Validation of the overset meshing method

NepTech
Intelligent sea mobility

Version	Date	Written by	Validated by
1	28/03/2025	Tanguy TEULET	Clément ROUSSET

Table of contents

Summary

This report presents a comprehensive validation study of the overset meshing method using NepTech's digital testing basin. The study compares the overset approach to the traditional mesh deformation method on a high-speed regime, covering a Froude number range from 1.2 to 2.7. The findings confirm that the overset method provides accurate results, making it a reliable alternative for cases involving large motions, strong dynamic interactions, or breaking waves, where mesh deformation is impractical. Despite its higher computational cost, the overset approach ensures robust and automated meshing, reinforcing the capabilities of NepTech's digital testing basin for complex CFD simulations.

Nomenclature

- F_n $[-]$, Froude number.
- CFD, Computational Fluid Dynamic.
- EFD, Experimental Fluid Dynamic.
- LCG ; TCG ; VCG $[m]$, coordinates of the centre of gravity: lateral, transversal and vertical.
- V $[m/s]$, ship speed.
- μ $[Pa.s]$, dynamic viscosity.
- ρ $[kg/m^3]$, density.

Figures

Tables

1. Introduction

Context and importance of the overset method

In numerical ship hydrodynamics simulations, the overset mesh method enables the handling of complex configurations involving moving bodies. Unlike traditional meshing techniques that rely on mesh deformation to follow displacements, the overset method uses multiple overlapping meshes, with interpolation ensuring data continuity between them.

The classical mesh deformation approach is efficient for moderate movements, but it reaches its limits in cases where large displacements occur, such as:

- High-speed vessels.
- Seakeeping simulations.
- Extreme scenarios, such as a ship.

In these situations, the overset method becomes essential, as it preserves mesh quality without excessive distortion. However, this comes at the cost of higher computational expense due to the additional calculations required for interpolation and the management of overlapping regions.

Validation of the overset method

One of the major challenges of the overset method lies in interpolation across mesh boundaries, which can introduce uncertainties in the numerical results. Poorly controlled interpolation can lead to errors in the transmission of physical quantities (such as pressure and velocity) between grids, ultimately affecting simulation accuracy and distorting hydrodynamic predictions.

To ensure the reliability of our simulations, it is crucial to validate our overset mesh by ensuring that:

- Interpolation is well-defined, with sufficient overlap to minimize accuracy losses.
- The chosen interpolation scheme provides an optimal balance between accuracy and stability.
- The observed discrepancies between simulations and reference results remain within acceptable margins.

We aim to validate NepTech's digital testing basin by comparing its results with well-established validation cases. This validation will confirm that our overset mesh implementation in NepTech's digital testing basin, is robust and that the automated simulations provide trustworthy and usable results for ship applications.

Implementation of the overset method in Fidelity Fine Marine

In Fidelity Fine Marine, the overset method is handled by the ISIS-CFD solver, which employs interpolation schemes to ensure proper information transfer between meshes. Two interpolation approaches are implemented:

- A least squares method based on a linear polynomial, which ensures formal second-order accuracy but can lead to numerical stability issues when the interpolation stencil is inadequate.
- A distance-weighted interpolation, which is more stable but less accurate.

By default, Fine Marine uses the least squares method, but it automatically switches to distance-weighted interpolation if the interpolation coefficient falls below a user-defined threshold. This approach ensures that numerical stability is maintained throughout the simulation.

2. Calm water

a. GPPH

The Generatic Prismatic Planing Hull (GPPH) hull was designed as a publicly available reference model to support research and development across government agencies, contractors, and academic institutions. Its prismatic design was specifically chosen to represent typical planning hulls while minimizing geometric complexities such as warp, rocker, and curvature in both transverse and longitudinal directions.

The objective of this study is to validate the overset meshing method (referred to as "GPPH_M1" in this report) by comparing it to a more traditional meshing approach, which relies on mesh deformation (referred to as "GPPH_M2") and is recommended whenever feasible.

This study does not aim to compare the results against experimental values, as such a CFD/EFD comparative report is already available on the NepTech website.

Since the simulation parameters are identical for both methods, only the meshing approach differs. Therefore, simulation details will not be discussed here, as they are already covered in the CFD/EFD comparative report available on the NepTech website.

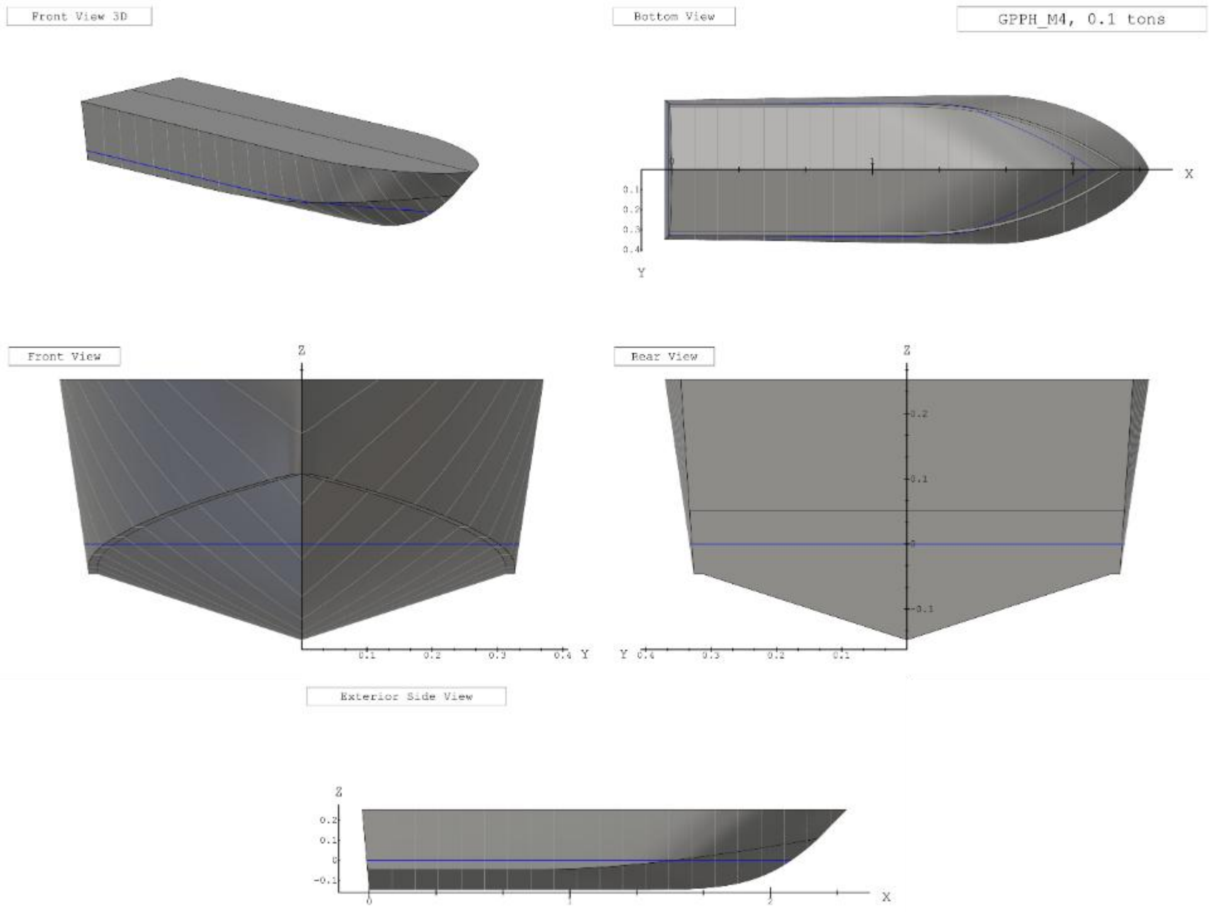


Figure 1: GPPH CAD model

b. 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 2 and 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 and Figure 5 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]	With overset	3.50	3.24	3.18	3.04	3.11	3.06	3.01
	Without overset	1.65	1.61	1.67	1.65	1.72	1.73	1.73

