y &= \Delta(\dot{v} + ur) \\ N &= I_{zz}\dot{r} \end{aligned} \tag{1}$$

where *X* and *Y* are the total external forces acting on the ship, respectively in surging direction and swaying direction, while *N* is the external yaw moment; then, *u*, *v*, *r* are the surge, sway and yaw velocities and finally, I_{zz} and Δ are respectively the inertia moment around *z* axis and the mass of the vessel. Further information and details about ship motions equations can be found in [5].

By solving the system of Equations (1) in the time domain, the ship velocities are found, from which, by numerical integration, the ship trajectory is finally obtained with respect to the earth-fixed coordinates system OX_0Y_0, coinciding with the body-fixed system at initial time of simulation.

In Equations (1), the considered external forces and moments acting on the ship are mainly due to the hydrodynamic action on the hull and wind action on the superstructures, as well as pulls of tugs and forces given by the different control means (propeller, rudder and side thrusters). The scheme of all the considered forces acting on the vessel, together with each reference frame, is shown in Fig. 3 (where the subscript *H* means hull, *RUD* means rudder, *PROP* means propeller and *W* means wind).

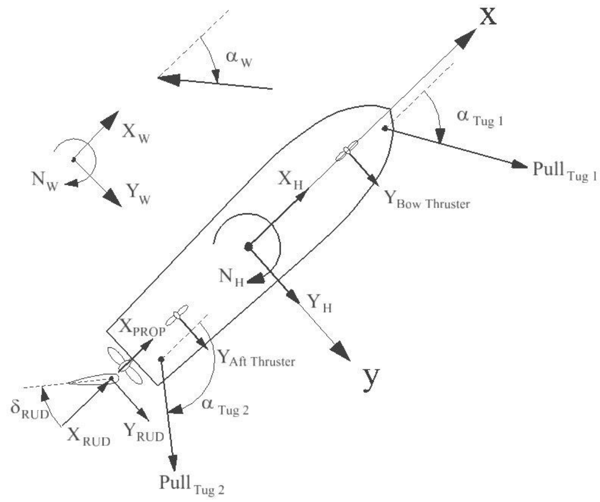

Fig. 3 Scheme of the considered forces acting on the ship

Regarding hull forces X_H, Y_H and moment N_H, the approach reported by Kang and Hasegawa in [6] has been implemented, in order to consider slow speed motions with high drift angles β (see Fig. 2) and contemporarily usual forces in correspondence to low drift angle motions, by means of a single unified mathematical model.

It has to be underlined that the employed model should be corrected in order to consider the different hull form of the actual ship with respect to those considered in [6]. At present, this was not possible due to the lack of experimental data (either with captive model tests or directly at sea trials). Nevertheless, it has to be also noticed that, at least in correspondence to some critical situations, like for instance the purely oblique towing, forces have been compared with some available experimental data, providing good results.

It is intention of the authors to obtain in the future more experimental data, in order to improve the accuracy of the current model.

II.2. Wind Action Equations

The wind action on the ship is modeled by the well-known following equations, representing the forces and moment due to the direction and speed V_W of the wind:

$$X_W = \frac{1}{2}\rho_A A_T V_W^2 C_X \tag{2}$$

$$Y_W = \frac{1}{2}\rho_A A_L V_W^2 C_Y \tag{3}$$

$$N_W = \frac{1}{2}\rho_A A_L L_{OA} V_W^2 C_N \tag{4}$$

where ρ_A is the air density, A_T and A_L are the transversal and longitudinal areas of the ship superstructures exposed to wind, while L_{OA} is the overall length of the ship. The aerodynamic coefficients C_X , C_Y and C_N take into account the wind direction and the particular shape of the superstructures of the considered ship. This approach is rather standard in the usual studies of ship maneuverability and the coefficients are evaluated by means of experimental tests at the wind tunnel for the specific ship, or, in many cases, derived from some data available in literature for ships similar to the considered one. The first option, which clearly leads to considerably lower errors, is seldom applied because of the wind tunnel cost. For this reason, the second option can be adopted but it could lead to significant uncertainties in the numerical assessment of the wind forces, which in many operating scenarios play a very important role (being hydrodynamic forces very limited, due to ship slow motion during towing condition). In the present work, a third approach has been considered, evaluating forces by means of CFD method, as reported in the following.

Regarding sea current forces, they have been simply considered by means of an additional velocity, which is added to the ship speed over ground. This approach, somehow simplified, has been chosen for its straightforwardness, even if, for future applications, it is planned to move to direct computations of forces due to the combined effect of hull motions and sea current, following an approach similar to the one adopted for wind forces. At present, this was not considered due to the fact that current plays a rather low role in the considered scenario (ship maneuvering in harbor).

II.3. Propulsion System Equations

In the propulsion plant model, the following elements are considered:

- the main engine performance depending on the bridge lever position;
- the shaft line dynamics;
- the propeller performance.

