CFD Prediction of the Asymmetrical Shaft Unbalance During Ship Maneuvers

Diego VILLA [a,1] and Stefano GAGGERO [a],

[a] *Department of Electrical, Electronic and Telecommunications Engineering and Naval Architecture (DITEN), University of Genoa, Via Montallegro 1, 16145, GE, Italy*

Abstract. The paper explores the accuracy of a low-cost CFD based approach to evaluate the propeller load variation experienced during manoeuvring conditions. The proposed procedure is based on the inclusion, in the ship hydrodynamic analyses by RANS, of the propeller effect through a body-force approach calibrated on BEM calculation to realize a computationally efficient method. Numerical results have been compared with the literature available experimental data performed on the well-known DTMB5415 benchmark test case, where the thrusts experienced by both of her propellers during dedicated Captive Model Tests were recorded. Both pure drift and pure yaw tests have been considered in the numerical campaign to cover the entire kinematic conditions involved during standard IMO manoeuvres. To prove the effectiveness of the method, also a severe turning circle condition is evaluated. The comparison shows the maturity of these numerical calculations, even if based on a simplified approach, to correctly evaluate the propeller unbalance, opening the way to the application of the proposed method to investigate the causes of load variations in manoeuvre conditions and directly in manoeuvre simulations.

Keywords. Ship Maneuvering, Computational Fluid Dynamics (CFD), Propeller Body-Forces, Propeller Load Unbalance,

1. Introduction

The ship manoeuvrability problem gained increasing importance in the last years, mainly because of the introduction of the manoeuvrability Standards by the International Maritime Organisation [1]. Consequently, the ability to predict the manoeuvring characteristics of a vessel became a mandatory task already in the design stage. Many studies have been dedicated on these aspects, but only a few of them deal with twin-screw ships, therefore leaving the designers with a lack of experience when they are involved in the design of fast vessels. This problem is particularly relevant when the dynamics of the propeller forces during manoeuvres are of interest. In general, it is well-known that the marine propulsion systems can experience large power fluctuation during tight manoeuvres (with fluctuation also exceeding 100% of the steady values in the straight course). This is caused mainly by the ship speed loss and the complex inflow field experienced by the propeller. These aspects can be further stressed when twin-screw ships are considered, since they can experience different dynamics on each propeller shafts, gener-

[1]Corresponding Author: diego.villa@unige.it.

ating considerable load asymmetry (propeller unbalances), which, for particular propulsion layouts, can cause significant mechanical stresses to the engines or to other machinery. Few papers have already considered these aspects, and some examples are given in [2,3,4]. The problem, however, was tackled mainly from an experimental point of view, with the primary aim to feed a specific mathematical manoeuvre model used to simulate the standard manoeuvres. Even if an experimentally based approach can provide valuable data for a particular test case, it cannot be considered a practical method for new designs. On the other hand, with the increase of computational power, Computational Fluid Dynamic (CFD) has become a reliable tool also for engineering applications of industrial problems like those previously discussed. Some pioneers works in this direction were carried out by [5,6] and in particular in [6] some viscous computations (without the presence of the propeller) were considered to analyse the nominal wake fractions experienced by the two propellers to shed a light into the physical mechanisms on the basis of the propeller overload. Following this strand, and on the basis of successful application of coupled BEM/RANS method for the fast and accurate prediction of propeller performances in self-propulsion conditions, as in [7], in current work a BEM/RANS combined approach is proposed for the evaluation of unbalanced propeller loads in manoeuvring conditions. The method is systematically applied to the DTMB5415 test case and the results are validated using the experimental measurements available for pure drift, yaw and turning circle tests.

2. Numerical Methodology

The CFD simulations of the flowfield around the ship have been performed by means of the RANS open-source libraries provided by the *OpenFOAM* project [8]. This code is based on a finite volume method adopted to discretize the flow equations on an unstructured polyhedral mesh. Considering the involved flow velocities, the turbulence closure is realized with the widely adopted *Shear Stress Transport* $k-\omega$ approach. Among all the available time discretization approaches, a quasi-steady method (the Local Time-Stepping, LTS) has been preferred because of its computational efficiency to solve the evolution of the free surface to its steady configuration for constant ship velocity by handling a variable time-step for each cell (based on the local cell size and flow field characteristics). This approach is particularly suitable for pure advective equations like the one, based on the *Volume of Fluid* approach, used in the free-surface capturing scheme.