Table 1: Averaged number of cells

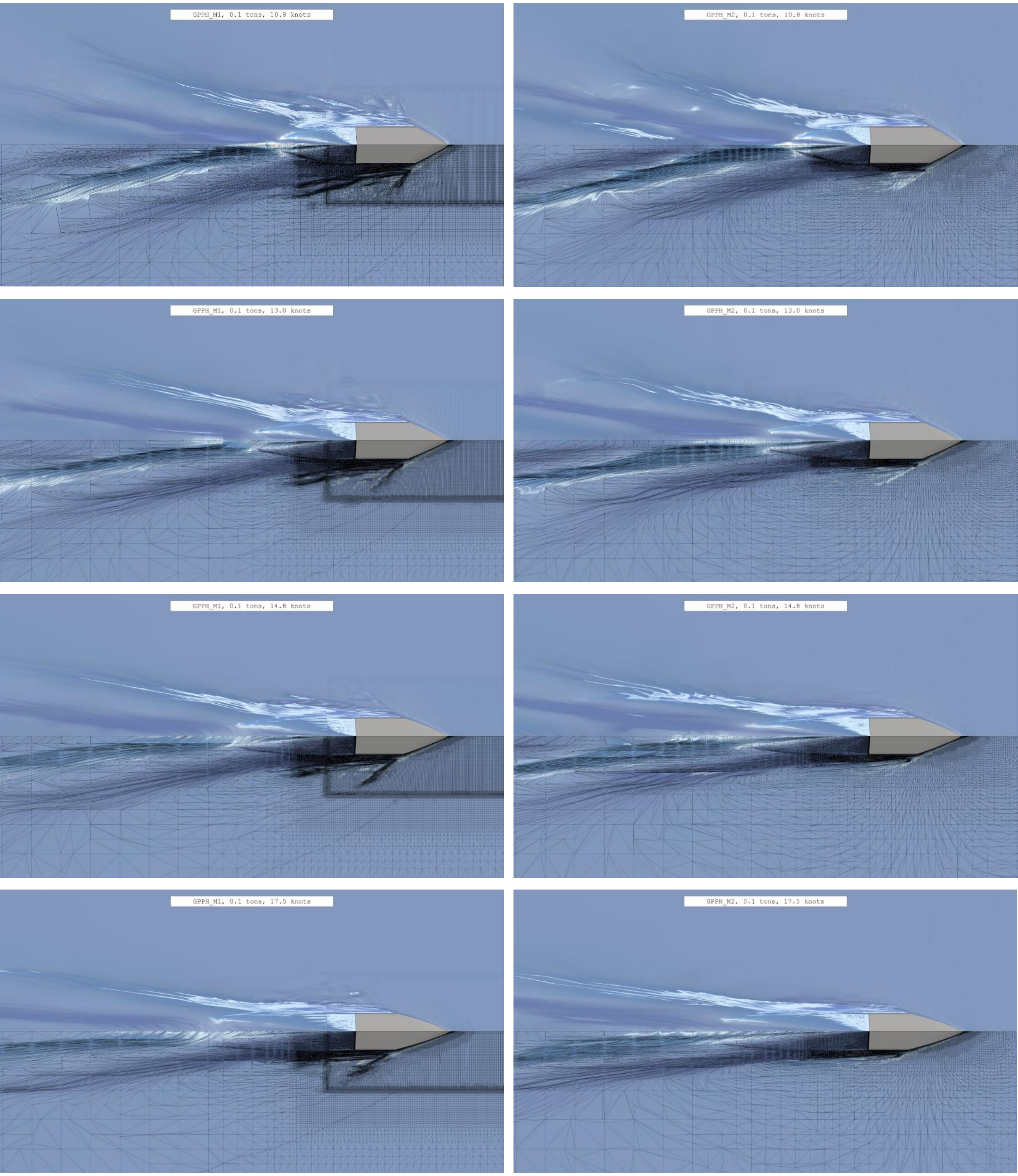


Figure 2: Free surface mesh from 10.80 to 17.50 knots (Right = Overset / Left = No overset)

