CFD VALIDATION STUDY:

GPPH - CALM WATER

NepTech
Intelligent sea mobility

Version	Date	Written by	Validated by
1	XX/XX/XXXX	Tanguy TEULET	Clément ROUSSET

Table of contents

Summary

This report provides a comprehensive validation study on the Generic Prismatic Planing Hull (GPPH) in calm water, using NepTech's digital towing tank. The study focuses on a high-speed regime, covering a Froude number range from 1.2 to 2.7. Key findings compare CFD results with experimental data, addressing resistance, vessel motions, free surface renderings, and computational time. The findings confirm that NepTech's automated digital towing tank is reliable and efficient for simulations of very high Froude numbers in record time, validating its capabilities for accurate predictions in similar flow types.

Nomenclature

- ❖ B_{WL} $[m]$, waterline beam.
- ❖ F_n $[-]$, Froude number.
- ❖ S_f $[m^2]$, waterline area.
- ❖ Δ $[kg]$, displacement.
- ❖ CFD, Computational Fluid Dynamic.
- ❖ EFD, Experimental Fluid Dynamic.
- ❖ LCG ; TCG ; VCG $[m]$, coordinates of the centre of gravity: lateral, transversal and vertical.
- ❖ LOA $[m]$, overall length.
- ❖ T $[m]$, draught.
- ❖ V $[m/s]$, ship speed.
- ❖ θ $[deg]$, static trim angle.
- ❖ μ $[Pa.s]$, dynamic viscosity.
- ❖ ρ $[kg/m^3]$, density.

Figures

Tables

1. GPPH

The GPPH hull was designed as a publicly releasable reference model to support research and development across government agencies, contractors, and academic institutions. Its prismatic design was specifically chosen to represent typical planing hulls while minimizing geometric complexities such as warp, rocker, and curvature in both transverse and longitudinal directions. By reducing these variables, the GPPH provides a simplified yet relevant test case for validating CFD predictions. This hull serves as a benchmark for hydrodynamic studies, enabling consistent comparisons across experimental and numerical simulations.

The paper "Experimental Results for the Calm Water Resistance of the Generic Prismatic Planing Hull (GPPH)" provides comprehensive towing tank results for the GPPH model.

This study is the sole reference for the present validation of NepTech's digital towing tank, as it provides the original, experimentally validated results. Moreover, the availability of detailed numerical data allows seamless integration into the validation process, ensuring accuracy and consistency in comparing numerical and experimental outcomes.

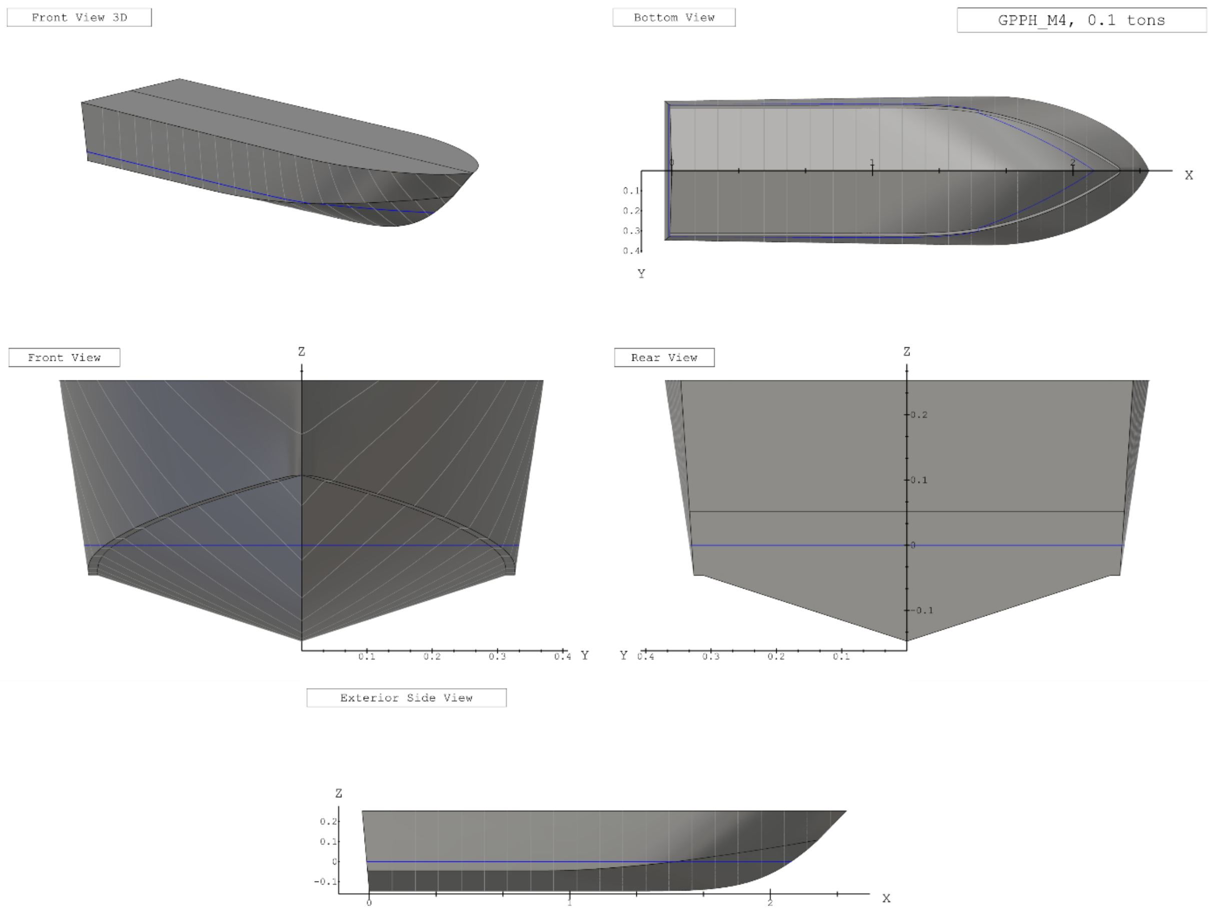


Figure 1: GPPH CAD model

2. Simulation setup

a. Sign convention

Heave: The heave values correspond to the dynamic elevation of the vessel at the centre of gravity, relative to its hydrostatic position, in the absolute reference frame with the vertical axis Z oriented upwards. A positive heave value thus corresponds to a hull rise, while a negative value indicates the hull sinking.

