

Free Surface Meshing and Domain Effects in LBM-Based CFD Simulations of DeepCWind Heave Plate Free Decay Tests

Miguel Gil¹, Alexia Torres¹, Juan Pablo Fuertes¹, Javier León¹

¹Public University of Navarre

Campus de Arrosadía s/n, Pamplona, Spain

miguel.gil@unavarra.es; alexia.torres@unavarra.es; juanpablo.fuertes@unavarra.es; javier.leon@unavarra.es

Abstract - This study investigates strategies to reduce computational cost in Computational Fluid Dynamics (CFD) simulations of free decay motion in floating platforms. Using the Simulia XFlow™ solver, a full-scale heave plate of the DeepCwind semi-submersible platform was analyzed under heave free decay conditions. Different mesh refinement levels (4, 5, and 6 levels) and channel lengths (1D, 2D, 3D) were tested to evaluate their influence on numerical stability, hydrodynamic force accuracy, and energy dissipation. The results reveal that simulations with 5 and 6 mesh levels introduce significant spurious velocities at the free surface, leading to unphysical amplitude growth and negative damping. In contrast, the 4-level mesh combined with an extended channel (3D) minimizes wave reflection and numerical instabilities, achieving accurate damping representation while reducing the computational cost by 74.7% compared to a high-resolution reference simulation. The proposed hybrid meshing strategy using localized refinement near the free surface proves effective in balancing accuracy and efficiency, making it suitable for preliminary design stages. The study also highlights the impact of simulation startup conditions on the initial cycles, suggesting future work to mitigate associated errors.

Keywords: free decay, CFD, free surface, heave plate

1. Introduction

In recent years, the offshore industry has been working both in the field of floating offshore wind energy and oil rigs, developing floating structures in deep water environments. In particular, in floating wind farms, the cost of the substructure represents a high percentage of the total budget, so the optimization of the structural design aimed at cost reduction and dynamic performance improvement is a key objective in this industry [1][2].

One of the fundamental aspects in the dynamic analysis of floating structures is the determination of their natural frequencies and hydrodynamic damping coefficients. For this purpose, it is common to perform free decay tests mainly in the heave, pitch and surge modes of motion. The numerical simulation of these tests provides relevant data for the dynamic characterization of the platform [3].

The presence of the free surface, air-water interface, plays a key role in such simulations. The oscillations of a floating body induce the radiation of surface waves, generating additional hydrodynamic loads associated with the added mass and radiation damping. These forces have a significant impact on the motion of the structure, which requires a correct modeling of the free surface in Computational Fluid Dynamics (CFD), since the proper representation of buoyancy, restoring and damping forces depends on it.

In offshore wind, linear potential models are traditionally used. However, certain modes of motion of the platforms present a damping strongly influenced by viscous effects, making CFD simulations have emerged as a complementary alternative capable of providing these data with greater speed and flexibility in the face of design variations [4].

In the field of computational tools, several CFD codes are available, both commercial and open source. Among the latter, OpenFOAM has established itself as one of the most widely used in offshore research, thanks to its openness and its ability to solve two-phase flows using the Volume of Fluid (VOF) method coupled to floating body dynamics with six degrees of freedom [4]. Numerous works have used OpenFOAM to simulate semi-submersible platforms under free decay conditions, validating their predictions against experimental data from the DeepCwind model, as was done in the OC5 and OC6 projects [1].

As for commercial codes, ANSYS Fluent and Star-CCM+ are widely used in the industry [2]. The latter has been employed in free decay simulations of semisubmersibles platforms, obtaining reasonable agreement with experimental data, especially with the DeepCwind model of the OC4 project. Another relevant software is XFlow, based on the

Lattice-Boltzmann method (LBM) with particle focusing, which has advantages in the treatment of large motions. Recent studies have compared XFlow and OpenFOAM in free decay studies, taking advantage of the complementary strengths of each approach [4].