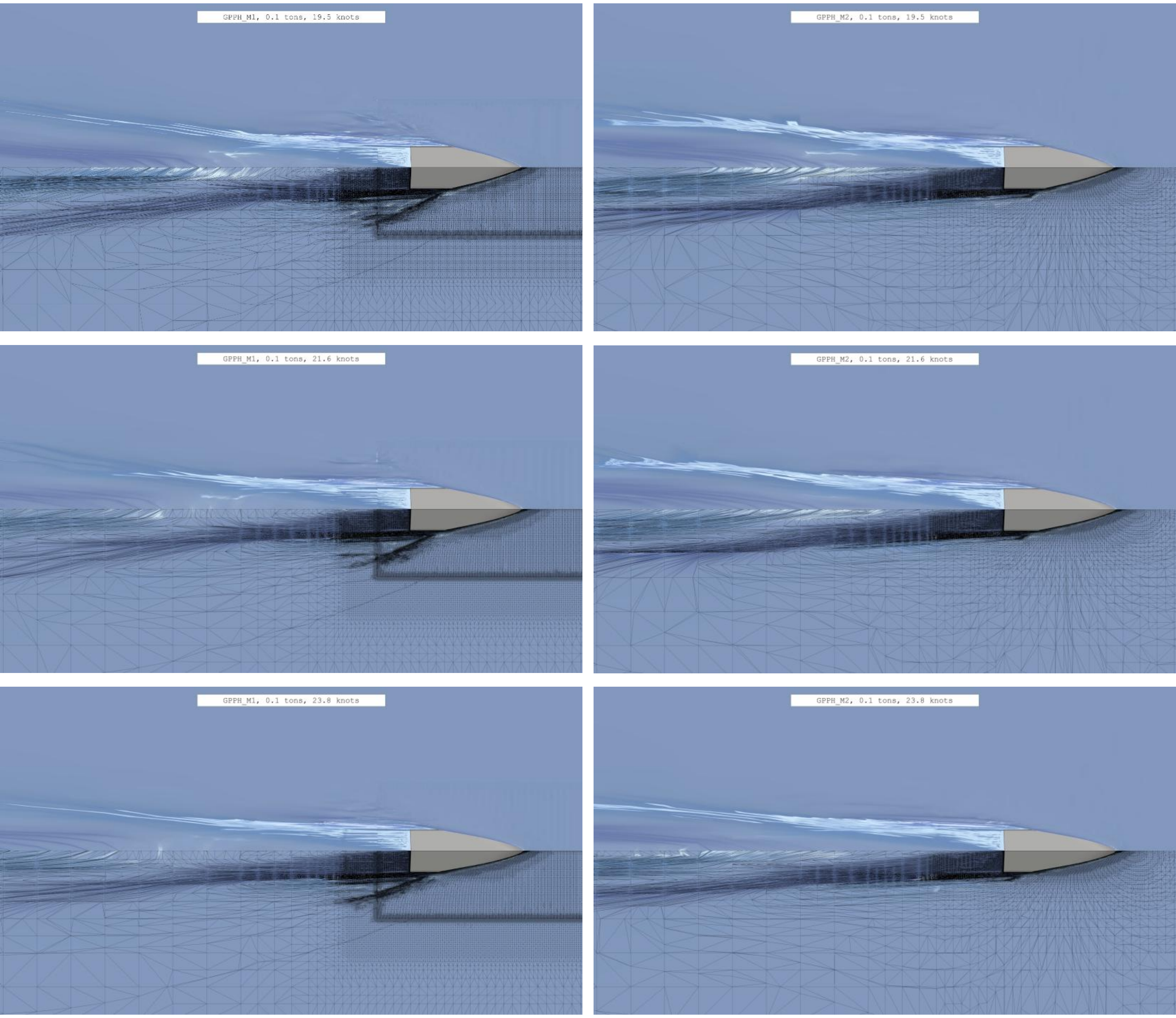


Figure 3: Free surface mesh from 19.50 to 23.80 knots (Right = Overset / Left = No overset)

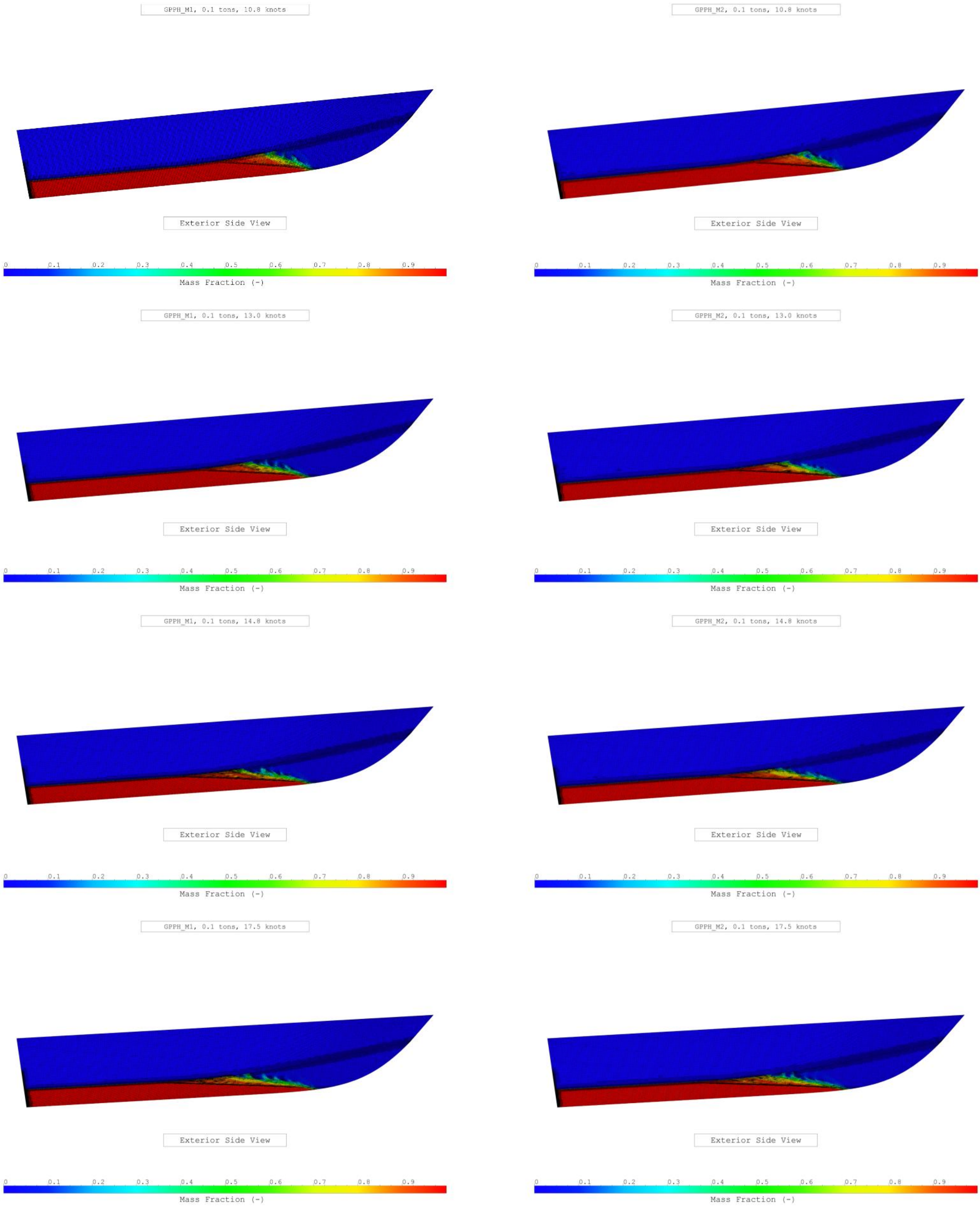


Figure 4: Bare hull mesh from 10.80 to 17.50 knots (Right = Overset / Left = No overset)

