Volume XXIV 2021
ISSUE no.1
MBNA Publishing House Constanta 2021

Scientific Bulletin of Naval Academy

SBNA PAPER • OPEN ACCESS

Hydrodynamic study of the flow developed around a bare hull ship in static drift motion

To cite this article: Oana MARCU, Elena-Gratiela ROBE-VOINEA and Dan OBREJA, Scientific Bulletin of Naval Academy, Vol. XXIV 2021, pg.48-53.

Submitted: 16.03.2021
Revised: 15.06.2021
Accepted: 22.07.2021

Available online at www.anmb.ro

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-21-I1-005
SBNA© 2021. This work is licensed under the CC BY-NC-SA 4.0 License

Hydrodynamic study of the flow developed around a bare hull ship in static drift motion

Oana MARCU[1]

Elena-Gratiela ROBE-VOINEA[1]

Dan OBREJA[2]

[1] "Mircea cel Batran" Naval Academy, Constanta, Romania
[2] "Dunarea de Jos" University of Galati, Galati, Romania
Oana MARCU, oana.marcu@anmb.ro

Abstract. The present study gives a Computational Fluid Dynamics (CFD) based insight into the three-dimensional incident flow developed around a very large crude carrier ship during static drift motion. The research proposes a set of virtual Planar Motion Mechanism (PMM) tests of "static drift" type conducted for a number of seven drift angles in the range of -9° to +9°. The emergence and development of vortical structures along the 1:58 KRISO Very Large Crude Carrier 2 (KVLCC2) tanker model are examined and explained, the influence of the considered drift angles being highlighted.

1. Introduction

As stated in Circular 1053 [1], marine casualties and pollution may be caused, among others, by the poor manoeuvring performance of the ship. This fact enforced the maritime regulatory bodies to consider the "development and implementation of standards for ship's manoeuvrability, particularly for large ships and ships carrying dangerous goods in bulk" [2]. It, thus, became mandatory to determine and optimize ship's manoeuvrability from the preliminary design stages. Although the measures imposed by International Maritime Organization (IMO) assume none but the determination of the manoeuvrability performance of the ship at the design stage, it is also important to be able to act for optimizing these performances. In this regard, it becomes a must to understand the three-dimensional, incident flow developed around a manoeuvring ship and to be able to use this understanding for obtaining the best manoeuvring solution.

With the purpose of responding to this need for knowledge, the present research depicts the results of a set of virtual PMM tests of "static drift" type conducted for the 1:58 KVLCC2 tanker model. The numerical tests are conducted for a number of seven drift angles in the range of -9° to +9°. The emergence and development of the vortical structures along the ship hull are examined and explained, the influence of the imposed drift angles being highlighted.

The scientific paper follows the direction drawn within the Gothenburg 2000 [3] workshop concerning the consideration of the CFD based approach for the study of the incident flow developed around the ship hull. Since then, major contributions to the subject were constantly brought by researchers such as Simonsen and Stern [9], [10], [11], Toxopeus [12], [13], Eca et al. [14], Marcu and Lungu [15], Fureby et al. [16] and Bhushan et al. [17]. By applying Reynolds averaged Navier-Stokes (RANS) type techniques the listed studies have addressed the flow around a ship hull with or without rudder and/or propeller in straight ahead and/or oblique motion providing a better understanding of the phenomena governing the asymmetric flow of a manoeuvring ship.

Addressing maneouvrability issues that are significant for full bodied ships with high block coefficient, the research considers the KVLCC2 chemical tanker.

Designed by Maritime and Ocean Engineering Research Institute (MOERI), the KVLCC2 tanker hull (Figure 1) was considered as a benchmark ship at all the workshops held on problems of verification and validation of ship manoeuvring simulation methods [3], [4], [5], [6], [7]. Full bodied ship with a complicated geometry that develops high speed and pressure gradients, the KVLCC2 tanker is specific to the class of single screw vessels with large bloc coefficient and bow and stern bulbs [7] to which it belongs.

