# APPLICATION OF OPENFOAM® TO THE STUDY OF WAVE LOADS ON OFFSHORE STRUCTURES IN SHALLOW WATER

*Utilisation d'OpenFOAM® pour l'étude d'impacts de vagues sur des structures offshores en eaux peu profondes*

Aurélien Thinat
aurelien.thinat@cfd-numerics.com

Stéphane Cordier[1], François Cany
stephane.cordier@stxeurope.com, françois.cany@stxeurope.com

## KEY WORDS

VOF, optimization, Offshore Wind Turbine, wave loads, shallow water, wave maker, moving mesh.

## ABSTRACT

*With the growth of the renewable energy market and especially the offshore wind energy sector, it becomes crucial to study the specific problems encountered in the design and installation of these systems. One of the most important design parameters in the optimization of the offshore structures, fixed or floating, is their behavior when subjected to waves. Up to now, theoretical and numerical methods have been found to have some shortcomings in the estimation of forces and moments generated by waves on offshore structures and experimental approaches have mostly been used. A CFD (Computational Fluid Dynamics) solution has been developed based on OpenFOAM® to simulate a wave tank in order to reduce the number of tests performed and possibly to avoid some of the limitations and costs of the experiments. A numerical model of an actual wave tank (Roger Brard wave tank at DGA/Th), has been generated according to its actual dimensions including the wave generator. The waves are generated similarly to the actual wave tank using a moving boundary of the domain which is programmed to duplicate the actual recorded wave generator motion. The grid of the domain adapts itself through a moving mesh. The validation of these simulations was conducted with a set of two different geometries of offshore wind turbine bases and two different wave solicitations for a total of four cases. We assess the viability of this numerical solution on three criteria: Ability of transporting a single wave's amplitude trough the wave tank without any numerical diffusion; Ability of producing a large number of waves on large time scale; Comparison of the numerical solutions with experimental data: forces, moments, wave amplitude.*
*This paper presents the numerical results which match well with the experimental data. Finally, a discussion of the future potential of this method is discussed, including the behavior of floating structures.*

## 1. INTRODUCTION

The renewable energy market growth leads industrials to explore the offshore wind energy potentials. By now, new Offshore Wind Turbine design validations are held in wave tank facilities which require a lot of time and resources.
The CFD (Computational Fluid Dynamics) computations can be used to couple an experimental approach with a numerical analysis to investigate different conditions and geometries in order to reduce cost and development time. To be effective, the numerical solution needs to be massively parallelizable to reduce the CPU time. Furthermore, the use of an open-source CFD software allows the simultaneous calculation of several different cases at no extra cost. The paper shows how the open-source CFD software package OpenFOAM® can provide the tools needed to perform the simulations of wave loads on offshore structures in shallow waters.

[1] Corresponding author

# 2. NUMERICAL MODEL

## 2.1 Geometric model and mesh

### *2.1.1 Numerical wave tank dimension*

The validation of the numerical solution is performed by comparison with experimental data obtained in experimental tests carried out in the Roger Brard wave tank facility of the DGA/Th. Thus, the numerical model proposes to simulate the exact geometry of this wave tank including the wave maker. The geometry is presented in the **Figure 1**.

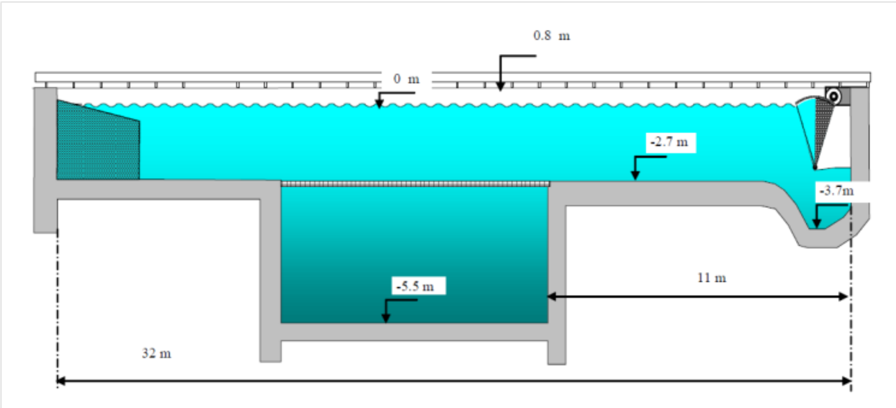


**Figure 1 :** Experimental facility of the Roger Brard wave tank.

The wave tank was divided into two sub-domains to avoid cell quality issues and the mesh size was adapted to the different elements of the physical environment: wave generator zone and offshore structure. Hence, two sub-domains were created: one wave generator region, and one offshore structure region. Then, the two sub-domains are connected using non-conform interface option (GGI interface) as described in the following section. The width of the numerical tank was reduced from 10 to 3 meters for an Offshore Wind Turbine model width close to 1 meter in order to save hexahedral cells without compromising the results with deflected waves. The numerical wave tank however, is longer than the real facility in order to add a sufficient wave dissipation area at its end.