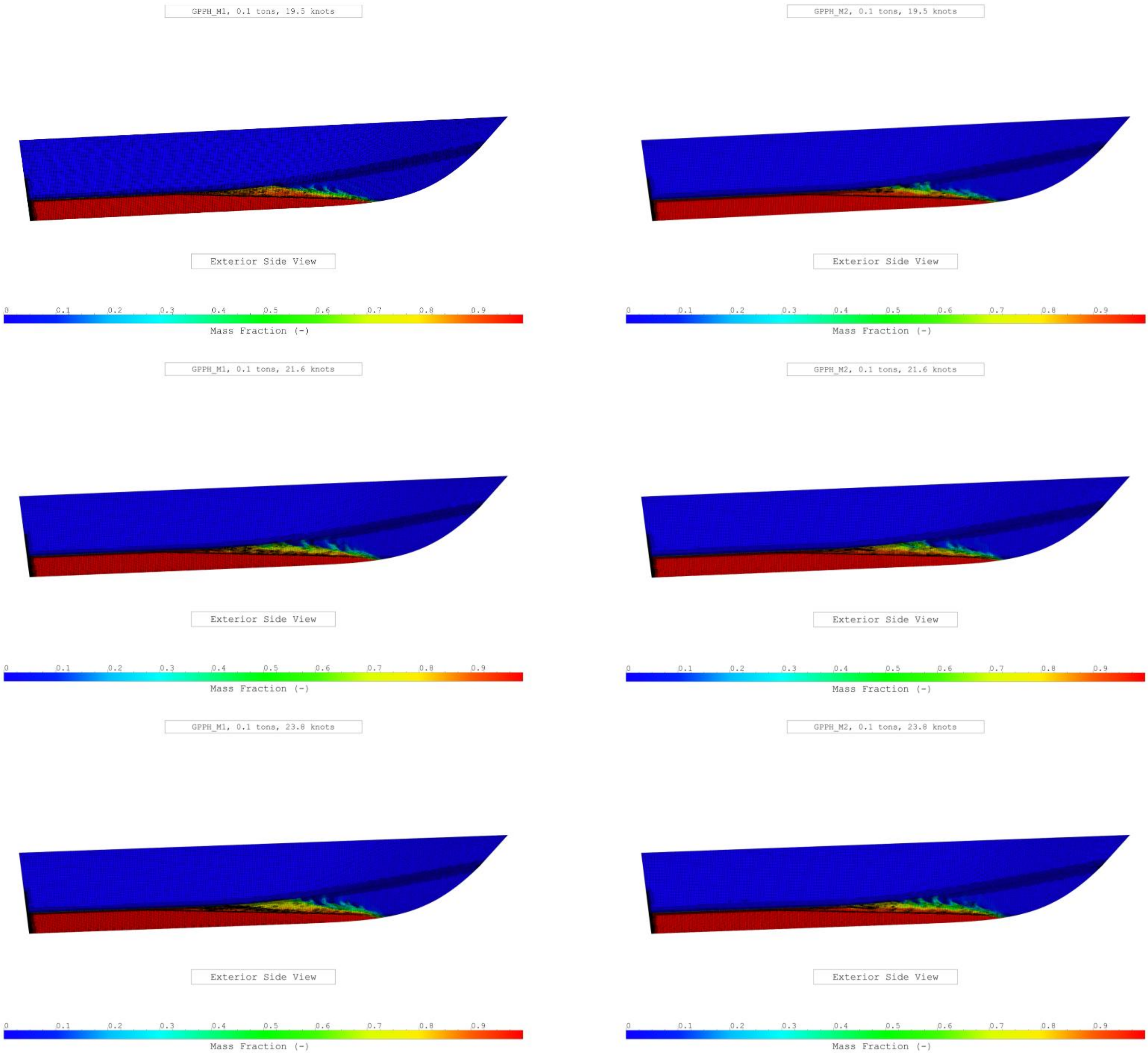


Figure 5: Bare hull mesh from 19.50 to 23.80 knots (Right = Overset / Left = No overset)

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:

<table>
<tr><td colspan="2">Ship speed $\mathbf{V}$ $[\mathbf{knots}]$</td><td>10.80</td><td>13.00</td><td>14.80</td><td>17.50</td><td>19.50</td><td>21.60</td><td>23.80</td></tr>
<tr><td colspan="2">Froude number $\mathbf{F_n}$ $[-]$</td><td>1.22</td><td>1.47</td><td>1.67</td><td>1.98</td><td>2.20</td><td>2.44</td><td>2.69</td></tr>
<tr><td rowspan="2">Averaged Courant number [-]</td><td>With overset</td><td>0.24</td><td>0.21</td><td>0.18</td><td>0.15</td><td>0.15</td><td>0.14</td><td>0.13</td></tr>
<tr><td>Without overset</td><td>0.19</td><td>0.16</td><td>0.13</td><td>0.11</td><td>0.10</td><td>0.10</td><td>0.09</td></tr>
</table>

Table 2: Averaged Courant number (Free Surface)

<table>
<tr><td colspan="2">Ship speed $\mathbf{V}$ $[\mathbf{knots}]$</td><td>10.80</td><td>13.00</td><td>14.80</td><td>17.50</td><td>19.50</td><td>21.60</td><td>23.80</td></tr>
<tr><td colspan="2">Froude number $\mathbf{F_n}$ $[-]$</td><td>1.22</td><td>1.47</td><td>1.67</td><td>1.98</td><td>2.20</td><td>2.44</td><td>2.69</td></tr>
<tr><td rowspan="2">Averaged Courant number [-]</td><td>With overset</td><td>1.33</td><td>1.31</td><td>1.32</td><td>1.29</td><td>1.30</td><td>1.30</td><td>1.32</td></tr>
<tr><td>Without overset</td><td>1.35</td><td>1.34</td><td>1.34</td><td>1.33</td><td>1.30</td><td>1.31</td><td>1.29</td></tr>
</table>

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:

<table>
<tr><td colspan="2">Ship speed $\mathbf{V}$ $[\mathbf{knots}]$</td><td>10.80</td><td>13.00</td><td>14.80</td><td>17.50</td><td>19.50</td><td>21.60</td><td>23.80</td></tr>
<tr><td colspan="2">Froude number $\mathbf{F_n}$ $[-]$</td><td>1.22</td><td>1.47</td><td>1.67</td><td>1.98</td><td>2.20</td><td>2.44</td><td>2.69</td></tr>
<tr><td rowspan="2">Averaged Courant number [-]</td><td>With overset</td><td>120.06</td><td>121.37</td><td>122.11</td><td>123.44</td><td>124.49</td><td>124.47</td><td>124.92</td></tr>
<tr><td>Without overset</td><td>119.95</td><td>121.28</td><td>122.34</td><td>123.66</td><td>123.79</td><td>124.62</td><td>125.16</td></tr>
</table>