The main propulsion behavior can be represented by the two following equations, one for the shaft line dynamics and the other relative to the engine governor:

$$2\pi\, J_p\, \dot{n}(t) = M_E(t) - M_{PROP}(t) \tag{5}$$

$$f(t) = k_i \int_t \left[n_c - n(t)\right] dt + k_p \left[n_c - n(t)\right] \tag{6}$$

where J_p is the total polar moment of inertia (engine plus propeller and shaft), M_E is the engine torque depending on the fuel flow f, M_{PROP} is the required propeller torque, n_c is the commanded shaft speed and k_i and k_p are the integral and proportional gains for the regulation process of the engine. By solving Equation (5), it is possible to calculate the actual shaft speed n depending on time t and then, by Equation (6), the engine fuel flow f can be found. For the considered propulsion plant, the shaft speed n is obviously equal to both propeller speed and engine speed. In the present simulator, the engine torque M_E is simply calculated by a numerical matrix, representing the engine torque as a function of rotational speed n and fuel consumption f, derived by the specific fuel consumption map, provided by the engine manufacturer. The time lag between input (fuel flow f) and output (torque M_E) is given by a proper transfer function, in order to simulate the dynamics of the engine and its main subsystems, as turbocharger. This approach, more fully described in [7], is quite simple and suitable only for a rough simulation of the engine dynamic behavior. It leads to a short computational time, but it does not allow a careful description of the thermal and mechanical stresses induced in critical elements of the power plant. To overlap this limitation, more detailed simulation submodels can represent the engine thermodynamics, mainly in order to analyze the behavior of much more variables of the engine (pressures, temperatures, blower speed, etc…). These kinds of engine models are not based on a simple numerical map, as previously described, but on several differential and algebraic equations ([8]-[11]).

The computation of the propeller thrust X_{PROP}, used in Equation (1), and torque M_{PROP} of Equation (5) is given by:

$$X_{PROP} = \rho\, k_t\, n^2 D^4 \tag{7}$$

$$M_{PROP} = \rho\, k_q\, n^2 D^5 \frac{1}{\eta_r\, \eta_m} \tag{8}$$

where D is the propeller diameter, η_r is the relative rotative efficiency of the propeller while η_m is the mechanical efficiency of the propulsive shaftline. Both the numerical values of these efficiencies are assumed constant during the whole simulation process, while the coefficients k_t and k_q are depending on propeller revolutions n and ship velocity u and they can be provided by the propeller open water tests or derived from systematic series available in literature. Further information about marine propeller theory can be found in [12].

The propulsion model includes the control system too, as previously introduced by Equation (6). The control system receives the input from the ship's Master, through the bridge lever position, which sets the proper value of n_c for the engine, in order to obtain the desired vessel performance. The scheme of this simple and usual control process is illustrated in Fig. 4.

Fig. 4 Engine governor action

In particular, the shown engine control consists of a PI governor (i.e. a controller based on proportional and integral action) of the shaft speed. In detail of Equation (6), the actual value of engine/propeller/shaft speed is compared with the engine speed commanded by the bridge lever position and then the engine fuel flow is adjusted by the PI action on the speed error, in order to reach and maintain the desired engine speed.

Regarding rudder forces, due to lift and drag, are represented by the following relationships:

$$X_{RUD} = 1/2 \rho A_R c^2 C_D \tag{9}$$

$$Y_{RUD} = 1/2 \rho A_R c^2 C_L \tag{10}$$

where ρ is the sea water density, A_R is the rudder area, c is the water flow speed considering propeller effect and ship motions, C_D and C_L are the drag and lift coefficients, depending on the rudder angle (one of the several inputs of the simulator), affected by local drift angle effect.

The mathematical model for the rudder system is described in [13], where the velocity c is calculated taking into account the water flow acceleration provided by the propeller on the rudder, as suggested in [14], [15].

Further improvements for the theoretical model have been specifically introduced on the basis of some data reported in [15] and in particular to consider the interaction between rudder and propeller during some different propulsion conditions of the vessel, as forward or reverse going, including relative transients (the so called “four quadrants propeller functioning”).

III. Wind Action Modeling by CFD Method and Validation

The aerodynamic coefficients in Equations (2), (3) and yawing moment in Equation (4) have been achieved by solving the fully turbulent Reynolds Averaged Navier Stokes Equations (RANSE) with a volume of fluid method already validated and successfully tested in previous studies [16]. For the proposed application, the numerical values of these coefficients have been compared with experimental data of a similar application, reported in literature [14]. The 3D model of the considered ship (a container ship) has been cleared of the small details and simulated as immersed in a uniform air flow with a wind speed equal to 20 m/s for three different values of ε angle of the approaching wind: 60°, 90° and 120° (where ε is positive counter-clockwise and it is zero for head wind). The three considered conditions are respectively illustrated in Figs. 5, 6 and 7.