Figure 1. KRISO Very Large Crude Carrier 2 [7]

Table 1 details the main dimensions for both full scale ship and 1:58 experimental model.

Table 1. Main dimensions KVLCC2

Dimension	full scale ship	1:58 model scale
Length between perpendiculars, L_{PP} (m)	320.00	5.5172
Maximum beam of waterline, B_{WL} (m)	58.00	1.00
Draft, T (m)	20.80	0.3586
Displacement, Δ (m^3)	312635	1.6023

2. Mathematical model

The numerical investigation of the three-dimensional flow developed around the bare hull KVLCC2 tanker ship in static drift motion is based on the use of a RANS solver that gives the mathematical solution of the incompressible Reynolds averaged Navier-Stokes equations. The nonlinear eddy viscosity type model – Explicit Algebraic Stree Model (EASM) of Gatski and Speziale [18] is used for closing the equations. Besides solving the “closure problem”, EASM is well known for capturing with high accuracy the turbulent character of the flow.

Dirichlet and Neumann boundary conditions are imposed on the geometric boundaries of the computational domain (Figure 2). The mathematical constraints of the Reynolds averaged continuity and momentum equations are formulated in terms of pressure, velocity, turbulent kinetic energy and turbulent frequency.

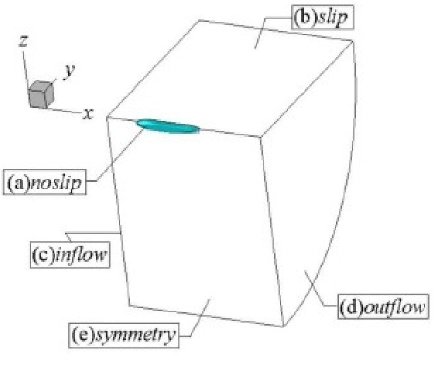

Figure 2. Boundary conditions [19]

3. Computational grid

In order to obtain the numerical solution of the virtual PMM tests, the study considers a 3D monoblock fully structured grid of H-O type (Figure 3) consisting of a number of 2750000 cells. An elliptical bidimensional technique is used for grid generation.

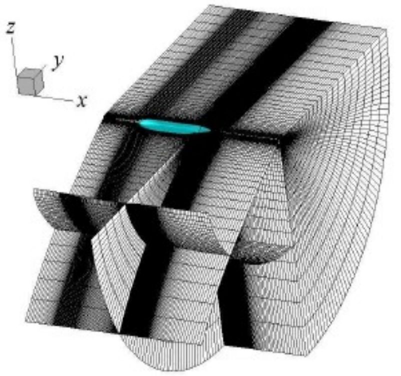

Figure 3. 3D Computational grid [19]

For capturing as well as possible the turbulent character of the flow, the computational grid is clustered near the solid boundary of the ship hull in the longitudinal and radial direction. By increasing cells density in the areas of interest it is facilitated an accurate solution for velocity and pressure gradients.

4. Results and discussions

For achieving the goal of the present scientific research which implies a better understanding of the three-dimensional incident flow developed around a manoeuvring ship, the paper presents the solutions obtained for a set of virtual PMM tests of “static drift” type.

4.1. *Modelling conditions*

The selection of the modelling conditions is based on the experimental data given by MOERI [4] for the 1:58 KVLCC2 model ship and implies seven drift cases obtained by varying the imposed drift angle with 3^o step between -9^o to $+9^o$. The velocity of the model is 1.047 m/s, the water temperature is 11.1^O C and the corresponding Froude and Reynolds numbers are equal with 0.142 and $4.6x10^6$ respectively. The sufficiently small value of the Froude number specific to this type of tanker ships with high block coefficient and low speed allows the neglect of the free surface. Likewise for simplifying the computational effort the motion of the model ship are restricted in the longitudinal and vertical direction.