Table 4: Averaged Y+

c. Free surface elevations

i. Same scale

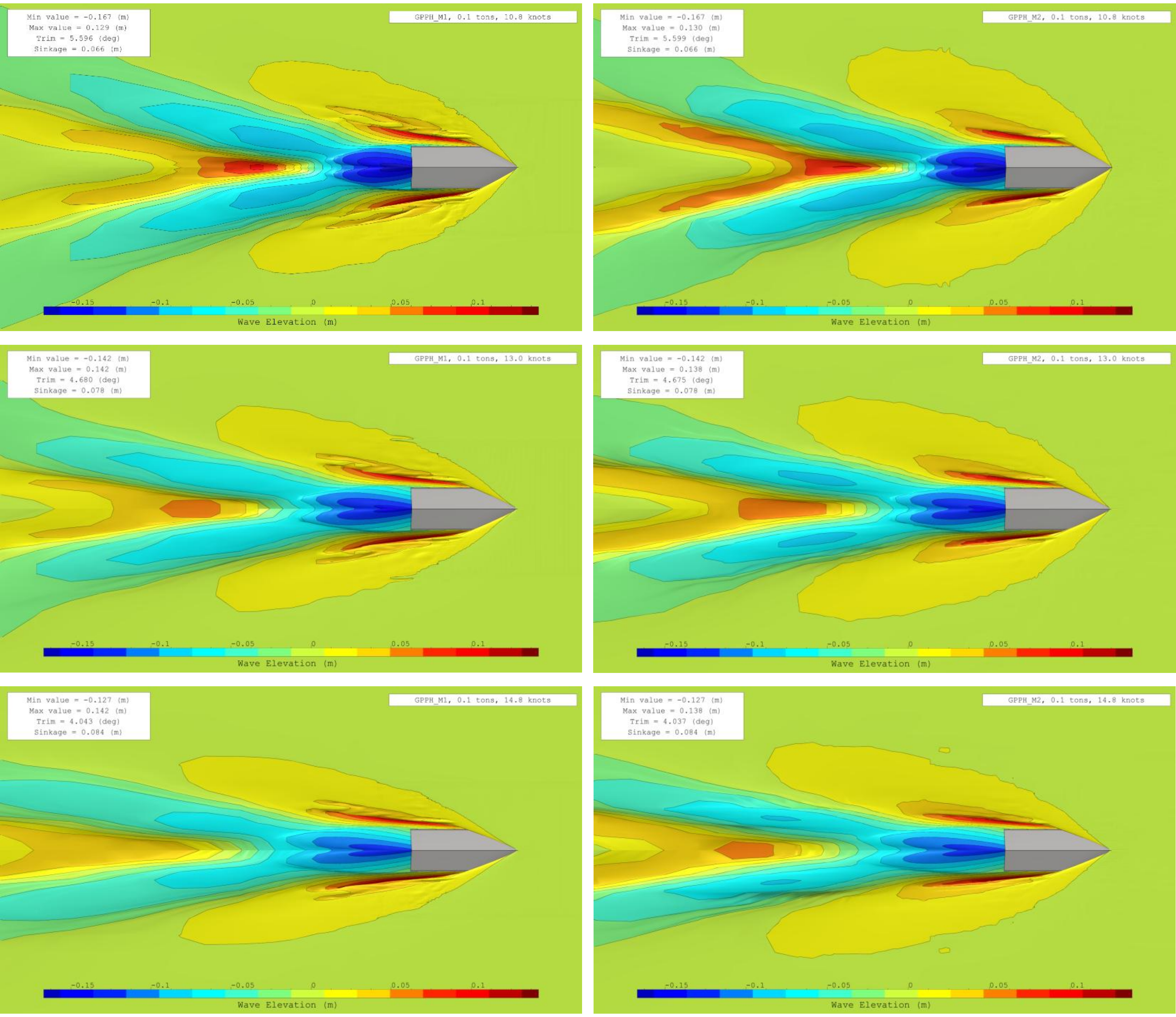


Figure 6: Free surface evolution (same scale) from 10.80 to 14.80 knots (Right = Overset / Left = No overset)

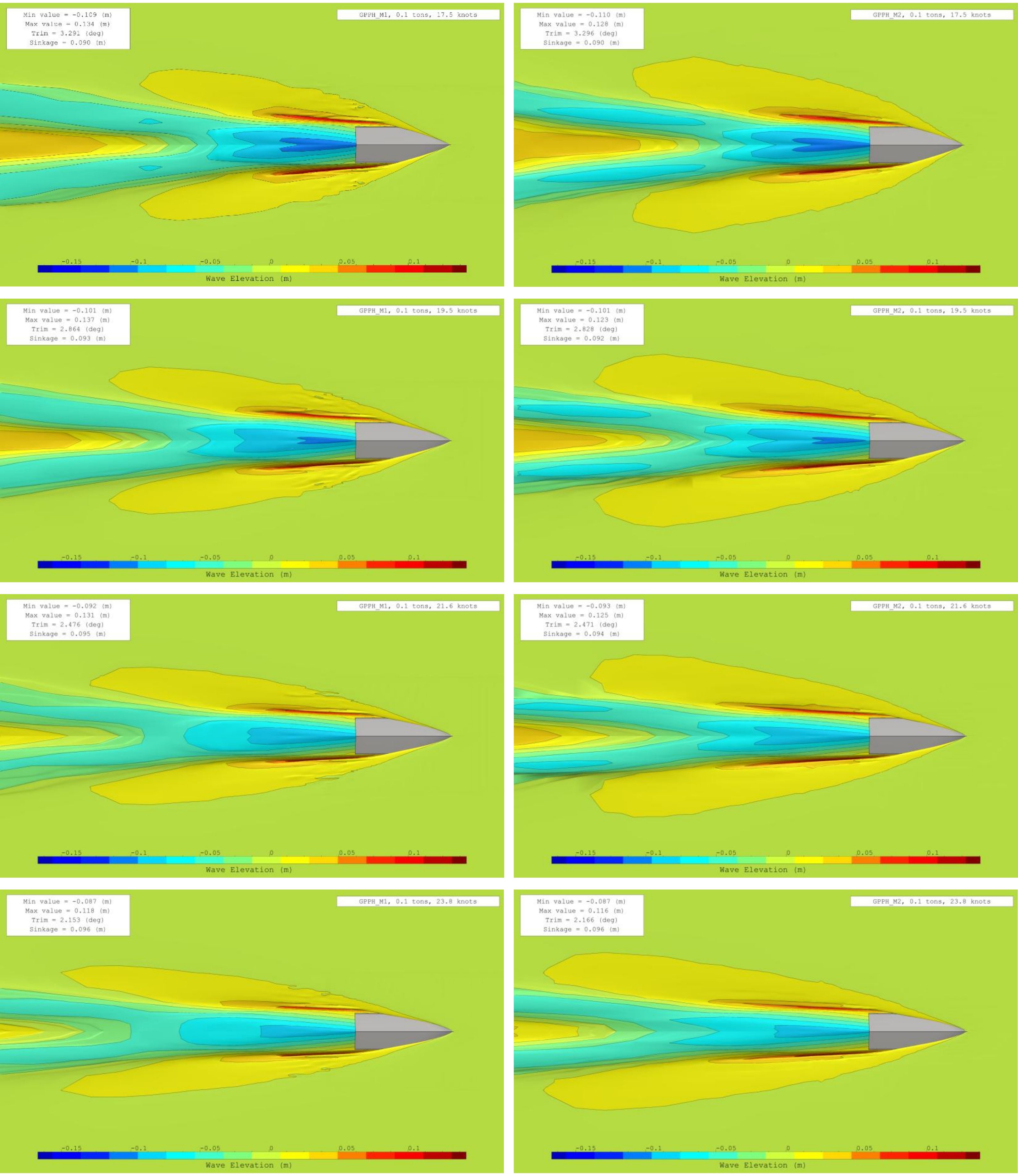


Figure 7: Free surface evolution (same scale) from 17.50 to 23.80 knots (Right = Overset / Left = No overset)

