March 2018

CFD Automated Self Propulsion Test

Gianluca GUSTIN [a,1], Gianpiero LAVINI [a,2] and Francesca MOCNIK [a,3]
[a] *Fincantieri S.P.A. - Direzione Navi Mercantili, Trieste, Italia*

Abstract. CFD simulations are becoming more and more reliable and the increasing computational power are making them a convenient tool to investigate complex phenomena and to reproduce the experimental tests. In the hydrodynamic design process the correct evaluation of the self propulsion performance is an essential item to access the true performances of the ship. This paper deal with a CFD methodology, developed in STAR-CCM+, that reproduces the self propulsion experimental test. The variation of propeller rpm has been automated in order to achieve the self propulsion condition. Ship motions and free surface are take into account to study the behavior of the propeller in the real wake and to compute the propeller efficiency and the absorbed power. A sliding rotating mesh encloses the propeller while the mesh motions, necessary to simulate correctly the ship trim and sinkage, are managed by a STAR-CCM+ proprietary algorithm. The towing force has been take into account in order to reproduce the load variation test. The numerical results are presented in this article and are compared with the experimental results. The good match proved that the numerical self propulsion simulations are getting an extremely useful tool in the hydrodynamic design.

Keywords. CFD, self propulsion, STAR-CCM+.

1. Introduction

Nowadays potential and viscous flow Computational Fluid Dynamics (CFD) codes are extensively used for design purposes, allowing experimental tests to be performed only in the final stages of the project. With reference to marine applications, CFD simulations can be used to predict the flow around hulls, appendages and propellers. The increasing of the computational power allows to investigate complex phenomena or reproduce the experimental tests. In the hydrodynamic design process the correct evaluation of the self propulsion performance is an essential item to access the true performances of the ship. Thus in this work a CFD automated Self Propulsion Test is described and the numerical results, in terms of thrust, torque and predicted rpm, are compared with the experimental results. Ship motions (sink and trim), free surface and rotation of the propeller are take into account in the CFD self propulsion test. Moreover the towing force is introduced to reproduce the self propulsion experimental test.

[1] gianluca.gustin@fincantieri.it
[2] gianpiero.lavini@fincantieri.it
[3] francesca.mocnik@fincantieri.it

2. The self propulsion test

The self propulsion test is an important test to verify that the designed ship (in particular hull and propellers) is able to achieve the contractual speed. In particular the target of the self propulsion test, both experimental and numerical, is to identify the revolution rate of the propeller in order to satisfy the following relations:

$$R_m = T_m + F_D$$

Measuring the torque absorbed by the propeller and its revolution rate is possible to asses the needed power:

$$P_{Dm} = 2\pi n_m Q_m$$

where:

R_m	is the total resistance of the model (including the appendages);
T_m	is the thrust of the propellers in model scale;
F_D	is the towing force;
PD_m	is the power absorbed by the propeller in model scale;
Q_m	is the torque generated by the propeller in model scale;
n_m	is the revolution rate of the propeller in model scale.

The towing force can be defined as follow:

$$F_D = 0.5\rho_m V_m^2 S_m (C_{F_m}\text{-}C_{F_S}\text{-}C_A)$$

where the coefficient of frictional resistance C_F is determined by the ITTC-1957 formula for model to ship correlation:

$$C_F = \frac{0.075}{(\log(Re)\text{-}2)^2)}$$

When the results of the propulsion test are either interpolated for the condition where the towing force exerted on the ship model F is equal to the scale effect on resistance F_D or when F_D is actually applied in the self-propulsion test, the corresponding model condition is called the self-propulsion point of the ship. Then, considering the scale factor λ, thanks to the model law, the following relations can be defined:

$R_S = (R_m + F_D)\lambda^3 \rho_m/\rho_S$	Full scale ship resistance
$P_E = R_S V_S$	Effective power
$P_{D_S} = P_{D_m}\lambda^{3.5}\rho_m/\rho_S$	Power delivered to the propellers
$T_S = T_m \lambda^3 \rho_m/\rho_S$	Thrust provided by the propellers
$n_S = n_m/\sqrt{\lambda}$	Rotation rate of the propellers in full scale
$V_S = V_m\sqrt{\lambda}$	Full scale ship speed

3. CAD model

The investigated geometries are relative to a typical medium size Fincantieri vessel with two conventional fixed pitch propellers, two rudders, two propeller shafts and the relative appendages, as visible in Figure 1. The propellers are rotating inward over the top. Tunnel thrusters are not been considered in this analysis.