Pitch: The pitch values correspond to the dynamic trim of the vessel at the centre of gravity, relative to its hydrostatic position, in the absolute reference frame where the transverse axis is Y. A positive trim corresponds to a bow-up attitude of the hull.

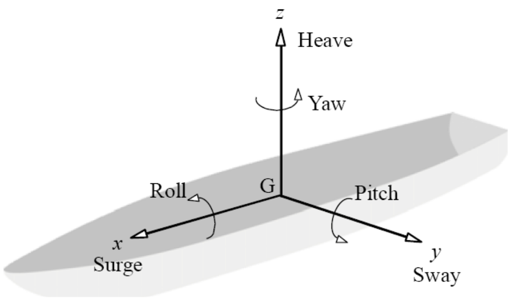


Figure 2: Sign convention illustration

b. Software's

Mesh: Hexpress™, version 12.1 developed by CADENCE

Resolution: Fidelity Fine Marine, version 12.1 developed by CADENCE

Solver: ISIS-CFD developed by CNRS and Centrale Nantes

Computing infrastructure: 2 virtual machines with 32 cores « C2D_STANDARD_32 », optimized for computation on Google Cloud Platform.

Post-processing:

- CFView™, version 12.1 developed by CADENCE
- Programming language Python version 3.11.6

c. Hypothesis

<u>Modelling scale:</u> model scale, with a symmetry plane along the vessel's median axis. This approach reduces computation time while maintaining identical results.

<u>Domain:</u> the dimensions of the simulation domain are conformed to International Towing Tank Conference (ITTC) recommendations, ensuring that the boundaries are positioned sufficiently far from the vessel to avoid any influence on the solution. It is crucial, especially for the outlet boundary, to place it in a way that prevents the reflection of the wave field generated by the vessel.

<u>Hydrostatic equilibrium:</u> the coordinates of the centre of gravity are defined as follows

$$LCG = 0.8440\, m;\ TCG = 0.0000\, m;\ VCG = -0.0097\, m$$

<u>Water:</u> corresponds to fresh water, which is

$$\rho_{water} = 995.56\, kg/m^3$$

$$\mu_{water} = 0.7972 * 10^{-3}\, Pa.s$$

<u>Air:</u> corresponds to air at a temperature of $15°C$, which is

$$\rho_{air} = 1.2256\, kg/m^3$$

$$\mu_{air} = 1.788 * 10^{-5}\, Pa.s$$

<u>Mesh precision</u>: this report presents the results from medium level meshes.

d. Numerical models

<u>Dynamic equilibrium:</u>

- The Quasi-Static (QS) method is used since we are interested in the vessel's dynamic equilibrium state. This method relies on a succession of predictions of the vessel's physical attitude to reach the dynamic equilibrium state in record time.
- Two movements of the vessel, heave and pitch, are left free to ensure convergence toward the vessel's dynamic equilibrium position.

<u>Flow:</u> The Reynolds-Averaged Navier-Stokes (URANS) equations are used to describe the flow, and they are coupled with the $k - \omega\, SST$ turbulence model as the closure model.

<u>Free surface:</u> The air-water interface is modelled using the Volume of Fluid (VoF) method. Adaptive Grid Refinement (AGR), developed by CNRS (French National Centre for Scientific Research) and Ecole Centrale de Nantes (French Engineering school), is used to model the free surface. This iterative process allows for dynamic adjustment of the mesh according to the solution's needs during the calculation, making refinement decisions based on the physics of the flow.

e. Validation

i. Mesh

Hull: The accuracy of the results regarding viscous resistance mainly depends on the mesh of the hull. This resistance is caused by the entrainment of a thin fluid film: the boundary layer. An appropriate mesh of the boundary layer is essential to correctly capture local phenomena such as viscous effects and rapid variations in fluid properties near the surface. It also allows for better capture and resolution of turbulent phenomena if they are present. The quality of the hull mesh also affects the fidelity of the 3D model representation. A clean and regular mesh improves the reliability of the simulation, making the simulated model more representative of the actual vessel.

Figure 3 illustrates the hull mesh configurations used for the various speeds considered in the study.

Free surface: The accuracy of the results regarding pressure resistance mainly depends on how the air-water interface is captured during simulation. This resistance is induced by the wave field generated by the vessel, and the quality of the mesh for the latter plays a crucial role in this accuracy. The use of AGR allows dynamically adapting the mesh based on the generated wave field, achieving maximum precision, as it is one of the most advanced and reliable methods to date and reducing computation time by converging more quickly toward the dynamic equilibrium state.

Figure 4 illustrates the free surface mesh configurations used for the various speeds considered in the study.