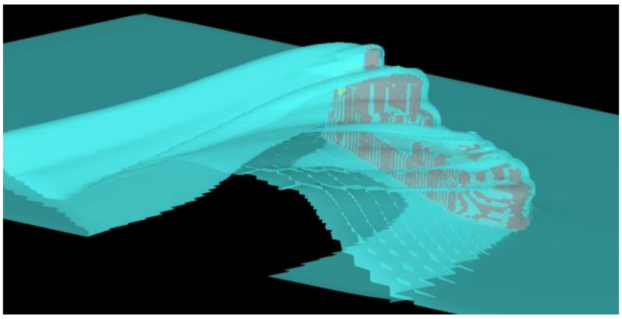

Fig. 5 Isosurface of given turbulent energy in the wind flow wake (for $\varepsilon = 60°$)

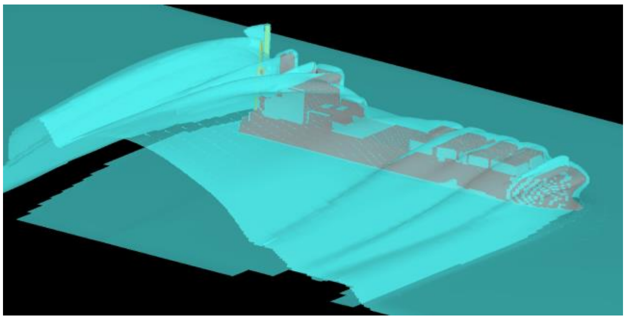

Fig. 6 Isosurface of given turbulent energy in the wind flow wake (for $\varepsilon = 90°$)

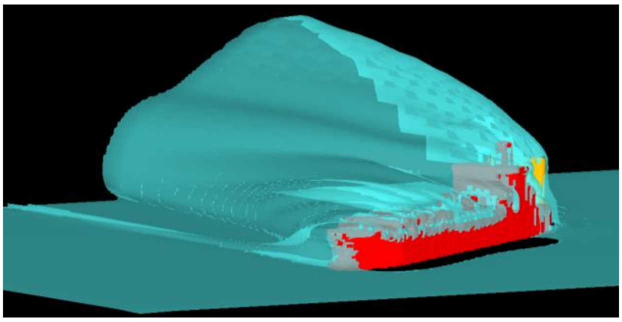

Fig. 7 Isosurface of given turbulent energy in the wind flow wake (for $\varepsilon = 120°$)

In Figure 8, for the second analyzed condition ($\varepsilon = 90°$), the pressure field on the upwind side (over pressure area) and on the leeward side of the ship (depressurized area) are shown.

By integrating the pressure distribution over the ship areas and correcting it for the effect of the typical velocity distribution in the wind boundary layer profile, it is possible to achieve the wind forces and the moment acting on the ship. The dimensionless results, i.e. the wind coefficients of Equations (2), (3) and (4), are compared in Fig. 9, 10 and 11 with the experimental data reported in literature [14] for a similar application (i.e. a Ro-Ro ship with the silhouette shown in Fig. 12, on which wind tunnel tests have been carried out).

The comparison shows a good agreement between CFD results and the wind tunnel data for the C_X and C_Y representation, while a higher discrepancy is evident for the C_N computation in the case of ε = 60°. However, this latter significant difference can be explained by the fact that the two considered ship profiles, although similar, are not exactly the same. In particular, for the ship layout reported in literature, the longitudinal position of the centre of the windage area is located aftward with respect to the real ship considered in this study.

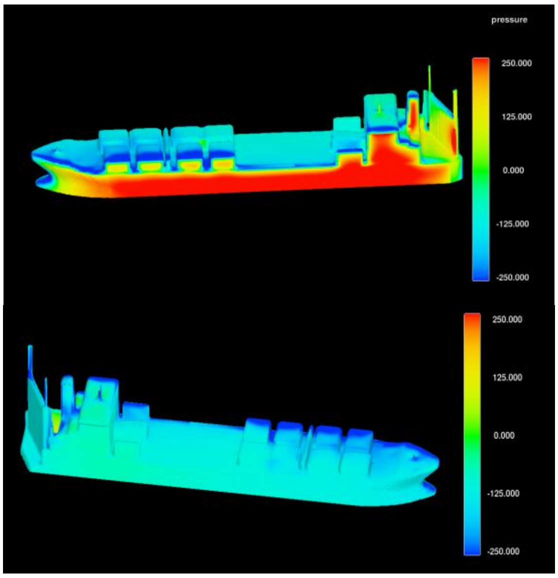

Fig. 8 Predicted pressure field for ε = 90° on the upwind side (top) and on the leeward side (bottom), wind speed: 20 m/s

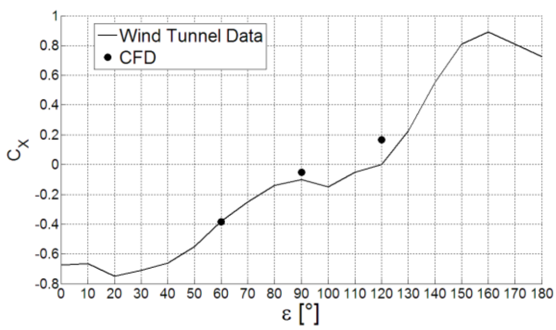

Fig. 9. Wind coefficient C_X: literature data vs CFD method.