Validation of free decay CFD simulations is commonly performed by comparison with physical tests in experimental facilities, mainly surge tanks [5]. In these, scale models of the platform are used, properly calibrating its buoyancy and mooring system. The tests consist of moving or tilting the model in a given degree of freedom and recording its oscillatory response using sensors or optical motion capture systems [6]. For example, in the OC6 Phase Ia project, experimental tests were performed with the DeepCwind semi-submersible at 1:50 scale, including free pitch, roll, and surge decay tests, and the results were used to validate multiple CFD simulations [1]. Similarly, experimental data from the OC5 project have been widely used as a reference in the validation of numerical models.

In general, recent studies show acceptable qualitative agreement between CFD simulations and physical tests, both in terms of oscillation frequency and damping rates [4]. Numerous publications in the last five years present detailed comparisons between the two approaches, supported by open databases established in international projects such as OC4, OC5 and OC6 of the IEA Wind program.

Despite advances, free-decay CFD simulations on floating platforms still face significant challenges. Prominent among them is the high computational cost: a high-fidelity three-dimensional simulation can require on the order of hundreds of thousands of CPU hours, compared to potential models that consume up to 100,000 times fewer resources.

Another major challenge is the adequate resolution of free surface and local phenomena. Accurately capturing the air-water interface requires vertically refined grids, even if the waves generated in the free decay have low amplitude. In addition, it is necessary to extend the computational domain or incorporate effective absorption regions to avoid interference from reflections. Similarly, accurate representation of areas with viscous separation or vortex generation requires fine meshing, low y^+ values and, in many cases, the use of calibrated wall functions, which introduces additional uncertainties.

Finally, a lack of systematic quantification of numerical uncertainty has been identified in many previous studies. In [5] they addressed this point through a campaign of 20 simulations varying mesh refinement and time step, concluding that discretization uncertainty dominates the total error, while iterative uncertainty turns out to be negligible once residuals of the order of 10^{-8} are reached.

In summary, while free decay CFD simulations have proven to be robust tools for validating and analyzing the dynamic behavior of floating structures, they still face significant challenges in terms of computational cost, spatial resolution, representation of viscous effects, coupled modeling, and numerical verification. In this paper, Simulia XFlow is employed in the study of different (meshing) strategies and free surface approaches at the air-water interface that allow to decrease the number of lattice elements without losing damping accuracy and making it computationally faster.

2. Load case description

The study focuses on a 3 m heave free decay test applied on a full-scale heave plate of the DeepCWind platform. It is composed of a base cylindrical column of 24 m diameter and 6 m high, and an upper column of 12 m diameter and 24 m high, with a total mass of 4,405e6 kg. Only the heave degree of freedom is enabled to completely isolate the motion and avoid coupling effects.

The mesh consists of a first layer (Level 0) with a target element size (TRS) at the periphery of the geometry of 0.22 m and which is kept constant throughout the study. Increasing the number of mesh levels allows reducing the number of elements and therefore the computational cost, although it may alter the results. The study alternates meshes composed of 4, 5 and 6 levels, which alters the coarse mesh size (RS) and which are quoted in the results according to the nomenclature N_x , where x defines the number of levels. The time step referred to level 0 is 4.68 ms and remains constant for all the meshes studied.

The domain length is responsible for the appearance of unwanted reflection waves that alter the behavior of the heave plate. Three domain lengths defined according to the nomenclature $1D=309.76$ m, $2D=422.4$ m and $3D=535.04$ m are studied. The domain height remains constant and equal to 239.36 m, and the depth is equal to the length to ensure symmetry.

Finally, an overlapping region is introduced in the free surface and centered with respect to the geometry with an element size of 0.44 m. This has a dimension of 35.2 x 35.2 x 3.52 m. It is intended to correctly capture the radiation effects present at the air-water interface, keeping a limited number of elements and reducing the computational cost. The results of these simulations are compared with respect to a base model (ref) where the free surface is meshed with a size of 0.44 m over the entire length and depth of the domain, maintaining the same layer height. Fig. 1 shows an overview of the meshes used, as well as the domain dimensions.