Values:

Ship speed $\mathbf{V}$ $[\mathbf{knots}]$	10.80	13.00	14.80	17.50	19.50	21.60	23.80
Froude number $\mathbf{F_n}$ $[-]$	1.22	1.47	1.67	1.98	2.20	2.44	2.69
Averaged number of cells [*10^6]	1.62	1.58	1.63	1.59	1.65	1.66	1.66

Table 1: Averaged number of cells

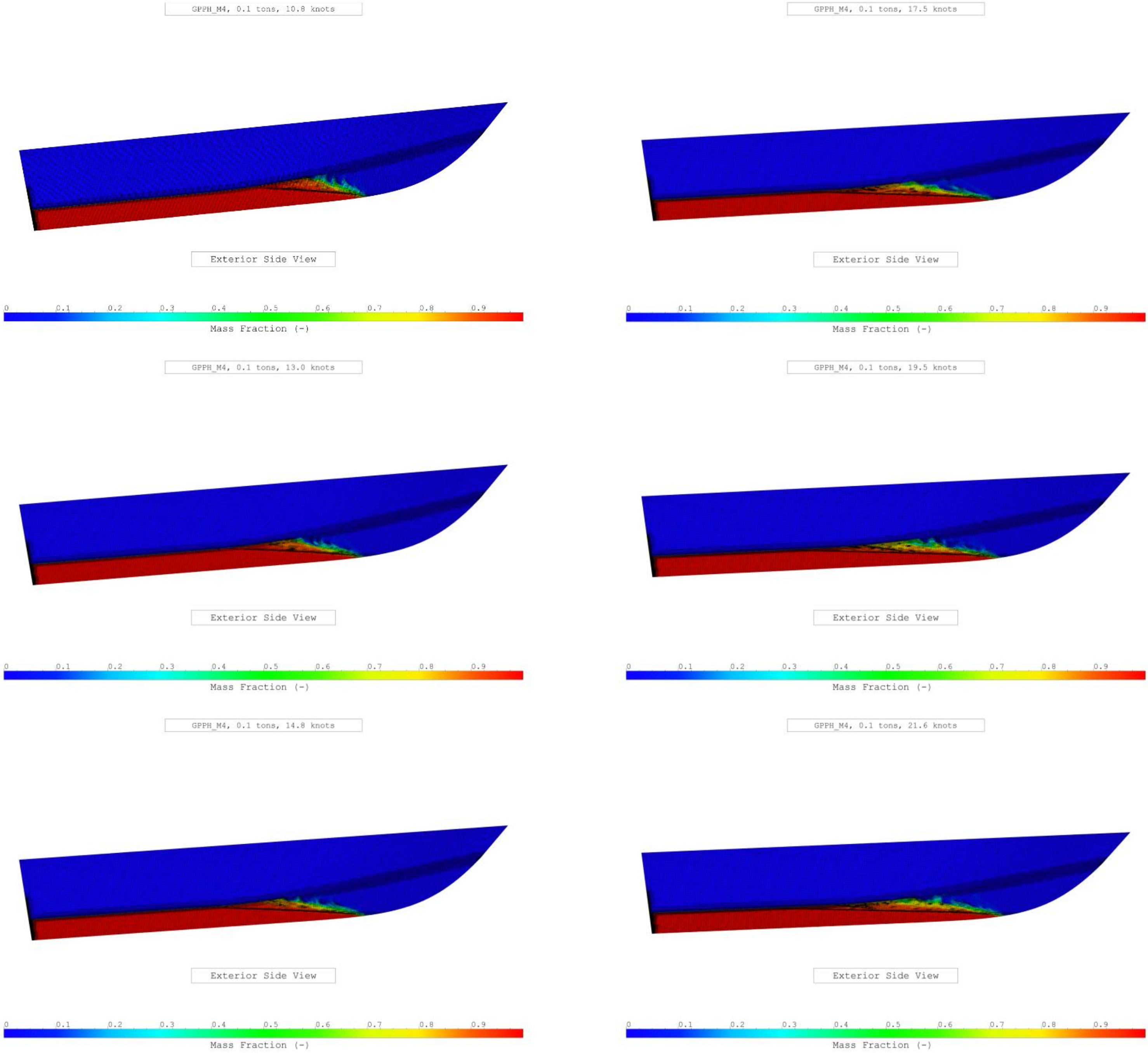


Figure 3: Bare hull mesh from 10.80 to 21.60 knots

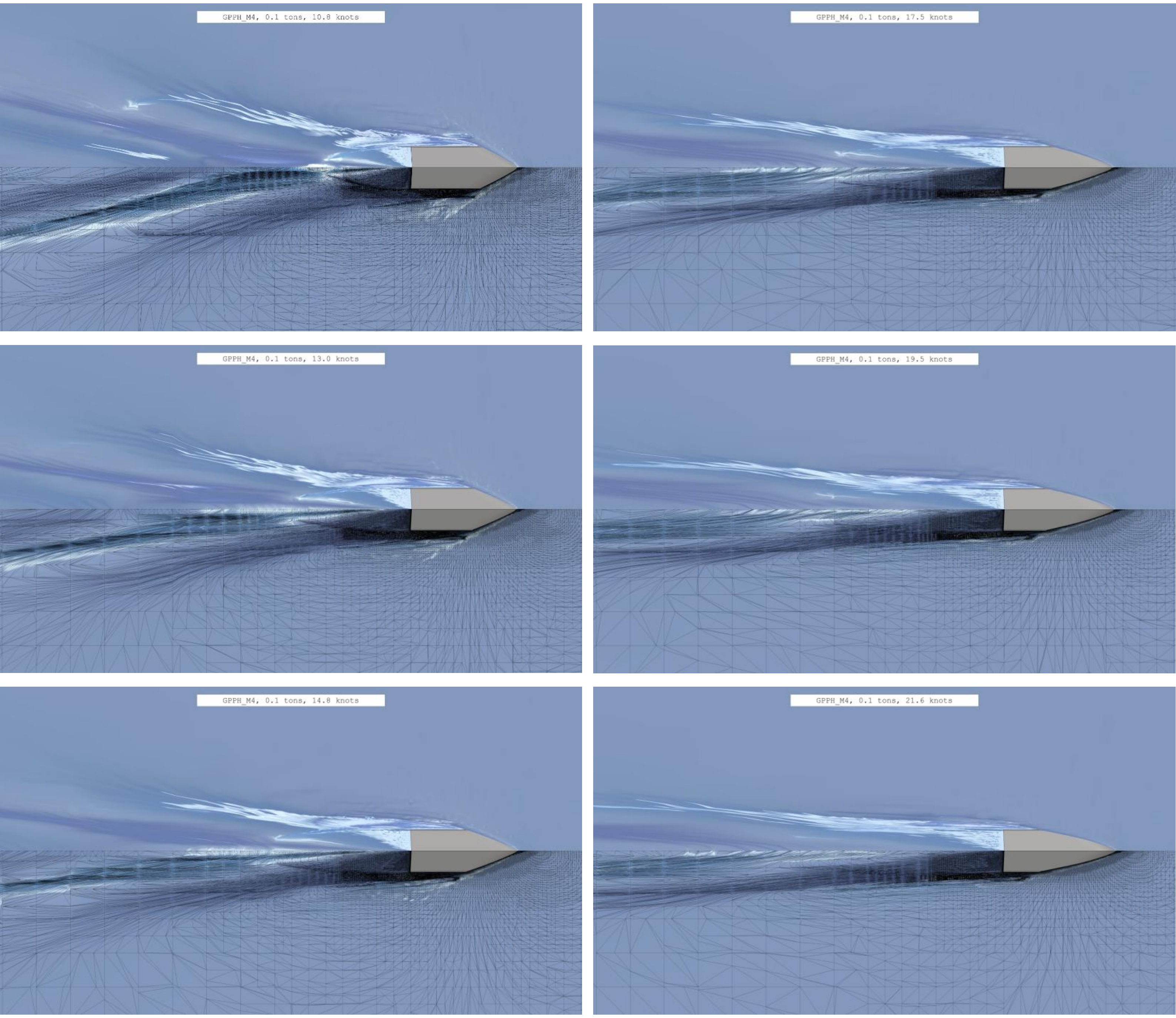


Figure 4: Free surface mesh from 10.80 to 21.60 knots

ii. Courant number

Description: The Courant number, also called the CFL (Courant-Friedrichs-Lewy) number, is a crucial parameter in computational fluid dynamics (CFD). It measures the numerical stability of the discretization scheme used in the simulation. An inappropriate Courant number can lead to numerical instabilities, compromising both convergence and the accuracy of the results. In CFD, the Courant number is related to the size of the numerical time steps. It is calculated by comparing the speed of fluid particles with the size of the cells in the simulation domain.

Recommended values: For typical resistance simulations, it is recommended to keep the Courant number below or close to 1 to ensure maximum accuracy and reliability. Local spikes in this parameter may occur, but it is essential to control them to maintain numerical stability and the quality of the results.

Values:

Ship speed $\mathbf{V}$ $[\mathbf{knots}]$	10.80	13.00	14.80	17.50	19.50	21.60	23.80
Froude number $\mathbf{F_n}$ $[-]$	1.22	1.47	1.67	1.98	2.20	2.44	2.69
Averaged Courant number [-]	0.18	0.16	0.13	0.11	0.10	0.10	0.09

Table 2: Averaged Courant number (Free Surface)

Ship speed $\mathbf{V}$ $[\mathbf{knots}]$	10.80	13.00	14.80	17.50	19.50	21.60	23.80
Froude number $\mathbf{F_n}$ $[-]$	1.22	1.47	1.67	1.98	2.20	2.44	2.69
Averaged Courant number [-]	1.36	1.34	1.34	1.33	1.31	1.30	1.28

Table 3: Averaged Courant number (Hull)

iii. Y+

