



Development, engineering, production and life cycle management of improved FIBRE-based material solutions for the structure and functional components of large offshore wind energy and tidal power platforms

D6.5 (WP6): DEVELOPMENT OF THE CAE MODELS

Responsible Partner: COMPASSIS

Contributor(s): TSI

DOCUMENT INFORMATION TABLE

CONTRACT NUMBER:	952966	
PROJECT ACRONYM:	Fibregy	
PROJECT COORDINATOR:	CIMNE	
DOCUMENT RESPONSIBLE	Compass IS	TSI
DELIVERABLE TYPE:	Report	
DOCUMENT TITLE:	Development of the CAE models	
DOCUMENT ID:	DX.X	
DISSEMINATION LEVEL:	PU: Public CO: Confidential, only for members of the consortium	
FILENAME:	filename	
STATUS:	First version	

Authoring & Review

PREPARED / REVIEWED BY				
Name	Role	Partner	Date	Comments
Joel Jurado	Creator	Compass IS	17/06/2022	
Reviewer 1	Reviewer			

EXECUTIVE SUMMARY

This document describes the CAD/CAE models of the W2Power platform and TIDETEC turbine housing, both in FRP (Fibre Reinforced Polymer). This task is part of the work-package 6 of the FIBREGY project aiming at generating the two complete 3D CAE models of the structure of the two different concepts of large Offshore Wind and Tidal Stream Platforms (OWTPs): W2Power floating wind turbine platform and TIDETEC's turnable tidal turbine. It will consist on applying the proposed modifications on the geometry from WP4, assigning the materials' and structural properties, defining the loads applied on the model and boundary conditions, and finally the generation of the required mesh.

TABLE OF CONTENTS

1.	INTRODUCTION	4
2.	W2POWER REAL SCALE PLATFORM MODEL.....	4
2.1.	CAD MODEL (GEOMETRY) FOR FEA	4
2.1.1.	GEOMETRY MODIFICATION	4
2.2.	CAE MODEL FOR FEA	6
2.2.1.	ENTITIES DEFINITION	6
2.2.2.	LOCAL AXES	8
2.2.3.	MATERIALS ASSIGNMENT	10
2.2.4.	LOADS ASSIGNMENT.....	12
2.2.4.1.	SELF-WEIGHT:.....	13
2.2.4.2.	HYDROSTATIC LOADS:.....	14
2.2.4.3.	WAVE LOADS:.....	15
2.2.4.4.	WIND LOADS:	15
2.2.4.5.	MOORING LOAD:.....	16
2.2.5.	CONDITIONS ASSIGNED	16
2.2.5.1.	DYNAMIC CONDITIONS:	16
2.2.5.2.	CONSTRAINTS	17
2.3.	FEM MESH GENERATION AND REFINEMENT FOR FEA.....	17
3.	TIDETEC TURBINE HOUSING MODEL.....	20
3.1.	CAD MODEL(GEOMETRY) FOR FEA	21
3.2.	CONDITIONS ASSIGNED	22
3.3.	FEM MESH GENERATION AND REFINEMENT FOR FEA.....	25
4.	CONCLUSIONS	27
5.	REFERENCES.....	28

1. INTRODUCTION

This document describes accurately the work conducted for defining the 3D CAE models of the structure of the W2Power platform and TIDETEC turbine housing. The deliverable encompasses the geometry definition, the materials properties assigned, application of loads, or boundary conditions defined. In addition, the meshes used for simulating both devices are included.

These two CAE models are required to demonstrate the different numerical models developed in WP3, as well as they are supporting the calculations and simulations conducted in WP4.

2. W2POWER REAL SCALE PLATFORM MODEL

The CAD model referred to the geometry of the W2Power platform is shown. Next, the assignment of properties and conditions to the CAE model are described. Finally, the description of the mesh is explained.

The model of the W2Power belongs to the structural elements of towers and floating platform. Any internal device is disregards, as well as the wind turbine, which is modelled by coupling OpenFAST with Tdyn-Ramseries.

2.1. CAD model (geometry) for FEA

The CAD model has been obtained after an iterative process of the design. The current geometry is based on the original steel-based design of the W2Power platform, at which several modifications have been performed. and it is enough as to adequately represent all the relevant structural elements. The model corresponds to the floating wind turbine platform.

