ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2015

DESIGN OF A WAKE-ADAPTED PROPELLER FOR A CONTAINERSHIP

Noufal Paravilayil Najeeb
University of Liege
EMSHIP Master Student
Place du 20-Aout, 7, 4000, Liège, Belgium,
E-mail:noufalparavilayil@gmail.com

Mihaela Amorăriţei
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, 47
Domneasca Street, , 800008, Romania,
E-mail:mihaela.amoraritei@ugal.ro

ABSTRACT

The paper is focused on the design a wake-adapted propeller for a 1200 TEU container ship. The aim is to select the main components of the propulsion system: main engine and propeller, to achieve ship propulsive performances. The main objective is to design an optimized propeller and to verify that it is the best available solution. In-house codes based on B Wageningen series and lifting line theory have been used to design the wake-adapted propeller. Numerical investigations have been performed to analyse the hydrodynamic performances of the designed propeller in open water and behind the portcontainer ship hull.

Keywords: wake-adapted propeller, Shipflow, numerical simulation.

1. INTRODUCTION

The main challenge faced by a Naval Architect is to make an efficient design meant to achieve a good balance between parameters such as increased speed, greater cargo capacity, minimum fuel consumption, better propeller, maneuvering capabilities, etc.

Large vessels require greater power to propel at higher speed carrying more cargo, which means large fuel consumption. Therefore, the design has to be more energy efficient, to minimize required engine power installation and to reduce fuel oil consumption, not only to save operational cost but also to reduce environmental pollution which has become more stringent under IMO regulations setting emission standards. Most of the ongoing research in marine industry deals with the building of future vessels more energy efficient by hull optimization, installing energy saving devices, propeller optimization, etc.

The paper is focused on the design of a wake-adapted propeller, for a 1200 TEU container ship. The aim is to select the main components of the propulsion system, namley main engine and propeller, to achieve ship propulsive performances. The propeller is the main component of a propulsion system playing an important role in the interaction between ship hull and main engine, with direct influence on propulsive performances and ship operational costs. One of the main objectives is to design an optimised propeller and to verify that it is the best available solution.

For main engine selection and propeller design, the containership resistance has been analysed by empirical formulae (Holtrop), CFD analysis (Shipflow) and experimental data. In-house codes based on B Wageningen series and lifting line theory have been used to design a wake-adapted propeller. The engine has been selected by its power (MCR) to overcome ship resistance (including sea margin). The type of propeller selected is a fixed pitch propeller as it is the most suitable one for the container ship since the service speed for the operational route is fixed.

Numerical investigations have been performed to analyse hydrodynamic performances of the designed propeller in open water and behind the hull. The step by step procedure carried out for this paper is depicted in Figure 1.

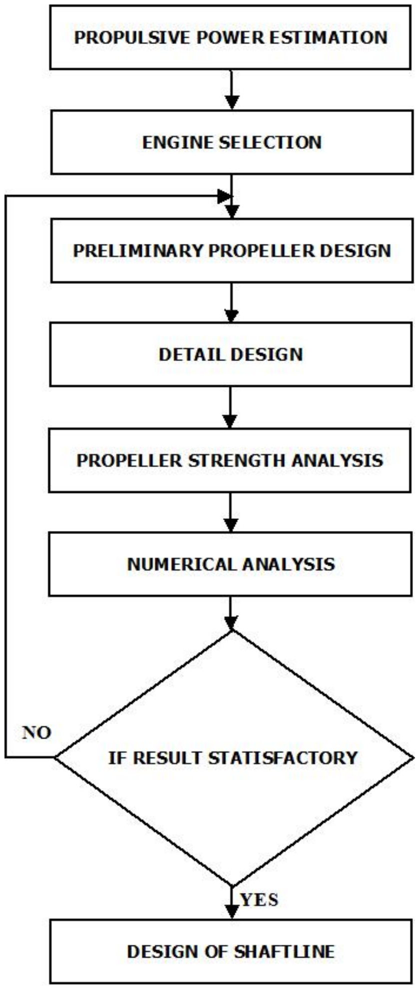

Fig.1. Flowchart showing paper objective

The vessel is a conventional container ship with bulbous bow, transom stern, single rudder and single screw propeller. The propulsion machinery with main engine is located at the aft part of the ship. The principal dimension of the vessel has been finalised after studying various hull forms to obtain as per the owners' specification as given in Table 1.