Description: In the naval field, managing the Y+ parameter is crucial in computational fluid dynamics (CFD) simulations. Y+ measures the quality of the boundary layer resolution along the submerged surfaces of ship hulls by evaluating the distance between the first mesh point and the wall relative to the boundary layer thickness. Maintaining an appropriate Y+ is essential to ensure reliable results in predicting resistance, drag, lift, and other critical hydrodynamic phenomena. An improper Y+ can lead to significant errors in the prediction of forces, drag coefficients, and other key parameters.

Recommended values: For typical resistance simulations, it is recommended that the Y+ value be between 30 and 300. This value may be lower depending on the choice of boundary layer modeling. Local spikes in this parameter may occur, but it is essential to control them to maintain numerical stability and the quality of the results.

Values:

Ship speed $\mathbf{V}$ $[\mathbf{knots}]$	10.80	13.00	14.80	17.50	19.50	21.60	23.80
Froude number $\mathbf{F_n}$ $[-]$	1.22	1.47	1.67	1.98	2.20	2.44	2.69
Averaged Y+ [-]	120.10	121.25	122.25	123.70	123.78	124.68	124.67

Table 4: Averaged Y+

3. Results

a. Geometry and hydrostatic

The numerical model used for the CFD simulations demonstrates a strong alignment with the experimental model used in the towing tank tests in terms of hydrostatic characteristics, ensuring a reliable comparison between numerical and experimental results. Key parameters such as displacement, draft, and static trim angle are well-matched, reinforcing the validity of the numerical approach. These hydrostatic properties significantly influence flow behaviour around the hull, directly impacting resistance predictions and hydrodynamic performance.

While most parameters show excellent agreement, two notable differences should be considered. The waterline beam in the CFD model is 6.06% larger than in the experimental model, which may increase the wetted surface area and alter wave generation. Conversely, the waterline area in the CFD model is 5.88% smaller, potentially affecting the distribution of pressure along the hull. These discrepancies, although small, can have contrasting effects on total resistance: a wider beam could slightly increase frictional resistance, while a reduced waterline area might modify wave-making resistance.