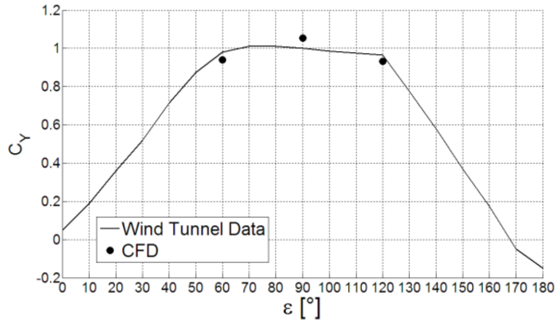

Fig. 10. Wind coefficient C_Y: literature data vs CFD method

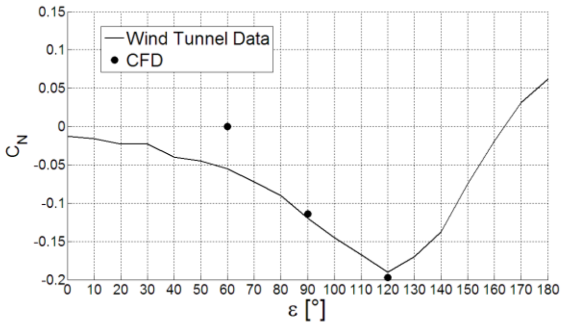

Fig. 11. Wind coefficient C_N : literature data vs CFD method.

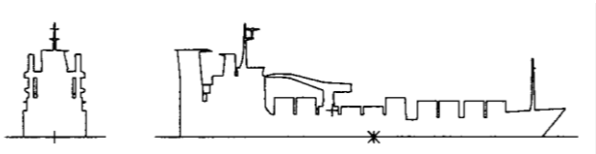

Fig. 12. Exposed area for the superstructure model used in wind tunnel tests [14]

IV. Case Study

In order to test the level of reliability of the developed theoretical model, hereinafter a particular application, regarding the unberthing and towing/escorting operations of a container ship is presented. The main characteristics of the towed container ship and of the two employed tugboats, are reported in Table 1.

TABLE I
CASE STUDY

Main Characteristics	values
Ship overall length	220 m
Ship displacement	27000 tons
Ship design speed	18 knots
Maximum pull of tugboat 1	600 kN
Maximum pull of tugboat 2	400 kN
Wind speed/direction	40 knots/SE

The effect of harbor structures, as dams or breakwaters, on the vertical and horizontal wind distribution has been taken into account as well, thanks to an accurate CFD analysis in which the geometry of these structures has been modeled in detail. From this point of view it has been enlightened the importance of the distance between the harbor breakwater and the operation area, for given wind direction and speed.

In particular, the upstream wind speed profile (i.e. the wind profile without the structure influence), as a function of the vertical dimension z, in literature is generally represented by the following equation:

$$V_W(z) = V_{W\,h}\left(\frac{z}{h}\right)^{\frac{1}{F}} \tag{11}$$

where $V_{W\,h}$ is the wind speed at a considered height h above the sea level and F is a proper coefficient depending on wind speed intensity (for instance, possible numerical values of N can be 5 for light winds up to 10 for strong winds).

The presence of a breakwater, whose geometry is illustrated in Fig.13, causes a variation of the wind profile expressed by Equation (11). By CFD method application, it has been possible to find the new wind distribution at several distances from the considered breakwater.

The CFD results are illustrated in Fig. 14, where the wind speed variation, around the modeled structure, is shown by a color scale (the analysis shown in the picture is valid for wind speed V_{Wh} = 18 m/s corresponding to h=10 m).

The same results are illustrated by using another visualization in Fig. 15, where the comparison among some wind speed vertical distributions, corresponding to various distances from the breakwaters, is shown. The comparison is carried out with the upstream profile (represented by the continuous solid line in the picture).

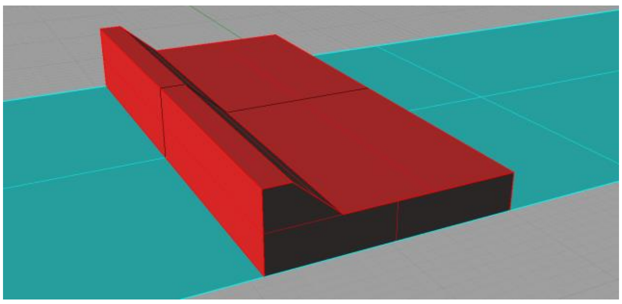

Fig. 13. Geometry of the considered breakwater.