Table 1. Main particulars of containership

Parameters	Units	Values
L_{PP}	[m]	145.9
L_{WL}	[m]	150.51
B_{WL}	[m]	23.25
T_F	[m]	7.3
T_A	[m]	7.3
TEU	[-]	1200
Design Speed	[Knots]	20
C_B	[-]	0.65

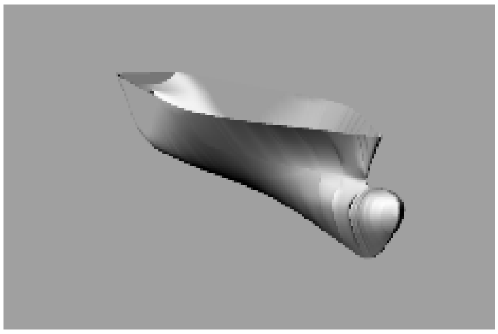

Fig.2. Containership surface created in Rhino

2. SHIP HULL RESISTANCE

An important parameter required to design the propulsion system is ship resistance, which can be described as the hydrodynamic force exerted by the fluid opposing its motion at a given velocity. In this work the resistance of the container ship has been analysed using different methods such as empirical formulae, CFD analysis and experiment.

Holtrop & Mennen method uses empirical formulae to estimate resistance coefficients, based on systematic series or statistical regressions to experimental data.

Experimental investigations involve testing of a small scale model of actual Ship (geometrically similar) in towing tank at various speeds to measure its total resistance which can be extrapolated to compute full scale resistance through Froude similitude. The experimental test results presented here have been based on towing tests conducted at ICEPRONAV Engineering Galati.

2.1. CFD Analysis

Resistance calculation using CFD technique can be done using different methods, namely RANSE, potential flow methods. By using RANSE method, the total resistance of the vessel including the viscous resistance can be determined by solving Reynolds averaged Navier-Stroke equations. Whereas potential flow solvers are an implied form of NS equation considering the fluid to be non-viscous and can only compute the wave resistance, the viscous pressure and frictional resistance can be found by ITTC 1978 procedure or using the boundary layer method. RANSE solvers have the significant advantage compared to potential flow solvers but they need a lot of computation time and resources. Due to these reasons, resistance computation has been carried out using the potential flow solver of Shipflow – XPAN based on a surface singularity panel method and frictional resistance using the boundary layer solver – XBOUND.

Different stages of CFD analysis such as pre-processing, computation and post processing have been carried out as follows.

The pre-processing has been carried out using XMESH module of Shipflow, which generates panels for XPAN computation. There are different automatic mesh modes depending on mesh density which are classified as very coarse, coarse, medium and fine being the most refine form. As given in Table 2, a grid convergence study has been carried out to find the wave resistance (Cw) coefficient for different mesh modes at the design speed of 20 Knots.

Table 2. Grid convergence study result

Mesh Modes	No.of Panes	No.of Nodes	Cw
Coarse	2622	2853	0.00044
Medium	6666	7030	0.00081
Fine	10396	10852	0.00078

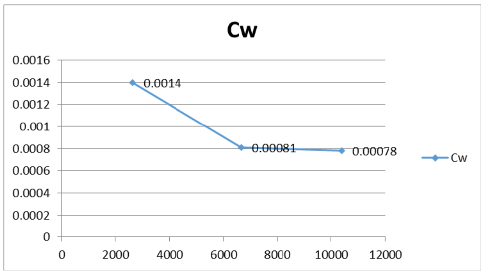

Fig.3. Cw plotted for different mesh modes

Variation in Cw between fine and medium mesh modes is very small i.e. convergence is obtained for medium. So for easiness and faster computation, grid generation for body and free surface has been carried out with automatic mesh mode using medium mesh (the total number of panels is 6666).

The free surface has been limited to 2.5 times the length between perpendiculars (LBP) along x-direction where the upstream is about 0.5 times LBP and downstream is as long as LBP. While the free surface in y-direction is taken as 0.7 times in portside as well as starboard side.

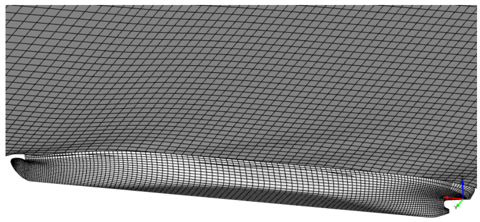

Fig.4. The discretization of hull & free surface

The computation using XPAN module involves solving non –viscous momentum equations at each discretized panel determining flow strength which are solved to get physical quantities such as velocity, pressure, etc. The potential flow theory agrees with linear superposition, where the flows generated by each panel are considered as components of the potential flow. The integration of pressure over the wetted surface area gives the wave making resistance.