As a result, it remains uncertain whether the CFD simulations will slightly overpredict or underpredict the resistance compared to the experimental data. However, the close agreement in hydrostatic characteristics supports the credibility of the numerical model and ensures that the comparison remains meaningful and reliable.

Main particulars		EFD	CFD	Difference [%]
Length overall	$\boldsymbol{LOA}\ [\boldsymbol{m}]$	2.414	2.414	0.00
Beam at waterline	$\boldsymbol{B_{WL}}\ [\boldsymbol{m}]$	0.627	0.665	6.06
Draft	$\boldsymbol{T}\ [\boldsymbol{m}]$	0.148	0.147	0.67
Displacement	$\Delta\ [\boldsymbol{kg}]$	101.514	101.510	-0.00
Static trim angle	$\boldsymbol{\theta}\ [\boldsymbol{deg}]$	0.127	0.132	3.94
Waterline area	$\boldsymbol{S_f}\ [\boldsymbol{m^2}]$	1.261	1.191	-5.88

Table 5: Geometry and hydrostatic

b. Resistance

Figure 5 illustrates the progression of the GPPH resistance across different advance speeds in the top graph and table. The middle table shows the absolute differences between CFD and EFD in the international system of units, while the bottom table displays the relative difference between CFD and EFD as a percentage:

$$E\%\,CFD = \frac{CFD - EFD}{EFD} * 100$$

The resistance error in the CFD simulations ranges from -9.14 N to -4.32 N, corresponding to a deviation of -3.39% to +2.68% compared to experimental measurements. On average, the CFD results underestimate the resistance by approximately 3%. The near-constant offset observed between the numerical and experimental resistance curves suggests that this discrepancy is primarily due to differences in hydrostatic characteristics rather than numerical instability or turbulence modelling errors.

At these high Froude numbers, the flow dynamics become highly complex, with intense turbulent interactions, wave-breaking effects, and unsteady flow structures playing a significant role. Given the intricacy of these physical phenomena, achieving an accuracy within 3%, likely attributable to slight hydrostatic variations, is an excellent result.

	Speed [knots]	Froude number [-]	● Total resistance [N], GPPH_M4	■ Total resistance [N], GPPH_EFD
0	10.80	1.22	157.10	161.42
1	13.00	1.47	154.17	159.56
2	14.80	1.67	159.07	165.38
3	17.50	1.98	177.49	184.42
4	19.50	2.20	199.57	206.80
5	21.60	2.44	227.24	234.91
6	23.80	2.69	260.69	269.83

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [N]
0	10.80	1.22	-4.32
1	13.00	1.47	-5.39
2	14.80	1.67	-6.31
3	17.50	1.98	-6.93
4	19.50	2.20	-7.23
5	21.60	2.44	-7.67
6	23.80	2.69	-9.14

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [%]
0	10.80	1.22	-2.68
1	13.00	1.47	-3.38
2	14.80	1.67	-3.82
3	17.50	1.98	-3.76
4	19.50	2.20	-3.50
5	21.60	2.44	-3.27
6	23.80	2.69	-3.39

Figure 5: Evolution (up), difference [N] (middle) and difference [%] (bottom) of total resistance

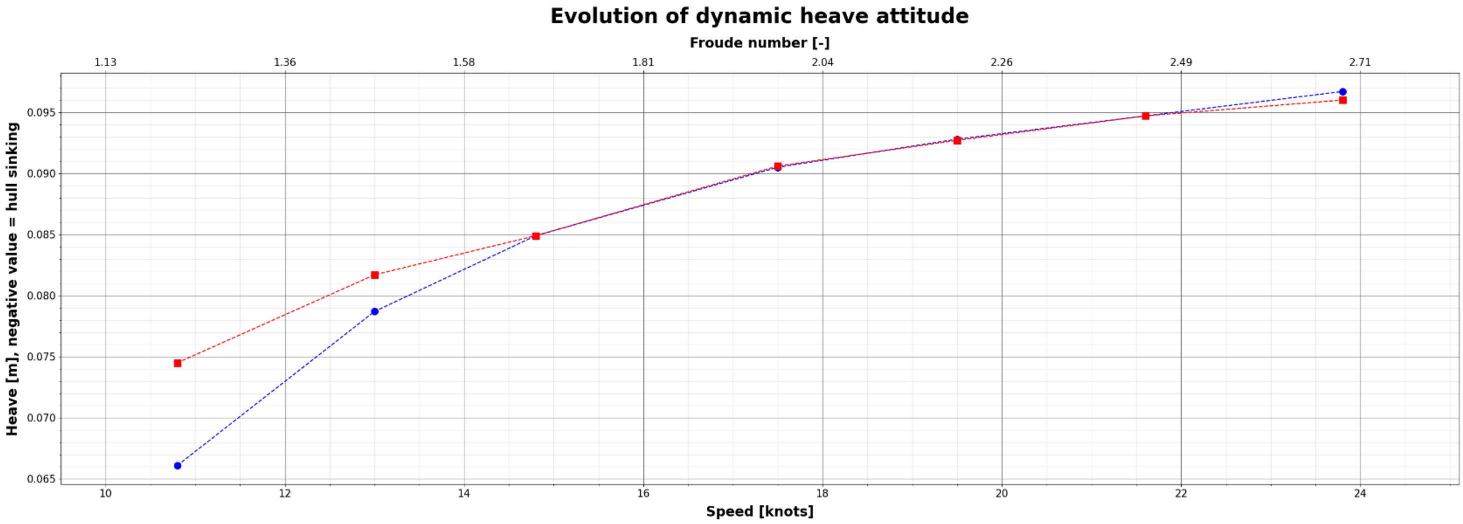
c. Motions

i. Heave

Figure 6 illustrates the progression of the GPPH dynamic heave response across different advance speeds in the top graph and table. The middle table shows the absolute differences between CFD and EFD in the international system of units, while the bottom table displays the relative difference between CFD and EFD as a percentage.