2.1.1. Geometry modification

First of all, geometry repairs and modifications were conducted. The repairing tasks have the aim to improve the original geometry for obtaining a better meshing. Examples of this task are the following:

- Joining of separated lines of the same surface.
- Elimination of drawing auxiliary geometries not belonging to the platform geometry, such as surfaces and lines used for trimming.
- Review of any repeated entity, such as points, lines or surfaces. In this case, a tolerance of 1 mm was defined. As a consequence, any entity closer than 1 mm to other is collapsed, avoiding duplications.

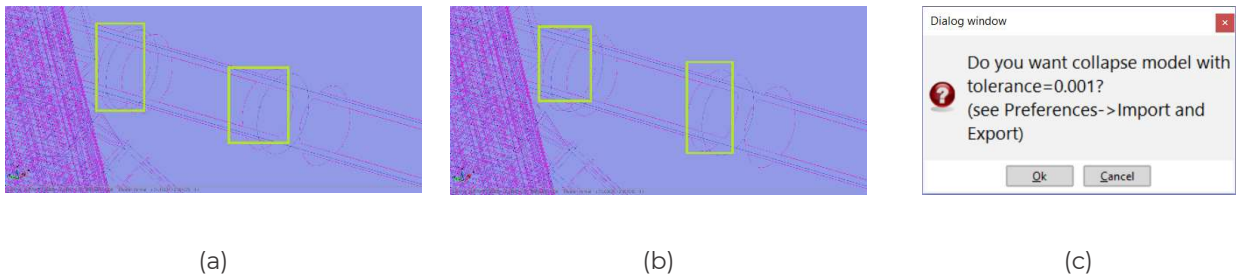


Figure 1. Activities for repairing the CAD model. (a) Lines divided by points. (b) Lines belonging to the same surface are joined. (c) Dialog window for collapsing any entity.

Once the model geometry is repaired, the modifications proposed from the engineering team are implemented. The regions modified in comparison to the original design are shown on the following pictures.

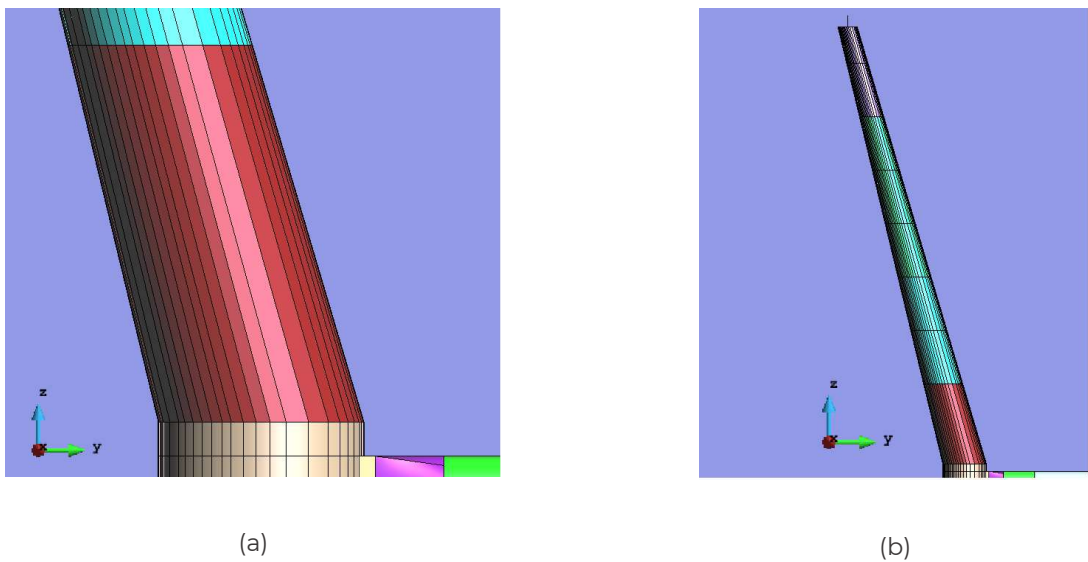


Figure 2. Geometry modification in the connectional between towers and platform transition. A conical transition was proposed by the engineering team. (b) Increased diameter on FRP towers.

