EEXI Technical File

for an Exemplary Container Vessel

Reference Number: 00000xxx

Draft Report

August 3, 2021

Contents

1. Preface

This documents provides all relevant information to hand in for the *Energy Efficiency Existing Ship Index* which will be referred to as the *EEXI* on the following pages. It gives an overview about the vessel's relevant information concerning the EEXI certification process.

For the calculation of the ship's reference speed V_{ref}, CFD simulations were employed in order to provide a power prognosis in the EEXI-relevant floating condition. In order to verify the calculated power level, simulations have also been conducted with the original configuration at design draft, which have been compared with model tank tests from the newbuilding phase conducted by *a model basin*. The predictions are carried out based on the calculated open water characteristics of model propeller recalculated for full scale and the resistance and propulsion conditions.

The following changes were conducted regarding the vessel's hull, propeller and main engine:

- An engine power limitation (EPL) has been conducted to reduce the main engine's maximum continuous rating *MCR* from 51394 kW to 24000 kW.
- The vessel is fitted with a *MMG ESPRO* retrofit propeller, which leads to a reduction of power demand of abt. 10% compared to the original propeller.
- The vessel is fitted with a *Retrofit Bow*, which leads to an additional power reduction of up to 23% depending on the operating speed compared to the original bow shape.

2. Project Specification

2.1. Vessel Information

Ship Data	
Length between perpendiculars L_{pp}	$xxx.x\ m$
Breadth B	$xx.x\ m$
Height H	$xx.x\ m$
Design Draught at FP/AP T_D	$xx.x/xx.x\ m$
Scantling Draught at FP/AP T_S	$xx.x/xx.x\ m$
EEXI Draught at FP/AP T_{EEXI}	$xx.x/xx.x\ m$
Displacement (at Design Draught) ∇_D	$xxxxx\ m^3$
Block Coefficient (at Design Draught) c_B	$x.xxxx$
Shaft line efficiency η_M	0.99
Shaft height above baseline	$x.xx\ m$
Classification Notation	–

2.2. Main Engine Information

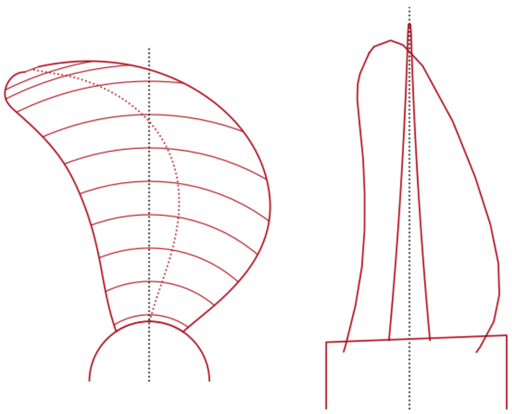
Original Engine Data	
Engine	1 x Wärtsila 9RTA96C-B
MCR P_B / N	51394 kW @ 102.0 min^{-1}
Derated Engine Data	
MCR P_B / N	24000 kW @ 79.1 min^{-1}

2.3. Auxiliary Engine Information

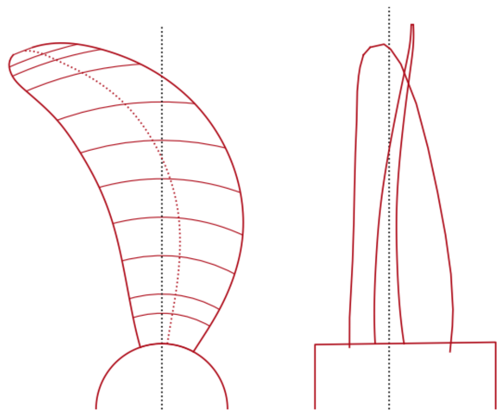
Engine Data	
Engine	5 x MAN 7L32/40
MCR P_B / N	3206 kW @ 720.0 min^{-1}

2.4. Propeller Information

Propeller Main Dimensions	Original Design	Redesign
Diameter D	$xxxx\ mm$	$xxxx\ mm$
Number of blades z	x	x
Material	Cu 3	Cu 3
Pitch ratio $(P/D)_{hyd}$	$0.xxxx$	$0.xxxx$
Expanded area ratio EAR	$0.xxxx$	$0.xxxx$
Direction of rotation	clockwise (right-handed)	