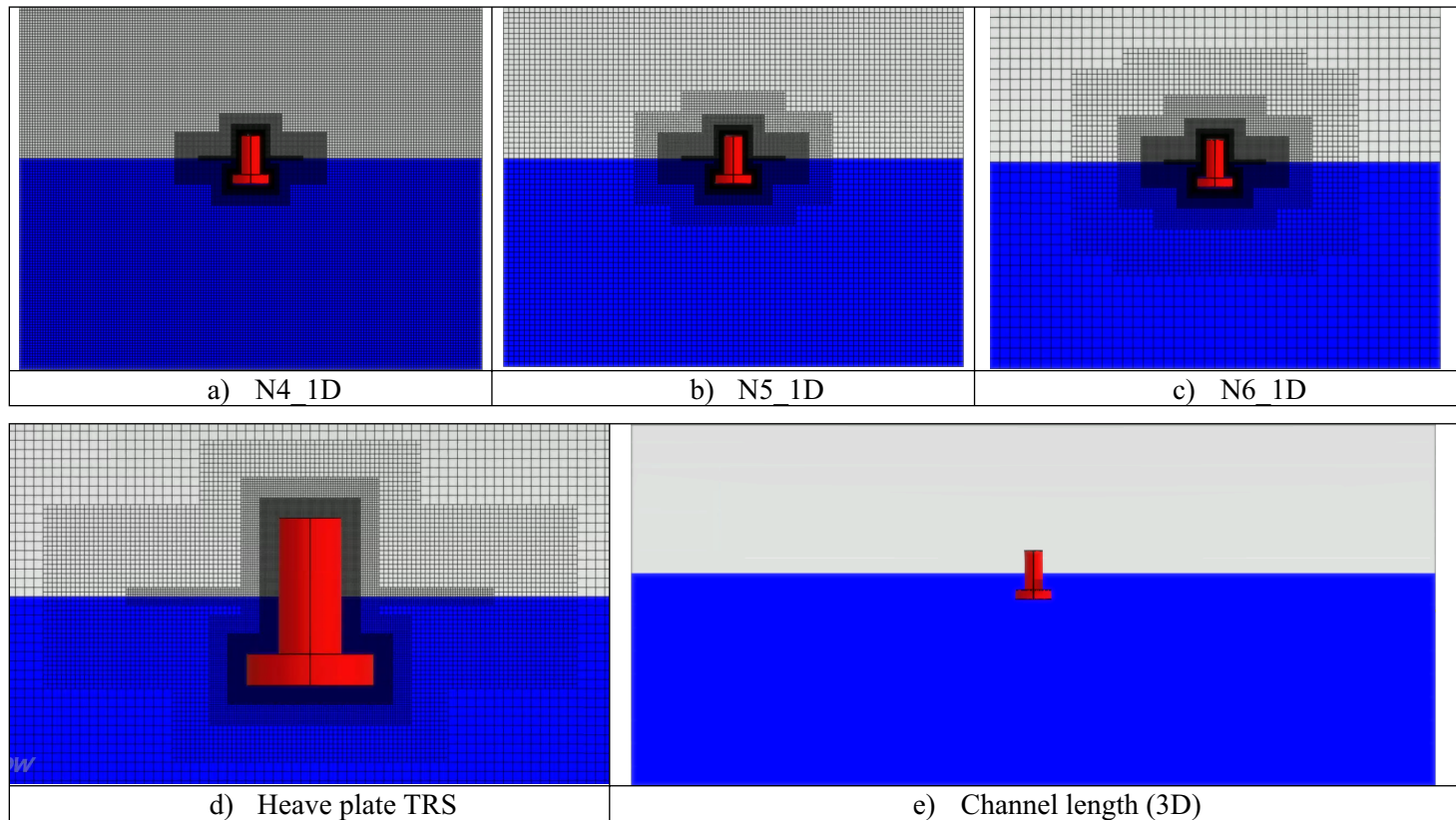


Fig. 1: Domain structure for simulations of a) 4 levels, b) 5 levels and c) 6 levels. Detailed view of the mesh structure (d) and channel dimension in simulation with domain 3 (e).

The properties of the mesh and the domain of the simulations are shown in Table 1, as well as the total simulation time.

Table 1: Grid properties for all simulation configurations.