The wave generator zone mesh (400,000 cells) has been generated using the block mesh meshing tool provided with OpenFOAM®. A 30-centimeter wide refinement cell zone was defined around z=0m to capture the free surface. The **Figure 2** presents the mesh defined in the wave maker sub-domain.

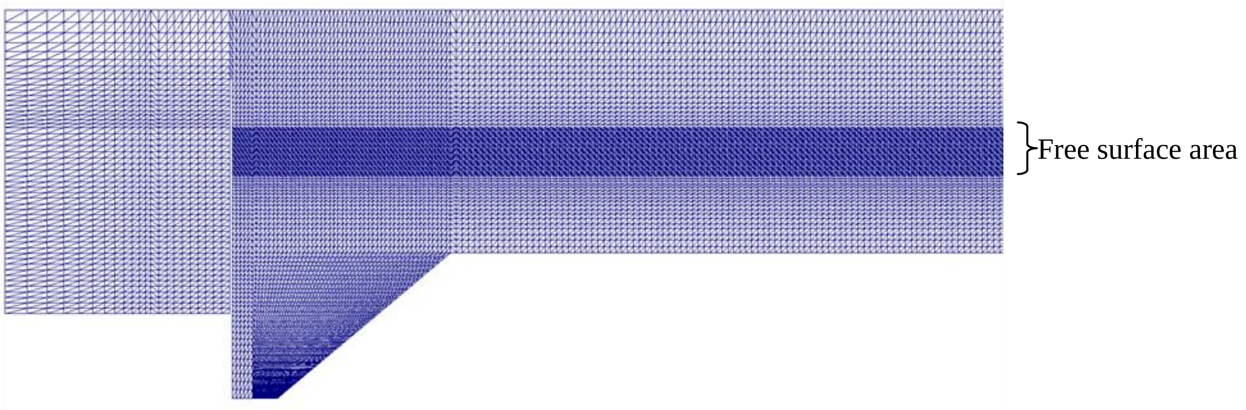


**Figure 2 :** Numerical wave tank's wave maker sub-domain mesh. A refinement cell zone was defined to capture the free surface interface.

The offshore structure mesh has been built using the embedded OpenFOAM® automatic mesh generation application SnappyHexMesh. The basic principle of this tool is to build a custom mesh with several refinements regions. This custom mesh is then snapped around the CAD geometry to get a body fitted mesh. The **Figure 3** presents the downstream sub-domain mesh, which includes the offshore structure. Several refinement zones were defined to capture the free surface elevation and the pressure distribution on the Offshore Wind Turbine's structure to be used for comparison with experiments.

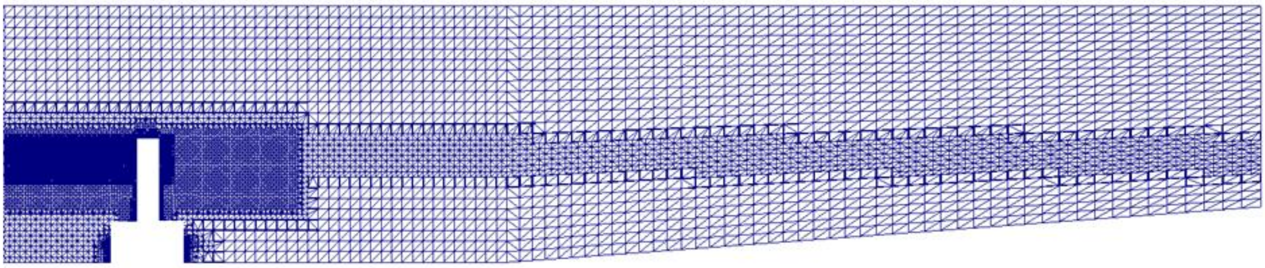

**Figure 3 :** Numerical wave tank's downstream sub-domain mesh.

### *2.1.3 GGI Interface*

The final mesh leads to an unstructured mesh with an interface between the two sub-domains. This interface is managed with a GGI algorithm included in OpenFOAM® [1]. Cell's sizes shown on **Figure 4** were imposed to avoid as maximum of interpolation errors as possible.