Post processing of the result has been carried out using CAESES GUI. From Figure 5, we can identify that there are different sets of wave systems such as bow, stern waves generated due to the motion of the ship. Wave height is given in non-dimensional form with respect to the length between perpendiculars.

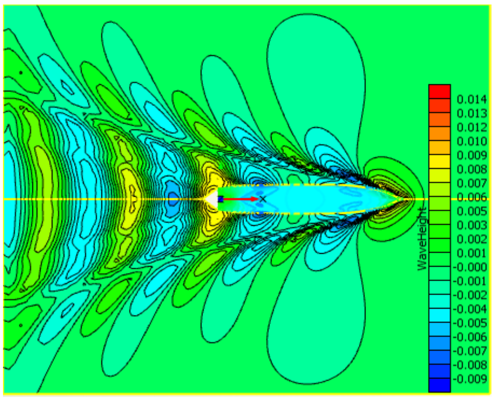

Fig.5. Wave generated due to ship motion

2.2. Results and comparison

Table 3. Comparison of ship resistance (in kN) from different methods. (EMP–empirical method, EXP-experimental tests, CFD-numerical investigation)

Speed(Kn)	EMP	EXP	CFD
16	398.9	345	322.3
17	472.2	394.5	366.4
18	556.2	468.3	438.3
19	654.7	569.6	535.9
20	778.7	721.5	692.2
21	927.4	953.7	910.9
22	1079.1	1233.1	1140.3

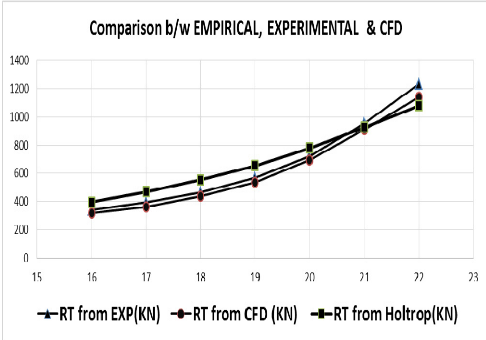

Fig.6. Comparison of resistance

The comparison of all the three methods is done as given in Figure 6. The resistance from the experimental and CFD analysis follows the same trend while the value from the empirical analysis is higher than others. From the three methods, the resistance predicted by Experimental data, which is 722 KN at design speed of 20 knots, is selected to start propulsive power estimation.

3. MAIN ENGINE SELECTION

The propulsive power estimation is carried out to select a main engine capable of generating sufficient torque, which is to be transmitted to propeller converting thrust to overcome the resistance at the design speed. The required brake power of the engine has been determined using ship resistance value from experimental tests and taking into account quasipropulsive coefficient components: (η_R - relative rotative efficiency, η_H - hull efficiency, η_o - propeller open water efficiency).

The open water efficiency has been determined through statistical method using Wageningen B propeller charts. Wageningen B propellers are a series of propeller models tested at MARIN in Wageningen. They are a group of propeller charts consisting of polynomial curves expressing thrust (K_T) and torque coefficients (K_Q) in terms of the number of blades (Z), the blade area ratios (BAR), the pitch diameter ratio (PR) and the advance coefficients (J).

For given BAR, Z polynomials of expressing thrust and torque coefficient for different PR (0.5 to 1.4) with respect to J (0.1 to 1.5) have been developed using Oostveld equations [2]. This analysis has been carried out for different Z=3,4,5 for BAR (minimum required to avoid cavitation as per Aufen Keller relation).

The propeller characteristics are determined for advance ratio at the point of intersection between K_T curve and parabolic curve defined by K_T /J^2 for all PR (A C++ code has been developed to find maximum efficiency of all intersections for the given diameter). Brake power and corresponding RPM are obtained with respect to η_o

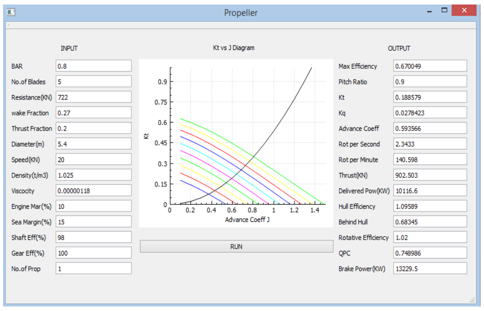

Fig.7. Brake power & RPM for Z=5

From the statistical analysis carried out for Z=5, the brake power and RPM required are 13230 KW and 141 respectively.