Simulation name	Resolved scale (m)	Grid Levels	Target resolved scale (m)	Wake refinement (m)	Domain length (m)	Remeshing region length (m)	Total number of cells in Million	Simulation time
N4-1D	1.76	4	0.22	-	309.76	35.2	7.15	68 h 18 min
N5-1D	3.52	5			309.76		3.65	64 h 10 min
N6-1D	7.04	6			309.76		3.29	61 h 4 min
N4-2D	1.76	4			422.4		10.8	72 h 20 min
N5-2D	3.52	5			422.4		4.11	64 h 5 min

N6-2D	7.04	6			422.4		3.35	65 h 38 min
N4-3D	1.76	4			535.04		15.57	80 h 38 min
N5-3D	3.52	5			535.04		4.71	62 h 29 min
N6-3D	7.04	6			535.04		3.42	58 h 47 min
N4-3D-ref	1.76	4			535.04		27.93	318 h 7 min
				0.44		-		

3. Results

Fig. 2 shows first the results obtained for the 9 simulations arranged in such a way that the effect of the number of levels as a function of the domain size can be analyzed. The decrease in amplitude of the heave motion is analyzed for each half-cycle drop. First of all, the appearance of cycles with negative amplitude decrease is highlighted, especially in the simulations performed with 5 and 6 grid levels. This implies an increase in the total height of the platform with respect to the draft point, which is not realistic from a purely energetic point of view since it implies the appearance of positive damping hydrostatic forces. This effect is caused by the occurrence of spurious velocities at the free surface, as will be explained later. Simulations performed at N4 experience fewer cycles with amplitude amplification, and the reduction is noticeable as the channel dimension increases. This effect is due to the reflection of the wave on the channel walls, so this effect is practically dissipated in the 3D channel.

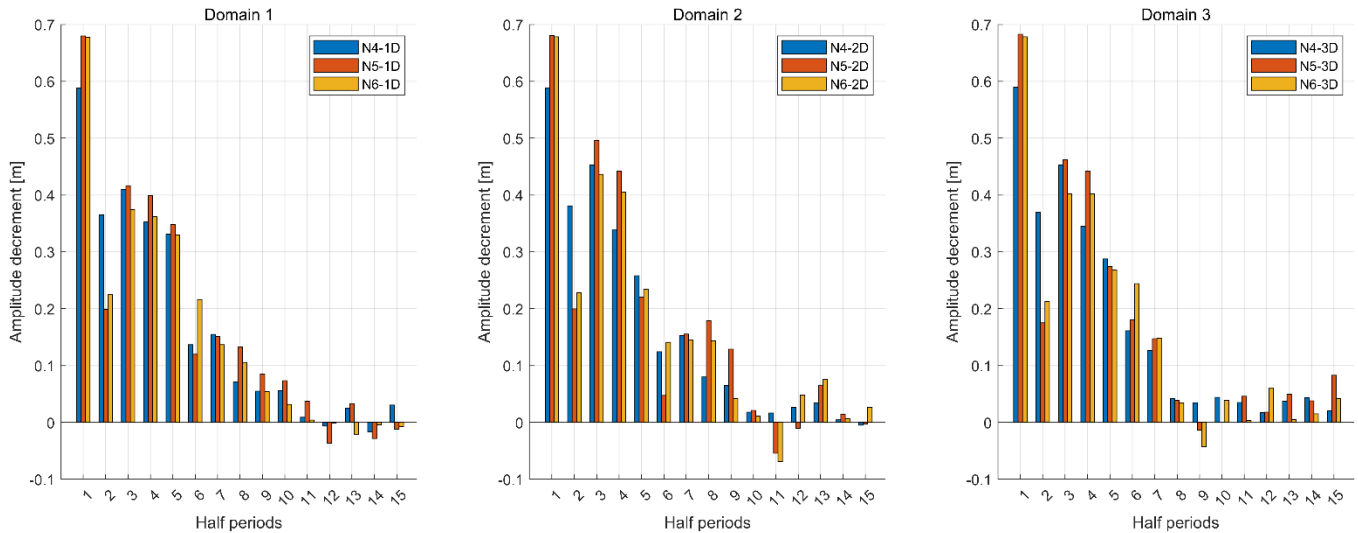


Fig. 2: Decrease in amplitude per half-cycle for 1D (left), 2D (middle) and 3D (right) simulations