a) b)

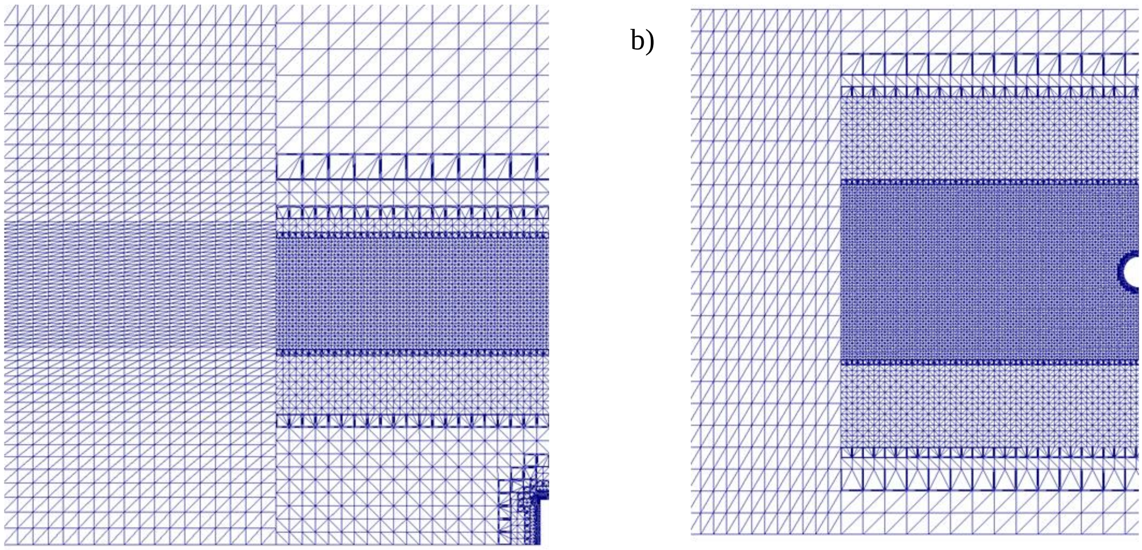

**Figure 4 :** Visualizations of the GGI interface between the two sub-domains: a) y normal view of the mesh. b) z normal view of the mesh.

The mesh size for the given offshore structure is aboutt 500,000 cells and ensures good mesh quality with maximum skewness of 9.57 and a non-orthogonality limited to 65. We show hereafter in **Figure 5** a global view of the simulated domain with the different boundary conditions.

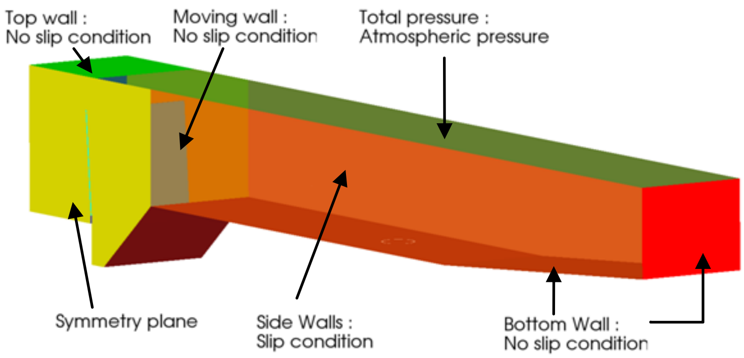


**Figure 5 :** Boundary conditions used in the wave tank numerical model.

## 2.2 VOF Algorithm

There are two main methods to compute the free surface: the interface tracking method and the front capturing method. The former treats the free surface as a sharp interface whose motion is followed by moving the grid and the free surface. The latter, commonly used, is performed on a fixed grid and the shape of the interface is determined by the fraction of each near-interface cell that is partially filled. In this paper, the front capturing method has been used as a volume of fluid method ("VOF"). The "VOF" method is a two phase surface compression method that solves the Navier-Stokes equations and an advection transport equation for the volume fraction:

$$\frac{\partial c}{\partial t} + \vec{\nabla} \cdot (c\vec{u}) = 0, \quad (1)$$

where c is the volume fraction :
c = 1 in pure water,
c = 0 in pure air.