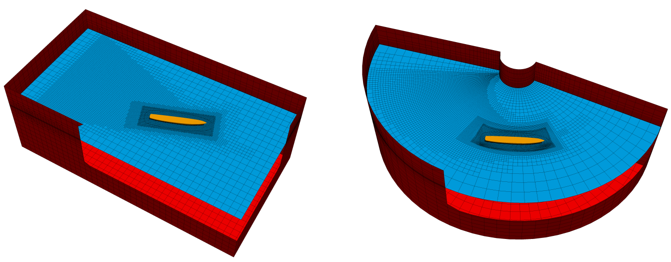

Figure 1. Overview of the meshes and domain sizes adopted for drift (left) and yaw tests (right).

The mesh generation is based on an in-house-developed procedure, already successfully applied for the numerical prediction of the manoeuvring behaviour for fast ships [9]. It is based on a combination of the *cfMesh* tool with *blockMesh*, the standard structured meshing tool of the *OpenFOAM* library. Figure 1 shows two examples of the mesh arrangements adopted depending on the ship manoeuvre under investigation. In any case an hex-dominant mesh of about 4 million cells has been used to discretize the entire domain, which shape has been selected to better accommodate the undisturbed flow field and with sizes compliant with the ITTC prescriptions [10]. The aim is to minimize the influence of the boundaries on the solution while keeping, at the same time, the computational effort to a reasonable level affordable also for industrial applications. This setup was adopted with success for the numerical solution of most of the typical ship hydrodynamic problems, ranging from pure towing tests (see [11,12]) to the self-propulsion condition [13] or complex captive model tests (see [14,15,16]). All these analyses included the presence of the free surface but since the prediction of the propeller forces is only barely influenced by the free surface itself [17,18], current analyses were carried out under the double-model assumption to further reduce the computational time without sacrificing the accuracy of propeller calculations.
There are two possible strategies when the propeller performances in behind hull conditions, or the propeller influence on the ship hydrodynamics, have to be computed in a RANS simulation. The first consists of including the propeller geometry into the viscous RANS solution by means of sliding or overlapping meshes and then by simultaneously solving the hull and propeller flow field. This is the most physically consistent approach but it requires a significant computational effort because of the different flow scales (time and space) involved for the hull and the propeller problems. The second approach, that from the computational point of view adds instead a negligible increment to the overall effort, consists of including the propeller effect into the viscous solver as a source of volume forces. This approach, differently from the previous one, requires the knowledge of the forces generated by the propeller in the actual flowfield and, then, it is particularly suitable for the evaluation of the influence of the propeller functioning on the ship rather than for the direct prediction of propeller forces, the solution of which requires a dedicated solver. For these reasons it has been extensively used to investigate in an efficient way the propeller/rudder interaction and the rudder forces, crucially influenced by the propeller slipstream [19,20]. In the present activity this second approach is used in combination with a Boundary Element Method [21,22] to obtain, starting from the fixed rate of revolution of the propellers during maneuvers (and in this case given by experiments), an estimation of the delivered thrust and the absorbed torque of the propulsors. Since during severe manoeuvrings the propeller inflow changes based on its working condition (due to self-induced velocities and hull/propeller interactions), the procedure already adopted to compute the self-propulsion functioning [7] has been utilized to feed the propeller solver with appropriate effective wakes. A simplified estimation of the self-induced propeller velocities, given by a dedicated analyses of identical body forces working in an equivalent uniform inflow, is subtracted from the "total" velocity field (i.e. that in correspondence of the body forces distribution in behind hull condition) of the ship. This provide a spatial non-uniform effective wake to be used for the estimation of propeller unsteady performances using the BEM and for iteratively update the process until convergence (in terms of estimated forces or wake characteristics).