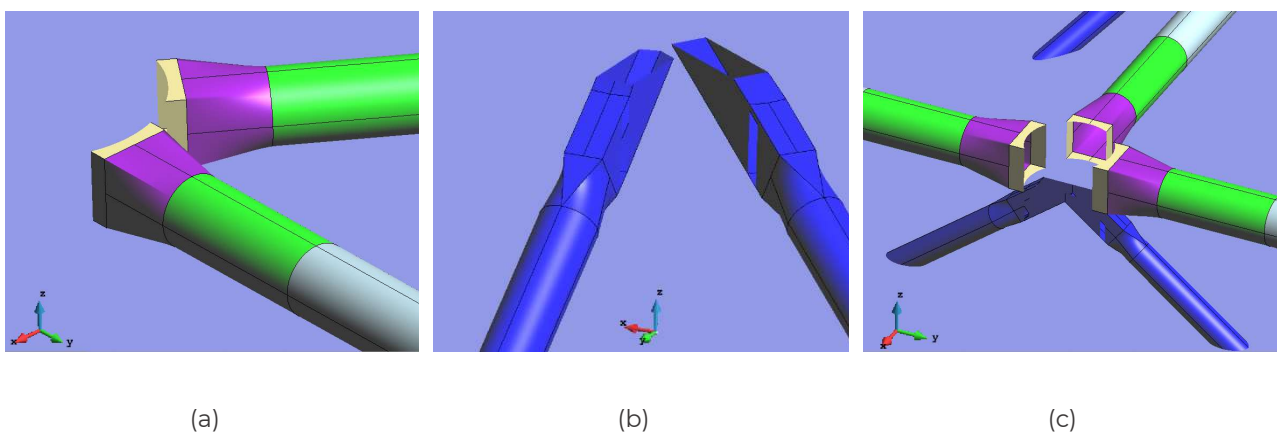
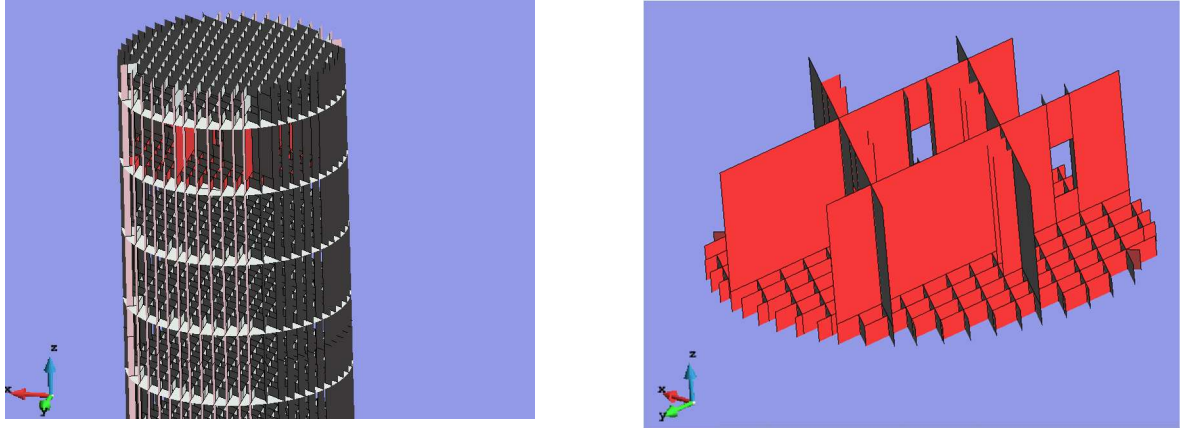


Figure 3. Geometry of braces. (a) Detail of the transition between horizontal braces and columns. (b) Detail between diagonal braces and columns. (c) General view of braces.



(a)

(b)

Figure 4. Internal structure of column D. (a) General disposition of internal structure. (b) Detail of the horizontal bulkhead of column D.