4.2. *Computational results*

Of utmost importance, the research starts with validating the numerical methodology [19]. The experimental results used for the CFD-EFD (Experimental Fluid Dynamics) comparison are provided in [3], [8] and [20] and depict the axial velocity distribution in several planes longwise the tanker ship as well as the pressure distribution on the ship hull [19].

After proving its effectiveness for both straight ahead and “static drift” motion [19], the CFD approach is further used to penetrate the asymmetric flow field developed around the bare hull ship engaged in a static type manoeuvre.

Figure 4 presents the axial velocity distribution in the propeller plane for each of the seven defined drift cases, whereas Figure 5 depicts the axial velocity distribution, in the longitudinal direction, for -9^o, 0^o and 9^o drift angle.

Figure 4 (a) reveals the symmetry of the flow with respect to the centerline and the “hook”-like shape of the contour lines in the propeller disc area highlighting the presence of two aft bilge vortices.

As Figure 5 (b) shows the two contra-rotating vortical structures appear in the bilge area, downstream of *0.7Lpp* and evolve in an axisymmetric manner in relation to the longitudinal plane of symmetry of the ship, their influence being felt far downstream. The vortices are favoured by the accentuated narrowing of the ship hull in the corresponding region and by the pronounced curvature of the surface due to the presence of the stern bulb. It is, thus, reduced the magnitude of the axial velocity component in the propeller disc. The reached values are equal with half or less than half the ship speed.

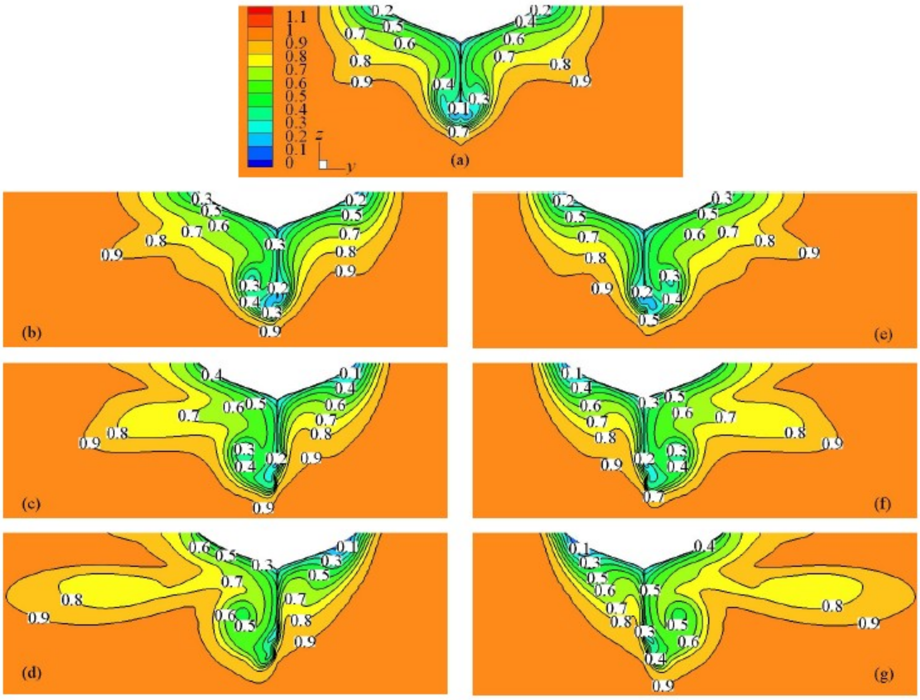

Figure 4. Axial velocity distribution in propeller plane (*0.9825Lpp*) for 0° (a), -3° (b), -6° (c), -9° (d), 3° (e), 6° (f), 9° (g) drift angle [19]

In the same analyzed figures, the study captures the appearance and evolution of two other vortex formations. They are identified in the starboard and portside margin area of the axial velocity contours, are contra-rotating and axisymmetric and have a development similar with the bilge vortices taking birth around *0.7Lpp* and evolving far downstream.