The engine has been selected for its power (MCR) to overcome ship resistance (including sea margin). Maximizing propeller efficiency is often associated with large diameters with low rotation rate. Taking this as a starting point, a low speed diesel engine has been selected to avoid the necessity of a reduction gear. The main characteristics of the selected engine are:

Engine Manufacturer	-	MAN
Model	-	K60MC-S
No of Cylinders	-	7
Brake Power	-	13860 kW
RPM	-	150 rpm

4. PROPELLER DESIGN

The propeller is one of the main components of a ship propulsion system, which rotates fluid around it, resulting in generation of opposing hydrodynamic forces from fluid transferring to the hull and propelling it forward. The propeller plays an important role in the interaction between hull and main engine, with direct influence on ship propulsive performances and ship operational costs.

Propeller design has been accomplished in a sequence of three stages: preliminary design, detailed design and hydrodynamic analysis of the final propeller. In the first stage, the main characteristics of the propeller such as diameter, rotation rate, blade area ratio will be obtained using B Wakening series data. These are used as initial data for the next stage, namely the detailed design performed via the lifting line theory. The objective of this stage is to find the propeller geometry for a radially varying distribution of the loading. In the final stage using CFD methods, the propeller will be investigated in steady and unsteady conditions to compare the results with the lifting line theory.

4.1. Preliminary Design

The propeller must be designed to consume the totality of the engine delivered power at a given rotation rate. The power delivered to the propeller has been calculated taking into account Sea Margin and Engine Margin requirements.

Using delivered power and engine RPM, an optimum diameter has to be obtained through statistical analysis. Required propeller clearance to avoid or minimize vibrations problems at the stern of the ship has to be checked for optimum diameter using DNV rule requirement.

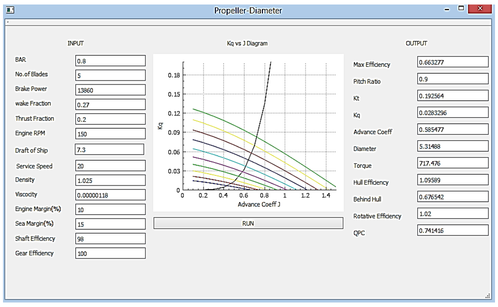

Fig.8. Optimum diameter for Z=5

Ship speed estimation was carried out to meet the owners' requirement that the vessel should be able to achieve a certain speed under power service less than maximum continuous power MCR of the engine. Checking that the service speed complies with the design speed of the vessel is crucial in the design. A C++ code has been developed to compute the P_D, RPM and Pitch ratio, etc. for different speeds (results plotted in Figure 9).

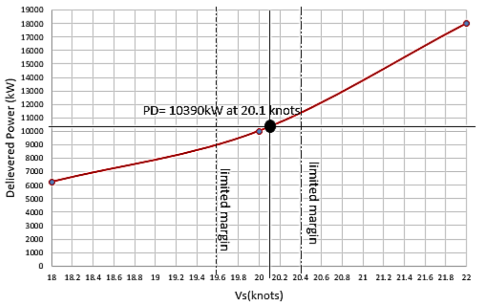

Fig.9. Delivered power vs Velocity

The main geometrical parameters and the hydrodynamic characteristics of the optimal B Wageningen propeller are presented in Table 4, based on preliminary design.

A cavitation check has been carried out using Burill diagrams, which is sufficient during preliminary design stage. The cavitation obtained is about 2.85%, which is acceptable in the case of an ocean going container ship.

