VII International Conference on Computational Methods in Marine Engineering
MARINE 2017
M. Visonneau, P. Queutey and D. Le Touzé (Eds)

Towards CFD guidelines for planing hull simulations based on the Naples Systematic Series

Simone Mancini*, Fabio De Luca* and Anna Ramolini#

* Department of Industrial Engineering
Università degli Studi di Napoli "Federico II"
Via Claudio 21, 80125, Naples, Italy
e-mail: simone.mancini@unina.it, fabio.deluca@unina.it, web page: http://www.dii.unina.it

TU Delft Faculty of Aerospace Engineering
Delft University of Technology
Kluyverweg 1, 2629 HS Delft, Netherlands
e-mail: a.ramolini@student.tudelft.nl, web page: http://www.lr.tudelft.nl/en/

ABSTRACT

Due to their higher motion amplitudes and instabilities, numerical simulations of planing hulls using Computational Fluid Dynamics (CFD) codes are more difficult than that of displacement ships. Indeed, for an accurate evaluation of the hydrodynamic performances of planing craft, the high-fidelity estimation of the pressure field around the hull is crucial. For this reason, validations and comparisons with experimental data are still important to identify the guidelines for both simulation settings and mesh generation. In this paper, two commercial packages will be compared focusing on a resistance case for the parent hull model (C1 hull) from the Naples Systematic Series (NSS) at four Froude numbers (*Fr*).

The NSS is a new systematic series of hard chine hulls intensively tested in planing and semiplaning speed range, De Luca et al. [1]. It has been chosen for the hull form: it is characterized by a warped bottom and a sectional area curve significantly different from the prismatic hulls. These differences amplify the difficulties in finding out the exact pressure distribution on the bottom and, consequently, make the evaluation more stringent.

The Unsteady Reynolds-Averaged Navier-Stokes flow solvers results are validated using these benchmark experimental data. Also, grid independence, iteration, and time-step convergence analysis for response variables (resistance coefficients, wetted surfaces, and dynamic trim angles) follow the recommendations published in the verification and validation (V&V) study from De Luca et al. [2]. Hence, the two software are more compared on different features such as the mesh deformation, the overset method, and the correction of numerical ventilation classically observed below the hull. The results show that both software can provide consistent values and that new guidelines are now identified to improve the reliability of the simulations.

Nomenclature

B_{WL}	Water line breadth (m)	S_W	Wetted surface (m^2)
COG	Center Of Gravity (m)	S	Numerical Simulation result
E	Comparison error	V	Hull speed (m/s)
D	Experimental data	Δ	Displacement (kg)
Fr	Froude number	***NSS***	Naples Systematic Series
L_{WL}	Water line length (m)	***V&V***	Verification and Validation
L_{PP}	Length between perpendicular (m)		
L_K	Wetted keel length (m)		
C_t	Total drag coefficient		
C_f	Frictional drag coefficient		
F_X	Longitudinal drag (N)		
Re	Reynolds number		

1. Introduction

It is known that simulating high speed flows around planing hulls is not a trivial task with the available CFD codes, and therefore several strategies were investigated in De Luca et al. [2]. This publication investigated the analysis of the flow around a planing bare hull model taken from the Naples Systematic Series. The results were compared against experimental data obtained in the towing tank at the Naval Division of the *Dipartimento di Ingegneria Industriale* of the *Università degli Studi di Napoli "Federico II"*. The available experimental results were provided at speeds ranging from 2.0 to 7.5 m/s, with an interval of 0.5 m/s, while the simulations were ran for four different speeds, namely 4.0, 5.0, 6.0 and 7.0 m/s. Two ways of approaching the problem were tested and finally recommended, mainly differing from the meshing strategy point of view, split into two categories:

1. "Single deforming mesh", also called "weighted deformation"
2. "Overset mesh", also called "overlapping grids" or "chimera approach"

From this analysis (De Luca et al. [2]), that tested the overset grid at different levels of coarseness other than the single mesh, it resulted that the coarsest case of the overset grid was the best compromise between accuracy and CPU time. In fact, the single mesh and finer overset yielded small improvements of the results but required around 400 s per timestep, against the 90 s per timestep of the coarsest overset case. These conclusions were obtained using STAR-CCM+ from Siemens.

The idea of this present publication is to try to replicate the conclusions using FINE/Marine from NUMECA International using both the single mesh deformation and the overset method. On top of the single mesh deformation, FINE/Marine also proposes a combination of the deforming mesh with Savitsky prediction. The idea is to be able to mesh the vessel in the final estimated position and speed up the simulation. Hence, this strategy differs with respect to the initial position of the boat, while the overset strategy requires a whole different mesh.

In all cases, the authors tried to replicate as much as possible the meshes used in CD Adapco STAR CCM+, in order to provide a fair comparison since there was no clear way to share the mesh between the two softwares. All approaches yielded satisfactory results, and they will be compared in a more detailed way in the following sections.

2. Test case description

The model used for the experimental testing is the parent hull of the Naples Systematic Series. The name of this model is C1. The other four models of the series were obtained by systematically scaling breadth and depth by the same reduction factors, without changing the transversal shape. The main feature of this series is that its hulls have a warped bottom and their sectional area differs from the one of the classic prismatic hulls. Figures from 1 to 4 show the C1 model in different views. In Table 1 all the relevant information about the model is reported.

Table 1. Model C1 hull data.

Quantity	Value
L_{WL} [m]	2.4
B_{WL} [m]	0.743
D [m]	0.46
Δ[kg]	106.7
S_{WS} [m^2]	1.7
L/B	3.23
x COG from bottom of transom [m]	0.943
z COG from bottom of transom [m]	0.193
Deadrise x/L=0 [deg]	13.2
Deadrise x/L=0.5 [deg]	22.3
Deadrise x/L=0.75 [deg]	38.5

Figure 1. Model C1 Y view.

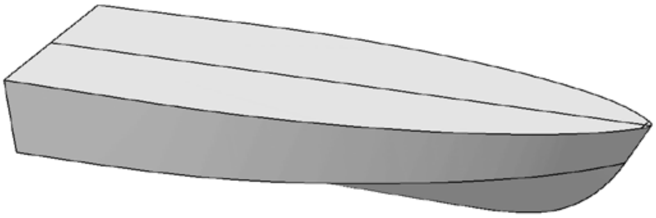

Figure 2. Model C1 3D view.