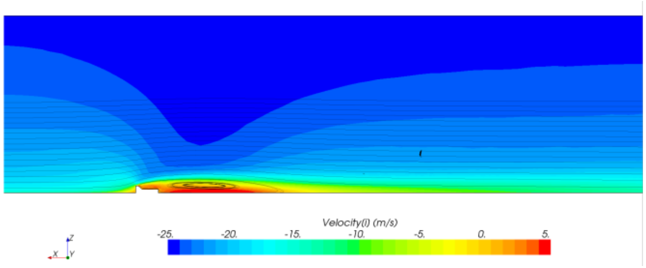

Fig. 14. CFD results representing the wind speed around the modeled breakwater

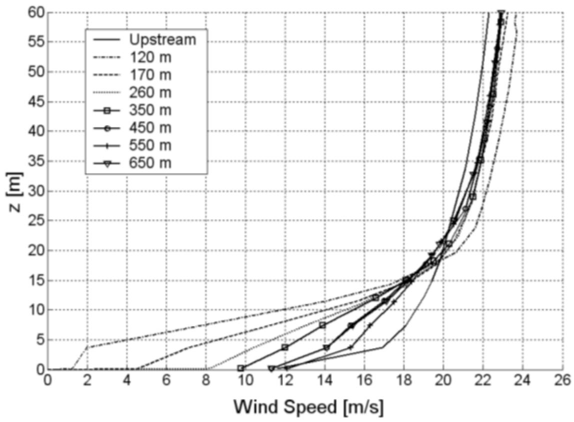

Fig. 15. Wind vertical profiles at different distances from the breakwater

For each profile the mean value of the wind speed has been found adopting the following expression:

$$\overline{V_W^2} = \frac{\sum A_{Li} V_W^2(z_i)}{\sum A_{Li}} \tag{12}$$

In fact, it has been thought to divide the longitudinal ship windage area A_L by i-longitudinal portions A_{Li}; each portion has been taken at different z_i positions of the ship vertical axis.

Once the mean value of the wind speed has been calculated by Equation (12), the speed reduction, with reference to the upstream value, can be found at several horizontal distances between breakwater and ship.

The obtained relationship is shown in Fig. 16 and it is used by the developed simulator for the wind forces calculation.

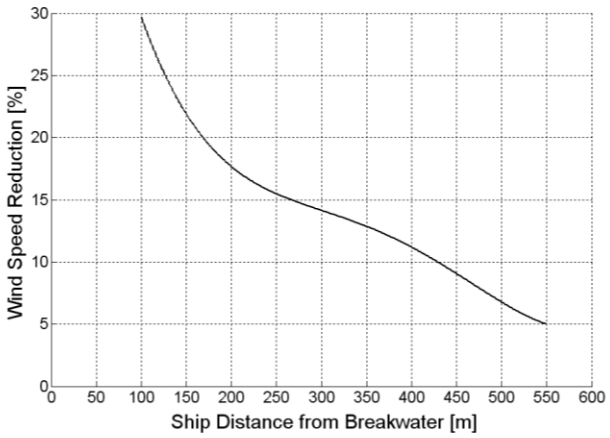

Fig. 16. Wind speed reduction at several ship distances

IV.1. Graphical and Numerical Results

As an example, a very critical towing condition has been simulated; in fact, it has been assumed an unexpected failure in the starting system of the main engine, so the ship can be moved only by the actions of the two tugboats and side thrusters. The considered weather conditions are also very severe (wind speed equal to 40 knots).

During the simulation process, all the results are recorded in numerical arrays that can be displayed during on-line or off-line simulation, as a function of the time array. Main numerical input for the considered ship manoeuvre are illustrated in Fig. 17. In particular, the time histories of pulls of the two employed tugboats, in terms of intensity and direction relative to the ship, together with the thrusts given by the two side thrusters, are reported in Fig. 17.

Simulation output can be represented in a 3D virtual marine scenario. The visualization of the complete manoeuvre with the relative time scale, gives to the simulation user an exact idea of the time that ship master has at disposal to take any decision during a critical situation as that one simulated. The animated sequence is shown from Fig. 18 to Fig. 25, illustrating some snapshots captured from the developed simulator.