3. Case study: the DTMB5415M

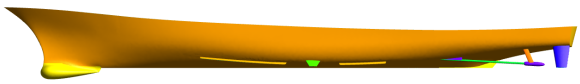

Figure 2. Side view of the DTMB5415M fully appended hull shape.

The selected ship for this analysis is the *DTMB5415*, a fast twin screw/twin rudder ship initially conceived as a surface combatant for the Navy in the '80 but finally used as a literature benchmark test case. The hull geometry includes a sonar dome, a transom stern and two shaft lines supported by two V-brackets each. In addition, the fully-appended configuration is equipped with a bilge keel interrupted by the presence of the fin stabilizer on each side. This setup, which is the one used in present analyses, is shown in Figure 2. Table 1 summarizes the main particulars of the hull and its appendages.

Table 1. Ship and appendages main data at full- and model-scale.

Ship				**Appendages**			
Data	**Unit**	**Ship**	**Model**	**Data**	**Unit**	**Ship**	**Model**
Lambda	[-]	1	35.48	Propeller Diameter	[m]	6.15	0.173
Length	[m]	142	4.00	Pitch ratio at 0.7R [m]	[-]	0.865	0.865
Breath	[m]	19.06	0.537	Number of blades	[-]	5	5
Draft	[m]	6.15	0.173	Rudder Area	[m^2]	15.83	0.012
Wet Surface Area	[m^2]	71.97	0.057	Stabilizer Area	[m^2]	6	0.004
				Bilge Keel Length	[m]	28.4	0.800

The model-scale size has been selected in compliance with the experimental campaigns conducted at MARIN and collected in [23,24,25]. In these activities, captive model tests using Planar Motion Mechanism were carried out at different kinematic conditions to measure all the ship hydrodynamic coefficients. Among all the available experimental data, those including the records of the force transducers mounted on both shafts are of particular importance since they permit a validation of the proposed numerical procedure. Table 3 summarizes the selected conditions. All the tests were conducted at 18 knots of approaching speed (equivalent to a Froude number of 0.248), exploring pure drift and pure yaw tests. Additionally, also a free-running test (the turning circle at 35 degrees of rudder angle with 18 knots of approaching speed, after stabilization of all the kinematic quantities) has been considered as representative of a real navigation condition.

Table 2. Analyzed test conditions for the DTMB5415M at 18 knots (equivalent to Fn=0.248)

Type of Test	**Ship Speed** [kn]	**Drift Angle** β [°]	**Yaw rate** r' [-]
Pure Drift	18	[0 - 20]	0
Pure Yaw	18	0	[0 - 0.6]
Stabilized Turning Circle	14.13*	11.6	0.492

* reduced speed from the 18 knots of approaching speed.

4. Results

4.1. Global manoeuvre forces and moments

Virtual captive model tests have been preliminary compared with the available experimental measurements in terms of global forces and moments of the ship. Two types of captive model tests have been considered: the pure drift and the pure yaw. These two types of tests (where the model is towed varying the ship drift angle or its yaw speed, respectively) represent simplified kinematic conditions encountered by a ship during typical manoeuvres. As already shown in previous papers [9], using appropriate grids capable of capturing all the relevant hydrodynamic phenomena around the ship ensures accurate predictions of forces ad moments, usually within the experimental uncertainty. With the adopted mesh and numerical setup, results for the fully appended *DTMB5415* are collected and compared with experiments, in terms of non-dimensional longitudinal and lateral forces and longitudinal and vertical moments, in figure 3. The prediction of pure drift tests (on the left side of the figure) show a very good agreement with experiments: both the linear part and the non-linear contribution to the forces and moments are well captured. It is worth mentioning that all these simulations have been carried out including the actual dynamic ship attitude (sink and trim) as the result of current forces and moments equilibrium at fixed the heel angle, as in the experimental campaign. Regarding the non-dimensional longitudinal component (X'), only the hydrodynamic hull force has been reported, without considering the force reduction on the anchor point given by the propellers forces. Pure yaw tests show, instead, a slight underestimation of the yaw moment, mainly ascribable to an underestimation of the non-linear part. It is worth noting, however, that the experimental uncertainty associated to this type of tests is usually greater than those observed for pure drift; this is also confirmed by the relatively higher level of "noise" visible in the proposed experimental data. For the aims of the present work, in any case, the attained accuracy can be considered appropriate and comparable to the state of the art for these kinds of simulations.