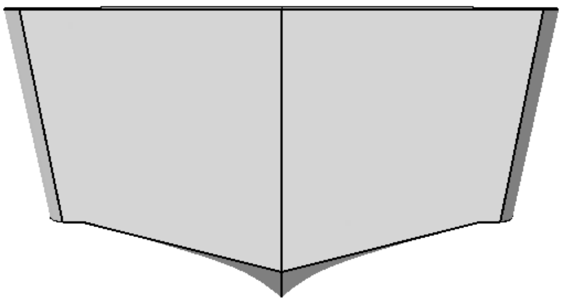

Figure 3. Model C1 X view from transom.

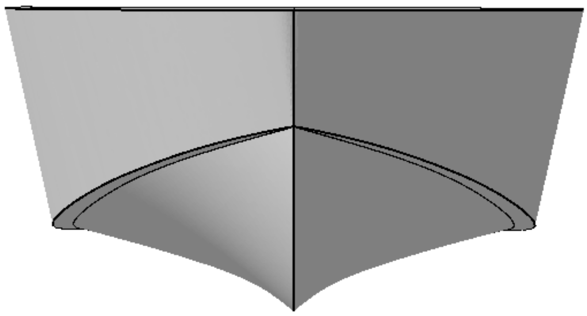

Figure 4. Model C1 X view from bow.

Given the symmetry of the model with respect to the *Y* axis (*y* COG=0) the simulations have been performed on half of the ship in order to decrease the computational time required.

3. Meshing strategy

This section discribes the meshing strategy, created using the unstructured NUMECA's mesh generator HEXPRESS.

3.1. Single mesh

The single deforming mesh is built using one single domain, its size is determined following the recommendations as shown in Figure 5:

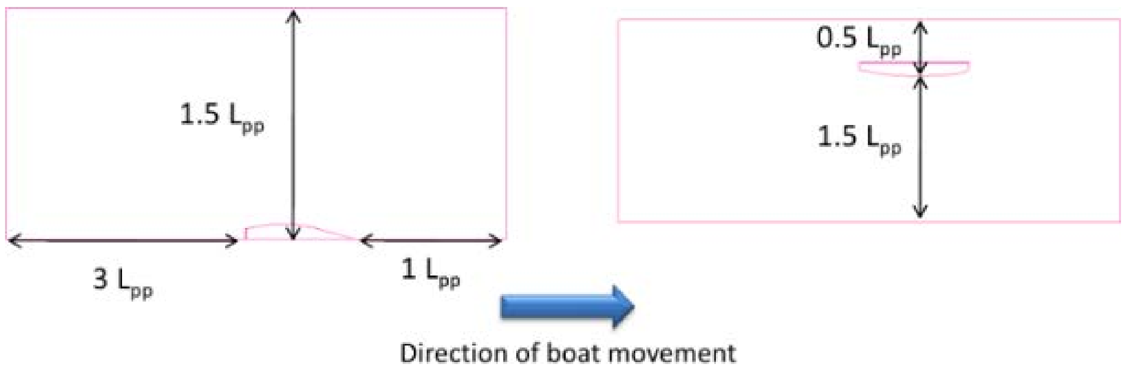

Figure 5. Domain size for single mesh.

Table 2 compares the single mesh base size and total number of cells of the FINE/Marine and STAR CCM+ meshes.

Table 2. Base cell size and total number of cells of the meshes generated with the two softwares

	FINE/Marine	STAR CCM+
Base size [m]	0.133	0.1128
Total number of cells [-]	1 703 724	1 855 777

A first box refinement is defined around the boat, with target cell size 0.039 m in all directions. This box is shown with a dash blue line in Figure 6.

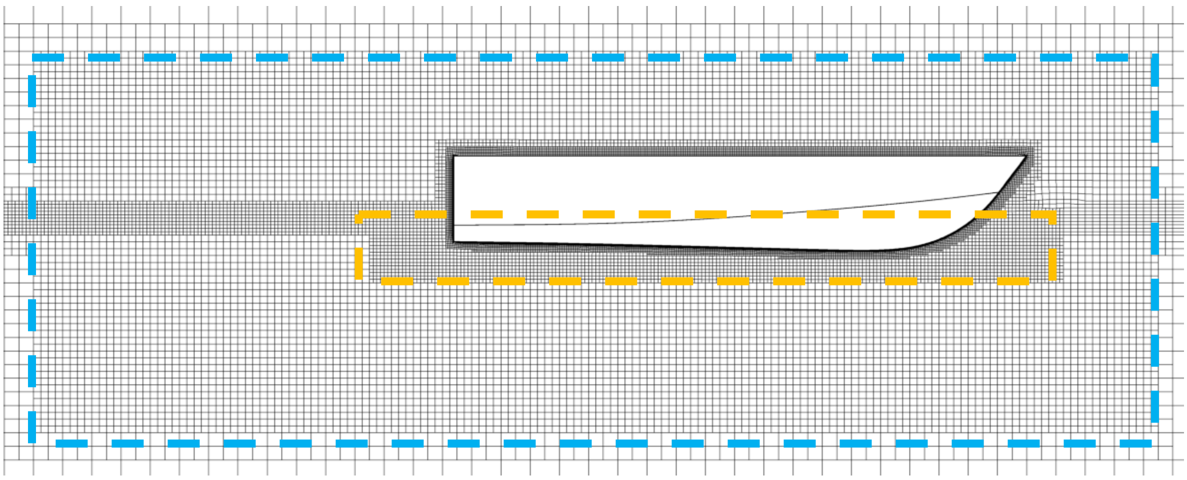

Figure 6. Box refinement around the model.

The undersea refinement can also be seen in Figure 6 with a dash orange line as a smaller box located at the bottom of the hull: this box will have one extra refinement compared to the other box and therefore its target cell size will be 0.0196 m. Thanks to this box it will be easier to capture the interaction between the free surface and the hull.
The free surface will also need to be refined, especially in the *z* direction, to capture the air-water interface. For this purpose an internal surface at the free surface level will be created, and the refinement will be applied only in the z-direction, with target cell size 0.0196 m. On the same internal surface more refinements are needed in the x and y direction. The wake is thenrefined by creating lofted surfaces (Figure 7). The target cell size in *x* and *y* direction will be also 0.0196 m.