(a) Original Design



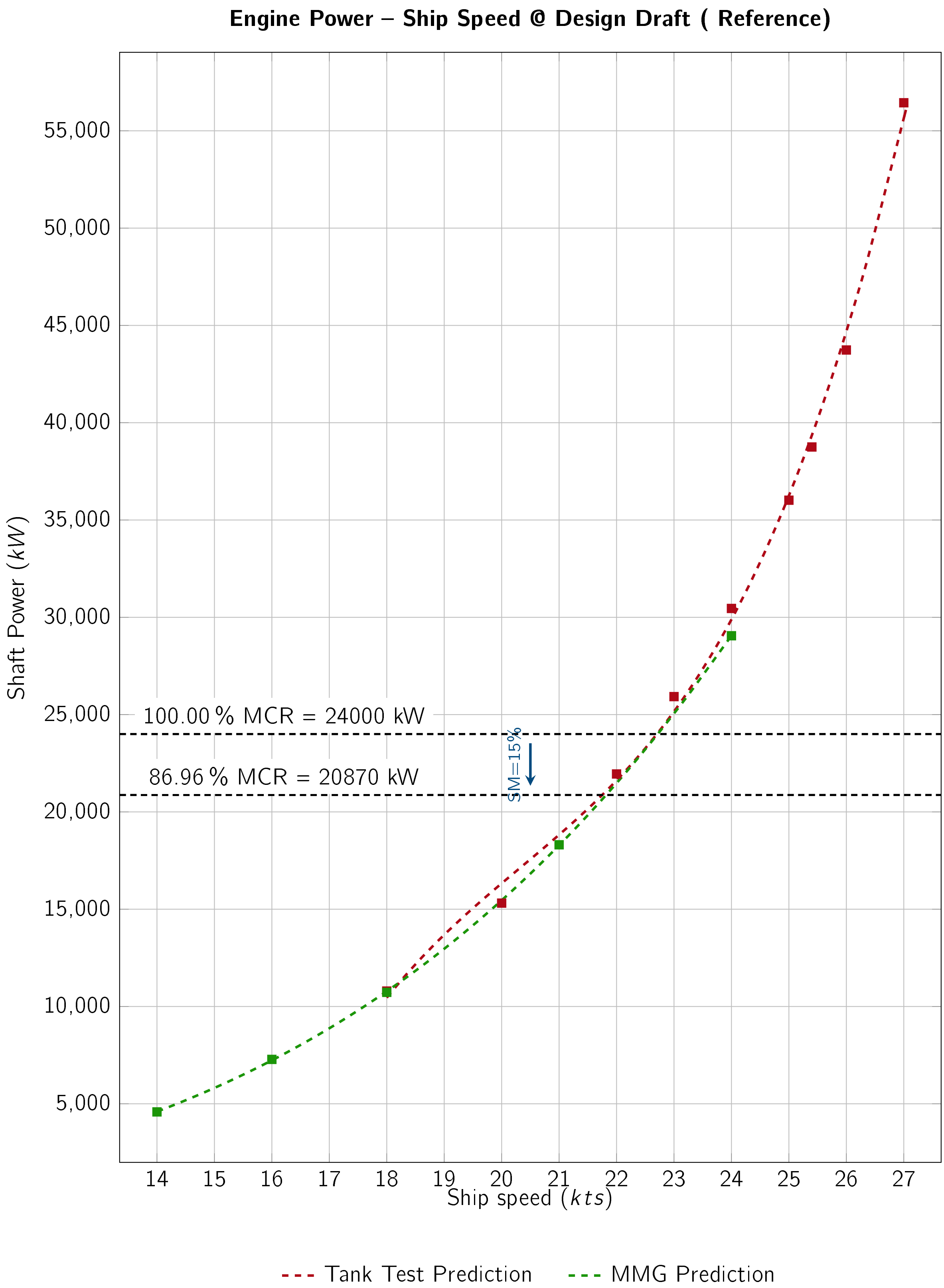
(b) Redesign

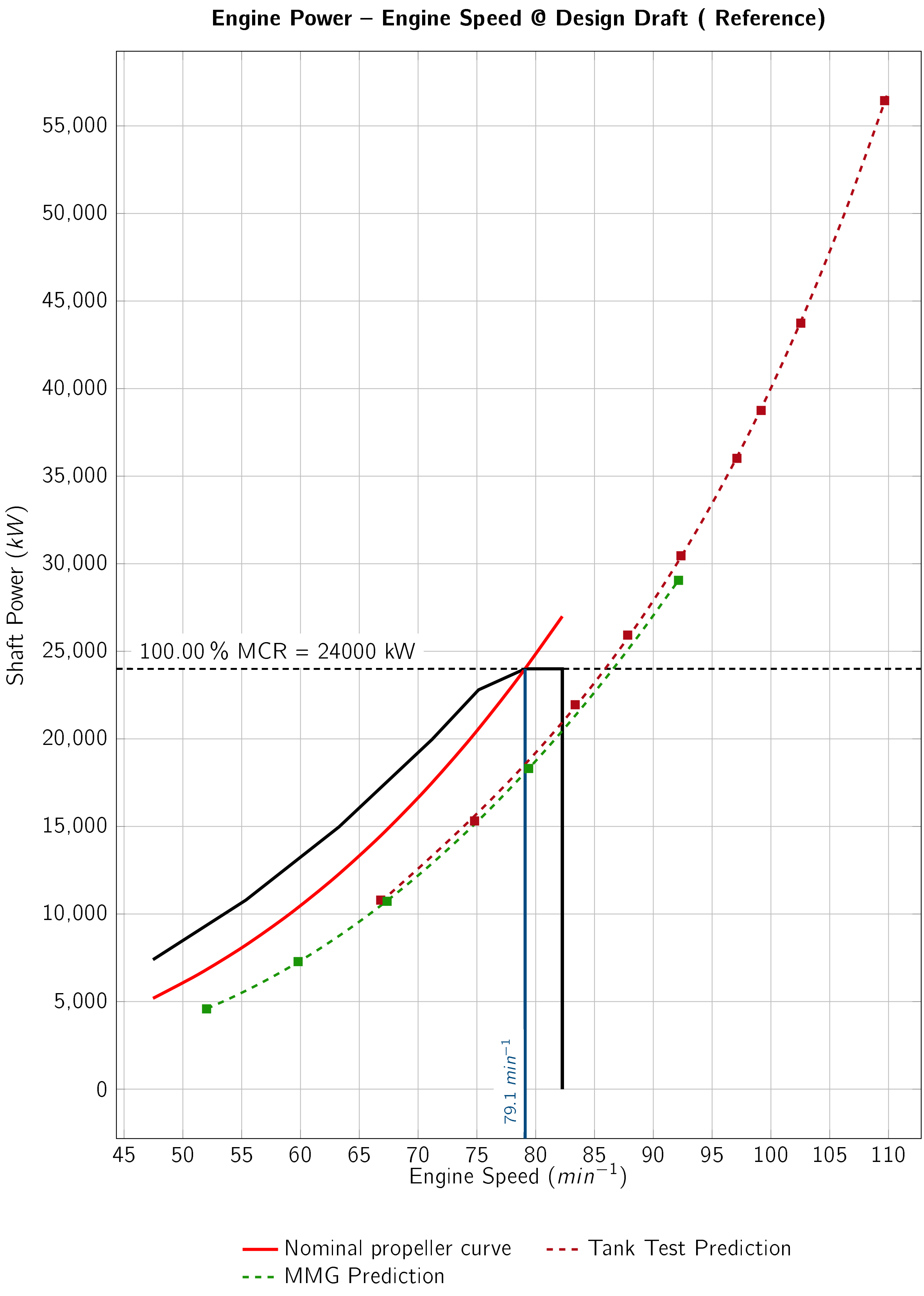
3. Verification of the Reference Power Level

The following table shows the MMG power prediction at Design Draft (12 m) for the original propulsion configuration compared to the prediction which was provided by *a model basin)* during the new building stage of the vessel class. As it can be seen in the following table and also on the diagrams which are presented on the following pages, the results are similar in both, power level and shaft speed.

Powering Prediction

Tank Test Prediction			**MMG Prediction**		
V_S (kts)	N_S (min^{-1})	P_B (kW)	N_S (min^{-1})	P_B (kW)	Deviation (%)
14.00	-	-	52.02	4583.1	-
16.00	-	-	59.80	7282.0	-
18.00	66.82	10793.0	67.36	10726.7	0.61
20.00	74.80	15313.0	-	-	-
21.00	-	-	79.40	18310.0	-
22.00	83.36	21949.0	-	-	-
23.00	87.82	25925.0	-	-	-
24.00	92.35	30455.0	92.14	29052.1	4.61
25.00	97.11	36018.0	-	-	-
25.40	99.17	38753.0	-	-	-
26.00	102.54	43737.0	-	-	-
27.00	109.67	56442.0	-	-	-


Engine Power – Ship Speed @ Design Draft (Reference)
55,000
50,000
45,000
40,000
35,000
30,000
25,000
20,000
15,000
10,000
5,000
Shaft Power (kW)
100.00 % MCR = 24000 kW
86.96 % MCR = 20870 kW
SM=15%
14 15 16 17 18 19 20 21 22 23 24 25 26 27
Ship speed (kts)
Tank Test Prediction
MMG Prediction


Engine Power – Engine Speed @ Design Draft (Reference)
100.00 % MCR = 24000 kW
79.1 min⁻¹
Shaft Power (kW)
Engine Speed (min⁻¹)
Nominal propeller curve
Tank Test Prediction
MMG Prediction