The challenge is to solve the equation (1) by limiting the numerical diffusion and keeping a bounded solution. Several schemes have been developed for this purpose: the HRIC method from Peric [2] or the CICSAM method from Ubbink [3]. In OpenFOAM®, the convective-only equation (1) is solved using the dedicated scheme MULES (for "Multidimensional Universal Limiter for Explicit Solution"). The equations are solved using a PISO ("Pressure Implicit with Splitting of Operators") algorithm. This model has been validated in combination with mesh morphing options for naval application [4].

## 2.3 Wave generation

### *2.3.1 Input data in the numerical model*

The numerical wave tank is used as an exact reproduction of the experimental facility and the wave maker motion used is the exact same as that used on the real wave generator device. A phase shift at $t_0$ was introduced to allow a soft initialization of the fluid flow by starting with a vertical position of the wave maker. These operations are presented on the **Figure 6**.

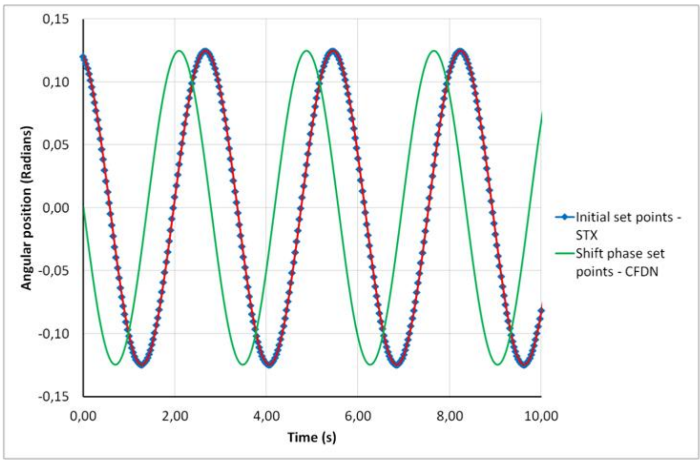


**Figure 6 :** Model input for the moving mesh set points with an initial shift phase of π/2.

The time steps used for the simulation were automatically adapted by the solver to limit the global courant number of the model to 0.8. Moreover, a specific courant number was applied for the VOF interface that represents 1/10$^{th}$ of the global courant number and which ensures good interface capturing.

### *2.3.2 Robustness of the method*

The main difficulty in moving mesh applications is to ensure the mesh quality is preserved over a long time period. The mesh deformation is performed in OpenFOAM® by using a morphing algorithm that adapts the cells around the moving boundary condition on which the displacement is applied. Tests have been performed to assess this on the wave generator model. One can attest of the robustness of the moving mesh method over a complete oscillation on the **Figure 7**. After a complete cycle, the mesh stands in its exact initial position. This robustness guarantees that the model can be used on large periods of time and thus makes it possible to use this solution to get averages of the efforts applied on the offshore structure over multiple cycles.