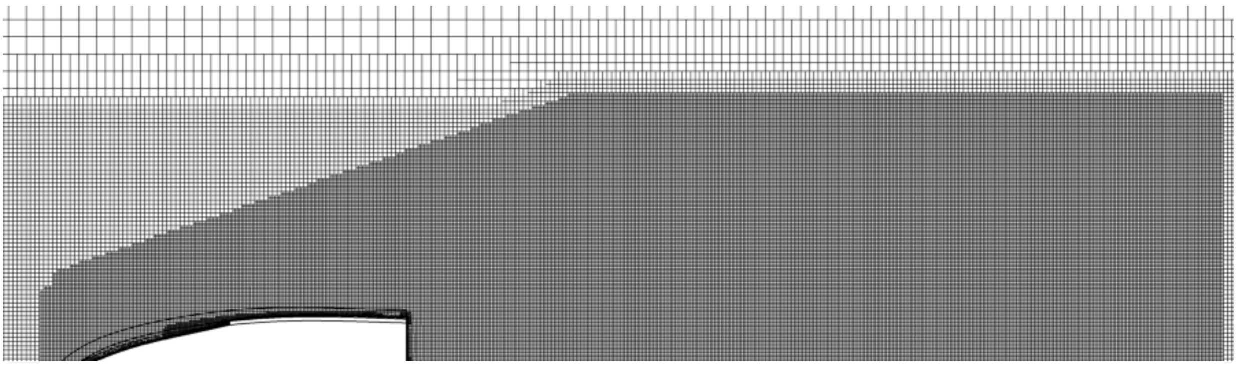

Figure 7. Wake refinement.

For all the solid surfaces a target cell size of 0.0165 m has been chosen, while the curves have one extra level of refinement, with target cell size 0.011 m. This is done to ensure that the shape of the boat is correctly captured even in the most difficult areas, e.g. at the bow where 5 hull lines converge in the same point. The meshed half boat is shown in Figure 8.
Once all refinements are defined, it is necessary to add viscous layers in correspondence of all the solid patches (except for the deck) to accurately resolve the boundary layers. 11 layers will be applied to the hull and the transom.

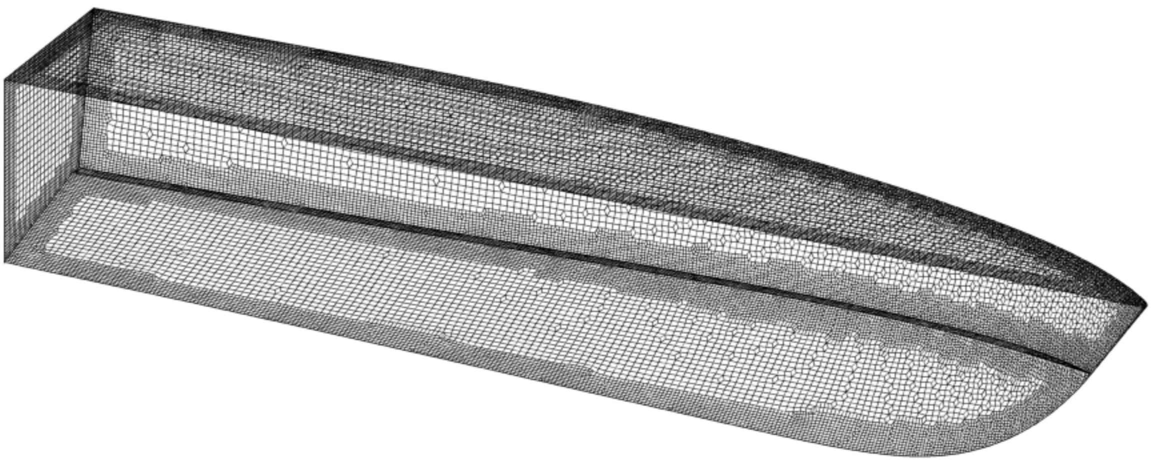

Figure 8. Meshed half boat.

3.2. Overset mesh

The overset mesh creation is a bit more challenging than the one of the single mesh, since it requires the creation of two computational domains: the background and the overlapping domain. The first one consists in a box of large dimensions, while the second one is smaller and contains the boat. The idea underlying the overset technique is that the overlapping mesh rigidly follows all the motions of the ship, while the background domain just follow the forward motion of the boat. The advantage of this technique is that there no mesh deformation ensuring the optimum mesh quality all along the simulation. On the other hand, the drawback lays in the fact that there has to be communication between the two domains, through an interpolation across the domain boundaries of the smaller domain. In order to limit the effect of the interpolation to the minimum the meshes have to be built so that the size of the cells at the boundary of the overlapping domain is as close as possible to the one of the background domain in that area.
First of all, the domain sizes are defined following what is shown in De Luca et al. [2], reported in Figure 10 for clarity. Once the domains are created, the same refinements presented for the single mesh are applied, with the only difference that some refinements will belong to the overset domain and some to the background. The free surface refinement will belong to both: an internal surface needs to be created in both domains. The wake refinement will be present only in the background domain, while the surface, curve and undersea refinements will be added in the overset one. Another refinement is added in the background domain, to comply with the requirement of same cell size in the area around the boundaries of the overset domain. This is done by inserting in the background domain a box refinement around the overset domain with target cell size 0.04 m. Figure 11 shows how the requirement is well met throughout the boundary, having a 1-to-1 ratio between the cell sizes of the two domains across the interface.