4. Calculation of the attained EEXI

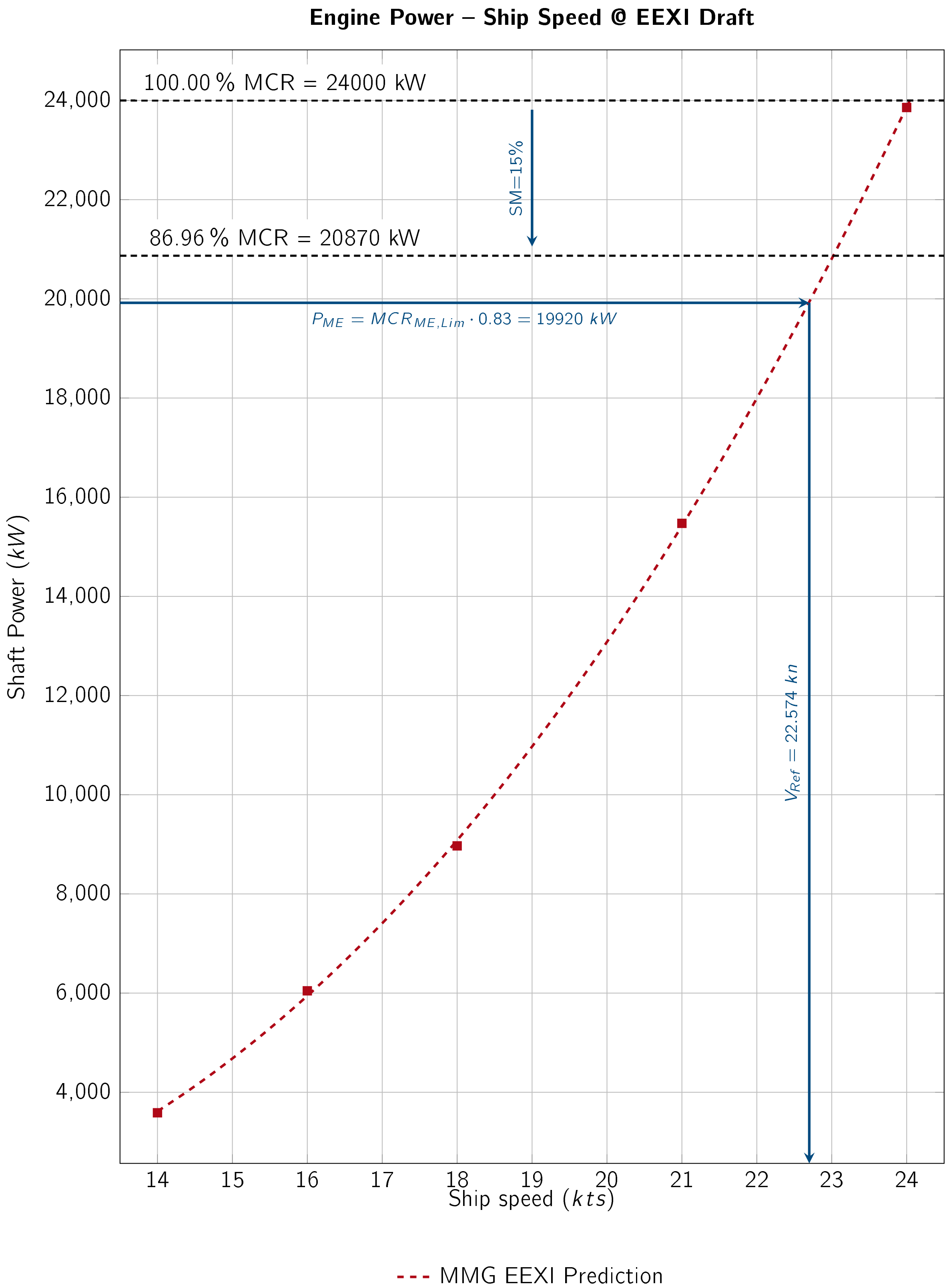
4.1. Overview of Input for EEXI Calculation

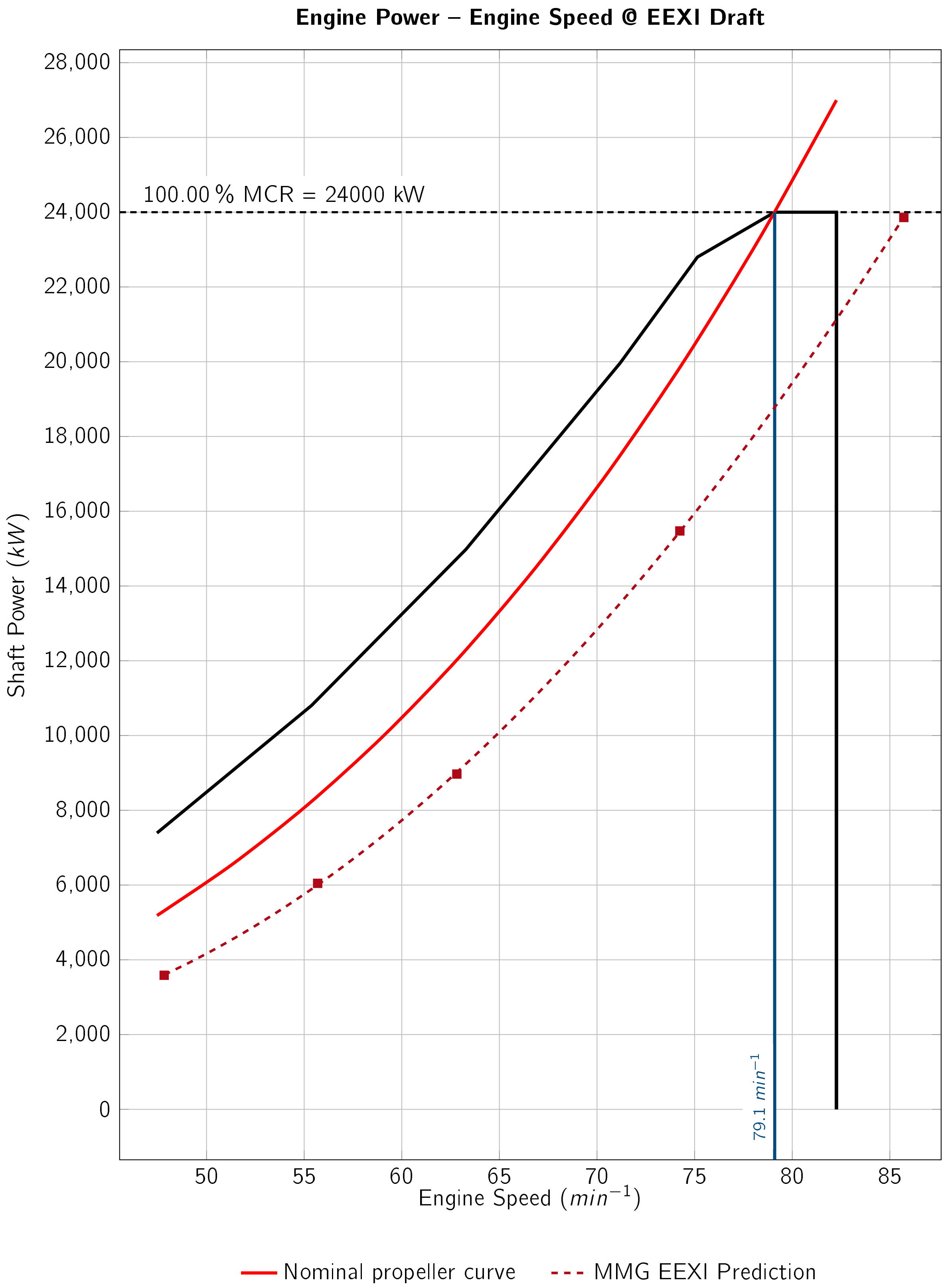
Basic Data	
Type of Ship	Container Vessel
Capacity	37330.16 t
Reference Speed V_{ref}	22.574 kts
Main Engine	
Maximum Continuous Rating MCR_{ME}	51394.0 kW
Limited Maximum Continuous Rating $MCR_{ME,Lim}$	24000.0 kW
Reference Main Engine Power P_{ME}	19920.0 kW
Type of Fuel	Diesel Oil / HFO
CO2 Conversion Factor C_{FME}	$3.114 \frac{t\ CO2}{t\ Fuel}$
Specific Fuel Oil Consumption SFC_{ME}	$171\ \frac{g}{kWh}$
Auxiliary Engines	
Reference Auxiliary Engine Power P_{AE}	850.0 kW
Type of Fuel	Diesel Oil / HFO
CO2 Conversion Factor C_{FAE}	$3.114 \frac{t\ CO2}{t\ Fuel}$
Specific Fuel Oil Consumption SFC_{AE}	$200\ \frac{g}{kWh}$