Table 4. Result from Preliminary Design

Propeller Type	B5-Wageningen	
Optimum Diameter	D [m]	5.31
Maximum open water efficiency	ηo [-]	0.663
Thrust Coefficient	K_T [-]	0.193
Torque Coefficient	K_Q[-]	0.028
Blade Area Ratio	EAR	0.8
Number of Blades	Z	5
Pitch Ratio	P/D	0.9

4.2. Detail Design

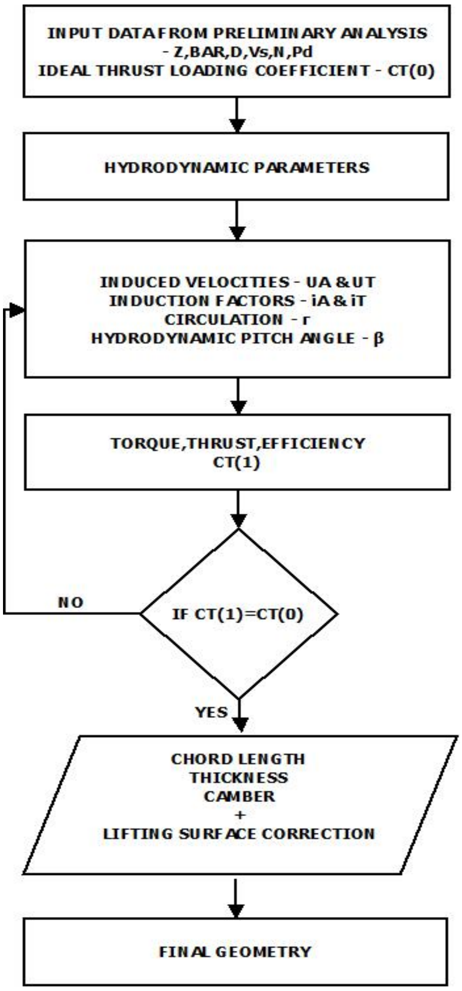

Fig.10. Schematic representation of the lifting line method to generate final geometry

The main objective of the detail design is to find the complete geometry of the propeller for a radially varying distribution of the loading. A wake-adapted propeller has been designed using an in-house code using the lifting line theory with lifting surface corrections.

The lifting line model assumes the blade sections to be replaced by a single line vortex whose strength varies from section to section. The line, which is continuous in the radial direction along which vortices act, is termed as the lifting line, and is normally considered to pass through the aerodynamic centres of the section. Since the flow is disturbed due to the presence of unsymmetrical blade sections, a circulation of streamline around the section is generated as shown in Figure 10. The lift generated by the propeller blade can be mathematically expressed in strength of circulation of vortices around the lifting line.

The step by step process of the lifting line method is explained in Figure 10.

The results regarding the geometry and hydrodynamic performances of the designed wake adapted propeller are presented in Tables 5 and 6.

Table 5. Geometrical Characteristics of the final Propeller (using NACA 66 profile)

r/R	Chord (mm)	Thickness (mm)	Camber (mm)	Pitch (mm)
0.2	1454.6	255.5	0	3622.1
0.3	1609.9	218.9	82.3	4174.6
0.4	1749.3	185	54.3	4328.8
0.5	1866.7	152.9	43	4373
0.6	1952.1	123.5	34.4	4403.7
0.7	1987.1	95.8	28.8	4426.1
0.8	1933.8	69.4	24.8	4451.7
0.9	1685.5	44.7	22.2	4447
1	0	2.3	10.6	4443.7

Table 6. Hydrodynamic Characteristics of the final Propeller

Velocity (Kn)	20.0005
Thrust (kN)	904.1
Torque (kNm)	693.1
K_T	0.19
K_Q	0.027
H	0.6
Adv Coeff J	0.5452
P/D	0.824

5. NUMERICAL ANALYSIS

CFD analysis of the propeller is used to study the hydrodynamic performance of the propeller, to understand local flow details including both steady as well as unsteady flow and to select the most promising candidate design for model testing.

Numerical analysis using SHIPFLOW software has been done in three different stages:

1) Resistance Computation
2) Open water test (steady flow)
3) Self-propulsion (unsteady flow).

5.1. RANSE for Propeller Analysis [5]

In RANSE solver – XCHAP, the effect of the propeller is taken into account by a force field computed by the lifting line method depending on circulation strength induced onto the flow due to the presence of the propeller.

The propeller is embedded in a cylindrical grid and the interaction with the surrounding grids for the hull is managed by the overlapping grid capability of the solver. When the flow passes through the propeller its linear and angular momentum increases as if it had passed through a propeller with an infinite number of blades. This results in an efficient method to predict the propeller hull interactions i.e. wake and thrust deduction factors similar to a self-propulsion test.

In SHIPFLOW, the body forces are computed with a built in lifting line propeller analysis program. The computation of axial and tangential propeller induced velocities is done using Lerb & Goldstein method. The axial and tangential force components acting on a blade section are related to the local thrust, and torque depends on the circulation, which varies in both the radial and circumferential directions of the propeller disk.