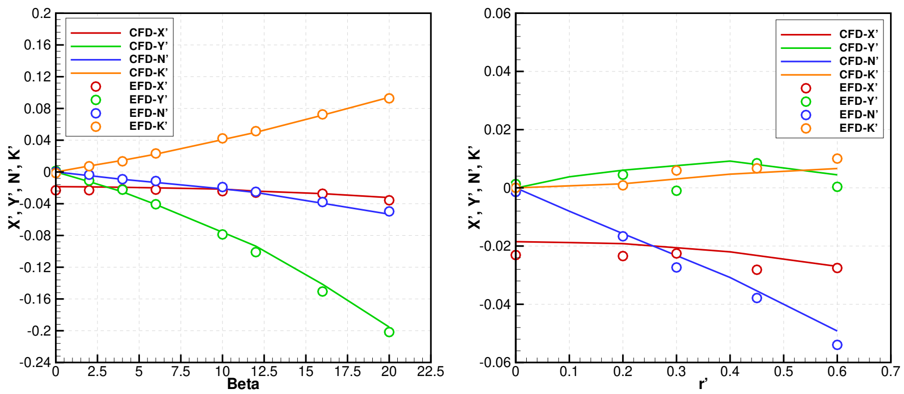

Figure 3. Comparison (CFD vs EFD) of the global ship forces for pure drift (right) and pure yaw (left) tests at a speed equivalent of 18 knots.

4.2. Propeller forces

During a manoeuvre, a propeller can experience higher or lower loads compared to those of the straight course. This can be ascribed, generally, to a speed loss of the ship during the manoeuvre and, locally, to the differences in the inflow to the propeller itself. In the case of twin-screw ships, each propeller can experience opposite loading conditions due to a combination of mutual interactions and interactions with the ship wake. Even if manoeuvre conditions are often transient situations for the ship, then not of particular interest for the propeller designer, they significantly impact the design of the propulsion plan layout especially if the entity of the load variations can generate relevant stresses to the propulsion system. Intuitively, the inward propeller, that is located in the hull shadowed region, should experience lower inflow velocity due to the stronger hull wake, while the outward propeller should operate in a more uniform, open water like, and fast velocity field which should result into an inward propeller heavily overloaded and in an outward one slightly unloaded. This, instead, is not the behaviour of the propeller forces measured during the pure drift tests for the selected test case. As shown in figure 4, which collects the thrust variation for the two propellers during the drift and yaw tests with respect to the values in straight course, when a very high drift angle is considered, an evident opposite trend is observed: the inward propeller generates a lower thrust, the outward, instead, a significantly higher one.

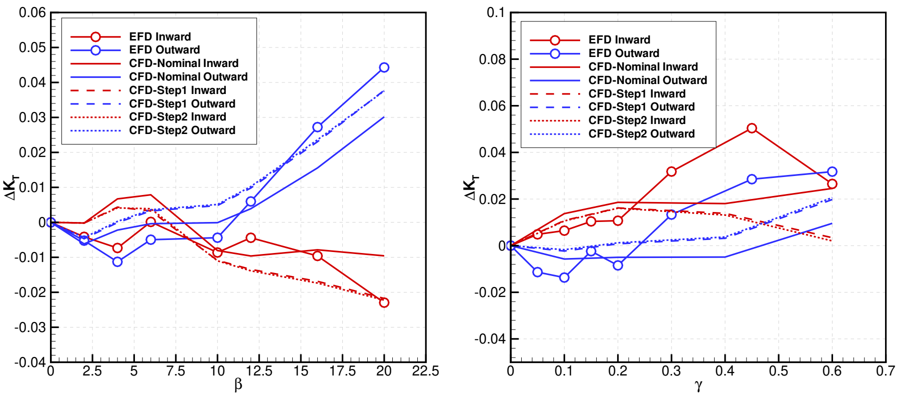

Figure 4. CFD and EFD non-dimensional thrust variation for pure drift (left) and pure yaw (right) tests. Intermediate steps of the iterative numerical procedure shown with the dashes lines.