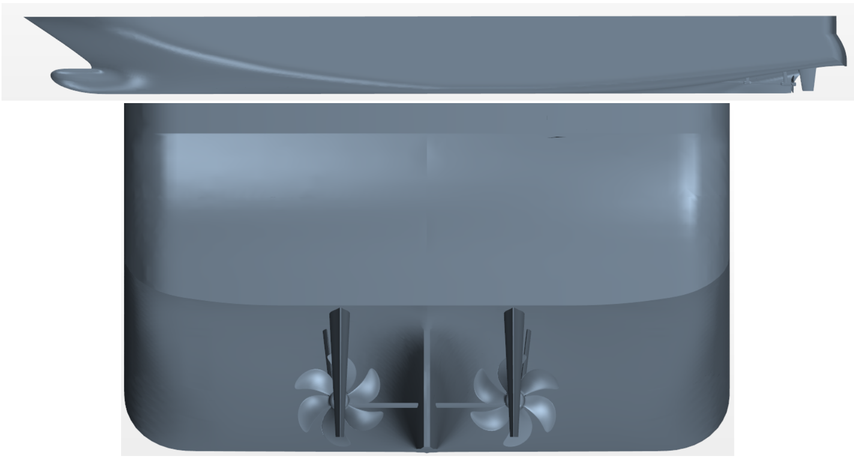

Figure 1. Views of the tested Fincantieri Cruise Ship.

4. Computational domain

To reproduce the towing tank self propulsion test, the simulations have been carried out in model scale. The computational domain is necessarily composed of an *external domain* and a *rotating domain*. The external domain includes all the model, while the rotating domain includes just the propeller.

Considering the length between the perpendicular of the model (L_{pp_m}), the external domain, built as a box, extends $1L_{pp_m}$ ahead from the bow, $2L_{pp_m}$ on the side of the model, $2L_{pp_m}$ behind and below the model and $0.5L_{pp_m}$ above of the model. Because the ship and the flow around it are symmetrical, a symmetry plane has been applied as boundary condition at the diametrical plane. The plane behind the model has been considered as a pressure outlet. All the other surfaces has been considered as velocity inlet.

The rotating domain is composed by a cylinder and it extends 1.2 diameter of the propeller while the thickness is the lowest possible to accommodate the propeller and the hub. All the face of the rotating domain work as interfaces.

The resulting computational domains are shown in Figure 2 and 3.

5. Meshing

In order to achieve a reasonable duration of the simulation, the mesh has to be as light as possible. In fact the time step is strongly related with the size of the

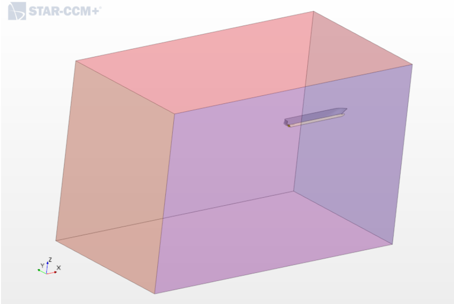

Figure 2. View of the external computanional domain.