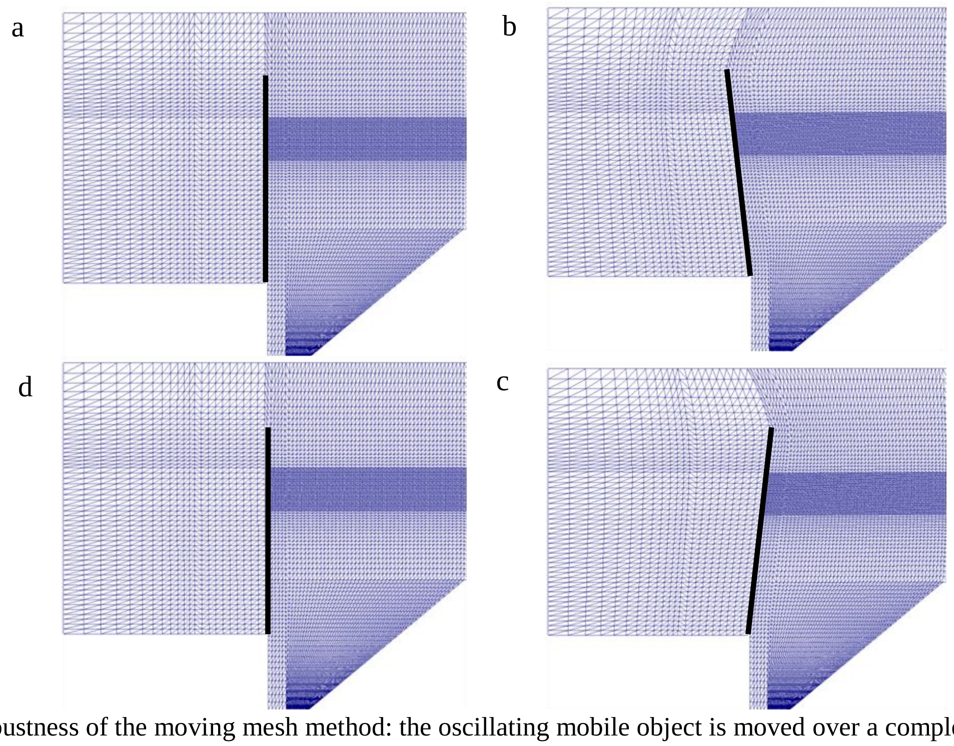


**Figure 7:** Robustness of the moving mesh method: the oscillating mobile object is moved over a complete period. a) Initial position. b) Minimum angular position. c) Maximum angular position. d) End of the period.

# 3. NUMERICAL MODEL VALIDATION

## 3.1 Wave's amplitudes

The first part of the numerical results evaluation effort consists in verifying that the waves generated are conserved in terms of amplitude along the complete numerical domain. This test has been performed for a wave corresponding to a sinusoidal wave generator motion without any obstacle in the tank. One can see on the **Figure 8** that the wave height and shape are conserved at different time steps along the wave tank.

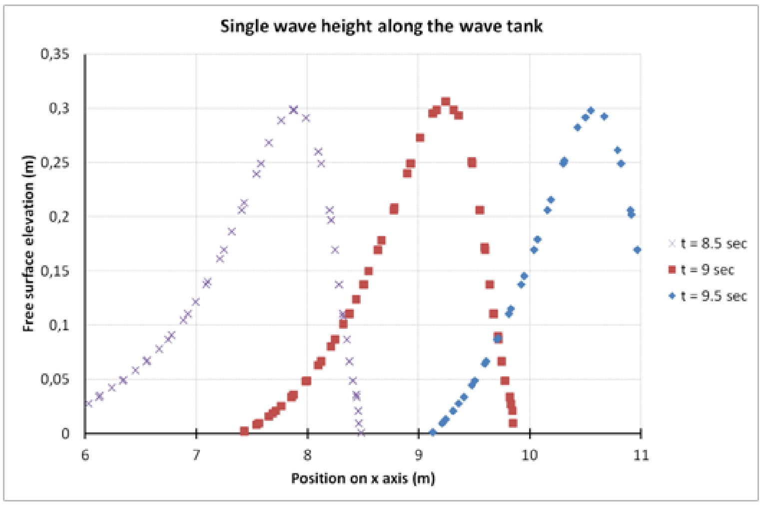


**Figure 8:** Transport of a single wave along the wave tank model.

### *3.1.2 Generation of several waves*

The evaluation of the numerical method continues with its ability of generating several waves in a single run of the numerical wave tank model. The **Figure 9** shows a first result of the waves generated over 25 seconds of physical time in order to verify the stability of the model.

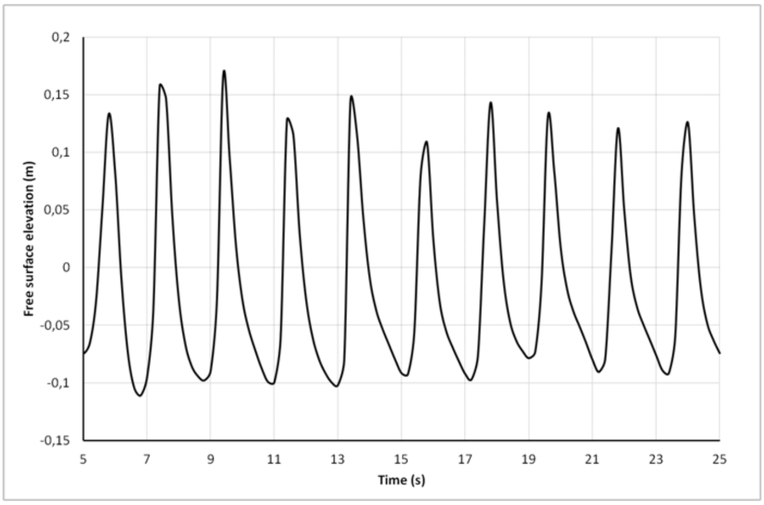


**Figure 9:** Generation of multiple waves in a single run over 25 seconds of physical time.

One can notice some dispersion in wave height which are due to the turbulent flow generated in the close vicinity of the wave maker in the zone between the wave maker and the ramp which has a sharp edge transition to the tank bottom, whereas the actual tank has a smooth transition.