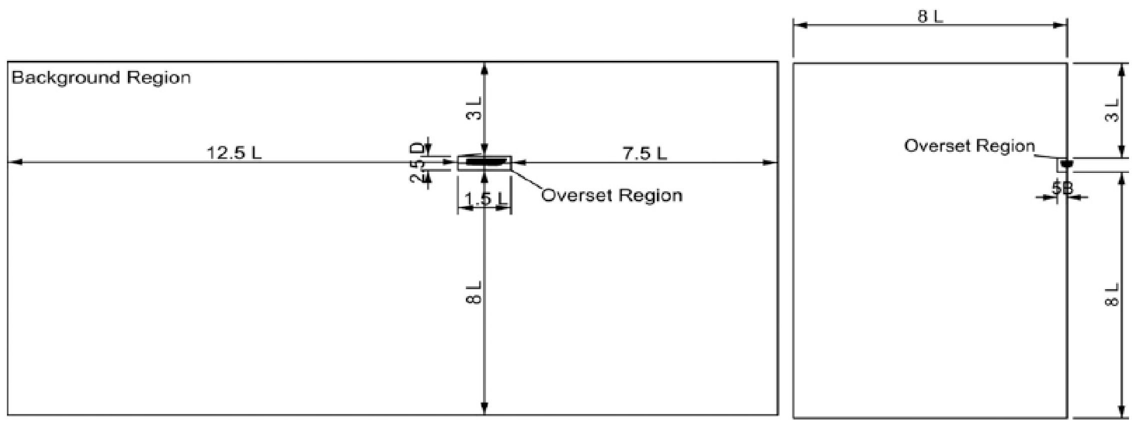

Figure 10: Background and overset domain sizes.

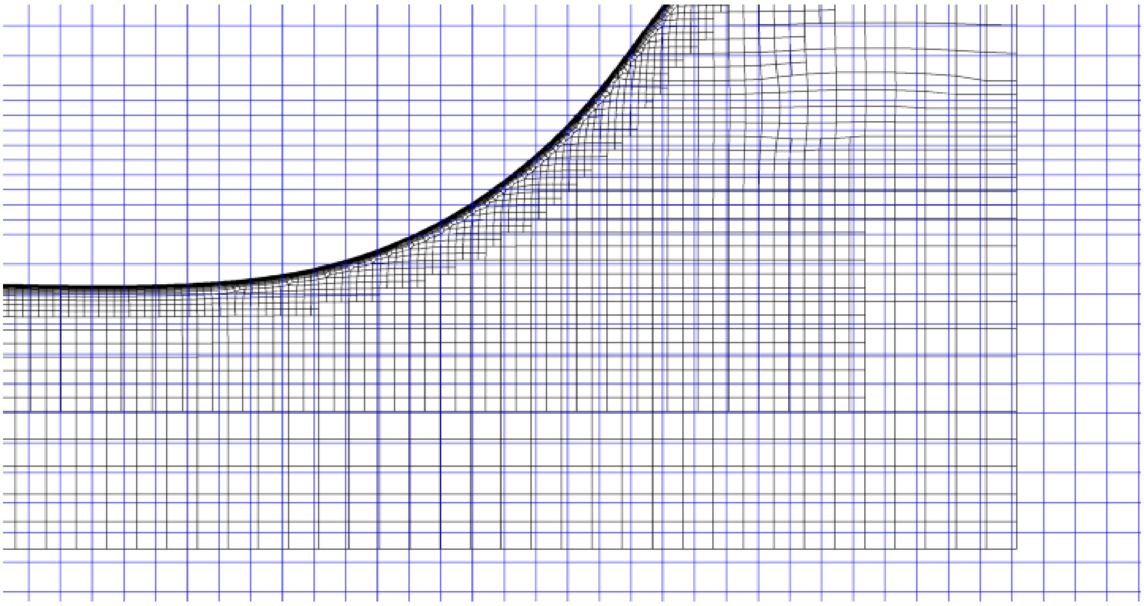

Figure 11: Close up of overset mesh interface area between overset and background region.

Table 3 and Figure 12 show the comparison between the mesh generated by HEXPRESS and STAR CCM+. As there was no easy way to share the mesh between the two softwares, the size of the refinement areas was not known precisely and had to be estimated, and this led to a variation in the total number of cells. Another factor that might have contributed to have different number of cells is that HEXPRESS uses a refinement diffusion that can also be anisotropic, as it can be observed in the top right of Figure 7, while it is not the case for STAR CCM+.

Table 3. Base cell size and total number of cells of the meshes generated with the two softwares.

	FINE/Marine	STAR CCM+
Overset base size [m]	0.592	0.9
Background base size [m]	2.476	2.299
Total number of cells [-]	1 196 583	813417

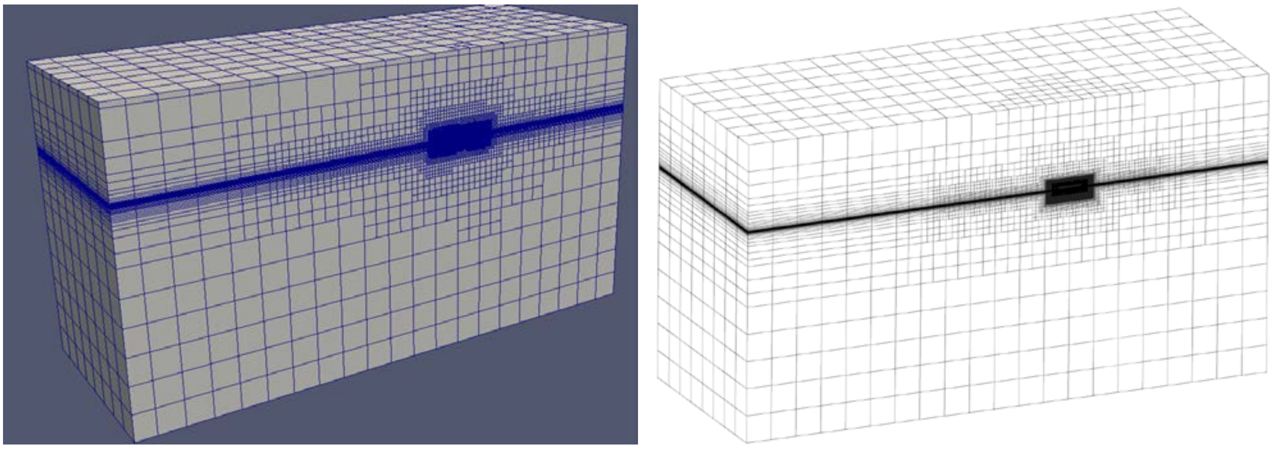

Figure 12: Overset mesh created with STAR CCM+ (left) and with HEXPRESS (right).