As presented in [9] for the *Esso Osaka* ship the two pairs of swirls have the same origin but different evolution. The “hook”-like pair of vortices originate in the bilge area, advance towards the bottom of the ship, meet in the propeller plane and continue their route, in a symmetric manner, downstream of the hull. The second pair of vortices originate in the same bilge region but grow along the lateral edges of the axial velocity contours.

Following the same goal, the numerical study continues with analyzing Figure 4 (b) to (g) and Figure 5 (a) and (c). It is observed the symmetry with respect to the centerline of the axial velocity distribution for the cases defined by equal but opposite drift angles as well as the manner in which the incidence of the flow influences the evolution of the aforenamed turbulent structures. The intensity of the vortices identified in the starboard and portside edge area increases with increasing the drift angle in the bord opposite to the incident flow and diminishes in the incident bord.

Going further with the introspection of the turbulent flow the paper proposes a more detailed investigation of the cases governed by the highest imposed drift angles (-9° and +9°). Figure 4 (d) and (g) and Figure 5 (a) and (c) highlight in the bord opposite to the incident flow a strong vortex formation which, by comparison with the straight ahead case, arises earlier, at *0.4Lpp*. In addition to the case of zero drift angle, Figure 5 (a) and (c) also capture a new circular formation that seems to be generated in the bow area and adds to the utmost lateral structure amplifying its intensity. As for the second vortex, symmetrically developed along the ship hull for the 0° case, it considerably changes its intensity but still retaining its origin. The axial velocity representations emphasize the phenomenon, revealing the fact that it can be no longer found in the propeller plane.

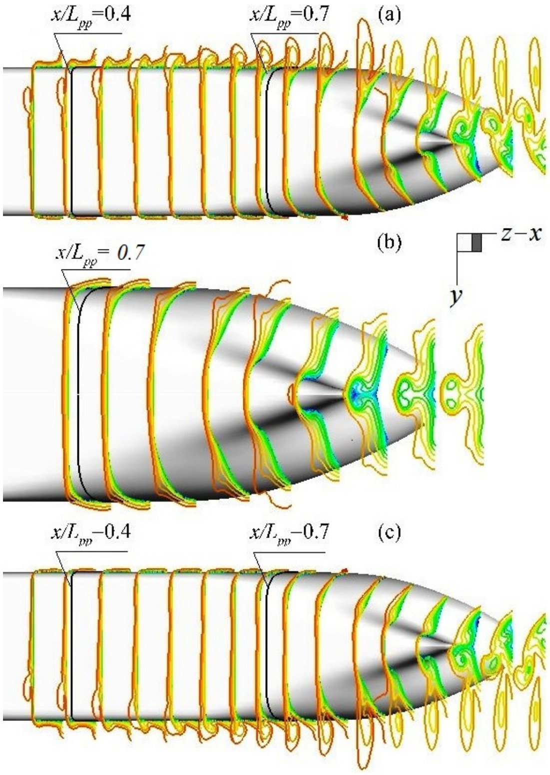

Figure 5. The axial velocity distribution, in longitudinal direction, for -9° (a), 0° (b) and 9° (c) – view on the bottom of the hull [19]

Similarly, the "hook"-like vortices significantly change their symmetry and magnitude. For the maximum 9° imposed incident angle, it is observed a strong development of the starboard vortex, the portside turbulent structure shown in the straight ahead situation not appearing to manifest. The -9° case shows a "mirror"-like manifestation.

For smaller drift angles as given by ±3° and ±6°, the evolution of the three-dimensional turbulent flow follows the pattern identified for ±9° reducing its intensity in the board opposite to the incidence but allowing for a small magnitude vortical structure in the propeller disc area in the incident board.

The presented results highlight the influence of a non-zero drift angle, the circular formations identified for the straight ahead case being strongly modified by the presence of the incident flow. This in combination with the highly changing shape of the tanker hull creates a particular turbulent area unfavourable to the propeller-rudder system operation. It is, thus, recommended an investigation of the "static drift" type flow developed around the hull-propeller-rudder configuration.