## 3.2 Comparison with experimental results

### *3.2.1 Comparison of wave amplitude*

The next step in order to validate the methodology is to compare the numerical results with experimental data sets. The test case is based on an offshore wind turbine which has been evaluated for different wave periods and amplitudes.

The **Figure 10** shows the numerical waves amplitude normalized by the theoretical wave's amplitude at measurement point located at 50 centimeters in front of the Offshore Wind Turbine after the first 13 seconds of simulation. Due to the phenomenon described before concerning the near field wave generator flow, and some wave reflection occurring after a long period, only the 20 first seconds of this simulation will be taken into account for the pressure force average calculation.

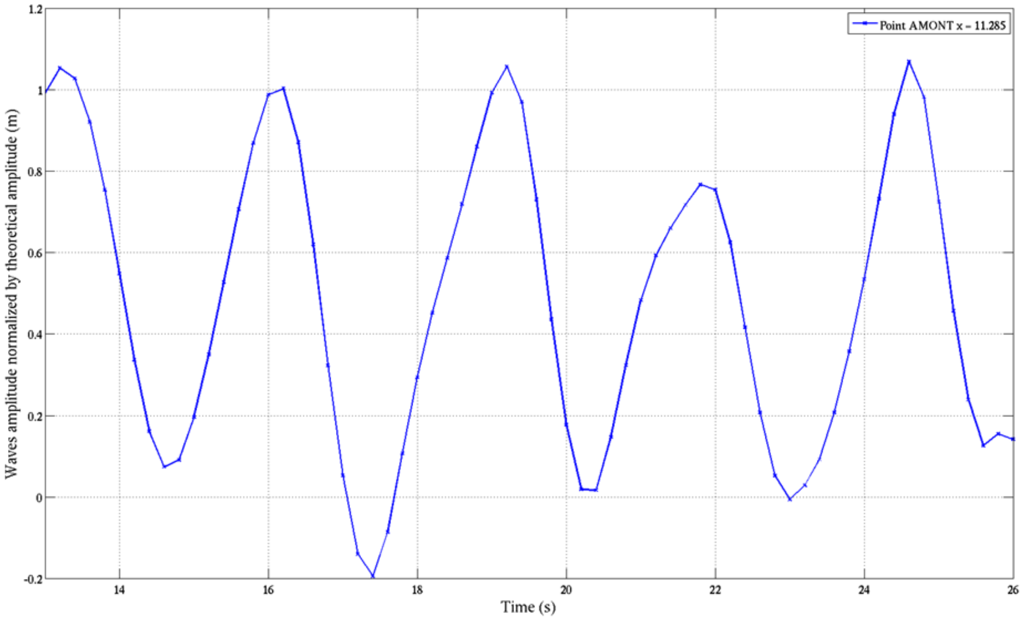


**Figure 10**: Waves amplitudes normalized by the theoretical wave amplitude.

### *3.2.2 Comparison of forces and momentum*

The pressure forces and the momentum applied on the offshore wind turbine base calculated with the numerical model are shown on **Figure 11** for a given case. As explained previously, we only take into account the three waves which impact between $t_0$= 60s and $t_1$ = 110s. The results are normalized by the experimental data. The pressure forces Fx is over evaluated by 25%. The moment MFy computed by the model is very close to the experimental measurements with results located in a 20% range.