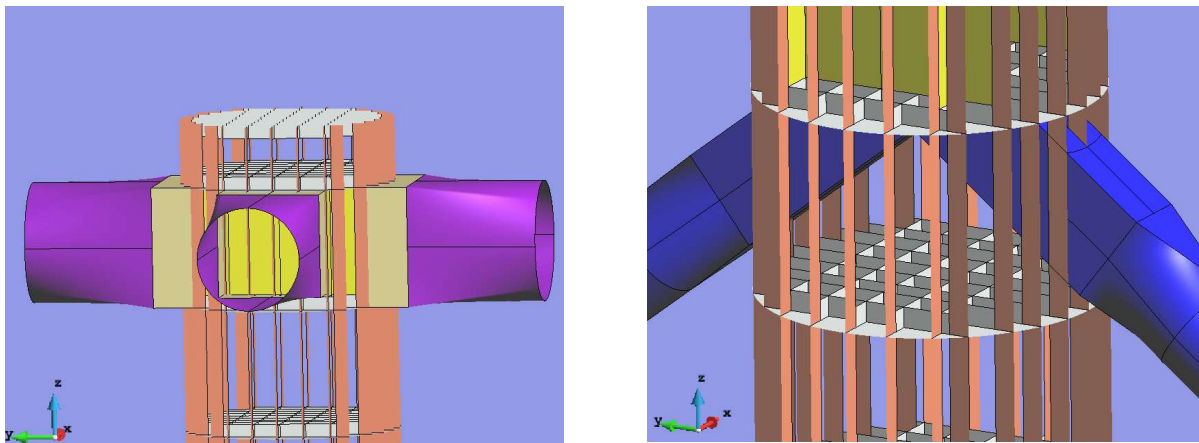


Figure 5. Detail of the intersection between braces and columns. (a) Horizontal braces and column D.

(b) Diagonal braces and column D.