A similar behaviour is predicted by the CFD simulations themselves using the simplified coupling between BEM and RANS described in previous section. Already with the nominal wake (i.e. without any action of the propellers, solid lines), which is a pure theoretical condition, the inward propeller is unloaded, the outward overloaded, despite the nominal wakes (wake fractions) computed by the CFD and collected in figure 5 show as expected significantly slower inflow to the inward propeller with respect to that to the outward one. The reasons of these discrepancies can be discussed comparing the local (i.e. not only axial wake fraction) and three-dimensional (i.e. including radial and tangential components) nature of the wakes to the propellers proposed in figure 6. For the inward propeller (on the left), rotating inward over the top, the increase of loading ascribable to the reduction of the axial component of the velocity is more than compensated by the tangential flow in the lower portion of the propeller disk, resulting in an averaged unloading of the propulsor. On the contrary, for the external propeller the tangential components of the wake contribute to a local loading of the blades in the same portion of the disk, with a net increase of the thrust provided on the outward shaft, visible also in the instantaneous pressure distributions over the blades of figure 7. This phenomenon is exacerbated when the propellers operate in the effective wakes, which in the end provide, as expected, predicted propeller performances closer to the experimental measures.

Also in pure yaw tests the two propellers show a similar behaviours. The inward propeller has a reduced load, the outward is overloaded. In this case a certain difference between measurements and calculations can be observed. It can be explained, at least partially, by the higher uncertainty associated to this type of experimental tests. In any case the load difference between the two shafts is very well captured and also the load "inversion" at the highest tested yaw rate ($\gamma' = 0.6$) recorded in the experimental campaign is reasonably predicted by the numerical analyses when true effective wakes are considered. This result, in particular, show the necessity to properly consider also the hull-propeller interaction (i.e. the effective wake fraction) to predict the thrust variation behaviour correctly. Adopting the nominal wake only could lead to non completely corrected propeller loading trends predictions, as those in [6].

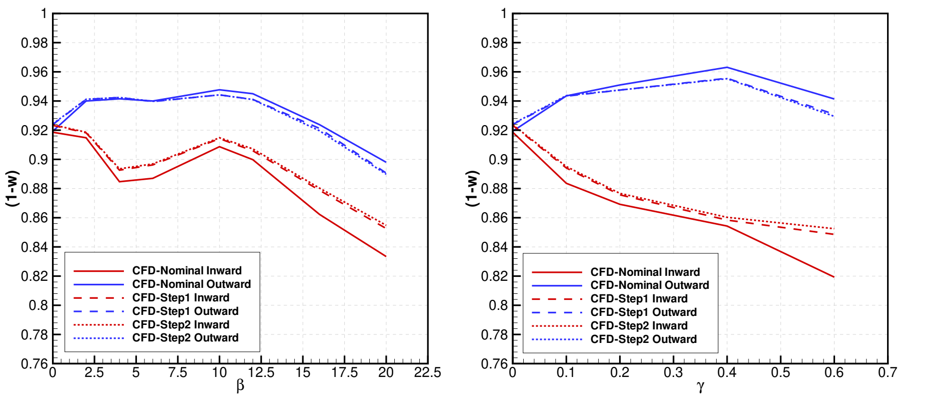

Figure 5. Nominal (solid) and effective (dashed) wake fraction for the inward (red) and the outward (blue) propeller in drift (left) and yaw (right) conditions.

To exacerbate the effect of the tangential velocities on the propeller load, the drift tests have been calculated also reversing the propeller rate of revolution (outward over the top). Figure 8 collects the thrust variations and the wake fractions in this new configuration for which, unfortunately, no experimental data are available. The computed effective wakes changing the drift angle have a very similar behaviour of the previous propeller configuration, with only appreciable variations with respect to the nominal wake (in terms of wake fraction) only for the outward propeller. A completely different combination of the tangential velocity with the propeller rotation determines very high values of loading of the inward propeller (doubled compared to the previous case)), and unloading for the outward one, partially recovered for very high drift angles. In addition, for the lower drift angles (smaller than 10°), a significant shaft unbalance occurs, not observed in the inward over the top configuration.