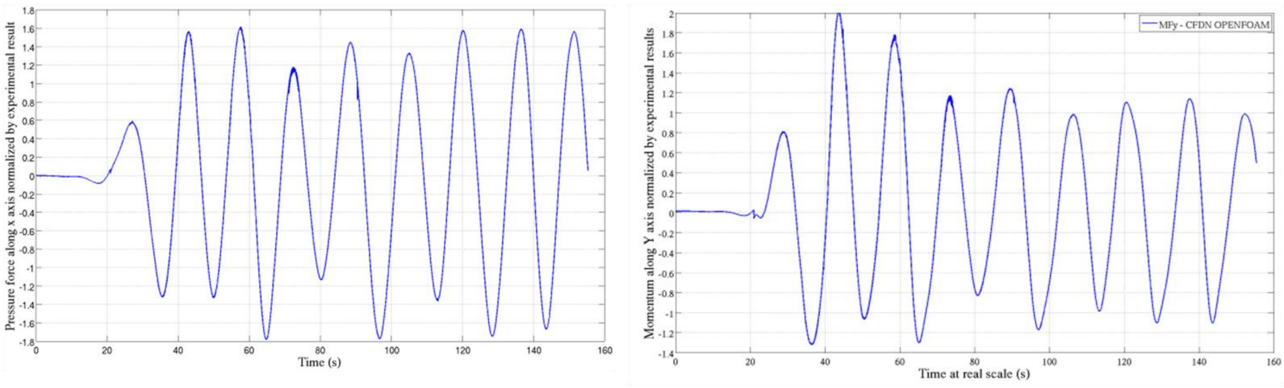


**Figure 11 :** Pressure force Fx and momentum My applied on the offshore wind turbines base normalized by experimental data.

### *3.2.3 Geometrical variations*

The model was used over a set of 4 different cases: 2 geometries submitted to 2 different wave series each. Averages of pressure forces and moments are computed and averaged over a set of 4 or 5 waves for each case. Results normalized by experimental data are shown at **Figure 12**. The pressure force Fx is generally well computed within a 8% range except for the case “windmill 1 – Consigne 2” which is over evaluated by 22% The numerical pressure forces Fz are in a 25% range of the experimental data. Finally, moments applied on the structures are under-evaluated in comparison of the experimental measurements.

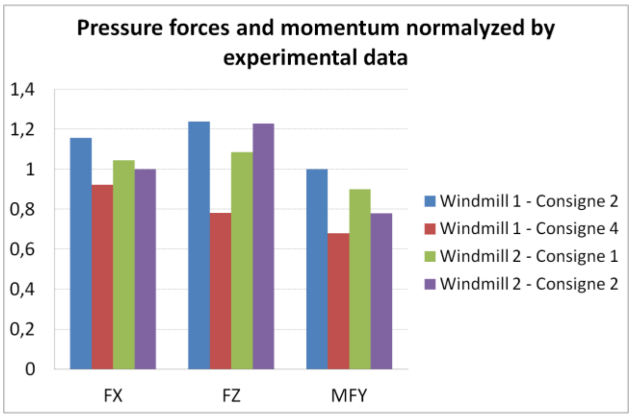


**Figure 12 :** Comparison of pressure forces and momentum with experimental data for different offshore wind turbines bases and wave solicitations.

## 3.3 Synthesis

A first series of calculations was performed for two different geometries and three different types of waves. The **Figure 13** shows in a synthetic form that the numerical model is generally in line with the measurements performed. These results show that a numerical model of a wave tank where the wave generation, propagation and interaction with a complex body are simulated in the same numerical domain is possible. Furthermore, this model provides reasonable estimates of the forces and moments which may be used in an industrial optimization loop.