The dynamic heave error between the CFD and experimental results ranges from -0.008 m to +0.001 m, corresponding to a deviation of -11.28% to +0.73% compared to the experimental measurements. Overall, the CFD simulations closely follow the experimental trend, with the curves nearly overlapping.

However, at Froude numbers of 1.22 and 1.47, a more noticeable discrepancy is observed. At these high Froude numbers, the flow dynamics become increasingly complex due to intense wave interactions and dynamic pressure variations along the hull. The heave motion in this regime is highly influenced by the balance between hydrodynamic lift and dynamic trim adjustments. Despite this localized discrepancy, the overall agreement between CFD and experimental data suggests that the numerical model effectively captures the dominant trends of dynamic heave across the tested conditions.



	Speed [knots]	Froude number [-]	● Heave [m], GPPH_M4	■ Heave [m], GPPH_EFD
0	10.80	1.22	0.0661	0.0745
1	13.00	1.47	0.0787	0.0817
2	14.80	1.67	0.0849	0.0849
3	17.50	1.98	0.0905	0.0906
4	19.50	2.20	0.0928	0.0927
5	21.60	2.44	0.0947	0.0947
6	23.80	2.69	0.0967	0.0960

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [m]
0	10.80	1.22	-0.008
1	13.00	1.47	-0.003
2	14.80	1.67	0.000
3	17.50	1.98	-0.000
4	19.50	2.20	0.000
5	21.60	2.44	0.000
6	23.80	2.69	0.001

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [%]
0	10.80	1.22	-11.28
1	13.00	1.47	-3.67
2	14.80	1.67	0.00
3	17.50	1.98	-0.11
4	19.50	2.20	0.11
5	21.60	2.44	0.00
6	23.80	2.69	0.73

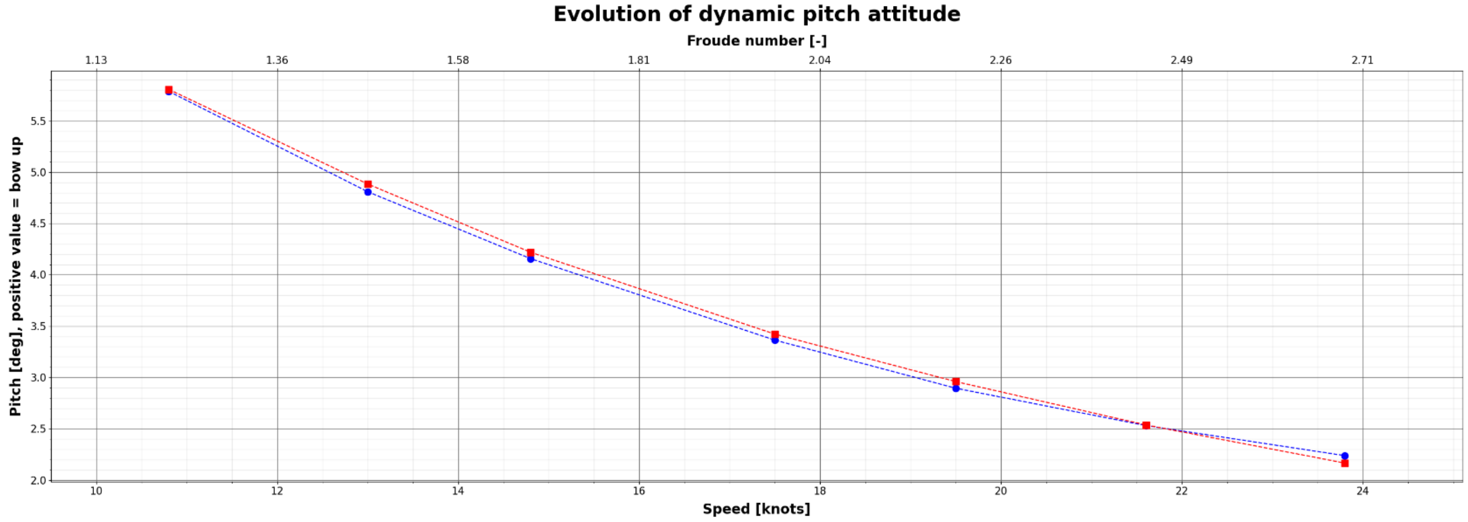
Figure 6: Evolution (up), difference [m] (middle) and difference [%] (bottom) of dynamic heave attitude

ii. Pitch

Figure 7 illustrates the progression of the GPPH dynamic pitch response across different advance speeds in the top graph and table. The middle table shows the absolute differences between CFD and EFD in degrees, while the bottom table displays the relative difference between CFD and EFD as a percentage.

The dynamic heave error between the CFD and experimental results ranges from -0.075 deg to +0.072 deg, corresponding to a deviation of -2.19% to +3.33% compared to the experimental measurements.

Overall, the CFD simulations closely follow the experimental trend, with the curves nearly overlapping, indicating a strong agreement in predicting the vessel's dynamic trim behaviour.