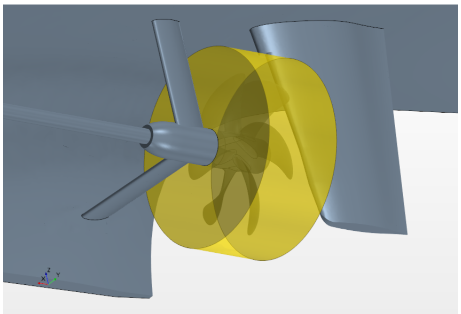

Figure 3. View of the roating domain.

mesh at the interface between the rotating domain and the external domain and also with the rotation rate of the propeller. In fact if we consider a size dx of the mesh at the interface, knowing the speed at the periphery of the rotating domain v, it is possible to compute an acceptable time step dt to allow a stable simulation as follow:

$$dt = \frac{dx}{v} = \frac{20dx}{\pi nd} = O(10^{-4})$$

where d is the diameter of the rotating domain and n is the rotation rate of the propeller, both in model scale.

The physical time resolved during the simulation is at least of $60s$. It is therefore evident that, with a so small time step, a mesh as light as possible becomes mandatory.

The mesh of the external domain consists of 4,574,273 cells and the average y^+ on the hull and on the appendages is 46.857. The set up of this mesh has been based on the Fincantieri experience, because no experimental resistance tests were performed on this hull. On the contrary, the mesh for the rotating domain has been built to achieve the best possible between the numerical and the experimental results of the open water test for this propeller at the advance coefficient J of $0.8 - 0.85$. This approach is not the best from the theoretical point of view, because in the open water test the rotation rate of the propeller is considerably higher than in the open water test to perform the mesh sensitivity analysis. Since the limited computational power available and the lower computational cost of the open water simulations, this approach is the only one possible to perform a mesh sensitivity analysis on the propeller. For the just mentioned considerations, three different meshes with different characteristics of the prism layer has been studied. These meshes are composed by 11 prism layer, for a total thicnkess of 0.00125 m and differ for the near wall cell thickness ($5e^{-6} - 1e^{-5}$). The results of this analysis are shown in Figure 4. The *mesh C*, characterized by a near wall thickness of $1e^{-5}$, has been chosen because it presents the lower errors in the range $J = 0.8 - 0.85$. Therefore the mesh applied at the rotting domain for the self propulsion simulation consist of 732,729 cells and the average y^+ on the blade is 0.759.

Some details of the mesh on the external and rotating domain are shown in Figure 5.

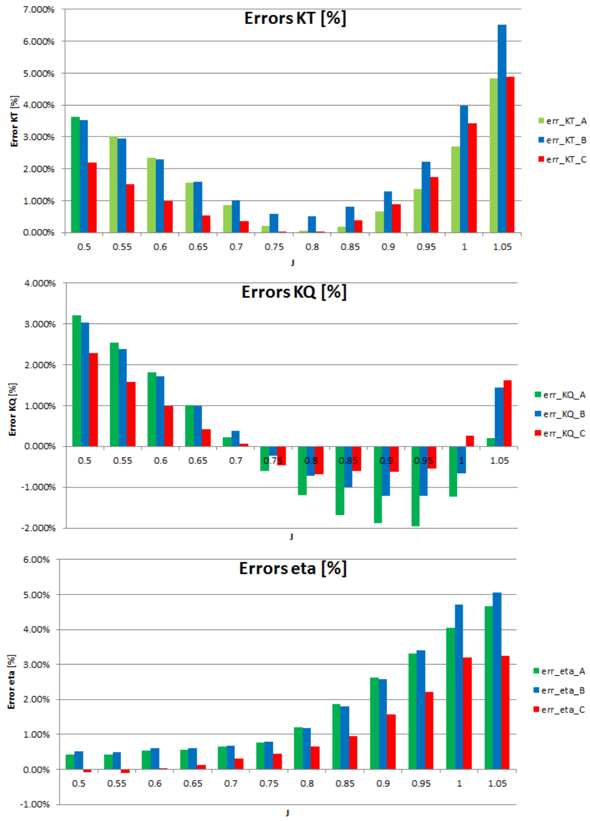

Figure 4. Deviations analysis of the numerical and experimental results of the open water test for the propeller for three different meshes.