ii. Independent scale

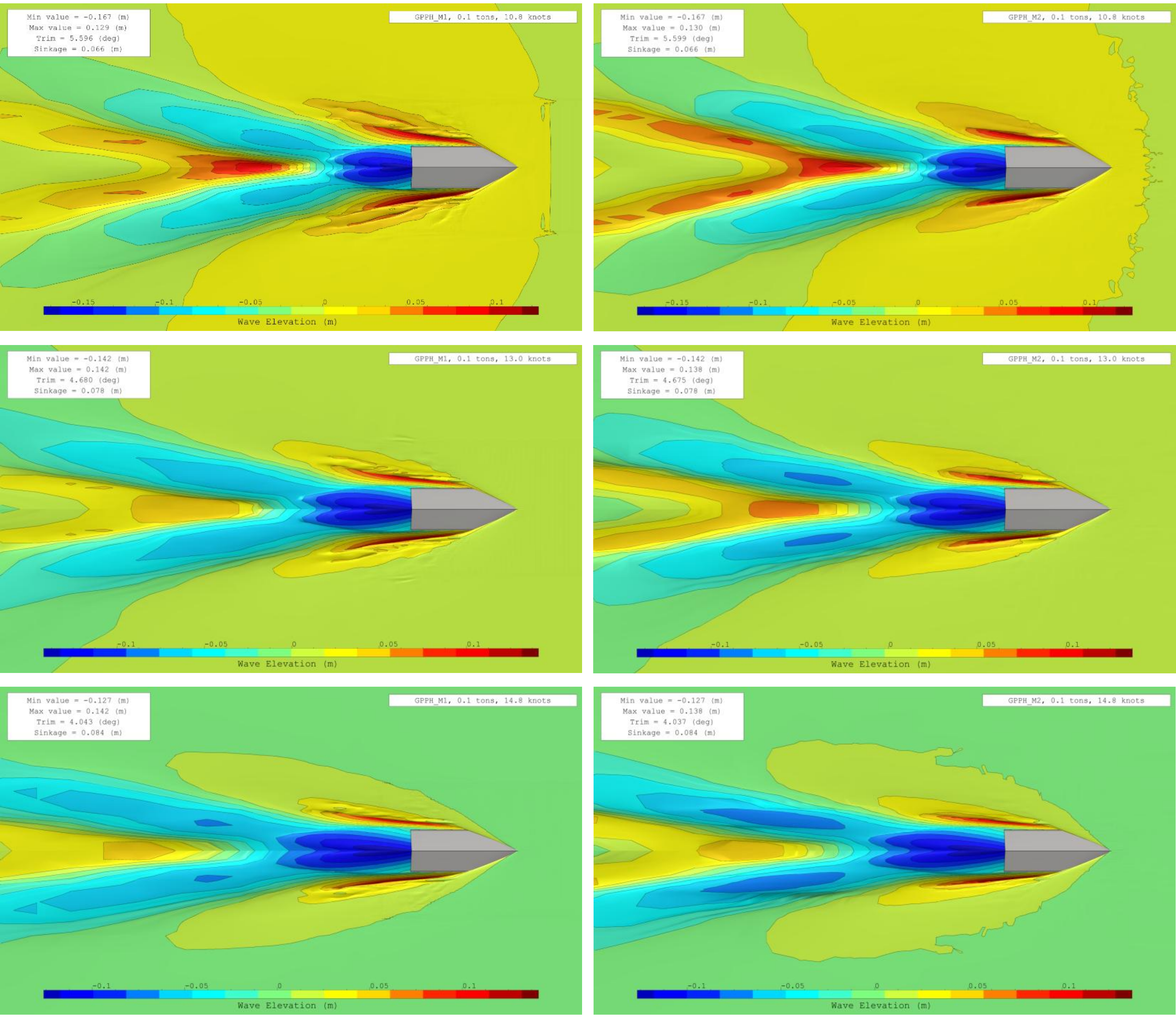


Figure 8: Free surface evolution (inde scale) from 10.80 to 14.80 knots (Right = Overset / Left = No overset)

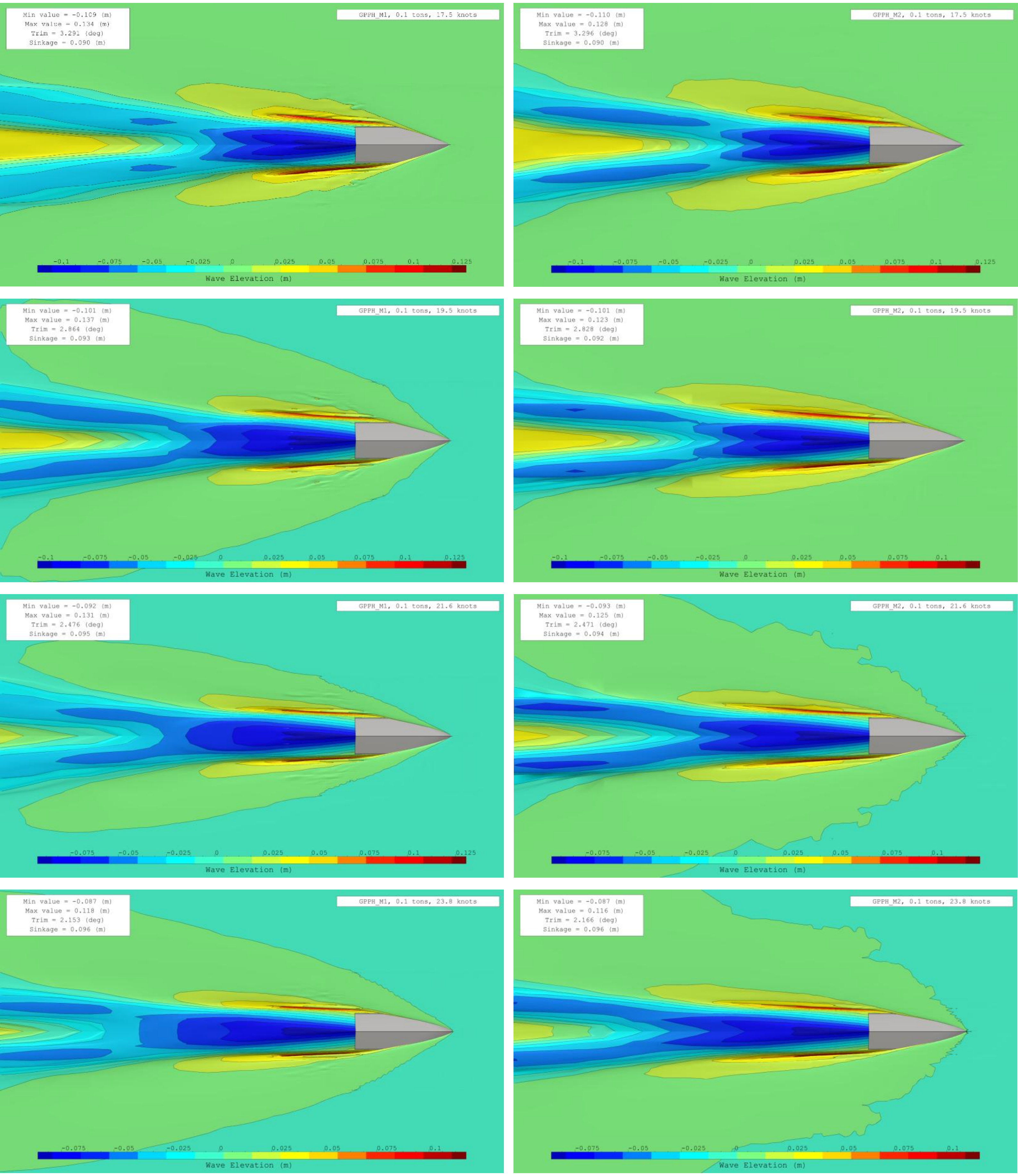


Figure 9: Free surface evolution (inde scale) from 17.50 to 23.80 knots (Right = Overset / Left = No overset)