5. Determination of EEXI Reference Speed

The reference speed is determined by conducting numerical flow simulations of the same kind as for design draught. The EEXI relevant draught is calculated to be $xx.x\ m$ on even keel, which corresponds to a capacity of 37330.16 t. This value is 70 % of the maximum capacity of this vessel class, which is 53328.8 t.

Powering Prediction		
V_S (kts)	N_S (min^{-1})	P_B (kW)
14.00	47.83	3586.6
16.00	55.69	6045.8
18.00	62.82	8968.9
21.00	74.24	15472.8
24.00	85.71	23858.6


Engine Power – Ship Speed @ EEXI Draft
100.00 % MCR = 24000 kW
86.96 % MCR = 20870 kW
SM=15%
$P_{ME} = MCR_{ME,Lim} \cdot 0.83 = 19920\ kW$
$V_{Ref} = 22.574\ kn$
Shaft Power (kW)
24,000
22,000
20,000
18,000
16,000
14,000
12,000
10,000
8,000
6,000
4,000
14
15
16
17
18
19
20
21
22
23
24
Ship speed (kts)
MMG EEXI Prediction


Engine Power – Engine Speed @ EEXI Draft
100.00 % MCR = 24000 kW
Shaft Power (kW)
28,000
26,000
24,000
22,000
20,000
18,000
16,000
14,000
12,000
10,000
8,000
6,000
4,000
2,000
0
79.1 min⁻¹
50
55
60
65
70
75
80
85
Engine Speed (min⁻¹)
Nominal propeller curve
MMG EEXI Prediction

5.1. Additional Features to be considered

Feature		relevant variable
Shaft Motor / Power Take In	None	$P_{PTI} = 0$
Innovative mechanical EET for main engine	None	$P_{eff} = 0$
Innovative mechanical EET for auxiliary engine	None	$P_{AEeff} = 0$
Technical/ Regulatory Limitation of Capacity	None	$f_i = 1$
Ice Class	DNV-E	$f_i = 1$
Cubic Capacity Correction	None	$f_c = 1$
Correction for Cranes	None	$f_l = 1$

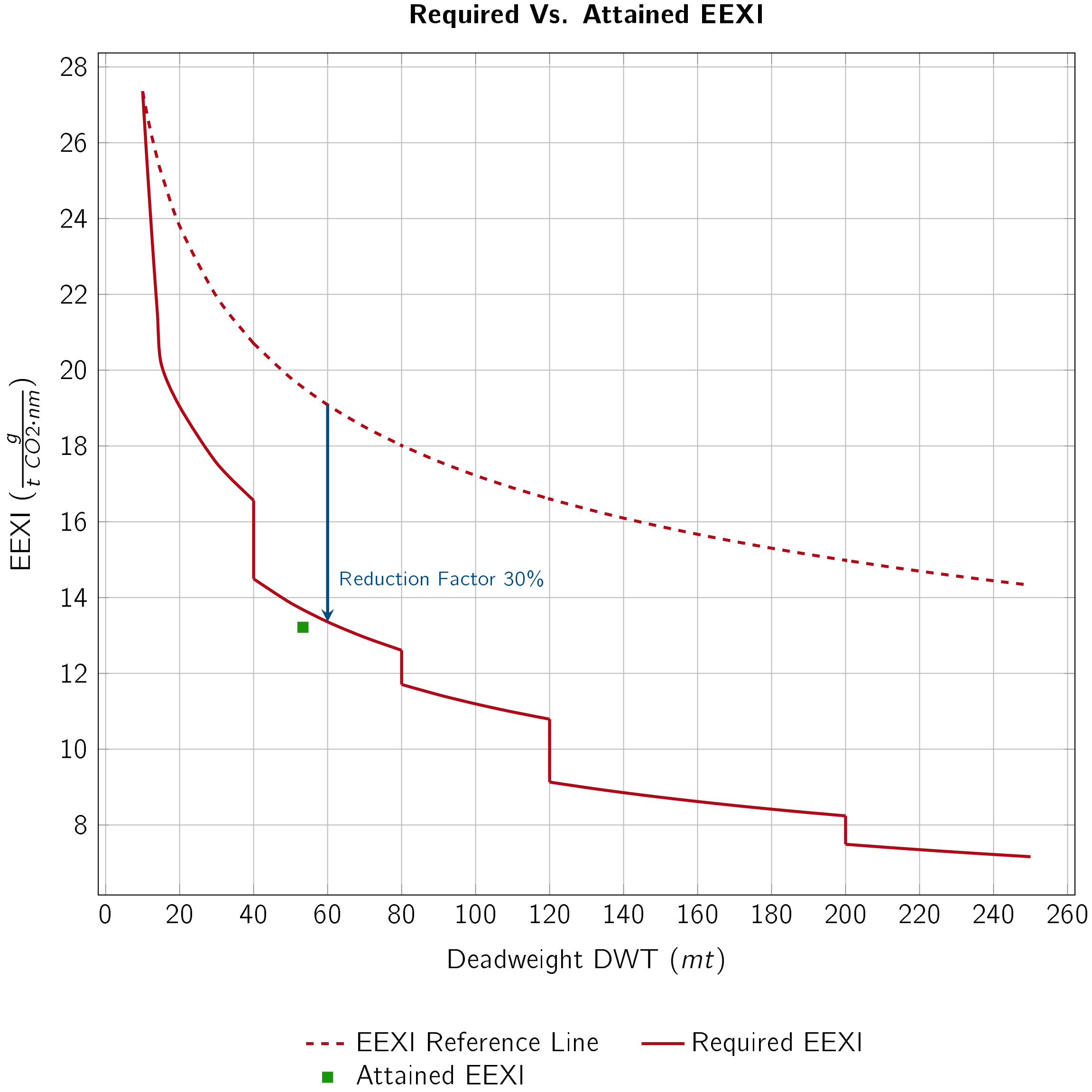
5.2. Calculation of the attained EEXI

$$EEXI = \frac{\left(\prod_{j=1}^{n} f_j\right) \cdot \left(\sum_{i=1}^{nME} P_{ME(i)} \cdot C_{FME(i)} \cdot SFC_{ME(i)}\right) + (P_{AE} \cdot C_{FAE} \cdot SFC_{AE})}{f_i \cdot f_c \cdot f_l \cdot Capacity \cdot f_w \cdot V_{ref} \cdot f_m} \tag{1}$$