5.1. Simulation of the Open water Test

The simulation of the open water test has been carried out in model scale on the complete propeller, placed in a cylindrical domain, using the MRF approach. The rotating speed of the propeller is 900 rpm, as per experimental test procedure used in MARIN. Therefore the variation of the advance coefficient J has been obtained changing the velocity at the inlet. In fact the upstream surface has been setup as a *velocity inlet*, the downstream surface was a *pressure outlet* and the mantle of the cylinder was a free slip wall. The cylinder has $25D$ length and $20D$ as diameter. The hub has been extruded up to the inlet surface and for the major art of it is fixed with a *free slip* condition, therefore its not covered with prism layer.

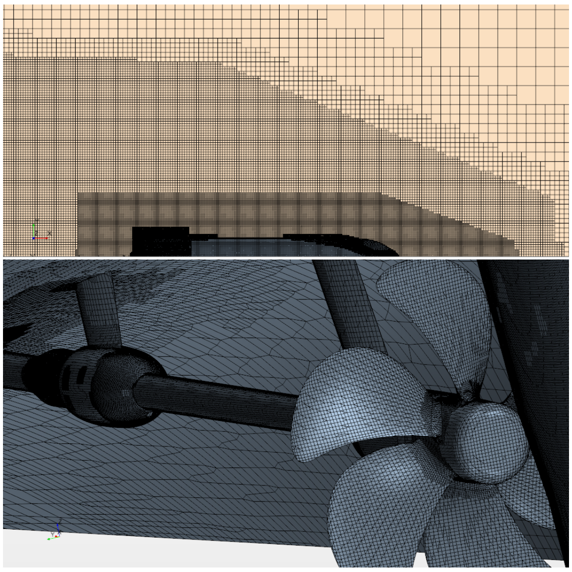

Figure 5. Section of the external domain mesh and view of the computational grid on the hull, appendages, rudder and propeller.

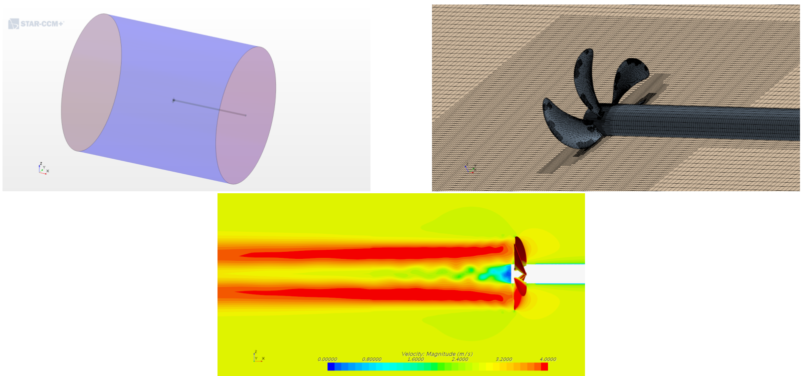

Figure 6. Overview of the computational domain used for the open water test, its mesh and a section of the velocity field.

6. Numerical strategy

The numerical simulations presented in this work were carried out with the STAR-CCM+ 12.02.011 (which will be referred from now as STAR-CCM+) commercial CFD solver. It employs the cell-centered finite volume method to transform the mathematical model into a system of algebraic equations. This transformation involves discretizing the governing equations in space and time. The resulting linear equations are then solved with an algebraic multigrid. In fact to accelerate solver convergence STAR-CCM+ employs the Algebraic Multigrid (AMG) method.

The Volume of Fluid approach has been used to numerical predict the wave patterns created by the ship.