3.3. Savitsky mesh

This mesh is equivalent to the deforming single mesh, the only difference is the initial position of the boat: while in first approach, the vessel was aligned with the *x*-axis, it is now translated and rotated around its center of gravity of an angle computed with the Savitsky estimation. This method, explained in detail in Savitsky [3], allows to estimate the final trim, sink, lift, drag and wetted area of prismatic planing hulls given some initial data such as speed, center of gravity, deadrise angle and mass. The more the hull is shaped like a triangle, the more accurate this estimation becomes. In the case treated in this paper the hull is warped and therefore the estimation will not be totally accurate. The advantage of this technique is that the computation can start already at the final speed, and since the initial position of the boat should be closer to the final one, convergence should be reached faster.
The Savitsky estimation method embedded into FINE/Marine requires a different mesh for each Froude number, since a different speed implies a different initial trim and sink. Therefore, 4 meshes were created for this case.Some adjustments were then made to have the same type of refinements as the other two meshes. The total number of cells in this case is around 1 700 000, similarly to the single mesh case.

4. Project settings

In this section the parameters chosen for the simulations are shown and explained. In Table 4 the fluid properties are summarized. The chosen time configuration is steady, since the flow surrounding the vessel is uniform and constant in time. The turbulence model used is *k-ω SST*, which is generally the standard one for marine applications. In De Luca et al. [2] the *k-ε* model was used, but the authors also showed in this paper that the differences in the results obtained with the two models were negligible (Figure 13).

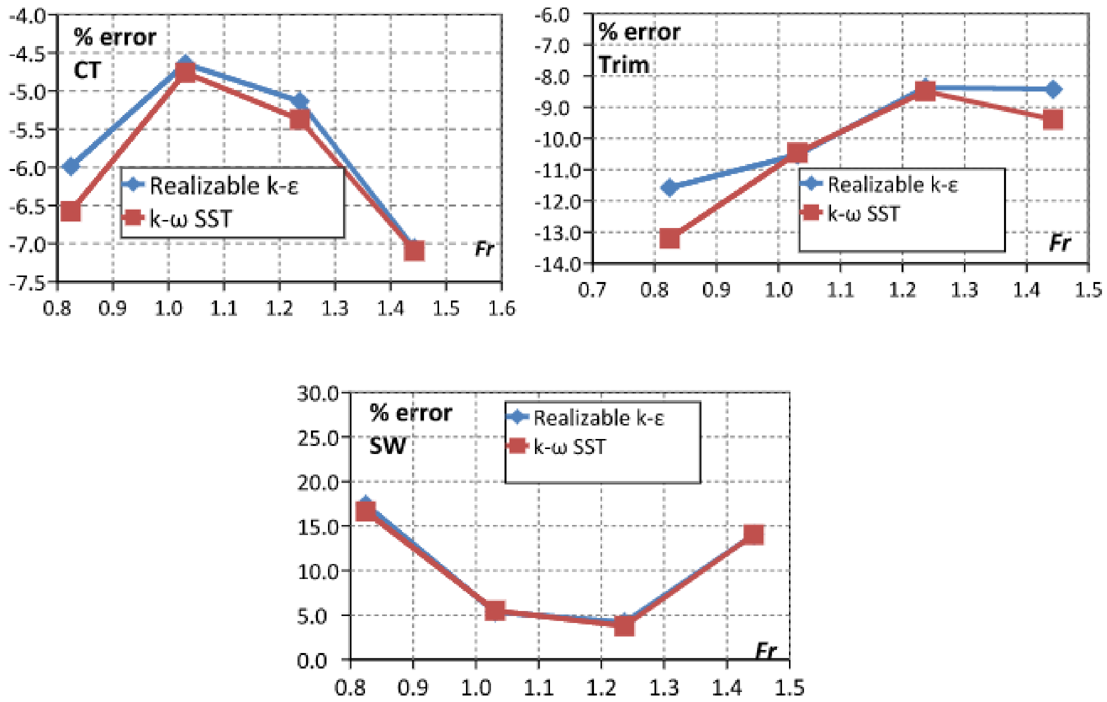

Figure 13: Results obtained with two different turbulence models with STAR CCM+.

Table 4: Fluid properties.

	Density [kg/m^3]	Dynamic viscosity [Pa s]
Water	997.56	0.00104
Air	1.205	1.81×10^{-5}

The applied boundary conditions are summarized in Table 5.

Table 5: Applied boundary conditions.

Patch	BC
Solid patches except Deck	Wall function
Deck	Slip (zero shear stress)
Inlet	Far field, Vx=0
Outlet	Far field, Vx=0
Top	Updated hydrostatic pressure
Bottom	Updated hydrostatic pressure
Side 1 (y=0)	Mirror
Side 2	Far field, Vx=0
All overset boundaries except mirror plane	Overset

Concerning the body motion in the single and overset case, the trim and sink motions are set to "solved" while the translation in *x* is imposed, with a half sinusoidal ramp motion law that brings the vessel from zero velocity to the desired speed in 200 timesteps. As it has been mentioned before, in the Savitsky case it is possible to start the computation with the vessel already at the desired speed, thus with no ramp. Therefore in this case the translation in *x* will be imposed at constant speed, and the Cardan angles for motion reference axis will be initialized with the value predicted by Savitsky. Since the ship is moving, the initial velocity of the flow is set to zero in all cases. An additional wave damping model is added to better reproduce the experimental data, since a damper was present in the towing tank, and also to reduce the oscillations in the results. This damping model is the one introduced by Choi and Yoon (see ref. [4]), and the damping areas are the ones illustrated in Figure 14.