d. Comparison

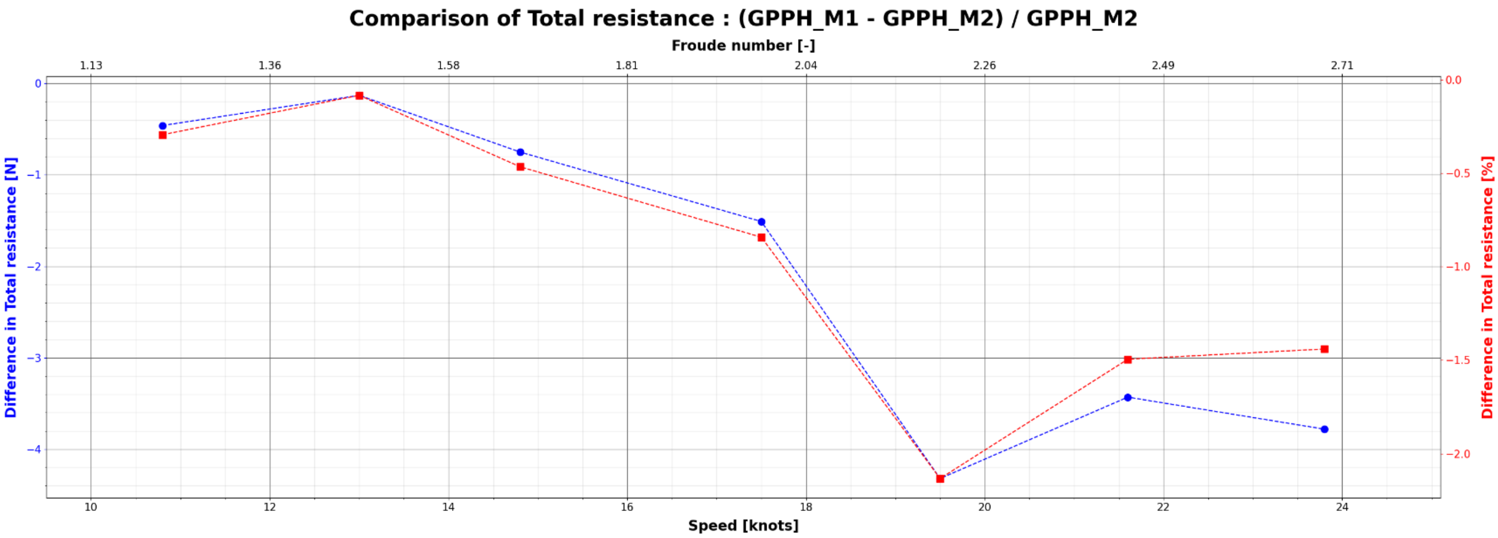
i. Resistance

Figure 10 illustrates the progression of the GPPH resistance across different advance speeds for both methods, along with the absolute difference between the methods in Newtons and the relative difference between the methods as a percentage:

$$E\% = \frac{GPPH\_M1 - GPPH\_M2}{GPPH\_M1} * 100$$

The difference between the two meshing methods is very small, on the order of just a few Newtons, with a maximum deviation of 2.13%. On average, across the seven studied speeds, the difference is approximately -1%.

The overset method tends to produce slightly lower resistance values compared to the mesh deformation method.



	Speed [knots]	Froude number [-]	GPPH_M2	GPPH_M1	Difference [N]	Difference [%]
0	10.80	1.22	156.33	155.87	-0.46	-0.29
1	13.00	1.47	155.16	155.03	-0.13	-0.08
2	14.80	1.67	161.00	160.25	-0.75	-0.47
3	17.50	1.98	179.20	177.69	-1.51	-0.84
4	19.50	2.20	202.50	198.18	-4.32	-2.13
5	21.60	2.44	229.38	225.95	-3.43	-1.50
6	23.80	2.69	262.41	258.63	-3.78	-1.44

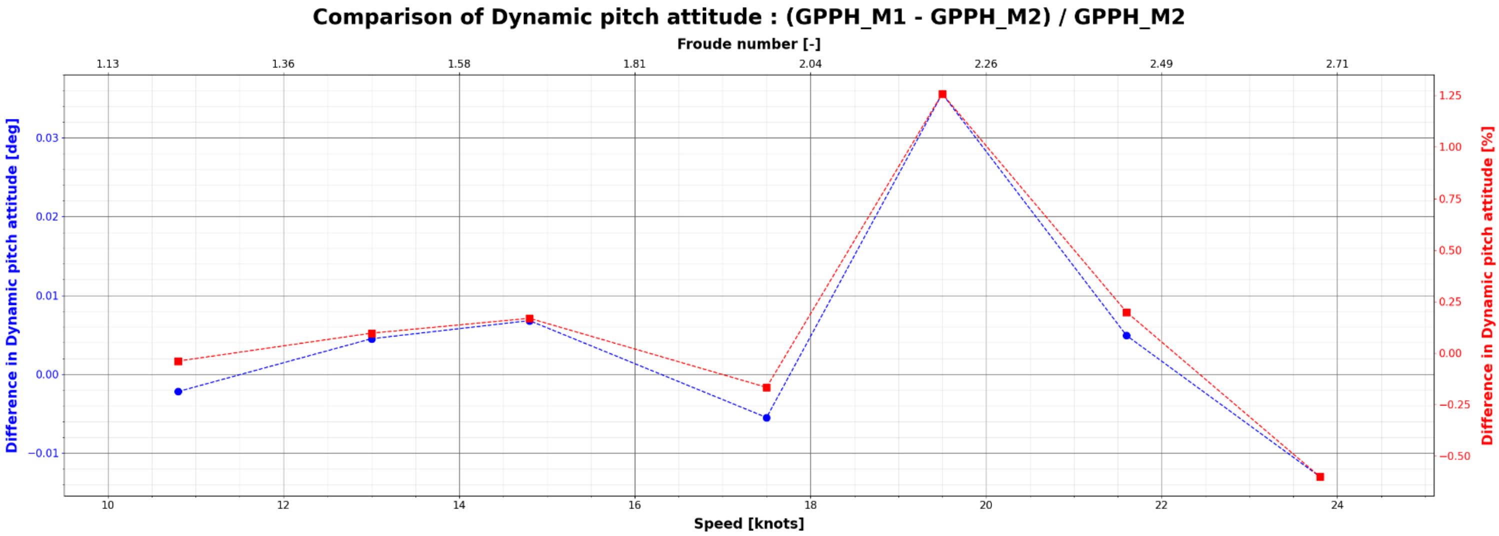
Figure 10: Comparison of total resistance

ii. Pitch

Figure 11 illustrates the progression of the GPPH dynamic pitch response across different advance speeds for both methods, along with the absolute difference between the methods in degrees and the relative difference between the methods as a percentage:

$$E\% = \frac{GPPH\_M1 - GPPH\_M2}{GPPH\_M1} * 100$$

The difference between the two meshing methods is very small, with a maximum deviation of 1.26%. On average, across the seven studied speeds, the difference is approximately +0.13%.