	Speed [knots]	Froude number [-]	● Pitch [deg], GPPH_M4	■ Pitch [deg], GPPH_EFD
0	10.80	1.22	5.7859	5.8054
1	13.00	1.47	4.8081	4.8832
2	14.80	1.67	4.1567	4.2191
3	17.50	1.98	3.3635	3.4218
4	19.50	2.20	2.8950	2.9597
5	21.60	2.44	2.5321	2.5376
6	23.80	2.69	2.2388	2.1666

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [deg]
0	10.80	1.22	-0.019
1	13.00	1.47	-0.075
2	14.80	1.67	-0.062
3	17.50	1.98	-0.058
4	19.50	2.20	-0.065
5	21.60	2.44	-0.006
6	23.80	2.69	0.072

	Speed [knots]	Froude number [-]	X_0 = GPPH_M4, [%]
0	10.80	1.22	-0.34
1	13.00	1.47	-1.54
2	14.80	1.67	-1.48
3	17.50	1.98	-1.70
4	19.50	2.20	-2.19
5	21.60	2.44	-0.22
6	23.80	2.69	3.33

Figure 7: Evolution (up), difference [deg] (middle) and difference [%] (bottom) of dynamic pitch attitude

d. Free surface elevations

i. Same scale

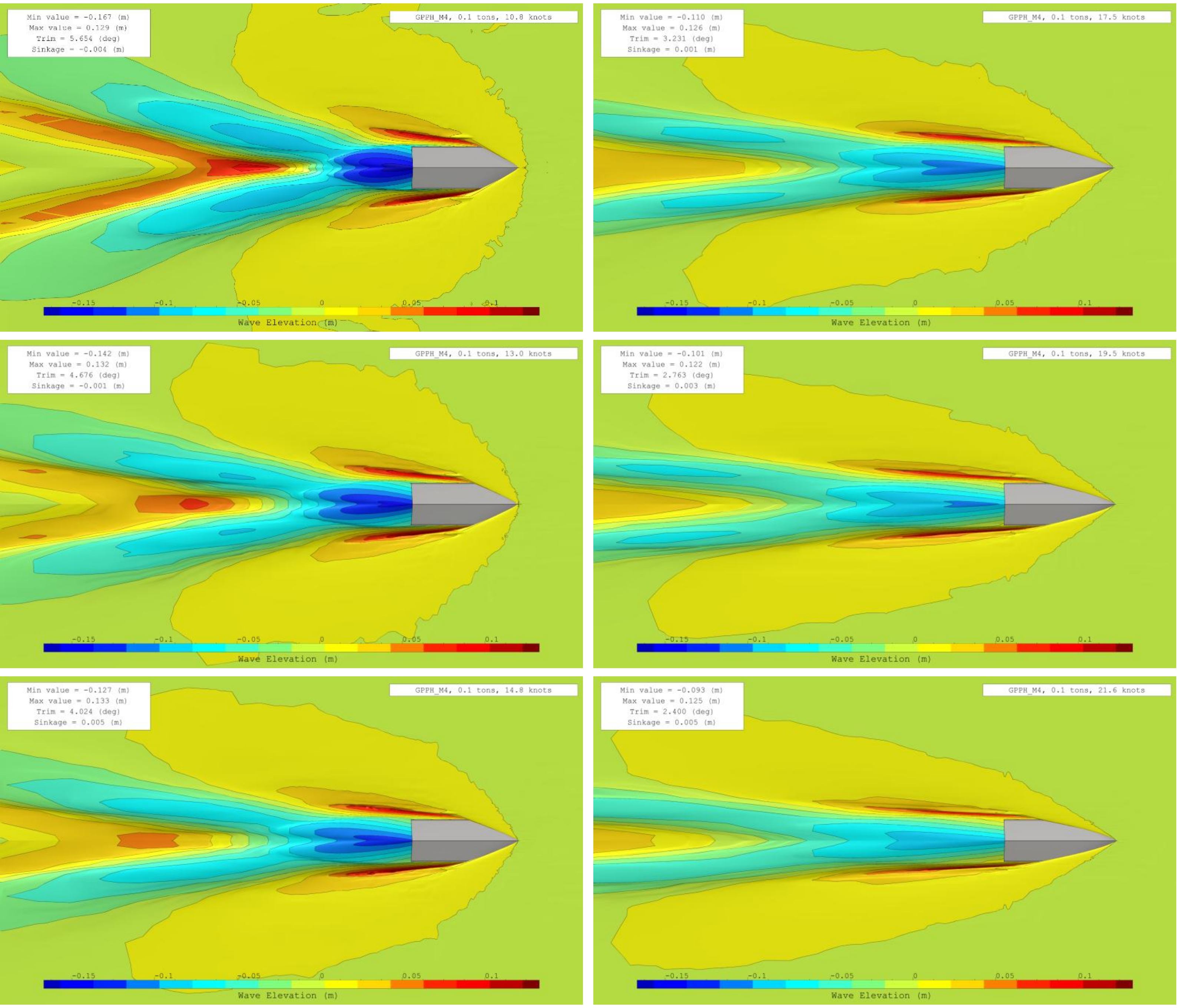


Figure 8: Free surface evolution (same scale) from 10.80 to 21.60 knots

ii. Independent scale

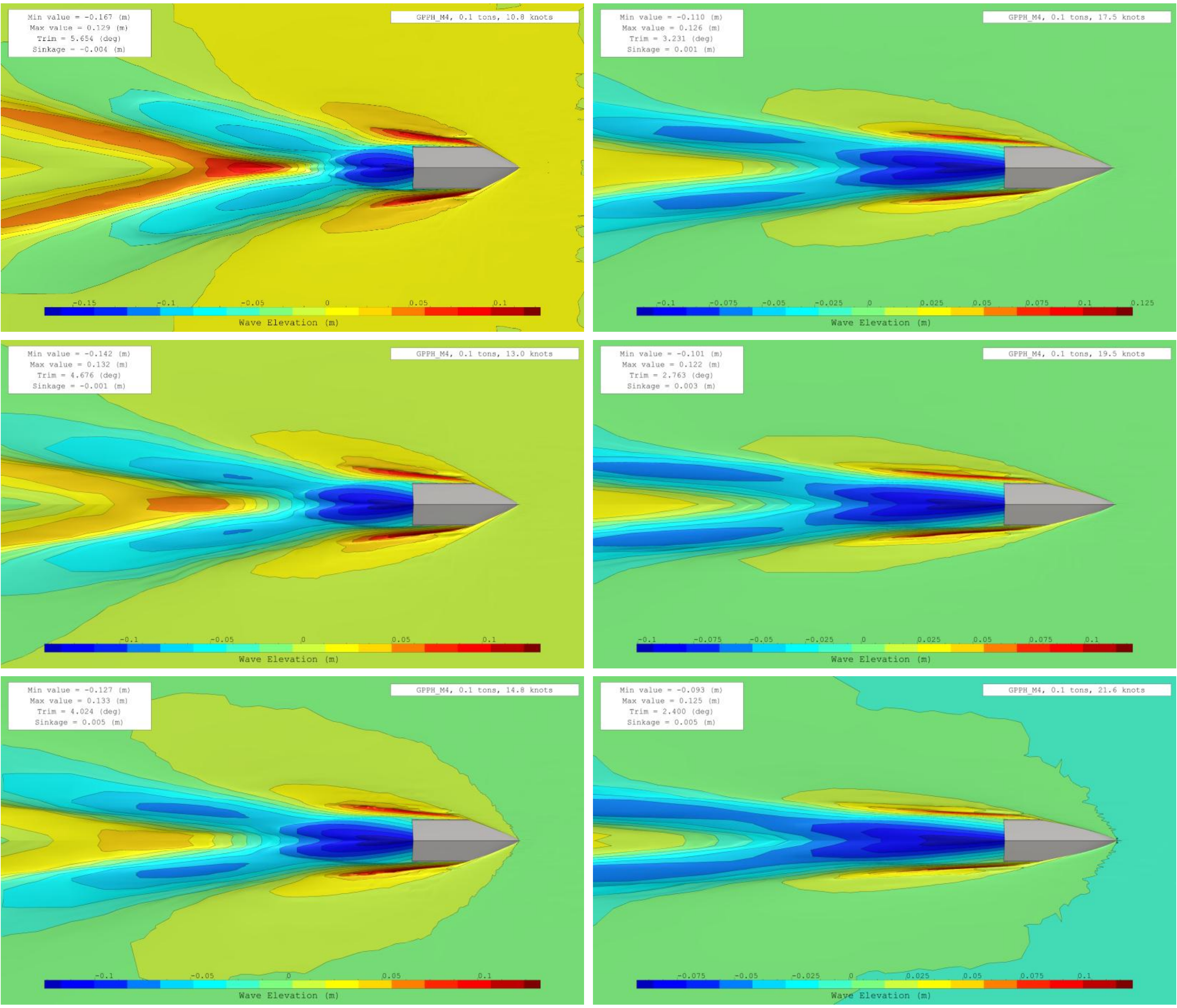


Figure 9: Free surface evolution (independent scale) from 10.80 to 21.60 knots

e. Computational time comparison

Figure 10 compares the computation times, in hours, across different mesh configurations. Since we run simulations on an optimal number of cores determined by the mesh cell count, it is crucial to consider the number of cores used.

Notably, for a medium mesh that produces more than acceptable results, we complete the entire resistance curve calculation in approximately 25 hours, which is exceptionally efficient for performing a total of 7 high-speed craft CFD simulations.

This remarkably low simulation time has been made possible through an in-depth study of various solution methods, which are outlined in a detailed report available on the NepTech website. The optimization of resolution techniques, along with efficient parallel processing strategies, has significantly reduced computational effort while maintaining accuracy, underscoring NepTech's commitment to advancing simulation efficiency in high-performance CFD modelling.

	Speed [knots]	Froude number [-]	● Core [-]	● Computational time [h], GPPH_M4