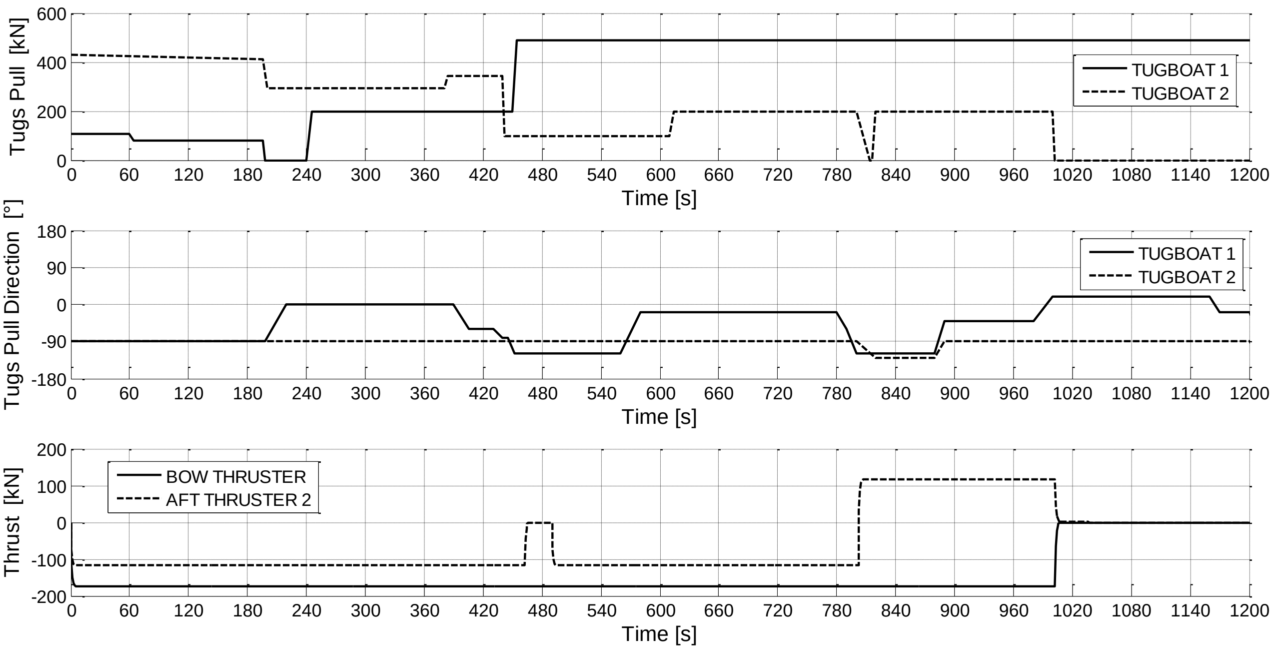

Fig. 17. Time histories of simulation input

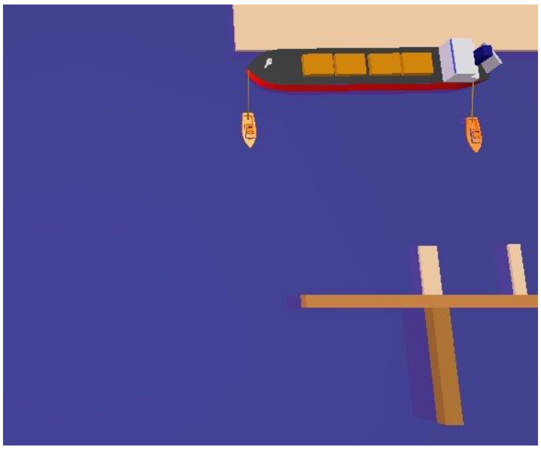

Fig. 18. Snapshot 1 (simulation start)

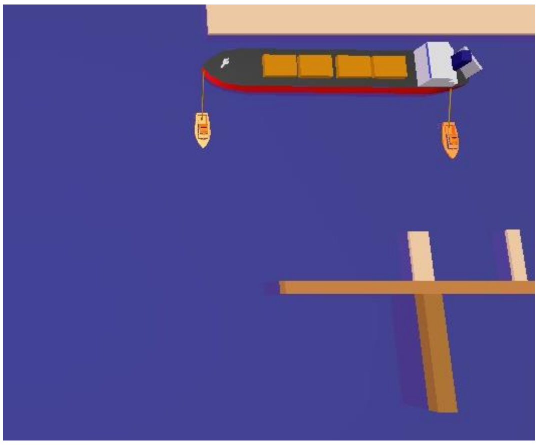

Fig. 19. Snapshot 2 (unberthing)