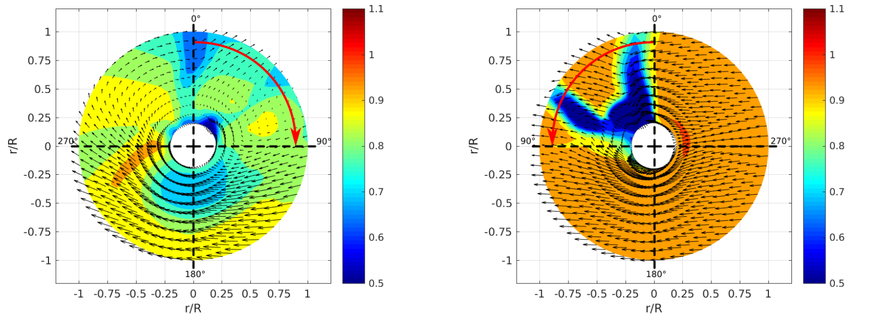

Figure 6. Nominal wakes (local distribution of axial, tangential and radial velocities) at the propeller plane for the inward (left) and the outward (right) propeller at 20° of drift angle.

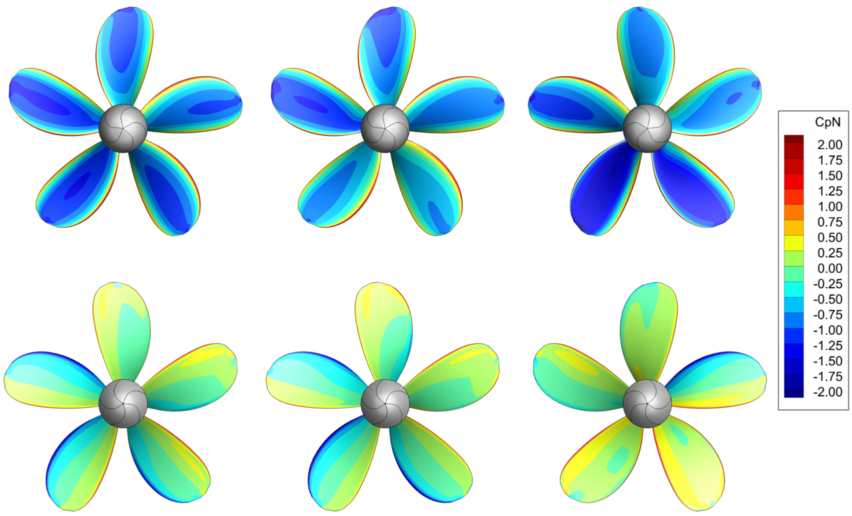

Figure 7. Pressure distributions over propeller blades. Back (top) and face (bottom) side. Inward propeller, nominal wake (left), straight course. Inward propeller, effective wake, drift angle of 20° (middle). Outward propeller, effective wake, drift angle of 20° (right).

As a final test, a free-running condition was explored. In particular, the stabilized turning circle manoeuvre with the rudder angle equal to 35° has been considered since it is representative of a real ship working condition: 11.6° of drift angle and a non-dimensional yaw rate of 0.492. For the present case, also a ship speed loss of 78.5% (equivalent to 14.13 knots) has been included. Table 3 clearly shows that the propeller unbalance is correctly predicted by the simulations: the inward propeller is slightly overloaded (fundamentally as a result of the ship speed reduction at constant rps), while the outward propeller is heavily overloaded with an increase of the thrust exceeding the 107% of the straight course condition. In addition, it is also evident, as already highlighted in previous cases, that the propeller unbalance is inevitably under predicted when the nominal wake fraction only is considered, while more consistent predictions are possible when the interaction effects (i.e. the effective wake) are accounted for.