The computation of the body forces is embedded in an iterative procedure where first the current approximation of the velocity field is extracted at a representative propeller plane. The effective wake is thereafter obtained by subtracting the induced propeller wake. This is the responsibility of the propeller code and is computed by the circulation from the previous iteration in the lifting line method. The new circulation, forces and torques are computed in the effective wake. Thereafter the forces are distributed over the volume cells in the cylindrical grid. The body forces are added to the right hand side of the flow equations, where they are solved in the cylindrical component grid containing the propeller disk. At convergence the total wake computed by XCHAP and the lifting line method should match in the selected propeller plane.

5.2. Boundary Conditions

The boundary conditions used in XCHAP are inlet, outlet, slip, no-slip and interior. At the inlet, velocity and turbulence quantities are defined while the pressure gradient is set to zero. Whereas at the outlet plane, the normal velocity and turbulence gradient is considered too and the pressure quantity is set to zero. The slip condition is normally used for simulating symmetry conditions at flat boundaries. In this case, the normal velocity component and all the normal gradients of all the other quantities are zero. The slip boundary condition is applied for solid walls thus the velocity, the turbulent kinetic energy and the normal pressure gradient are set to zero. Turbulence model used is EASM (Explicit Algebraic stress model).

5.3. CFD Open Water Test

This test is carried out to determine thrust coefficient, torque coefficient and the efficiency of the propeller in steady flow condition. The computation is carried out, as due to interactive coupling between the RANS solver and the lifting line method through which body forces accelerating the flow can be determined. The propeller is placed inside a tunnel with a grid (121,51,51).

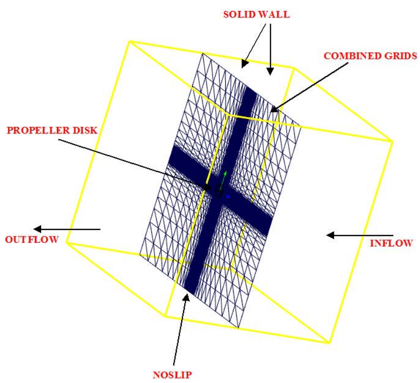

Fig.11. Boundary setup for open water test

Firstly, the computation has been performed for the B Wageningen propeller and the numerical results have been compared to the open-water characteristics for B series propellers given in polynomials.

In Figure 12 it can be seen that the range of values for K_T,K_Q,η_0 obtained through statistical and numerical analysis follows the same pattern and the differences obtained are in the range of 2-3%.

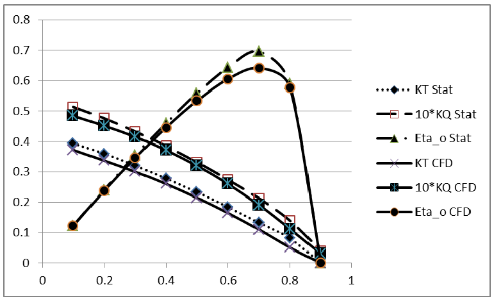

Fig.12. Comparison K_T,KQ,η_0 for numerical and statistical analysis of B propeller

The same CFD method has been applied to find the hydrodynamic performances in open water for the designed propeller. The results are plotted in Figure 13.

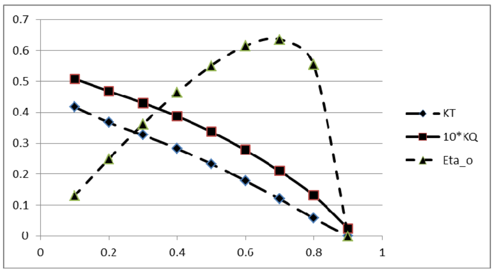

Fig.13. Open water characteristics K_T, KQ, η0 for numerical analysis of wake adapted propeller

5.4. Self Propulsion CFD Investigation

The self propulsion simulations were carried out to study the performance of the propeller behind the ship hull using the global approach where RANSE equations are solved over the entire fluid domain.