$$= \frac{1 \cdot 19920\ kW \cdot 3.114 \frac{t\ CO2}{t\ Fuel} \cdot 171\ \frac{g}{kWh} + 850.0\ kW \cdot 3.114 \frac{t\ CO2}{t\ Fuel} \cdot 200\ \frac{g}{kWh}}{1 \cdot 1 \cdot 1 \cdot 37330.16\ t \cdot 1 \cdot 22.574\ kts \cdot 1} \tag{2}$$

$$= \underline{\underline{13.216\ \frac{g}{t\ CO2 \cdot nm}}} \tag{3}$$

5.3. Comparison with required EEXI



A. Appendix: Performance Analysis of ship powering based on numerical simulations

A.1. Introduction

Under the name *Numerical Propulsions Simulation (NPS) MMG* embraces their simulation based power predition methods. These methods reflect the knowledge base of *MMG* regarding numerical power prognosis. The objective is to deliver optimal propulsion devices to the clients without having to perform model tests. In the meantime the *NPS*-procedures are a well accepted tool which has been used and validated in over 100 redesign and newbuilding projects.

The performance analysis *NPS* is generally conducted at *MMG* using two different approaches:

- prognosis based on model scale simulation
- prognosis on full scale simulation

A brief overview of the procedure for both approaches is shown in Figure 1. The assumptions and theoretical background the procedures are based on are given below.

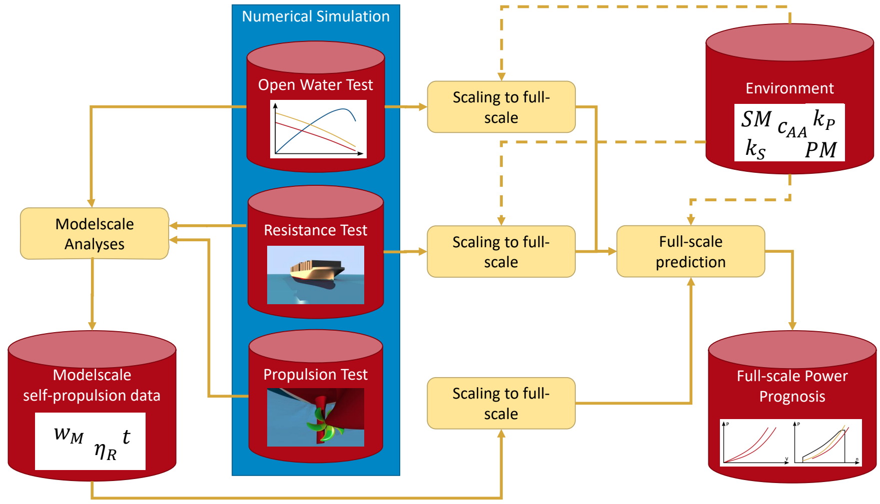
A.2. Simulation Model

MMG's propulsion performance prediction method is entirely based on *Computational Fluid Dynamics (CFD)* which makes it possible to take all relevant flow characteristics into account. For resistance and propulsion simulations, the commercial software package *Numeca FINE/Marine* is used.

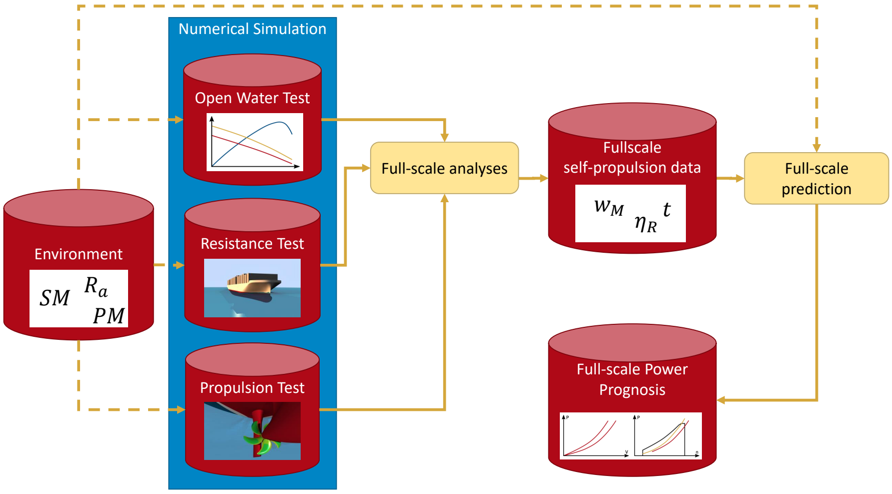
Numeca FINE/Marine uses steady and unsteady *RANS* equations to calculate the turbulent flow around the ship. *MMG's NPS* approach is usually based on unsteady calculations. Turbulence is modelled using Boussinesq hypothesis with Menters two-equation $k\Omega$-SST model. Resistance and propulsion simulations are following a multiphase simulation approach, hence the free surface between air and water is modelled using a Volume-of-Fluid (VOF) code. For details on the governing equations, see [3].

Convective fluxes are obtained using blending schemes, namely `AVLSMART` for momentum and turbulence equations and the `BRICS` scheme for the VOF equations. More information can be found in [3]. Under-relaxation is used to stabilise the simulation.

The simulations are conducted in a way that the domain and all patches are moving through the water like in the towing tank. Hence for all external boundaries, a far field condition is applied. Only the top and bottom boundaries of the fluid domain have a prescribed pressure boundary condition applied to make the equation system solvable. Solid boundaries like the ship, appendages and the propeller patches are defined as no slip conditions with wall functions for solving the viscous sublayer. Solid patches which have no hydrodynamic interaction (e.g. the deck of the ship) are treated as slip walls, so that they contribute less to the forces analysis. For full scale simulations, a sandgrain roughness model is applied on wall patches.



(c) Performance Analyses based on model scale simulation



(d) Performance Analyses based on full scale simulation

Figure 1: General procedures for powering prognosis based on numerical methods

Simulations are performed with multiple domains, e.g. the propeller is modelled in its own cylindrical domain which rotates at a given propeller speed. Non-conformal interfaces ensure that the field variables are transferred from one domain into another. The propeller, appendages and the ship are always treated as rigid bodies, thus deformation of the bodies is permitted. When enabling dynamic sinkage and trim in the simulation, the rigid bodies are transformed in the mesh using a mesh deformation technique.

The time step in the resistance simulation is defined by an experienced based formulae under consideration of characteristic length and speed of the ship. On the contrary, in propulsion simulations the rate of revolution of the propeller is characteristic for the flow's time scale. Thus the time step is calculated to be maximum 5° of propeller rotation per step.

A.3. Meshing

The faired *CAD* models of ship, propeller and appendages have to be transferred into a fluid domain representation. Therefore a CAD domain of the surrounding fluid is built and has to be discretized into finite volumes.