The amplitude decay in the simulations is shown in Fig. 3, so that the effect of the channel increase can be analyzed as a function of the number of mesh levels. As for the simulations with 4-level mesh, it is observed how the half-cycles with negative decrement are reduced as the channel dimension increases, and is completely eliminated in the N4-3D case for the simulation time studied. This reinforces the theory that this effect is a consequence of the occurrence of reflection effects and the increase of the channel length slows down their occurrence up to the study area. The increase in the domain implies an increase in the total number of elements. However, these are in the last layer, so this increase does not overly affect the computational cost, which is 4 hours for the N4-2D case and 8 hours for the N4-3D case, with respect to a total simulation time of 68 hours for N4-1D.

As for the 5 and 6 level meshes simulations, negative half-cycles are observed regardless of the number of mesh levels and channel size.

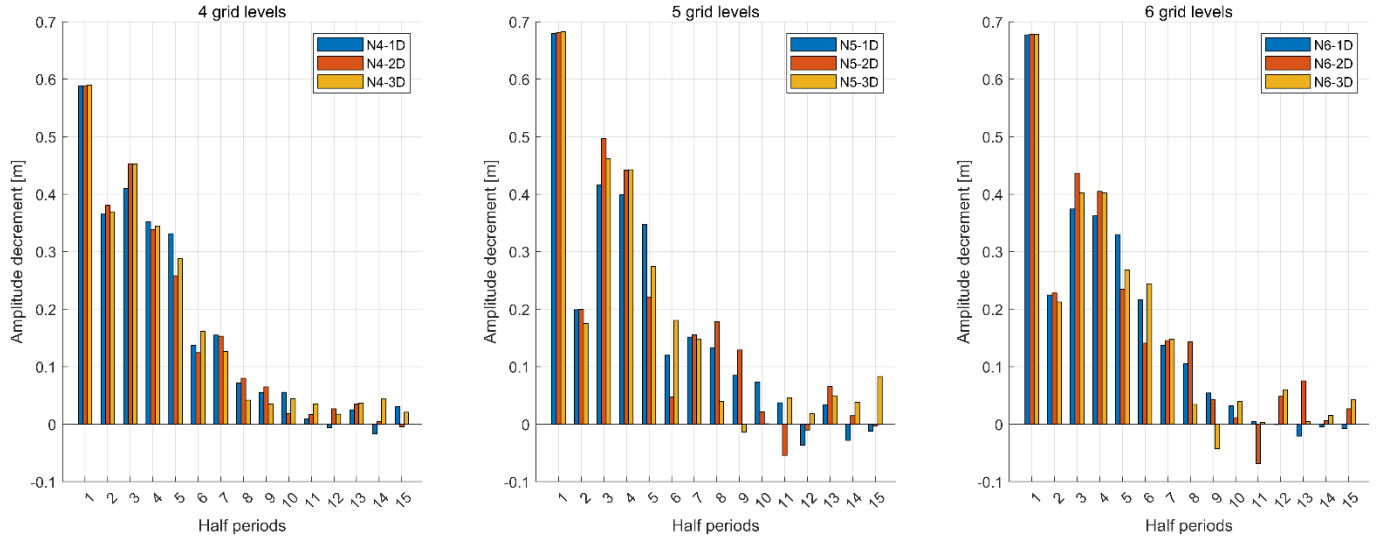


Fig. 3: Decrease in amplitude per half-cycle for simulations with N4 (left), N5 (middle) and N6 (right)

The decrease in amplitude is a consequence of energy dissipation by hydrodynamic damping forces [5], so its analysis provides information on the effects that occur in the simulation. Fig. 4 shows the evolution of the hydrodynamic forces present in the N4 simulations. It shows flattening of the sinusoidal peaks, which causes a distortion in the platform position and is related to the appearance of spurious velocities and radiation reflection effects. These dissipate as the channel length increases, delaying their appearance and reducing their intensity, so the N4-3D simulation seems to be the one that best represents reality.