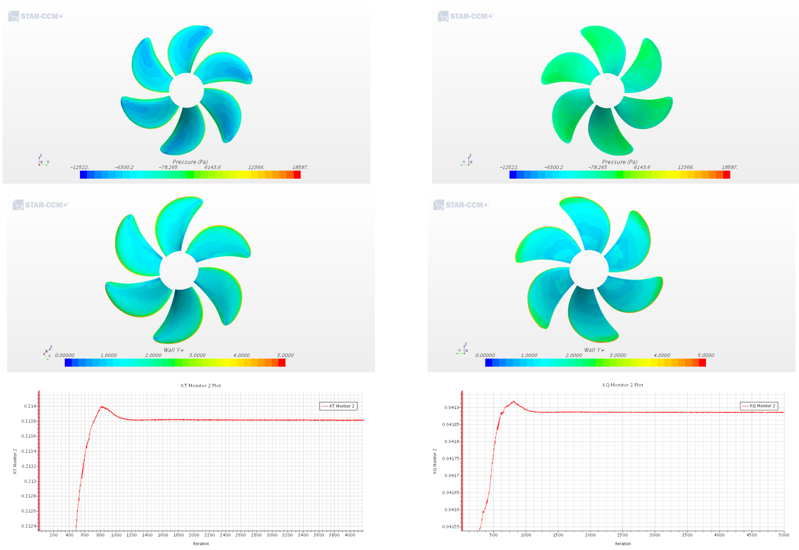

Figure 7. Pressure distribution on the propeller, y^+ contour and plot of KT and KQ for simulation relative to $J = 0.85$

The sliding grid interfaces connect the two domains and they allow the rotation of the domain containing the propeller.

To reproduce the ship motions (sink and trim) and to allow the rotation of the propeller, the STAR-CCM+ Dynamic Fluid Body Interaction (DFBI) and DFBI Superposed rotation proprietary models has been exploited. Thanks to these models is possible to simulate the sinking and trimming of the ship without morphing the mesh. In fact all the domain and its own mesh move in a new position accordingly to the forces acting on the hull. The DFBI Superposed rotation allows the rotation of the propeller following the other ship motions.

For the turbulence modeling the *Realizable* $k-\epsilon$ has been selected because generally it presents the lowest error in the comparison with the experimental results for a typical Fincantieri cruise vessels, as shown in Table 1.

Resistance (hull with appendages)	
Model	**Error [%]**
Realizable $k-\epsilon$	+0.54 %
SST	-1.05 %

Table 1. Typical errors for the resistance simulations.

To initialize the solutions in the rotating domain the Moving Reference Frame (MRF) approach has been chosen. In this model the Navier-Stokes equations are solved in the relative velocity formulation. In fact the initialization of the simulation is composed by different steps:

- The ship remains fixed for 10 s of physical time. In the following 10 s of physical time the model is gradually released.

- For the first 5 s of physical time the MRF is applied at the rotating domain, then the propeller starts to rotate thanks to the Superposed Rotation model.

The Table 2 summarizes the numerical model used in the self propulsion simulation.

Numerics	**Turbulence model**	**Multiphase**	**Ship motions**	**Propeller motion**
Implicit	RANS	Eulerian	MDOF	MRF (init.) +
Segregated	*Realizable* $k-\epsilon$	VOF	DFBI	Superposed
Unsteady	*all* y^+	Gravity	Sink and Trim	Rotation

Table 2. Numerical model used in the self propulsion simulation.

7. Automation

The research of the self propulsion condition is an iterative process, both in the experimental and in the numerical simulation. The recursive process in this analysis is managed with a Java macro that drives the solution to a convergent value of the rotation per minute (rpm) of the propeller in model scale. In particular this process is composed by the following steps:

1. a certain round of the propeller is completed at fixed rpm ;
2. the thrust is averaged on the last couple of round of the propeller;
3. the target revolution rate is computed, if $T + F_D > R$ then the revolution rate of the propeller is decreased, otherwise is increased;
4. the new rpm is achieved over a certain amount of settling round of the propeller;
5. when the thrust is stable enough, the process start all over again.

Has to be noted that in the simulation, the F_D is applied in the center of mass of the vessel, while in the experimental tests it is applied according to ITTC procedure.

The Figure 8 show the variation of the rotation rate of the propeller during the simulation.