For all *RANSE*-simulations, the *MMG NPS* workflow uses unstructured, hexaeder-dominant meshes, generated with the software *Numeca HEXPRESS*. A two-mesh approach is used for resistance and propulsion simulations. This means that resistance and propulsion simulation use the same mesh for the ship's geometric representation. Only the mesh which represents the geometrical shape of the propeller is changed. Hence the geometric representation of the ship is consistent for all resistance and propulsion simulations in one projekct and there are no mesh dependent deviations to be taken into account when comparing results.

The ship mesh has a cylindrical cavity in the position where the propeller should be placed. The propeller mesh is also cylindrical and fits into the cavity of the ship mesh. To ensure that field quantities are transported properly from one mesh to the other and vice versa, a sliding mesh interface is used. In the resistance simulation, instead of the propeller mesh an empty mesh which contains only a blind cap at the position of the stern tunnel is used.

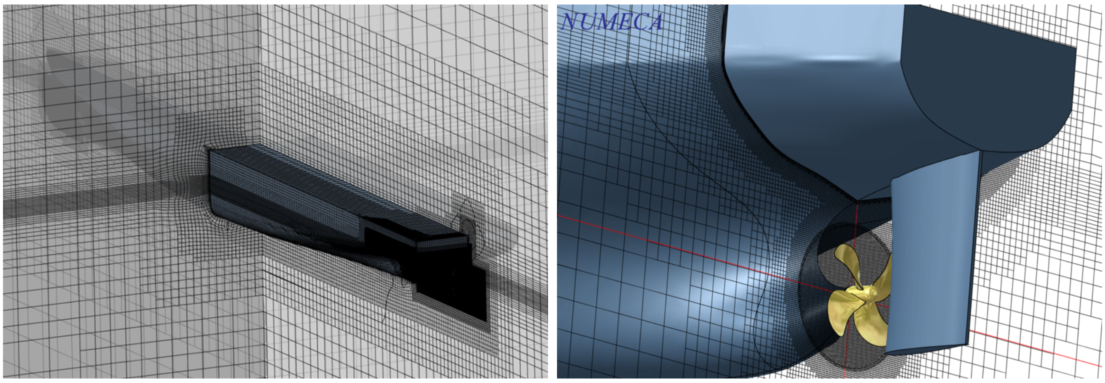
The ship domain mesh is especially refined in the following areas:

- all ship and appendage surfaces
- additional refinement of regions with high pressure gradients (e.g. bow and stern region)
- free surface area, in order to capture waves from the ship' movement correctly
- wake of the ship (inflow to the propeller domain)

Some of these refinements are only applied on the surface, while especially the freewater surface and wake refinement are volumetric refinements of cells. In order to capture the viscous forces correctly, boundary layers are applied to all underwater wall surfaces. Due to the usage of wall functions, a dimensionless wall distance of $y^+ = 30 - 100$ in average is applied to the surfaces. The prismatic boundary layer cells merge smoothly into the volume mesh. All geometric representations of bodies in the fluid domain are checked for consistency, manifoldness and knuckles. Further representation of known edges is checked (e.g. transoms, shaft brackets, rudder edges).

The propeller domain is a cylindrical mesh consisting of the propeller blades, the hub, cap and spacers. All wall patches are refined to soundly resolve the flow, the propeller blades are treated along leading edge, trailing edge and tip in order to correctly dissolve the pressure distribution over the blade. As wall functions are applied to the wall patches, the boundary layer is resolved with a $y^+ = 30 - 100$. Again attention is paid to the prismatic boundary layer so that it merges smoothly with the surrounding volume mesh. It is ensured that the geometry of the propeller is well represented, especially trailing/leading edges and propeller tips. The overall propeller grid cell size is finally adjusted so that it fits well with the cell size of the ship domain in the wake region. Thus discontinuous pressure distribution on the interface between ship and propeller mesh is omitted. The same meshing strategy holds for the blind hub cylindrical mesh which is used in resistance simulation instead of the propeller mesh.

Overall mesh size and refinement of the grid is continuously monitored by *MMG* with respect to the convergence of the results and grid dependent errors. The *MMG* meshing procedures are regularly evaluated and enhanced employing mesh dependency studies. An example mesh for *MMG NPS* is shown in Figure 2.



(a) Resistance simulation (b) Propulsion simulation

Figure 2: CFD meshes for performance analysis of ships

A.4. Control of Simulation

In the *MMG NPS* process all residuals and forces on the patches are monitored continuously during the simulation. There are several different convergence criteria which are checked before extracting the results:

1. flow has to be fully developed
2. residuals of the solving are below the defined criteria
3. forces and motions converged over time to a certain level

The first criterion is applied by visual inspection of the flow in at least two time steps. It is checked if the wave system of the ship is changing over the time and if there are unusual fluctuations or flow phenomena in the velocity and pressure fields, e.g. singularities, separation, cross flow, etc. Special attention is given to the wake of the ship where the propeller is working. The wall shear stresses and the value of dimensional wall distance y^+ is checked in order to not violate the requirements of the wall functions which would result in wrong viscous forces. Furthermore the VOF volume fraction values on the patches are checked to ensure that all underwater parts of hull, propeller and appendages are completely wetted and forces and moments on these patches are correctly computed.

For the second criterion the residuals are evaluated over time. Residuals should decrease to a defined level.

The third criterion should ensure convergence of forces, moments and motions of patches/bodies. This is very important since it ensures that evaluation of the forces and moments acting on propeller, hull and appendages can be correctly analysed. While on ship and appendages the resistance force

is of interest, for the propeller and other rotating devices the thrust and torque are of interest. These forces and moments change due to the inhomogeneous inflow over the degree of rotation. For this reason, the period which is used for averaging of the forces corresponds to an integer number of propeller rotations, e.g. 5.
The force of each body is averaged over a certain time. Each averaged value is compared to the last. If the rate of change of the force is below the *MMG NPS* margin of error, the resistance force is converged in the simulation.

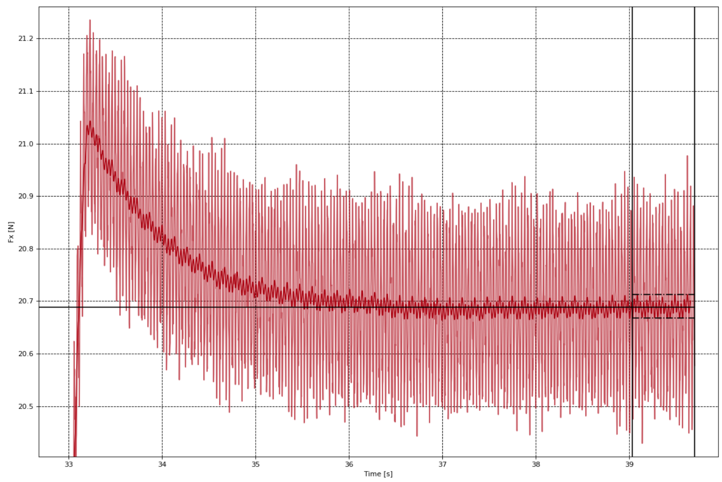


Figure 3: Example evaluation of propeller thrust from CFD: Black line is the averaged force, which is used for powering performance analysis

If all the discussed three criteria are fulfilled, the solution can be seen as approved and will be further processed for verification and validation with model test data and to predict powering performance of the vessel.