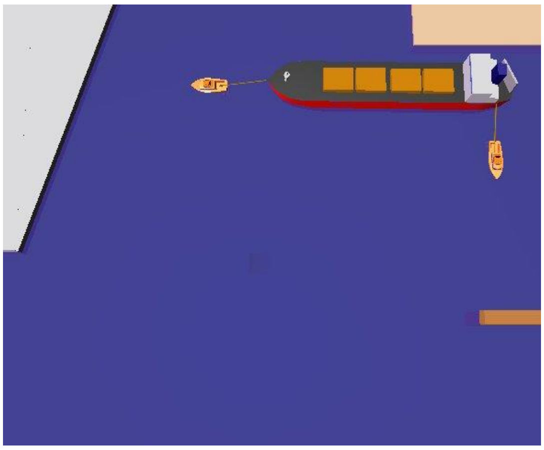

Fig. 20. Snapshot 3

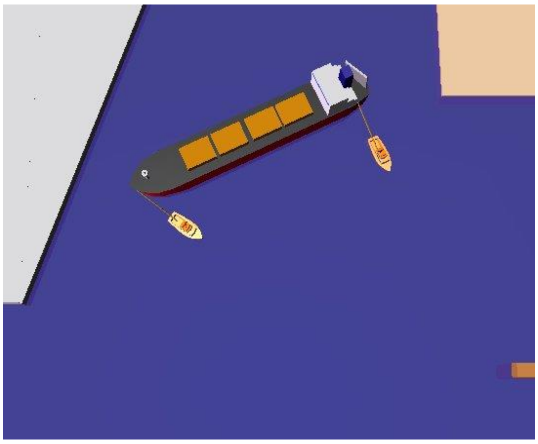

Fig. 21. Snapshot 4

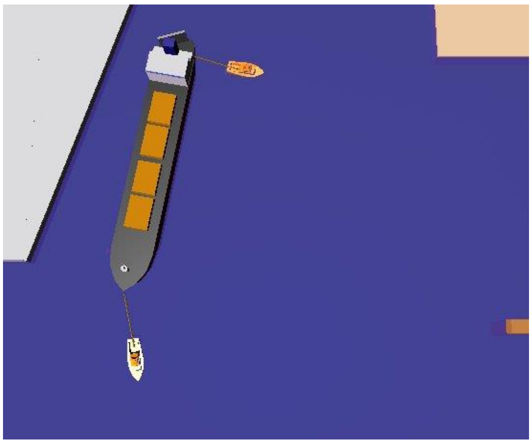

Fig. 22. Snapshot 5

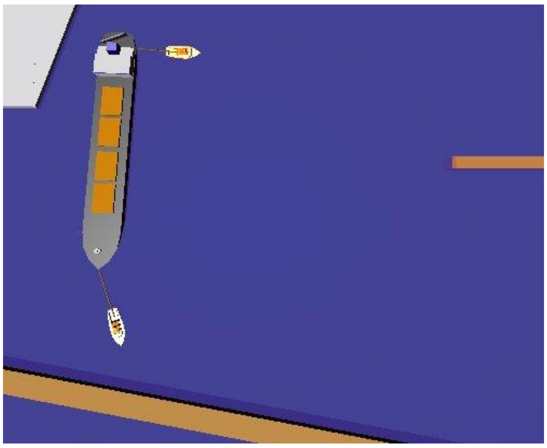

Fig. 23. Snapshot 6

Manuscript received and revised July 2013, accepted October 2013

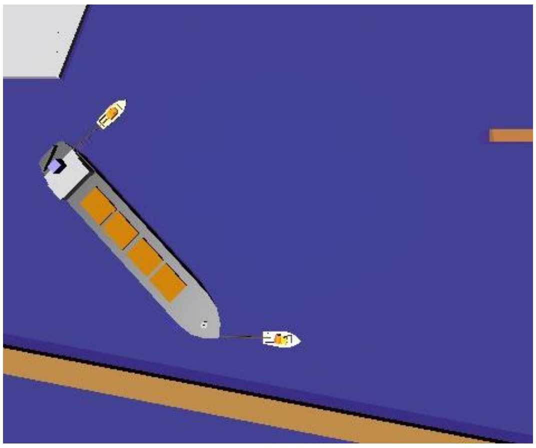

Fig. 24. Snapshot 7

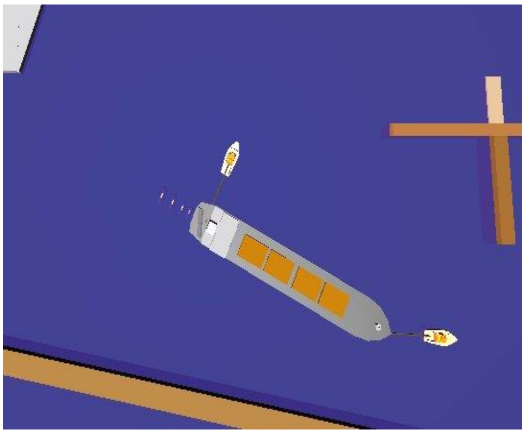

Fig. 25. Snapshot 8 (simulation stop)

V. Conclusions

In this paper, the complex problem regarding the simulation of a ship berthing/unberthing (with and without tugboats), under severe weather conditions, has been addressed.

In particular, a numerical simulation model has been developed by merging various theoretical submodels, adopted to represent different aspects as ship propulsion, manoeuvrability and effect of adverse environmental conditions.

The problem has been approached by integrating CFD modeling into time domain simulation. This last numerical technique has been used to model wind forces for a given ship, operating in a precise harbor. In fact, this method is able to represent the actual ship layout, including different loading conditions, providing more reliable results with respect to general literature data.

By this integrated procedure, it is possible to evaluate the behavior of a particular ship under different combinations of wind and sea current speeds and, as a consequence, how the tugboats could control the ship response.

The simulator is then structured to consider all the main parameters that can affect the ship towing operations, in order to investigate particular issues, as the proper use of the number and power of tugboats to be employed; a reliable simulation can be also adopted to post-process the dynamics of real marine accidents, in order to assess possible human errors.

References

[1] M. Altosole, G. Benvenuto, U. Campora, M. Figari, Real-Time simulation of a COGAG naval ship propulsion system, *Journal of Engineering for the Maritime Environment, 223 (1):47-62, 2008.*

[2] M. Altosole, G. Dubbioso, M. Figari, S. Michetti, A. Millerani Trapani, M. Viviani, Simulation of the dynamic behaviour of a CODLAG propulsion plant, *Warship 2010 Conference, London, UK, June 2010*.