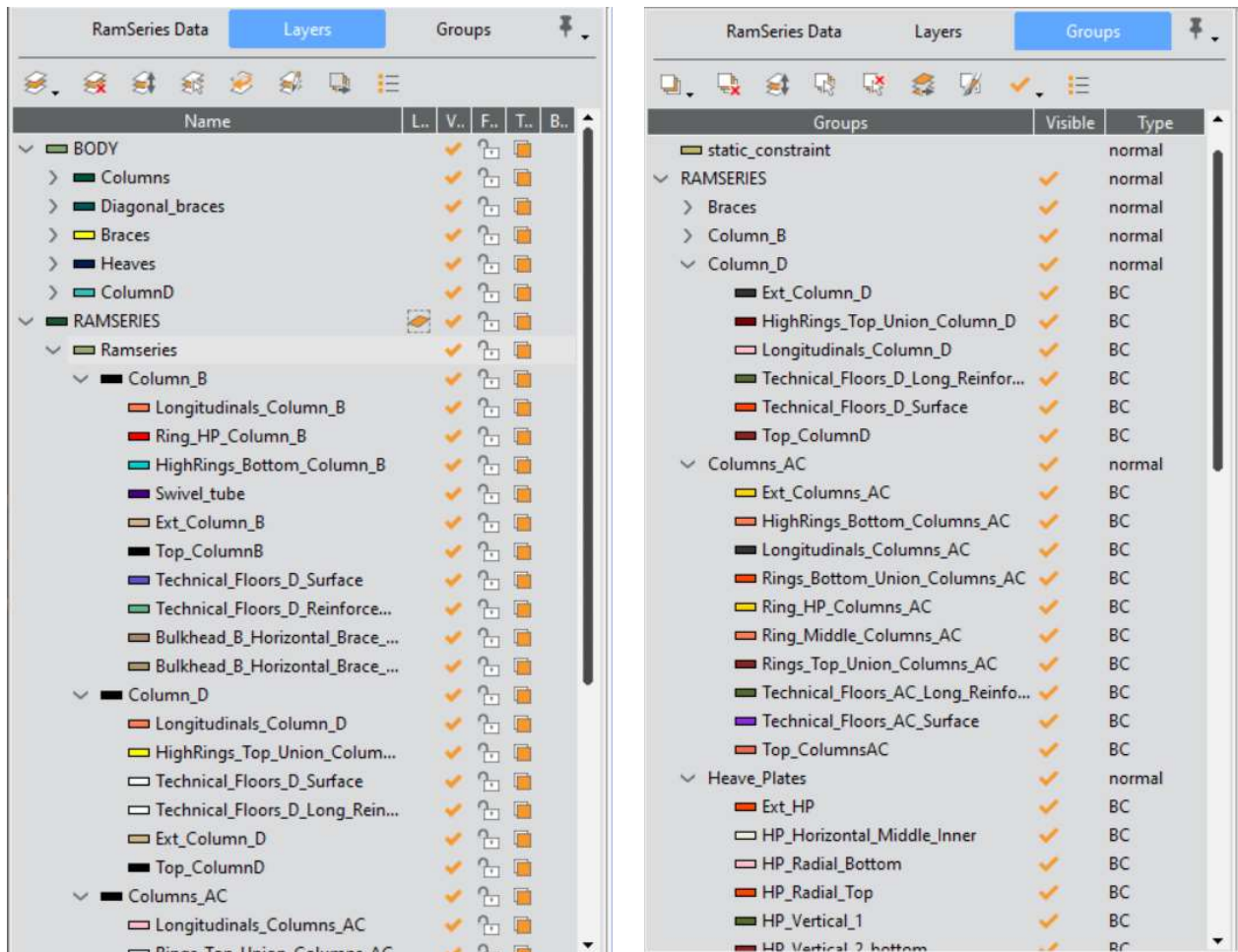
2.2. CAE model for FEA

Once the CAD model is ready, the CAE model is generated by applying the materials' properties, loads, and boundary conditions.

2.2.1. Entities definition

At this point, the different entities of the geometry (it means, surfaces and lines) are classified by layers and groups. This allows a faster visualization and definition of the different conditions of the model. Layers are used for splitting the drawing domain into separated regions. As a consequence, it allows to easy visualize or select entities in the graphical window, such as the external shell of the tower. They are useful for creating

or modifying the geometry. On the other hand, groups are used as a collection of entities to apply simulation conditions, like loads, material properties or boundary conditions.



(a)

(b)

Figure 6. Layers and groups definition. (a) Layers definition. (b) Groups definition.

In order to facilitate the assignation between Tdyn-Ramseries and SeaFEM, the geometry of the W2Power platform have been divided into Tdyn-Ramseries domain (Figure 7a) and SeaFEM domain (Figure 7b). Tdyn-Ramseries domain represents all the structural elements, while SeaFEM domain symbolise the underwater hull of the platform.

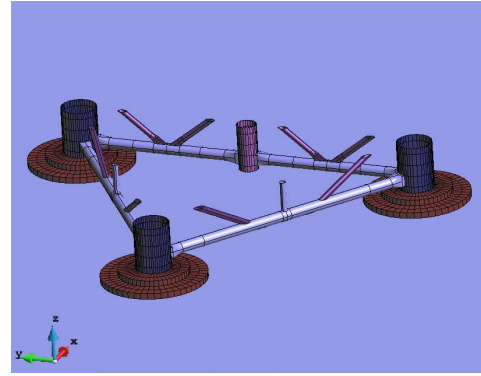
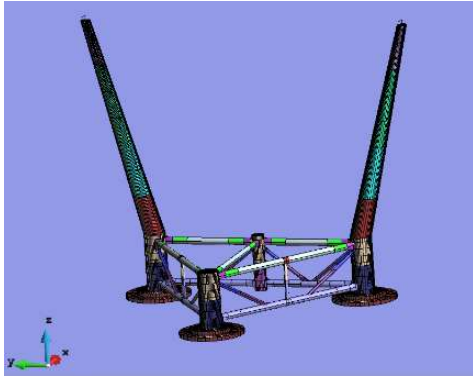
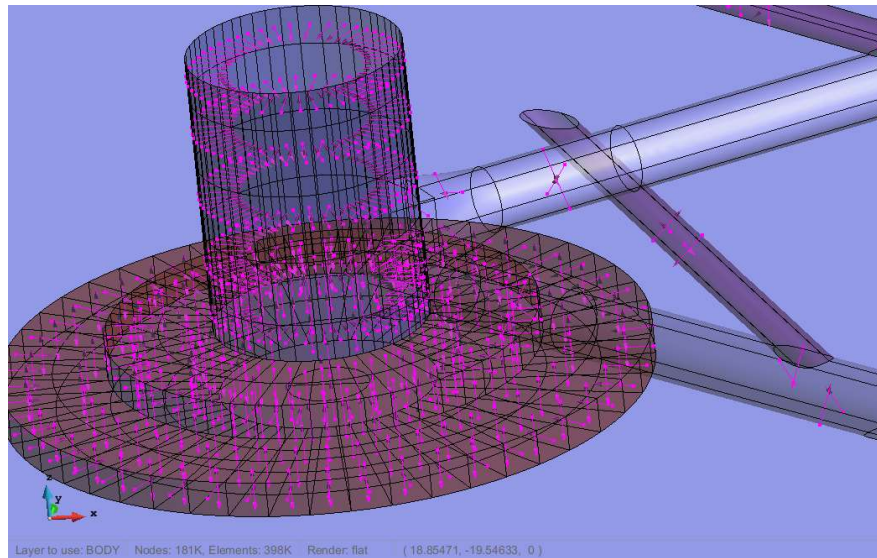