A.5. Validation of Performance Analysis

If available, the powering performance of the vessel using the numerical approach is validated by comparison to existing model tests. Thus the simulation will be based on the towing tank test procedure of the model basin which conducted the tests. This ensures that the results for model self-propulsion and consequently the full-scale prognosis can be directly compared, without disparities due to analyses or scaling procedures. The main prerequisites used are noted in the report.

Within the last centuries *MMG* designed propellers for a wide range of customers. Therefore a lot of knowledge and experience in testing and scaling procedures from all relevant model test basins around the world was gained. If no experimental data for validation is present, the performance analysis will be conducted according to the procedures recommended by the *ITTC* [1]. Details about this procedure are described in the following section.

A.6. Physical Properties of the simulation

If not mentioned differently, physical properties of modelscale simulations are the following:

- Density of water $\rho_W = 999.1026 kg/m^3$
- kinematic Viscosity of water $\nu_W = 1.1386 \cdot 10{-}6 m^2/s$

The physical properties in modelscale correspond to $15°C$ freshwater temperature. The properties for the full scale prognosis and simulations correlate with salt water properties for $15°C$ water temperature:

- Density of water $\rho_{W]} = 1026.021 kg/m^3$
- kinematic Viscosity of water $\nu_W = 1.1892 \cdot 10{-}6 m^2/s$

A.7. Open Water Test

Open water curves for the propeller are determined at *MMG* by using different CFD Methods, mainly

- Vortice Lattice Methods (VLM)
- Boundary Element Methods (BEM)
- Finite Volume Methods (FVM)

In the standard procedure, the full scale propeller is simulated and the coefficients of

- thrust coefficient $K_T = \frac{T}{\rho_W \cdot n^2 \cdot D^4}$
- torque coefficient $K_{QO} = \frac{Q}{\rho_W \cdot n^2 \dot{D}^5}$
- and open water effiency $\eta_O = \frac{J}{2\pi} \cdot \frac{K_T}{K_{QO}}$

are determined for the full scale propeller. Results are plotted as a function of the non-dimensional number of advance $J = \frac{v_a}{n \cdot D}$. At least for five different J values are determined which are in the range of the expected propulsion point. Frictional drag on the blade is determined by assuming a roughness of $k_P = 20 \cdot 10^{-6}\, m$. For powering prognosis based on model values, the full scale open water curve is scaled down using a reverse ITTC '78 [1] approach. Though usually the thrust coefficient K_T is not heavily affected by the reverse scaling, the torque coefficient K_{QO} is increased. This results in a reduced overall open water efficiency in the model test η_{O_M}.

When performing model scale simulations the process is applied vice versa. The model open water curves are scaled using the ITTC'78 procedure. In the model simulation the surface of the blade is assumed as beeing technically smooth, e.g. no roughness is applied to the simulation model itself. The propeller rotates within the simulation at very high speed ($n = 18 - 30\, rps$) in order to ensure that local Reynolds number Re_{LCH} at the blade section $r/R = 0.75$ is above $Re_{LCH} > 2 \cdot 10^5$ as recommended by the ITTC.

Multiple scaling procedures for open water tests exist be besides the ITTC recommended procedures. If agreed with the customer, other methodologies can be applied, e.g. HSVA strip method [4] or Lerbs-Meyne [2]. If applied, this is explicitly noted in the report.

In FVM mostly steady and unsteady RANS-Simulations are conducted. Steady state RANSE simulations are performed simulating the single blade and applying cyclic boundary conditions in order to consider interaction of blades. Additional unsteady simulations are performed with a completely meshed propeller model. The propeller mesh is rotating in a cylindrical outer domain by applying *Arbitrary Mesh Interface* (AMI) to the simulation-model. If not other mentioned in the report, open water simulations are conducted in a reverse condition. This means, that the shaft is placed upstream of the propeller and the a cap is arranged downstream, behind the propeller. Free surface effects are neglected in the simulation, thus a single fluid model is applied. RANSE simulations at *MMG* employ the well-known Menter $k\Omega - SST$ turbulence model to recognize viscous effects. On details of meshing the boundary layer, please refer to section A.3. Although it has to be noted, that in propeller open water CFD *MMG* not directly solving the sub-viscous layer, but applying wall functions instead. Therefore a non dimensional wall distance of $30 < y^+ < 100$ is standard.

A.8. Resistance simulation

Resistance of the ship is determined in *MMG NPS* by unsteady RANSE FVM simulations. The surrounding fluid is simulated using a Volume-of-Fluid (VOF) Method. Thus free surface effects like wave resistance are considered in the simulation. Viscous effects along the ship are taken into account by using the $k\omega - SST$ turbulence model. Wall functions are applied to model the viscous sublayer. Therefore a non dimensional wall distance y^+ between 30 and 100 is chosen. Details on the mesh setup for resistance simulations are described in section A.3. The ship can be simulated with all main appendages, like rudders, ducts and ESDs. Bilge keels are not modelled. Due to the moderate speed regime usually dynamic trim and sinkage has a negligable effect on the total resistance. Therefore the ship resistance simulation is conducted at given static draught conditions by standard. Nevertheless, in the *NPS* procedures it is also possible the allow dynamic trim and sinkage of the ship. Especially for very fast ships with $Fr > 0.25$ this might be necessary. It is noted in each report whether simulations are performed with or without dynamic trim and sinkage.

Model simulations are performed under Froude similarity. As Reynolds similarity can therefore not be met, the scale of the model is chosen so that the Reynolds number is larger than $Re > 2 \cdot 10^6$. The model is simulated with technically smooth surfaces, hence no roughness model is applied to hull and appendages. Model resistance is determined for five different ship speeds at least. The scaling of model values to full scale is performed in accordance with ITTC'78[1]. Therefore the total resistance of the model R_{TM} is determined as sum of forces acting on the hull in the ship's moving direction from the CFD simulation.

Using the model scale ship parameters, the following resistance coefficients are calculated:

- Total resistance coefficient: $c_{TM} = \frac{R_{TM}}{0.5 \cdot \rho v_M^2 \cdot S_M}$,
- Frictional resistance coefficient: $c_{FM} = \frac{0.075}{(\log 10(Re) - 2)^2}$,

- Residual resistance coefficient: $c_R = c_T - (1+k) \cdot c_F$,

where S_M is the wetted surface of the hull, v_M the model speed, the ship based Reynolds number is Re and k is the form factor. Mostly the same form factor k as given by model test reports is used. Thus in most cases $k = 0$ as form factor k is used normally for very plump ship hull forms only. If not given by model tests, *MMG* can derive the form factor independently by analysis of pressure drag from double body resistance simulations in combination with resistance simulation with free water surface.

By taking into account Reynolds number dependant effects, the frictional resistance coefficient has to be corrected for the full scale prognosis, while the froude dependent components of wave resistance are kept constant. Hence the full scale resistance can be determined by:

$$c_{TS} = (1+k) \cdot c_{FS} + c_R + \Delta C_F + c_{AA}$$

The frictional resistance coefficient c_{FS} of the full scale ship is determined using ITTC'57 frictional regression line. The roughness allowance ΔC_F takes into account roughness of the hull in full scale and is determined using the following formulae:

$$\Delta C_F = \left[105 \left(\frac{k_S}{L_{WL}}\right)^{1/3} - 0.64\right] \cdot 10^{-3}$$

The air resistance coefficient can be calculated:

$$C_{AA} = 0.001 \cdot \frac{A_T}{S}$$

where A_T is the lateral wind area of the ship. There are many other ways to take into accont roughness allowance and air resistance coefficient, so if other methods than the described method are used to calculate these values, it is noted in the report.