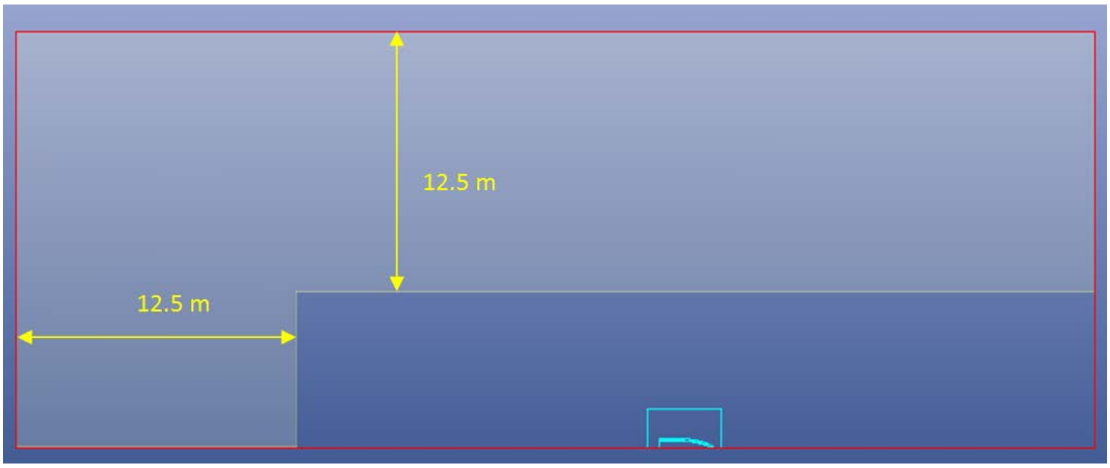

Figure 14: Damping areas.

The choice of the timestep was done following FINE/Marine recommendations and what has been done in De Luca et al. [2], namely:

$$\Delta t = 0.005l/V$$

where V is the hull speed and l is the L_{WL}. All computations were run until convergence, which means that a different number of time steps has been applied to each case with the only common requirement that the oscillations should be less than 1.0% of the final computed value.

5. Results and discussion

5.1 Comparison of results

In this section the results are presented in terms of percentage difference with respect to the experimental data. The difference is expressed as:

$$D = \frac{E-S}{E}$$

Where D is the difference, E the experimental value and S the value computed in the simulation. Figures 15 and 16 show the absolute values of the differences obtained with the various methods and

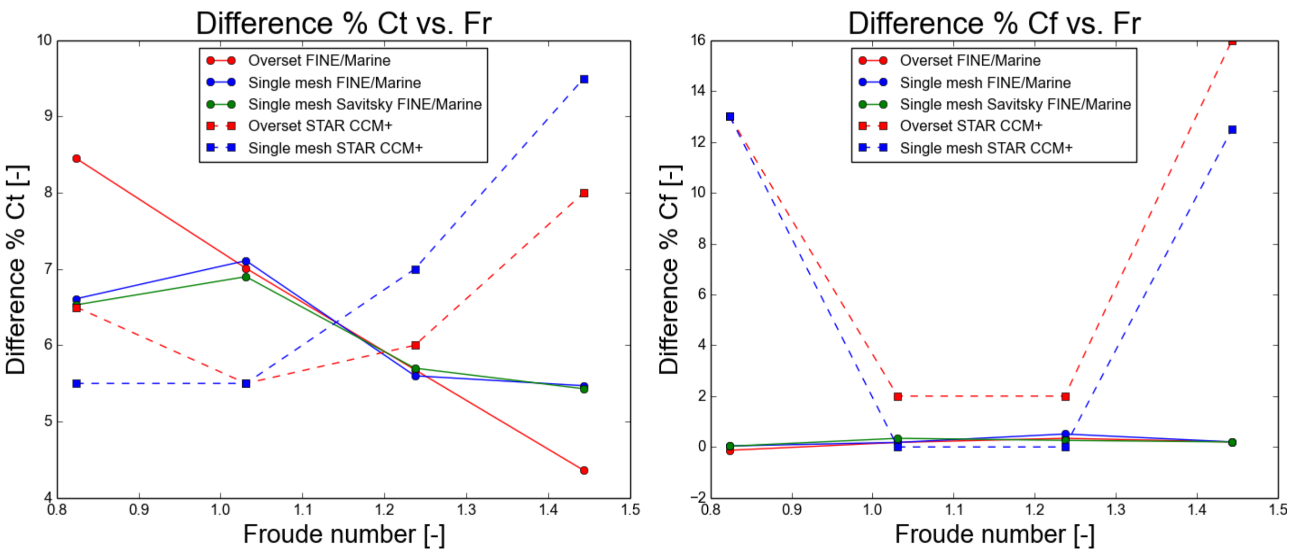

Figure 15: Percentage difference between CFD simulation and EFD for the different methods used. Ct (left) and Cf (right)

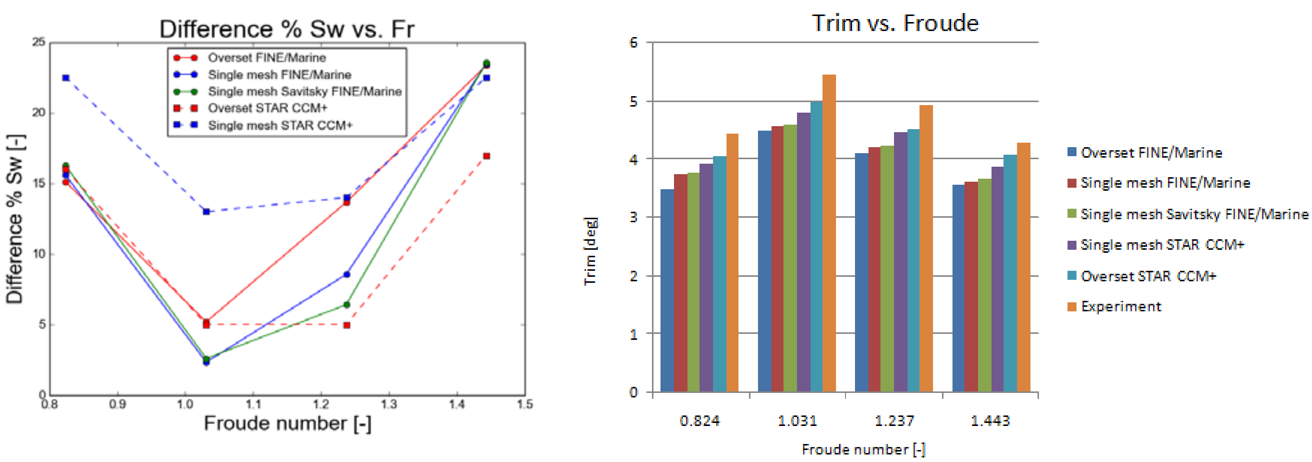

Figure 16: Percentage difference between CFD simulation and EFD for the different methods used for S_W (left), and CFD and EFD trim values (right)

the two different softwares. The analyzed quantities are the resistance coefficient (C_t), the frictional resistance coefficient (C_f), the wetted surface (S_W) and the trim angle (R_y). The two coefficients are computed respectively as:

$$C_t = \frac{Fx}{\frac{1}{2}\rho S_w V^2} \quad C_f = \frac{0.075}{(log_{10}Re-2)^2}$$