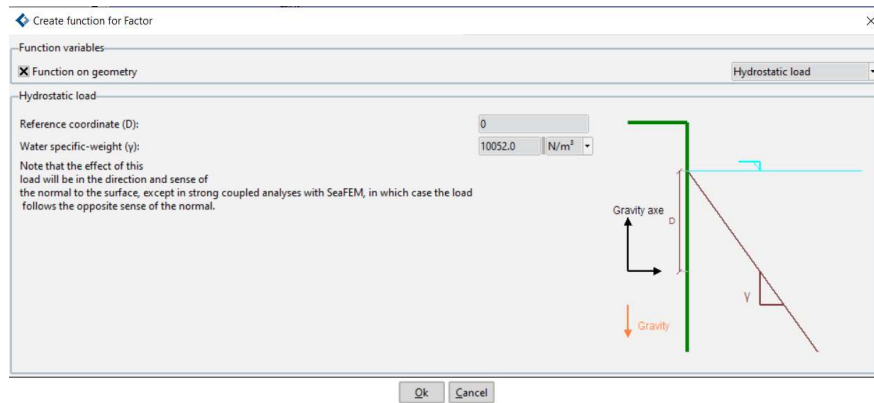
Figure 7. W2Power geometry domain. (a) Tdyn-Ramseries domain. (b) SeaFEM domain.

2.2.2. Local axes

The next step is checking the normal vectors in all the surfaces of the model. The importance of it is that in Tdyn-Ramseries environment, the hydrostatic pressure is applied in the same direction than the normal vector of the surface. Therefore, normal vectors should point inside the volumes, or in this case, inside the platform. In the next figure it is shown.



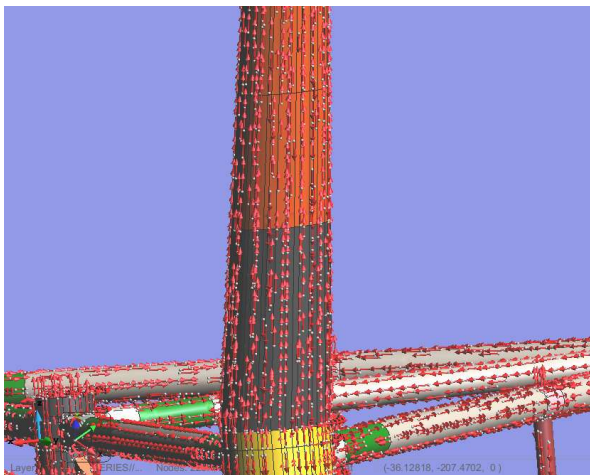
(a)