8. Results

The experimental test for this model were carried out at MARIN towing tank. The experimental values, used to compare with the CFD results, has been extracted from the load variation test for the same value of the towing force used in the CFD simulation. In fact, due to high computational cost, the CFD self propulsion test has been performed with a towing force computed as previously reported. Anyway, because the F_D is adopted in the simulation, with a larger computational power it is possible to perform a numerical load variation test. In fact the presented CFD method has run for 2 weeks on 180 cores to complete 150 s of physical time. The Table 3 presents the obtained results, compared with the experimental values.

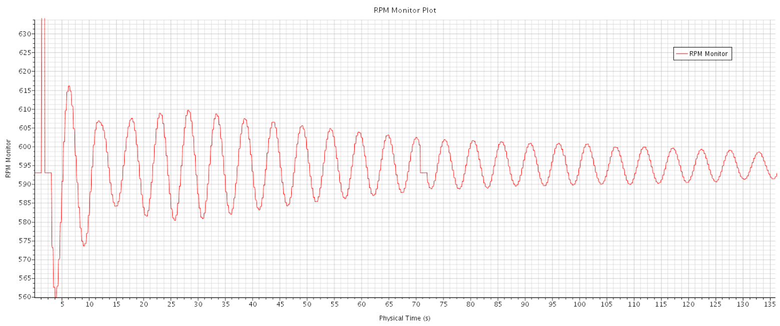

Figure 8. Plot of the variation of the rotation rate of the propeller during the simulation.

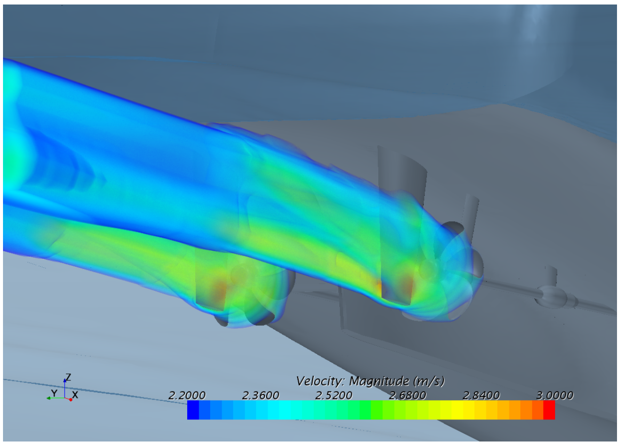

Figure 9. Velocity field in the wake of the propellers.

	Model scale		
ρ $[kg/m^3] = 998.9$	V $[m/s] = 2.084$	$F_D = 35.69$	$\lambda = 22$
	CFD values	**Exp. values**	Δ [%]
Thrust [N]	44.37	43.55	1.85%
Torque [Nm]	1.88	1.93	-2.90%
Rotation Rate [RPM]	593.25	587.24	1.02%

Table 3. Comparison between numerical and experimental results.

9. Conclusions

The presented CFD method reproduces the towing tank self propulsion test and can predict the propulsive performance of the vessel. The results are well comparable with experimental values. Because the towing force is used in the procedure, it is possible to replicate the load variation test as well.

To reproduce the towing tank self propulsion test, the simulations have been carried out in model scale but it still possible to apply the same procedure to a full scale CFD simulation. In this case the towing force will not be take into account and some adjustment to the mesh as an increasing of the computational cost will be expected.

References

[1] *Calm Water Model Test with Designed Propellers.* MARIN, 2014.

[2] *STAR-CCM+, User and Theory Guide.* Siemens, 2017.

[3] *Testing and Extrapolation Methods Propulsion, Propulsor Open Water Test*, ITTC - Recommended Procedures and Guidelines, 2002.

[4] *Practical Guidelines for Ship CFD Applications*, ITTC - Recommended Procedures and Guidelines, 2011.

[5] *Practical Guidelines for Ship Resistance CFD* , ITTC - Recommended Procedures and Guidelines, 2014.