Finally, it should be noted that the first cycle departs a lot from the sinusoidal shape, which induces an error at the start of the simulation. This is due to problems in the start-up that should be studied in depth, since it implies an error that affects the rest of the simulation. In further studies we intend to implement techniques to eliminate this effect at start-up and to analyze the overall computational cost with respect to the reference simulation.

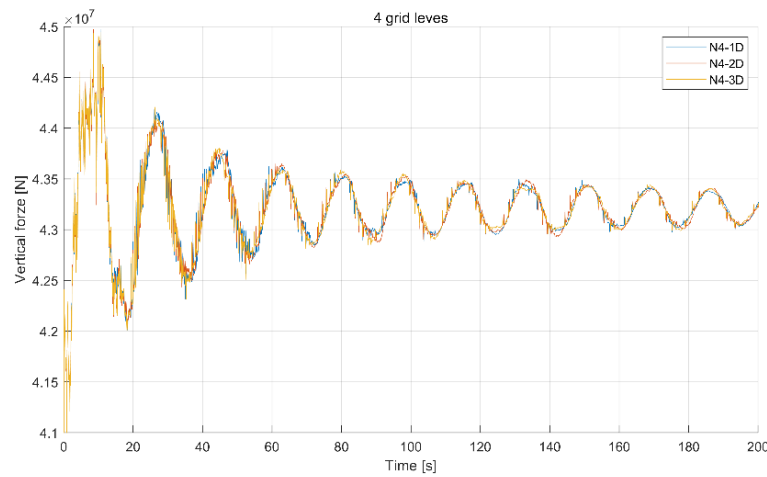


Fig. 4 Hydrodynamic forces in simulations with 4 mesh levels.

Fig. 5 shows the air-water interface at 50 s and 150 s of simulation for the reference model and the models with 4 and 6 mesh levels. The introduction of 4, 5 and 6 mesh levels causes a numerical instability in the interface zone, which can affect more or less depending on the length of the transition, as well as the number of levels in the mesh. This numerical instability results in the appearance of unreal, spurious velocity in the fluid. The first conclusion to be drawn is that a high number of levels, 5 and 6, cause instabilities both in the free surface and in the rest of the water volume. Therefore, these models are discarded since these instabilities can significantly influence the free motion of the heave plate producing an added numerical damping that, depending on how it is superimposed on the heave plate itself, can cause both over- and under-damping. However, in the 4-level mesh model it is observed that although the air-water interface is affected in the area of mesh size transition, the rest of the water volume is not affected. Likewise, both this case and the reference simulation (without spurious velocities) as well as the 4-level simulation, the area near the heave plate, where the greatest turbulence effects occur, presents the same velocity distribution at both 50 s and 150 s.