0	10.80	1.22	24	3.53
1	13.00	1.47	24	3.68
2	14.80	1.67	26	3.43
3	17.50	1.98	26	3.40
4	19.50	2.20	26	3.58
5	21.60	2.44	26	3.43
6	23.80	2.69	26	3.62

Figure 10: Computational time in hours

4. Conclusion

This report presents a validation study conducted to predict the calm-water resistance of the GPPH hull, a planing hull, comparing results obtained using NepTech's digital towing tank with available experimental data from the paper "Experimental Results for the Calm Water Resistance of the Generic Prismatic Planing Hull (GPPH)".

The findings demonstrate a strong correlation between the numerical and experimental results, with:

- A resistance error ranging from -9.14 to -4.32 Newtons, corresponding to -3.39% to +2.68% compared to the experimental measurements.
- A dynamic heave error from –0.008 m to +0.001 m, corresponding to a deviation of -11.28% to +0.73% compared to the experimental measurements.
- A dynamic pitch error from -0.075° to +0.072°, corresponding to a deviation of -2.19% to +3.33% compared to the experimental measurements.

The EFD/CFD differences are certainly due to be attributed to variations in hydrostatic characteristics between the model used for the tank tests and the model applied in CFD calculations.

This report thus confirms NepTech's capability to accurately and efficiently predict the dynamic behaviour of a planing hull. By employing a fully automated digital towing tank and leveraging advanced modelling tools, we conclude that simulations of similar planing flow types will be reliable.

The optimization of simulation methods has led to an exceptionally low simulation time, with the entire resistance curve calculation for 7 high-speed craft simulations completed in 25 hours, demonstrating the efficiency and robustness of NepTech's CFD approach for high-performance planing vessel design.

Bibliography

Lee, E., R. Weil, C., & Fullerton, A. (2017). *Experimental Results for the Calm Water Resistance of the Generic Prismatic Planing Hull (GPPH).* West Bethesda: Naval Surface Warfare Center Carderock Division.