Where F_x is the longitudinal force and Re is the Reynolds number based on the wetted keel length L_K, as defined in De Luca et al. [2]. Regarding the results obtained with FINE/Marine, all computational results agree well with the experimental data, as shown in Fig. 15 and 16. For all the mesh configurations that were tested, the difference between the experimental and the CFD data is lower for the frictional resistance coefficient than for the other parameters. The difference for this quantity is particularly low because this parameter only depends on the wetted keel length L_K, which is in almost perfect agreement with the experimental data in all computations. From the results it is possible to argue that the Savitsky approach gives the most satisfactory results, yielding the lowest differences in all computed quantities. As a general consideration it can be stated that even in the worst cases the differences are of totally acceptable entities for all the quantities: the highest difference percentages are found in the trim angle, but in these cases the difference in terms of degrees is never higher than 1 degree.

When comparing the results between the two softwares some observations can be made: FINE/Marine is reliable in computing the frictional coefficient (and therefore the wetted keel length), while STAR CCM+ yields better results in the estimation of the trim angle. Regarding the other analyzed quantities, the graphs show that the differences depending on the Froude number: for example, at low Froude it is preferable to use STAR CCM+ for the resistance coefficient, while FINE/Marine is more accurate at high Froude. The opposite can be stated for the wetted surface, where the overset mesh from STAR CCM+ gives the best estimate at high Froude while FINE/Marine is preferable for low Froude numbers.

5.2 Analysis of Savitsky prediction effects

In this section the Savitsky prediction will be analyzed and its effects on the computation will be shown. First of all it is worth mentioning the tool through which this estimation is performed: the C-Wizard. This plug-in for FINE/Marine is made to simplify the mesh and computation setup. It is made of Python scripts that automatically call existing macros and finish setup procedure with the minimal user input required. It has four main applications: resistance, seakeeping, open water and planing regime. In this case the planing regime will be chosen. This tool allows to import the selected geometry, input the necessary data for the computation and obtain the final trim and sink of the boat as an output using the estimation method from Savistky.

The estimation method consists in an iterative process for which a trim angle is initially guessed and plugged in simplified equilibrium equations, at the end of each cycle a check is made and the process is stopped when equilibrium is reached (given a certain tolerance). The approximation in this method lays in the fact that the simplified equations are based only on certain types of hulls (prismatic planing hulls), and therefore the estimation will be more or less realistic the more or less the shape of the hull is close to the prismatic one.

The aim of this technique is to reduce the computation time since the estimation places the boat in the estimated final position. Figures 17 and 18 compare the convergence histories of trim and drag with and without Savitsky estimation for 4.0 m/s.

From the graphs it can be argued that the Savitsky estimation guarantees a faster convergence to the same value for both quantities, as it is shown by the vertical lines. In the Savitsky case the drag force stabilizes to its final value after around 4 seconds of physical time of simulation, while in the single mesh case the value can be considered converged only after 6 seconds, as it can be noted in the close up of Figure 18. For the trim angle the difference is even more appreciable: with Savitsky convergence is reached after less than 4 seconds of physical time, while without the estimation only after around 6 seconds. The great improvement is due to the fact that the trim angle is initialized to a value that is closer to the final one, and therefore the latter is reached more quickly. Hence the Savitsky computation could have been stopped at 4 seconds of physical time, saving up to 33% CPU time.