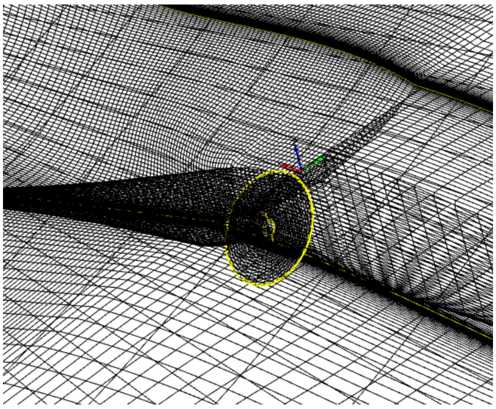

Fig.14. Overlapping grid setup

The pre-processing has been carried out using structured grids for defining the fluid domain and the boundary of the hull. Since in our case we aim to perform self propulsion of the hull with a propeller with a very complex geometry, overlapping grids are used for more complex geometries. XCHAP has an automatic algorithm for the cell classification. All cells in the overlapping grid need to be classified as fluid or outside. The hull grid and the fluid domain were dicretized, which consisted of 2.43 million cells. Including the additional overlapping grid components, the propeller grid and a local refinement grid in the stern, the total number of cells was 2.62 million.

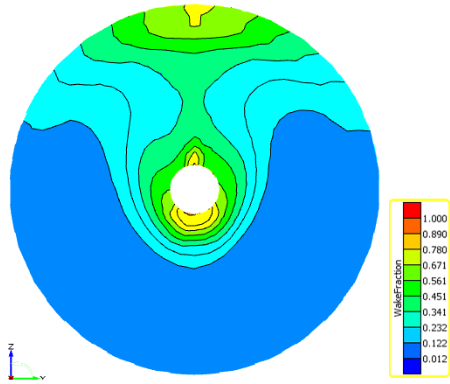

Fig.15. Effective wake distribution in propeller plane

Table7. Results of Numerical Self-Propulsion

Effective Mean Wake, w	0.277
Thrust Deduction Factor, t	0.184
Advance Coefficient, J	0.60
Thrust Coefficient, K_T	0.178
Torque Coefficient, K_Q	0.024
Propeller Speed (RPM)	141
Delivered Power (KW)	8234
Hull efficiency, η_H	1.183
Relative rotative eff, η_R	1.018
Propeller Efficiency, η_D	0.71

The rpm and delivered power obtained were 1% and 20 % lower in the computations compared to the initial estimation. Comparing thrust and torque coefficients determined during the detailed design stage, numerical analyses were different by 6.32% and

10.37% respectively. Regarding hull propeller interaction factors, the wake obtained from numerical analysis is in very good agreement with preliminary data while thrust deduction factor deviation is about 7.7 % from the initial values.

6. CONCLUDING REMARKS

The paper presents aspects regarding the design of a wake-adapted propeller for a 1200 TEU containership. One of the main objectives was to design an optimised propeller and to verify that it is the best available solution. In-house codes based on B Wageningen series and lifting line theory have been developed and used.

Once the propeller geometry has been defined, numerical investigations have been performed to analyse the hydrodynamic performances of the designed propeller, in open water and behind the port containership hull.

CFD methods have been used to predict ship hull resistance, propulsion coefficients and to predict the propeller hull interaction.

From the self propulsion simulation results, it can be concluded that numerical analysis using shipflow software is an efficient method to predict the propeller hull interactions i.e. wake and thrust deduction factors similar to the experimental self-propulsion test.

Moreover, CFD analysis has been used to study the hydrodynamic performance of the propeller, to understand local flow details including both steady and unsteady conditions and to select the most promising candidate design for model testing.

ACKNOWLEDGMENTS

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 and ICEPRONAV ENGINEERING Galati.

REFERENCES

[1]. **Bertram,V.,** *"Practical Ship Hydrodynamics"*, Butterworth & Heinemann, Oxford, 2000.

[2]. **Oosterveld, M.,W.,C., Oossanen,P.,** *"Further Computer-Analyzed Data of Wageningen B Screw Series",* 1975.

[3]. **Carlton,** *"Marine propeller and propulsion"*, Butterworth & Heinemann, London, 2007.

[4]. **Tocu, A.,M., Amoraritei, M.,** *Numerical study of the flow field around a ship hull including propeller effects."* , JNMR, 2008.

[5]. **Broberg, L.,Orych, M.** *"An efficient numerical technique to simulate the propeller hull Interaction."*, International Journal of Innovative Research & Development, 14, 2012.

[6]. *** Flowtech, Xchap theoretical manual

[7]. *** Flowtech, user manual

[8]. *** ITTC procedures, Performance Prediction Method, 1978.

[9]. *** DNV GL Rules and regulations for the classification of ships, Part 5, Chapter 6, 2015.

Paper received on December 15th, 2015