5. Concluding remarks

The CFD based study presents a detailed introspection of the turbulent flow developed around the bare hull KVLCC2 tanker ship for a number of "static drift" cases. The vortical structures that arise along the ship hull in the straight ahead and non-zero drift situations are identified and explained, the influence of the incident flow angel being emphasized. In order to better understand the three-dimensional, incident flow developed around a manoeuvring ship and to be able to use this understanding for obtaining the best manoeuvring solution, the present research must be continued with the consideration of the propeller-rudder ensemble.

Acknowledgments

I would like to express my gratitude to Professor Adrian Lungu my PhD Advisor, for his guidance during this research work.

References

[1] International Maritime Organization 2002 MSC./Circ. 1053 Explanatory notes to the standards for ship manoeuvrability

[2] International Maritime Organization 2002 Resolution MSC. 137(76) Standards for ship manoeuvrability

[3] Gothenburg 2000–A Worshop on CFD in Ship Hydrodynamics 2000

[4] SIMMAN 2008–Workshop on Verification and Validation of Ship Manoeuvring Simulation Methods 2008

[5] SIMMAN 2014–Workshop on Verification and Validation of Ship Manoeuvring Simulation Methods 2014

[6] Tokyo 2015–A Workshop on CFD in Ship Hydrodynamics 2015

[7] SIMMAN 2020–Workshop on Verification and Validation of Ship Manoeuvring Simulation Methods 2020

[8] Kim W J, Van S H, Kim D H 2001 Measurements of flows around modern commercial ship models *Exp. Fluid* **31** 567–78

[9] Simonsen C D and Stern F 2005a Flow pattern around an appended tanker hull form in simple maneuvering conditions *Comput. Fluids* **34** 169–98

[10] Simonsen C D and Stern F 2005b RANS manoeuvring simulation of Esso Osaka with rudder and a body–force propeller *J. Ship Res.* **49**(**2**) 98–120

[11] Simonsen C D and Stern F 2006 Flow structure around manoeuvring tanker in deep and shallow water *Proc. 26th ONR Symposium on Naval Hydrodynamics* (Rome, Italy)

[12] Toxopeus S L 2004 Validation of calculations of the viscous flow around a ship in oblique motion *Proc. of the First MARIN–NMRI Workshop* (Tokyo, Japan)

[13] Toxopeus S L 2005 Verification and validation of calculations of the viscous flow around KVLCC2M in oblique motion *5th Osaka Colloquium on Advanced Research on Ship Viscous Flow and Hull Form Design by EFD and CFD Approaches* (Osaka, Japan)

[14] Eça L, Hoekstra M, Toxopeus S L 2005 Calculation of the flow around the KVLCC2M *CFD workshop Tokyo* (Tokyo, Japan)

[15] Marcu O and Lungu A 2012 Numerical investigation of the flow around the KVLCC2 hull in static drift motion *AIP Conference Proceedings* vol 1479 pp 185–8

[16] Fureby C, Toxopeus S L, Johansson M, Tormalm M, Petterson K 2016 A computational study of the flow around the KVLCC2 model hull at straight ahead conditions and at drift *Ocean Eng.* **118** 1–16

[17] Bhushan S, Yoon H, Stern F, Guilmineau E, Visonneau M, Toxopeus S L, Simonsen S, Aram S, Kim S-E, Grigoropoulos G 2019 Assessment of computational fluid dynamic for surface combatant 5415 at straight ahead and static drift $\beta = 20$ deg *J. Fluids Eng.* **141(5)** 051101

[18] Gatski T B and Speziale C G 1993 On explicit algebraic stress models for complex turbulent flows J. Fluid Mech. **254** 59–78

[19] Marcu O 2012 *Contributions on the study of the flow developed around the hull-propeller-rudder configuration. PhD Thesis* (Galati: “Dunarea de Jos” University of Galati) p 205

[20] Gothenburg 2010–A Worshop on CFD in Ship Hydrodynamics 2010