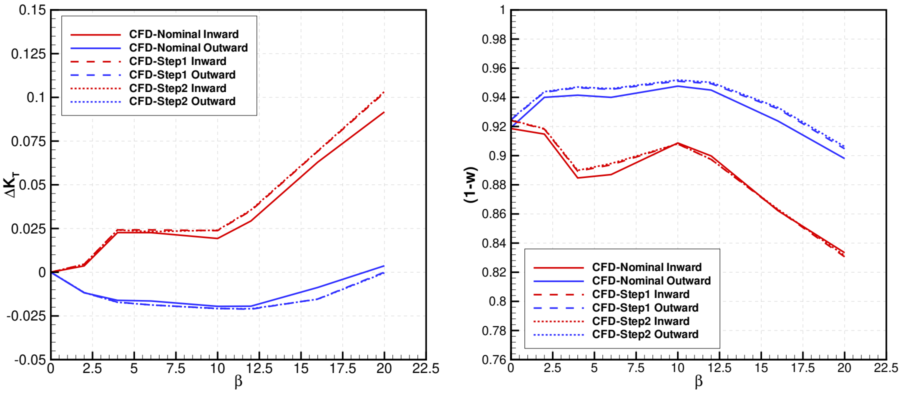

Figure 8. Drift tests imposing reversed revolution rate (outward over the top): non-dimensional thrust variations (left) and wake fractions (right).

Table 3. Thrust and torque variations during the stabilized turning circle.

Data	**Inward Prop.**		**Outward Prop.**	
-	ΔK_T	$\Delta 10K_Q$	ΔK_T	$\Delta 10K_Q$
Nominal	0.0488	0.0409	0.0908	0.1079
Step 1	0.0294	0.0189	0.1040	0.1215
Step 2	0.0256	0.0148	0.1041	0.1215
EFD	*0.0317*	-	*0.0928*	-

5. Conclusions

This paper deals with the application of a simplified approach to evaluate the propeller performances in a manoeuvre scenario using a computationally efficient combination of CFD simulations and BEM analyses. The method consists in the inclusion of a body-force field representative of the the propeller, and obtained by means of a potential flow code iteratively fed with the information extracted from the viscous simulation itself, in the solution of the flow around the hull. The combination of the two approaches provides, simultaneously, the propeller performance in the effective wake and the interactions coefficients between the propeller and the hull, which are valuable information for the design of the propulsion system. Current application encompasses, rather than the usual self-propulsion estimation for which very good performances have been already achieved, severe manoeuvring scenarios, including typical PMM tests (pure drift and pure yaw), and realistic conditions like the stabilized turning circle. Comparison with available model tests shown a very good agreement, proving the possibility of modelling all the relevant propeller related forces in an efficient numerical framework and highlighting the mechanism of shaft load unbalancing, not always clear by the analysis of the (predicted) nominal ship wakes only.

References

[1] IMO. Standards for ship manoeuvrability.Resolution MSC. 137 (76) (adopted on 4 December 2002). International Maritime Organization.

[2] Viviani M, Altosole M, Cerruti m, Arcangelo M, Dubbioso G. Marine propulsion system dynamics during ship manoeuvres. In: 6th International Conference on High-Performance Marine Vehicles HIPER'08; 2008. .

[3] Coraddu A, Dubbioso G, Mauro S, Viviani M. Analysis of twin screw ships' asymmetric propeller behaviour by means of free running model tests. Ocean Engineering. 2013;68:47-64.

[4] Kang D, Nagarajan V, Hasegawa K, Sano M. Mathematical model of single-propeller twin-rudder ship. Journal of marine science and technology. 2008;13(3):207-22.

[5] Atsavapranee P, Miller R, Dai C, Klamo J, Fry D. Steady-Turning Experiment and RANSE Simulations on a Surface Combatant Hull Form (Model 5617); 2010. .

[6] Mauro S, Dubbioso G, Broglia R, Muscari R. Investigation of Asymmetrical Shaft Power Increase during Ship Maneuvers by means of Model Tests and CFD; 2012. .

[7] Villa D, Gaggero S, Gaggero T, Tani G, Vernengo G, Viviani M. An efficient and robust approach to predict ship self-propulsion coefficients. Applied Ocean Research. 2019 nov;92(March):101862.