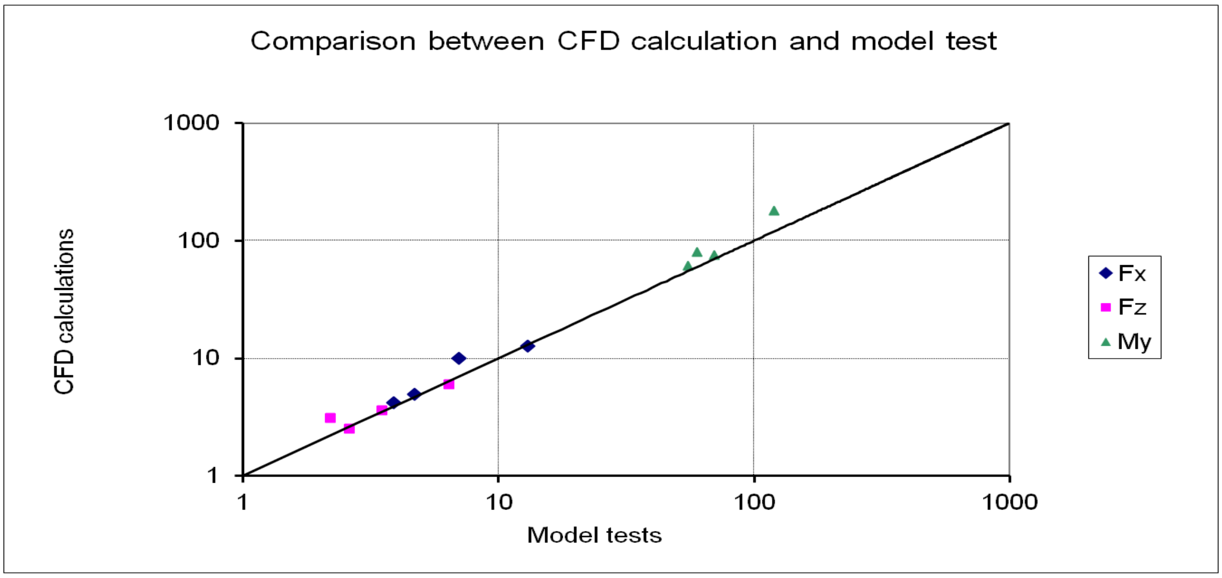


**Figure 13 :** Comparison between CFD calculation and model test

## 4. IMPROVEMENTS AND FUTURE DEVELOPMENTS

### 4.1 Stabilization of the waves amplitude

We have noticed that the amplitudes of successive waves are not identical. We expect to improve this issue by smoothing the sharp angle located at the end of the initial ramp near the wave generator as it is in reality. The vorticity generated by the sharp angle represents a source of variation in the wave generation.

### 4.2 Standardization of the wave dissipation area

The wave dissipation area revealed to be a crucial point of the numerical model in order to limit wave reflection on the downstream wall. The **Figure 14** shows the velocity field for two different wave dissipation models. In the first one, the wall reflects the pressure wave which tends to destabilize progressively the flow from the back to the front. In the second model which extends above the free surface, the waves are break up without reflection. The second configuration will be used for further developments.

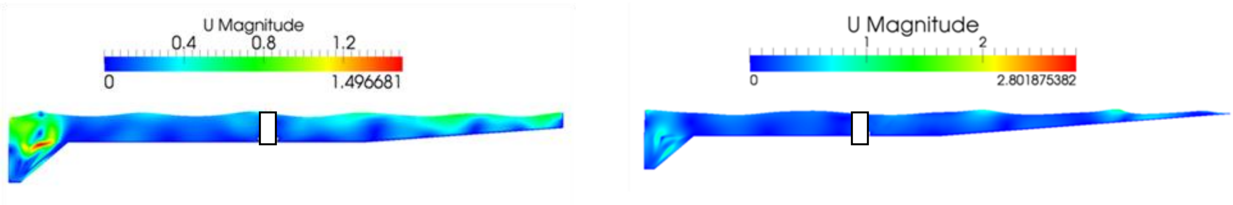


**Figure 14 :** Wave dissipation area at the wave tank's downstream.

### 4.3 Coupling with floating structures

The next main step of this methodology will be to introduce a floating structure instead of a static structure in order to perform numerical simulations of the behavior of floating objects and to study their seakeeping behavior. Such computations are already performed but with potential-flow solvers which have different limitations such as linear behavior and single body analysis.

## 5. CONCLUSIONS

A numerical methodology based on the open source CFD package OpenFOAM® has been developed to study the wave loads on offshore structures in shallow water by simulating the complete environment of a wave tank including the wave maker system and the structure to be tested. It has been shown that the methodology in its current state is able to produce waves over a large period of time. The results obtained are in good agreement with experimental measurements both in terms of absolute values and in the identification of trends between different designs.

The numerical model needs further development to improve the flow in the wave maker area and to reduce the reflected waves.

The CFD calculation appears to be a good way to reduce costs and time in an industrial process of design of an offshore structure in shallow water.

## REFERENCES AND CITATIONS

[1] Beauboin Martin, Jasak Hrvoje (2008). Development of a Generalized Grid Interface for turbomachinery simulations with OpenFOAM. Proceedings of the Open Source CFD International Conference 2008, Berlin, Germany.

[2] M. Peric & J.H. Ferziger. *Computational method for fluid dynamics*. 3rd edition, Springer, 2002.

[3] O. Ubbink. Numerical prediction of two fluids system with sharp interface. Imperial College, 1997.

[4] S Cordier, L Morand, J-M Roux, and E. Cortey. *Application of OpenFOAM® to hull form optimization at STX France.* Proceedings of the RINA Conference 2011.