	Speed [knots]	Froude number [-]	GPPH_M2	GPPH_M1	Difference [deg]	Difference [%]
0	10.80	1.22	5.599	5.596	-0.002	-0.04
1	13.00	1.47	4.675	4.680	0.005	0.10
2	14.80	1.67	4.037	4.043	0.007	0.17
3	17.50	1.98	3.296	3.291	-0.006	-0.17
4	19.50	2.20	2.828	2.864	0.036	1.26
5	21.60	2.44	2.471	2.476	0.005	0.20
6	23.80	2.69	2.166	2.153	-0.013	-0.60

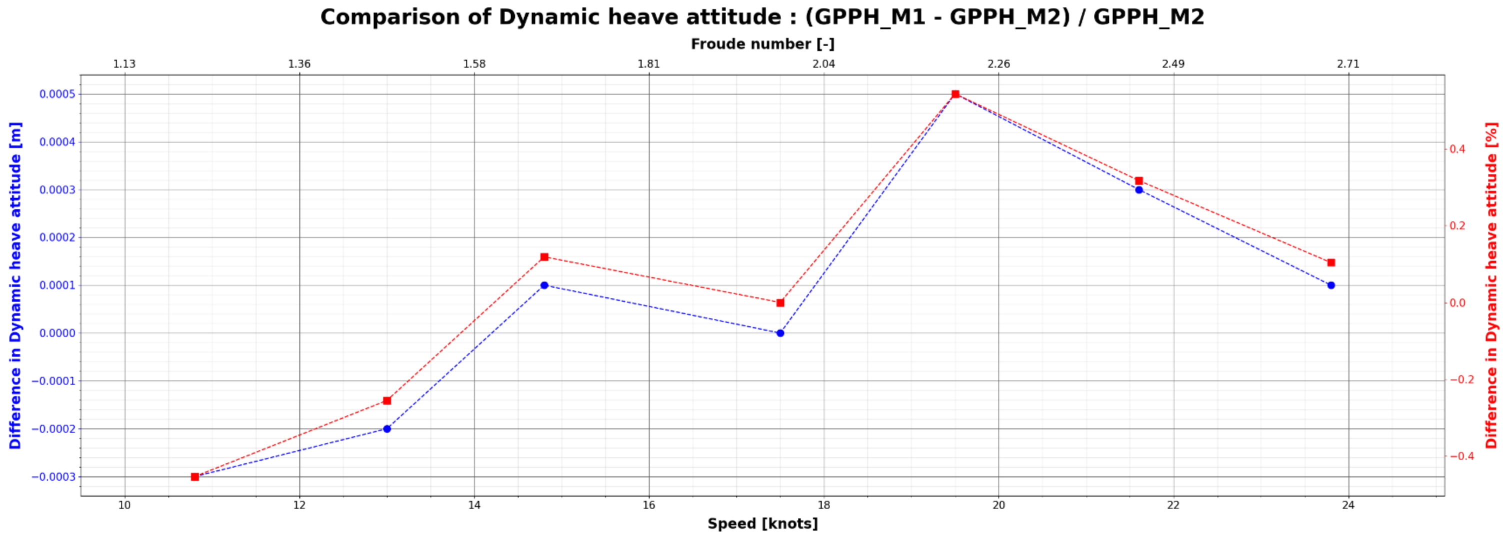
Figure 11: Comparison of dynamic pitch attitude

iii. Heave

Figure 12 illustrates the progression of the GPPH dynamic heave response across different advance speeds for both methods, along with the absolute difference between the methods in meters and the relative difference between the methods as a percentage:

$$E\% = \frac{GPPH\_M1 - GPPH\_M2}{GPPH\_M1} * 100$$

The difference between the two meshing methods is very small, on the order of millimetres, with a maximum deviation of 0.54%. On average, across the seven studied speeds, the difference is approximately +0.37%.



	Speed [knots]	Froude number [-]	GPPH_M2	GPPH_M1	Difference [m]	Difference [%]
0	10.80	1.22	0.066	0.066	-0.000	-0.45
1	13.00	1.47	0.078	0.078	-0.000	-0.26
2	14.80	1.67	0.084	0.084	0.000	0.12
3	17.50	1.98	0.090	0.090	0.000	0.00
4	19.50	2.20	0.092	0.093	0.001	0.54
5	21.60	2.44	0.095	0.095	0.000	0.32
6	23.80	2.69	0.096	0.097	0.000	0.10

Figure 12: Comparison of dynamic heave attitude

iv. Simulation time

Figure 13 compares the computation times in hours. Since simulations are run on an optimal number of cores determined by the mesh cell count, it is essential to consider the number of cores utilized when analysing performance.

The overset mesh method is significantly more computationally demanding, requiring approximately 114 hours to complete simulations for the seven studied speeds, compared to around 48 hours for the mesh deformation method.

Additionally, the overset method is more resource-intensive in terms of core usage, than the traditional mesh deformation approach, further increasing computational costs.

	Speed [knots]	Froude number [-]	● Core [-]	● GPPH_M2	■ Core [-]	■ GPPH_M1
0	10.80	1.22	24	6.63	32	16.68
1	13.00	1.47	24	6.48	32	18.00
2	14.80	1.67	26	6.80	32	15.92
3	17.50	1.98	26	6.67	32	15.48
4	19.50	2.20	26	6.93	32	15.78
5	21.60	2.44	26	7.00	32	16.25
6	23.80	2.69	26	7.27	32	15.90

Figure 13: Computational time in hours

3. Conclusion

This study aimed to validate the overset meshing method by comparing it to the traditional mesh deformation approach for simulating the Generic Prismatic Planing Hull (GPPH). The results confirm that both methods produce very similar hydrodynamic predictions, with only minor differences in resistance, pitch, and heave across all tested speeds.

The resistance comparison revealed a maximum deviation of 2.13%, with an average difference of -1.00%, indicating a slight tendency of the overset method to predict lower resistance values. Similarly, the pitch and heave variations remained minimal, with maximum deviations of 1.26% and 0.54%, respectively. These results demonstrate that the overset meshing approach provides accurate and reliable predictions comparable to the mesh deformation method.

However, the study also highlights the significant computational cost associated with the overset method. Simulation time increased from 48 hours (mesh deformation) to 114 hours (overset), and the overset approach required more computational cores due to the higher cell count. This increased cost must be carefully considered when choosing the most suitable meshing strategy.

Overall, these findings confirm that the overset meshing method implemented in NepTech's digital testing basin is accurate enough and can be reliably used in cases where large motions, breaking waves, or strong dynamic interactions make mesh deformation impractical. Despite its higher computational cost, the overset approach ensures robust and automated meshing capabilities, making it a valuable tool for complex CFD simulations.