[3] M. Altosole, M. Figari, G. Marcilli, M. Ratto, A. Spadoni, Ship control system wide integration and the use of dynamic simulation techniques in the Fremm project, *International Conference on Electrical Systems for Aircraft, Railway and Ship propulsion, Bologna, Italy, October 2010.*

[4] M. Altosole, M. Figari, M. Martelli, Time domain simulation for marine propulsion applications, *Proceedings of the 44th Summer Computer Simulation, Genoa, Italy, July 2012*.

[5] The Society of Naval Architect and Marine Engineers, *Principles of Naval Architecture, Vol. III* (Edward V. Lewis, 1989).

[6] D. Kang, K. Hasegawa, Prediction method of hydrodynamic forces acting on the hull of a blunt body ship in the even keel condition, *Journal of Marine Science and Technology, 12:1-14, 2007*.

[7] M. Altosole, M. Figari, Effective simple methods for numerical modelling of marine engines in ship propulsion control systems design, *Journal of Naval Architecture and Marine Engineering, December:129-147, 2011.*

[8] R. S. Benson, N. D. Whitehouse, *Internal Combustion Engines* (Pergamon Press International Library, Oxford, 1979).

[9] G. Benvenuto, U. Campora, A Computer Model for the Dynamic Simulation of Turbocharged Diesel Engine, *The 1st International Conference on Control and Diagnostics in Automotive Applications, Genoa, Italy, October 1996.*

[10] G. Benvenuto, U. Campora, A Calculation Procedure of the ‘Real’ Cycle for Four-Stroke Diesel Engines, *52° A.T.I. National Congress, Cernobbio, Italy, September 1997*.

[11] G. Benvenuto, U. Campora, G. Carrera, P. Casoli, A Two-Zone Diesel Engine Model for the Simulation of Marine Propulsion Plant Transients, *Second International Conference on Marine Industry, Varna, Bulgaria, October 1998*.

[12] The Society of Naval Architect and Marine Engineers, *Principles of Naval Architecture, Vol. II* (Edward V. Lewis, 1989).

[13] M. Viviani, G. Dubbioso, G. Soave, M. Notaro, R. Depascale, Hydrodynamic coefficients regressions analysis and application to twin screw vessels, *IMAM 2009 Conference, Instanbul, Turkey, October 2009*.

[14] J. Brix, *Manoeuvring Technical Manual* (Seehafen Verlag, Hamburg, 1993).

[15] A. F. Molland, S. R. Turnock, *Marine Rudders and Control Surfaces* (Butterworth Heinemann, 2007).

[16] S. Brizzolara, E. Rizzuto, Wind Heeling Moments on Very Large Ships. Some Insights through CFD Results, *The 9th International Conference on Stability of Ships and Ocean Vehicles, Rio de Janeiro, Brazil, September 2006.*

Authors’ information

[1] Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture (DITEN), University of Genoa, Italy.

[2] Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture (DITEN), University of Genoa, Italy.

[3] Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture (DITEN), University of Genoa, Italy.

[4] Department of Electrical, Electronic, Telecommunications Engineering and Naval Architecture (DITEN), University of Genoa, Italy.

Marco Altosole obtained his PhD degree in Naval Architecture and Marine Engineering from University of Genoa in 2005. He is currently Assistant Professor of the “Electrical, Electronic, Telecommunications Engineering and Naval Architecture Department” (DITEN) of Genoa University, Italy. His main research interests are Marine Propulsion Systems, Control Systems of Naval Vessels and Numerical Simulation in Marine Applications. He is the author of more than 50 scientific papers, presented at international conferences and journals.

Dario Boote is Professor of “Ship Construction” of the “Electrical, Electronic, Telecommunications Engineering and Naval Architecture Department” (DITEN) of Genoa University, Italy. He is President of the Bachelor and Master Courses in Yacht Design at La Spezia Campus. His research activity deals with Ship and Yacht Structures and Materials. He is the author of more than 100 scientific papers, presented at international conferences and journals.

Stefano Brizzolara, PhD in numerical hydrodynamics, is Assistant Professor of the University of Genoa. As Head of the Marine CFD Group, he is currently guiding different research projects for ONR, Italian Ministry of Defense and EU, all dealing with non-conventional high efficiency hull form and propulsors design, devised and optimised by CFD methods. His previous experience includes navy ships and propellers design in the hydrodynamic design office of Fincantieri Naval Business Unit and experimental research at the cavitation tunnel of the Italian Navy in Rome. He is the author of more than 70 scientific papers, presented at international conferences and journals.

Michele Viviani obtained his PhD degree in Naval Architecture and Marine Engineering from University of Genoa in 2004. He is currently Assistant Professor of the “Electrical, Electronic, Telecommunications Engineering and Naval Architecture Department” (DITEN) of Genoa University (Italy), where he is responsible of the cavitation tunnel. His main research interests are related to Cavitation Tunnel activities (and in general Marine Propeller Design) and Ship Manoeuvrability. He is the author of more than 80 scientific papers, presented at international conferences and journals.