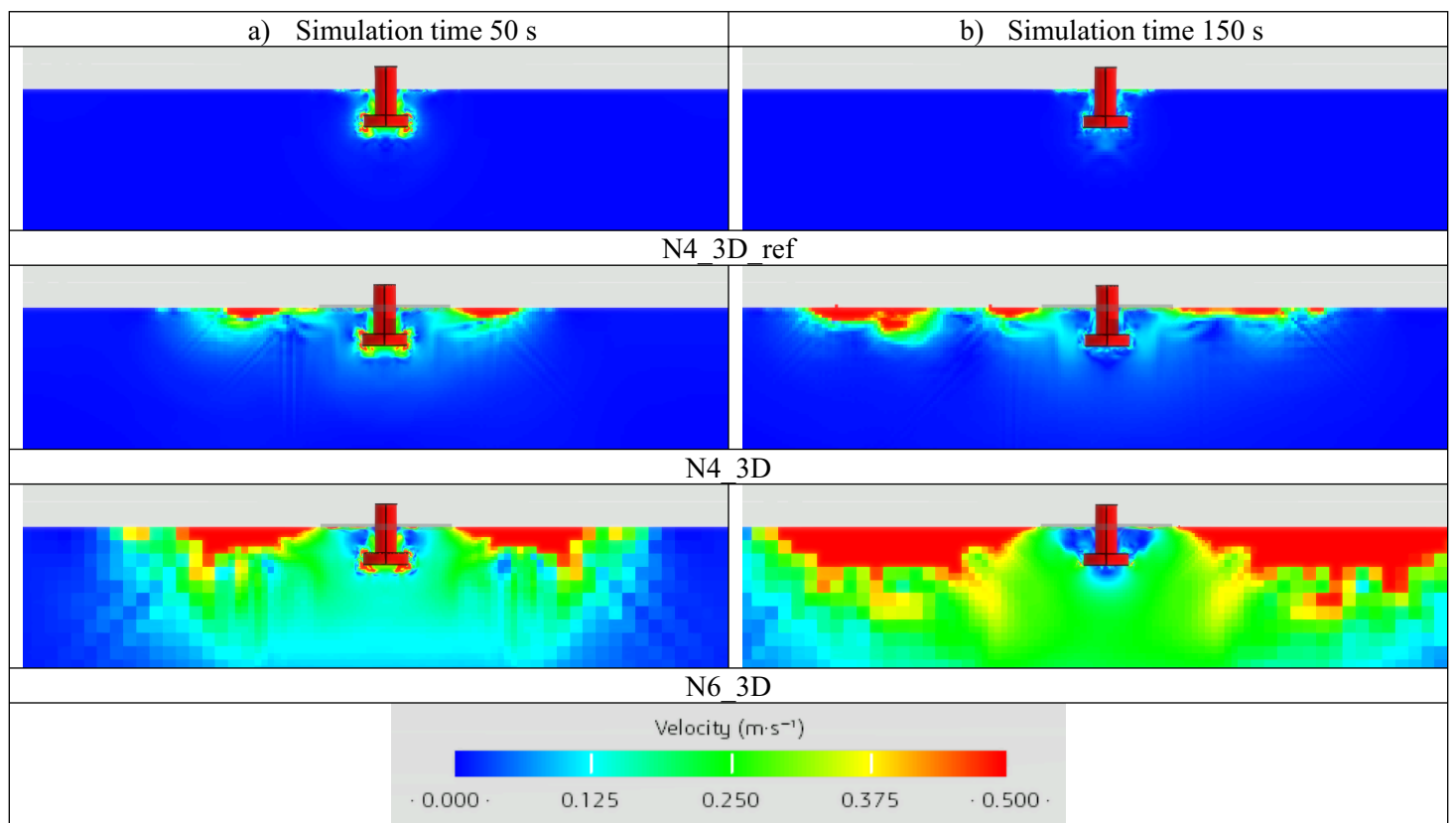


Fig. 5: Inestabilidades numéricas en la interfase

Finally, Fig. 6 shows a comparison of the vertical position of the platform and the vertical hydrodynamic force between the N4-3D and N4-3D-ref simulations. Both simulations share the same channel dimension, TRS and mesh levels, and differ in the meshing of the free surface. A slight difference is observed in the heave position, where N4-3D mainly experiences a lower decay in the first half-cycle, as a consequence of errors in the simulation start-up, which are reflected in the hydrodynamic force. Otherwise, the proposed simulation shows a correct behaviour, except for slight differences in total amplitude in the semi-cycles, especially in those moments where spurious velocities are present, modifying the sinusoidal shape of the resultant force. However, this simulation saves 74.7% in computational cost, so the implementation of this meshing technique may be convenient in approximation simulations.