If bilge keels are mounted to the full scale ship, they are considered in the resistance prognosis by increasing frictional components due to relative change in wetted surface:

$$c_{TS} = \frac{S + S_{BK}}{S} \left[(1+k)c_{FS} + \Delta c_F\right] + c_R + c_{AA}$$

Hence the total calm water resistance of the ship is determined by

$$R_{TS} = c_{TS} \cdot \frac{\rho}{2} v_{FS}^2 S$$

A.9. Propulsion Simulation

Propulsion simulations within the *MMGs NPS* procedure employ the very popular load variation method in order to determine the propulsion point. In the standard procedure for model and full scale condition usually one simulation is performed with underloaded and one with overloaded propeller. Hence the propulsion point can be found by linear interpolation between the results. The propeller speed for both runs is calculated by decreasing/increasing the propeller speed by around 10% from an estimated propeller shaft speed hich is taken from the preliminary power prognosis based on i.e. stock propeller model tests.

The simulations are performed under Froude similarity and hence Reynolds similarity cannot be met. Therefore, the model propeller is working in a slower wake of the ship (compared to full scale operation) and the propeller loading is too high compared to full scale. Hence in the towing tank tests in model basins the ship model is towed with a residual force F_D in order to decrease loading of the propeller due to relatively larger boundary layer. Even though the model cannot be directly towed in the numerical simulations, for the determination of the propulsion point in the simulations F_D has to be known. In the *MMG* procedures the resisidual force is calculated using the ITTC recommendation by

$$F_D = \frac{\rho_M}{2} S_M v_M^2 \left[c_{FM} - (c_{FS} + \Delta c_F)\right]$$

Several model basins use procedures which differ from this formulae to calculate F_D. If model test are available, the calculated F_D from the model test report can also be used for the analysis, because residual force has a huge impact on the results of the performance prediction.

The propulsion point for model scale simulation is found when the thrust of the propeller T_P is equal to the towing force of the ship R_{PM} substracted by the residual force F_D:

$$T_P = R_{PM} - F_D$$

where R_{PM} represents the towing force of the ship model determined in the propulsion simulations.

For the direct full scale prediction method, the procedure of correcting for the influence of the boundary layer thickness is not required. Hence the residual force F_D is not applied and the propulsion point can be directly interpolated for the condition $T_P = R_{PS}$. As the full scale ship surface is not technical smooth, a surface roughness model is applied to the hull and propeller patches in order to assume realistic roughness of all parts in full scale simulation.

Once the propulsion point is found for the model condition, the performance prognosis for the full scale ship can be derived. The *MMG* procedure in the *NPS* is almost similar to the ITTC'78 performance prognosis. The hull efficiency elements for the model scale can be found using results of resistance and open water simulations. With the assumption of thrust identity, the wake fraction w_{tM} for model scale is found by determination of number of advance J_{TM} in the open water curve of the model propeller at K_{TP}. The wake fraction in modelscale can be calculated:

$$w_{TM} = 1 - \frac{J_{TM} \cdot D_M}{v_M}$$

with model propeller diameter D_M and model ship speed $v_M = v_S/\sqrt{\lambda}$. Thrust deduction coefficient is consequently determined:

$$t = \frac{T + F_D - R_{TM}}{T}$$

As in the CFD simulation different tank conditions between resistance and propulsion test do not occur, corrections for different water conditions are obsolete. The relative rotating efficiency η_R is derived by the relation of propeller torque Q in open water and behind ship condition:

$$\eta_R = \frac{K_Q M}{K_{QO}}$$

While thrust deduction t and efficiency η_R do not have to be corrected to full scale, the wake fraction w_t is influenced by Reynolds number depending effects and needs some correction. There exist different scaling procedures for w, mainly derived by model basin experience. If not other mentioned in the report, the *NPS* uses ITTC'78 correction for full scale effective wake fraction:

$$w_{TS} = (t + 0.04) + (w_{TM} - t - 0.04) \cdot \frac{(1+k)c_{FS} + \Delta c_F}{(1+k)c_{FM}}$$

The full scale load of the propeller is then obtained from full scale open water characteristics by intersection of ship and propeller load curve. Ship load curve can be determined by:

$$\frac{K_T}{J^2} = \frac{S}{2D^2} \cdot \frac{c_{TS}}{(1-t)(1-w_{TS})^2}$$

The full scale advance coefficient J_{TS} and torque coefficient K_{QS} can be read at thrust identity point. Hence the full scale performance is

- propeller speed

$$n_s = \frac{(1 - w_{TS}) \cdot v_S}{J_{TS}\dot{D}}$$

- delivered power

$$P_D = 2\pi\rho D^5 n_S^3 \frac{K_{QTS}}{\eta_R}$$

- hull efficiency

$$\eta_H = \frac{1-t}{1-w_{TS}}$$

- total efficiency

$$\eta_D = \eta_O \eta_R \eta_H = \frac{P_E}{P_{DS}}$$

Usually the full scale data is not corrected using C_N, C_P corrections for rate of revolutions and power. Anyhow many performance predictions, especially from model basins, use these empirical corrections. Therefore for comparison with model basin prognosis these corrections have to be sometimes applied, resulting in

$$n_T = c_N \cdot n_s$$

$$P_{DT} = c_P \cdot P_{DS}$$

Once corrections are applied it is noted in the report.

Performing full scale simulations in the *MMG* procedure, no correction of wake fraction w has to be applied. Thus all scaling procedure is neglected and the propulsive factors are directly determined using full scale resistance data and full scale openwater curves.

The performance prognosis for calm water trial conditions using *NPS* can be used to predict severity operating conditions of the ship, e.g. due to wind or heavy weather. In the *MMG* procedures, the performance in this conditions is usually found by increasing resistance of the ship R_{TS}, the wake fraction w_{TS} and the propeller torque coefficient K_{QOS} in order to consider effects like added resistance, fouling of hull and propeller. For special working conditions like pulling, additional *NPS* simulations can be performed, which also consider the effect of operating condition on thrust deduction and relative rotative efficiency.

The delivered standard service prognosis for vessels is based on an assumed additional required power of the engine of SM=15%, called sea margin. Therefore the resistance is iteratively increased until the power demand of the propeller fulfills the sea margin level.

References

[1] ITTC. Performance, Propulsion 1978 ITTC Performance Prediction Method (7.5 – 02 03 – 01.4), 1999.

[2] K. Meyne. Experimentelle und theoretische Betrachtungen zum Maßstabseffekt bei Modellpropeller-Untersuchungen. *Schiffstechnik*, 1968. (in german).

[3] Equipe METHRIC Ecole Centrale Nantes. Theory guide: Fine/Marine 7.2, 2018.

[4] Heinrich Streckwall, Lars Greitsch, and Martin Scharf. An advanced Scaling Procedure for Marine Propellers. In *Proceedings of third International Symposium on Marine Propulsors (smp'13)*, pages 136–142, Launceston, Australia, May 2013.