[8] OpenFOAM - The Open Source CFD Toolbox - User's Guide, version 8. https://openfoam.org/; 2020.

[9] Franceschi A, Piaggio B, Tonelli R, Villa D, Viviani M. Assessment of the Manoeuvrability Characteristics of a Twin Shaft Naval Vessel Using an Open-Source CFD Code. Journal of Marine Science and Engineering. 2021;9(6).

[10] ITTC. ITTC–Recommended Procedures and Guidelines-Practical Guidelines for Ship CFD Applications. 7.5-03-02-03 (Revision 01). ITTC-Recomm Proced Guidel. 2011:15-8.

[11] Ferrando M, Gaggero S, Villa D. Open source computations of planing hull resistance. Transactions of the Royal Institution of Naval Architects Part B: International Journal of Small Craft Technology. 2015;157:83 – 98.

[12] Villa D, Furcas F, Pralits JO, Vernengo G, Gaggero S. An effective mesh deformation approach for hull shape design by optimization. Journal of Marine Science and Engineering. 2021;9(10).

[13] Gaggero S, Gaggero T, Tani G, Vernengo G, Viviani M, Villa D. Ship self-propulsion performance prediction by using OpenFOAM and different simplified propeller models; 2018. p. 195 – 206.

[14] Villa D, Viviani M, Gaggero S, Vantorre M, Eloot K, Delefortrie G. CFD-based analyses for a slow speed manoeuvrability model. Journal of Marine Science and Technology (Japan). 2019;24(3):871 – 883.

[15] Piaggio B, Villa D, Viviani M. Numerical analysis of escort tug manoeuvrability characteristics. Applied Ocean Research. 2020;97.

[16] Piaggio B, Villa D, Viviani M, Figari M. Numerical analysis of escort tug manoeuvrability characteristics – Part II: The skeg effect. Applied Ocean Research. 2020;100.

[17] Gaggero S, Villa D, Viviani M. An extensive analysis of numerical ship self-propulsion prediction via a coupled BEM/RANS approach. Applied Ocean Research. 2017 jun;66:55-78.

[18] Villa D, Gaggero S, Brizzolara S. Simulation of ship in self propulsion with different CFD methods: From actuator disk to potential flow/RANS coupled solvers. In: RINA, Royal Institution of Naval Architects - Developments in Marine CFD, Papers. London England; 2011. p. 1-12.

[19] Villa D, Viviani M, Tani G, Gaggero S, Bruzzone D, Podenzana CB. Numerical Evaluation of Rudder Performance Behind a Propeller in Bollard Pull Condition. Journal of Marine Science and Application. 2018 jun;17(2):153-64.

[20] Villa D, Franceschi A, Viviani M. Numerical Analysis of the Rudder–Propeller Interaction. Journal of Marine Science and Engineering. 2020 dec;8(12):990.

[21] Gaggero S, Vernengo G, Villa D, Bonfiglio L. A reduced order approach for optimal design of efficient marine propellers. Ships and Offshore Structures. 2020;15(2):200 – 214.

[22] Gaggero S, Villa D, Viviani M, Rizzuto E. Ship wake scaling and effect on propeller performances. In: Developments in Maritime Transportation and Exploitation of Sea Resources. vol. 1. CRC Press Balkema, Taylor Francis Group London; 2013. p. 13-21.

[23] Toxopeus S, van Walree F, Hallmann R. Manoeuvring and Seakeeping Tests for 5415M; 2011. .

[24] Stern F, Agdrup K, Kim SY, Cura-Hochbaum A, Rhee K, Quadvlieg F, et al. Experience from SIMMAN 2008—The First Workshop on Verification and Validation of Ship Maneuvering Simulation Methods. Journal of Ship Research. 2011 06;55:135-47.

[25] Yoon H, Simonsen CD, Benedetti L, Longo J, Toda Y, Stern F. Benchmark CFD validation data for surface combatant 5415 in PMM maneuvers – Part I: Force/moment/motion measurements. Ocean Engineering. 2015;109:705-34.