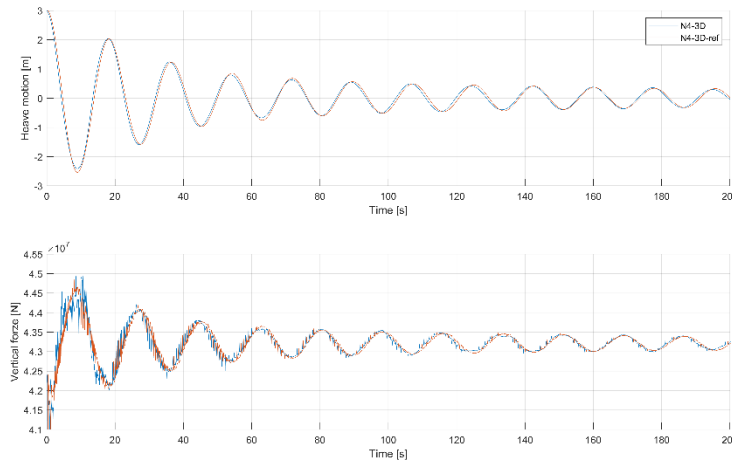


Fig. 6: Heave motion and hydrodynamic force comparison between N4-3D and N4-3D-ref simulations.

4. Conclusions

As shown in the previous section, one way to reduce the computational cost in simulations of large floating objects is by introducing different mesh levels combined with an increase in the dimensions of the simulation water channel. The 535.04 m long channel is able to dissipate reflection effects as well as minimize the effect of spurious velocities generated by the transition between meshes. Models with 5 or 6 mesh transitions, although they have a lower computational cost, present high spurious velocities that cause negative decays, indicating that they are invalid models. Regarding the reduction in computation time, the N4-3D model presents a reduction in computation time of 74.7 % with respect to the reference model N4-3D-ref. As a proposal for the future, it should be noted that a study of the simulation start-up would be interesting, since it has been observed that for all the simulations the first two decays do not correspond to reality. In several research works, the first two cycles are not taken into account and are automatically eliminated from the data processing, although their correct simulation is of interest, since they are the time instants where the platform velocities are maximum and the viscous effects take prominence.

References

- [1] Wang, L., Robertson, A., Jonkman, J., Kim, J., Shen, Z.-R., Koop, A., Borràs Nadal, A., Shi, W., Zeng, X., Ransley, E., Brown, S., Hann, M., Chandramouli, P., Viré, A., Ramesh Reddy, L., Li, X., Xiao, Q., Méndez López, B., Campaña Alonso, G., ... Yu, K. "OC6 Phase Ia: CFD Simulations of the Free-Decay Motion of the DeepCwind Semisubmersible". *Energies*, vol. 15, 2022. <https://doi.org/10.3390/en15010389>
- [2] Zhang, W., Calderon-Sanchez, J., Duque, D., & Souto-Iglesias, "Computational Fluid Dynamics (CFD) applications in Floating Offshore Wind Turbine (FOWT) dynamics: A review". *Applied Ocean Research*, vol. 150, 2024. <https://doi.org/10.1016/j.apor.2024.104075>
- [3] Medina-Manuel, A., Botia-Vera, E., Saettone, S., Calderon-Sanchez, J., Bulian, G., & Souto-Iglesias, A. "Hydrodynamic coefficients from forced and decay heave motion tests of a scaled model of a column of a floating wind turbine equipped with a heave plate". *Ocean Engineering*, vol. 252, 2022. <https://doi.org/10.1016/j.oceaneng.2022.110985>
- [4] Gil, M., Armañanzas, J., Torres, A., Fuertes, J. P., Campaña, G., Mendez, B., & Leon, J. "High fidelity CFD models comparison to potential flow method in the simulation of full scale floating platform under free decay tests". *Ocean Engineering*, vol. 331, 2025. <https://doi.org/10.1016/j.oceaneng.2025.121385>
- [5] Wang, Y., Chen, H.-C., Koop, A., & Vaz, G. "Verification and validation of CFD simulations for semi-submersible floating offshore wind turbine under pitch free-decay motion". *Ocean Engineering*, vol. 242, 2021. <https://doi.org/10.1016/j.oceaneng.2025.121385>

- [6] Rentschler, M., Chandramouli, P., Vaz, G., Viré, A., & Gonçalves, R. T. “*CFD code comparison, verification and validation for decay tests of a FOWT semi-submersible floater*”, *Journal of Ocean Engineering and Marine Energy*, vol. 9, pp. 233-254, 2023. <https://doi.org/10.1007/s40722-022-00260-z>