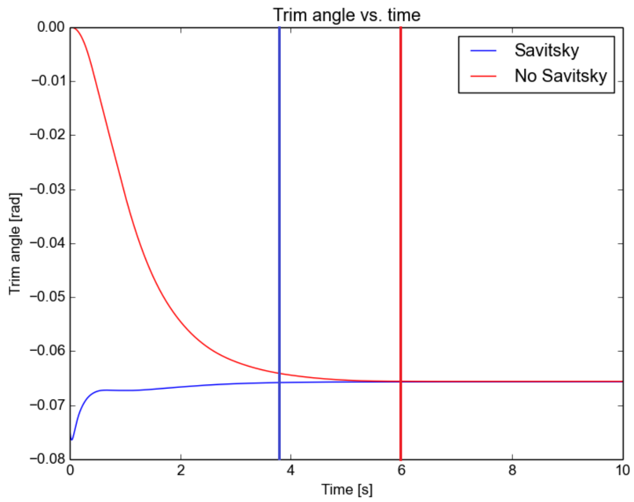

Figure 17: convergence history of trim angle at 4 m/s with and without Savitsky

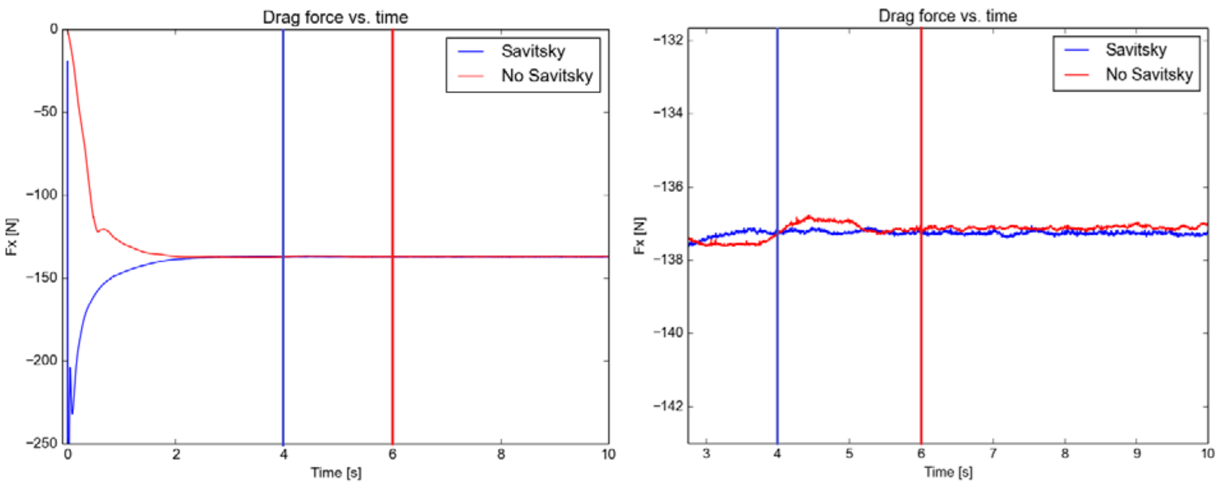

Figure 18: convergence history of drag force at 4.0 m/s with and without Savitsky (left) and close up (right)

5.3 Analysis of streaking correction effects

As it has been mentioned before, the computations on planing hulls are numerically more complicated than the regular ones, therefore the mass fraction can often result unphysical through a phenomenon called streaking, for which spray appears in areas where water should be present, as indicated by Ferziger and Peric [5] and by Andrillion and Alessandrini [6]. In order to avoid this source of error twonumerical corrections have been implemented in FINE/Marine These corrections are activated only where the viscous layers are present (hull and transom). In Figure 19 and 20 the difference in the mass fraction obtained with and without the corrections for the 4.0 m/s case can be appreciated.

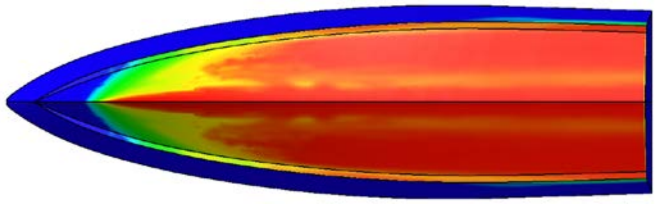

Figure 19: Z view of mass fraction without correction

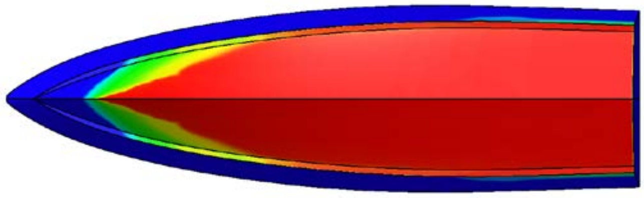

Figure 20: Z view of mass fraction with streaking correction

The figures show how the correction is working correctly and is improving the solution guaranteeing a physically meaningful mass fraction distribution underneath the ship hull. Similar results were obtained with STAR-CCM+. Hence, it seems that these numerical corrections are necessary in the guidelines for high speed boat simulations.

6. Conclusions

A hydrodynamic investigation on the C1 model of the Naples Systematic Series has been done. Two commercial CFD softwares and various mesh strategies were compared. For both softwares the results show good agreement with the benchmark experimental data obtained at the University of Naples Towing Tank, but it can be stated that there is still room for improvement since most of the percentage differences are in the range 0-20%. This result can be considered anyways satisfactory given the history of CFD computations on planing hulls, that shows how are they much more difficult to perform correctly than the regular ones, as shown in De Luca et al. [2]. Inside the computations carried out with the FINE/Marine software, investigations have been done to evaluate the usefulness of some tools such as the Savitsky estimation and the streaking correction. Regarding the former, it has been shown that this tool can significantly speed up the computation and allow to reduce the simulation time up to 33%. The streaking correction showed to help limit the streaking phenomenon, typical of high speed simulations, for which the mass fraction is not physical in some areas due to numerical ventilation beneath the hull. The correction allows to ensure the physical meaningfulness of the mass fraction and therefore a correct estimation of the efforts on the hull. Overall it can be stated that CFD is definitely a useful tool when combined with experimental data to have reliable information on the behavior of a high speed planing boat.

7. Recommendations

The results presented in this paper show that commercial CFD softwares can estimate quantities like trim angle, wetted surface and resistance coefficient. For this particular test case various combinations of software and mesh strategies were investigated and it can be argued that some combinations are particularly accurate for the computation of some quantities while not so much for other ones. For example, the combination FINE/Marine-Overset mesh yields the most accurate estimation of the frictional coefficient, while the combination STAR-Overset mesh is preferable for estimation of trim angle. Therefore, general guidelines should still be investigated but trends seems to already appear. The users can use this paper as a first guideline to decide what strategy to use depending on the quantity they want to focus on in their analysis.

REFERENCES

[1] F. De Luca, C. Pensa, "The Naples Warped Hard Chine Hulls Systematic Series – First Part: Resistance in Still Water", *Proceedings of 10th High Speed Marine Vehicles*, October 2014, Naples, Italy.

[2] F. De Luca, S. Mancini, S. Miranda, and C. Pensa, "An Extended Verification and Validation Study of CFD Simulations for Planing Hulls", *Journal of Ship Research*, Vol. 60, No. 2, June 2016, pp. 101–118.

[3] D. Savitsky. "Hydrodynamic Design of Planing Hulls", *Meeting of the New York Metropolitan Section of the Society of Naval Architects and Marine Engineers*, 1964.

[4] J. Choi, S. B. Yoon, "Numerical Simulations Using Momentum Source Wave-Maker Applied to RANS Equation Model", *Coastal Engineering,* Vol. 56 (10), 2009, pp. 1043 - 1060.

[5] J. Ferziger and M. Peric, "Computational Method for Fluid Dynamics", Berlin: Springer-Verlaag, 2003.

[6] V. Andrillion and B. Alessandrini, "A 2D+T VOF Fully Coupled Formulation for Calculation of Breaking Free Surface Flow," in *Proceedings of the 24th Symposium on Naval Hydrodynamics*, July 2002, Fukuoka, Japan.