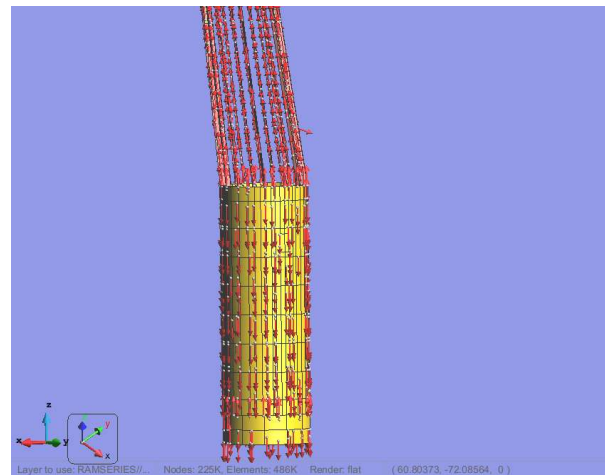
(b)

Figure 8. Normal vectors. (a) Normal vectors in heave plates, pointing inside the platform. (b) Description in Tdyn-Ramseries GUI regarding the definition of the normal vectors and hydrostatic pressure.

The normal vector is used as rotation vector for local axes. The definition of local axes in any composite model is mandatory, on the contrary, the material properties of the model are not well represented. Due to the anisotropic nature of composite materials, the *main* or *reference* direction in the geometry must be defined to properly represent this behaviour. The different plies of the composite use the reference direction for defining their orientation. By this means, a ply with 0° direction means that fibres are parallel to the reference direction. In figure Figure 9 are shown several captures of the definition of the local axes. Those zones considered that require higher stiffness, such as braces and towers, the definition of the local axes was specially studied.



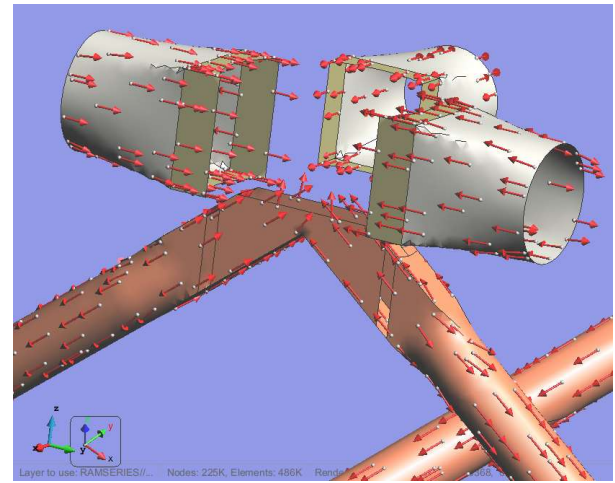
(a)



(b)



(c)



(d)

Figure 9. Main direction of local axes on the CAE model. (a) Detail on outer shell of the tower. (b) Detail on external shell of columns and longitudinal internal reinforcements on tower. (c) General view of local axes definition on braces. (d) Detail of local axes definition on the intersection between horizontal braces and diagonal braces.

2.2.3. Materials assignment

The materials considered for the analysis have been assigned according to section deliverable D4.1, being Hybon 2026 the glass fibre selected for designing the W2Power platform. The lay-up used two types of fabric: roving fabric ($0^\circ/90^\circ$) and unidirectional (0°) fabric. The different elements of the structure, with their thickness and material properties, have been assigned correspondingly to the parts in the geometry. The laminate stacking sequence (LSS) is obtained by defining the minimum percentage of one orientation not less than 12.5%. Assuming the orientations $[0/45/-45/90]$, it is concluded two laminates: the first one is a quasi-isotropic laminate using roving fabrics, and the second one is a symmetric balance angle ply laminate using unidirectional fabrics, which has a dominant orientation according to 0° , so as to optimise its properties. In Table 1, the percentages of each orientation for both laminates proposed are shown, as well as their lay-up.

Table 1. Laminate stacking sequence. Percentages used for each layer orientation.

Laminate	0°	90°	45°	-45°	Lay-up	Fabric used
Quasi-isotropic	25%	25%	25%	25%	$[0_3/45/-45/90/0_2]_{ns}$	Roving
Optimised laminate	62.5%	12.5%	12.5%	12.5%	$[0/45/-45/90]_{ns}$	Unidirectional

Those structural elements assumed to be more critical will be defined using the *optimised laminate*. On the other hand, those structural elements which are not critical, or will have a complex stress state, will be assigned with quasi-isotropic laminates.

The following images show the different properties applied over the geometry of the FRP W2Power platform, as well as the stacking sequence assigned.

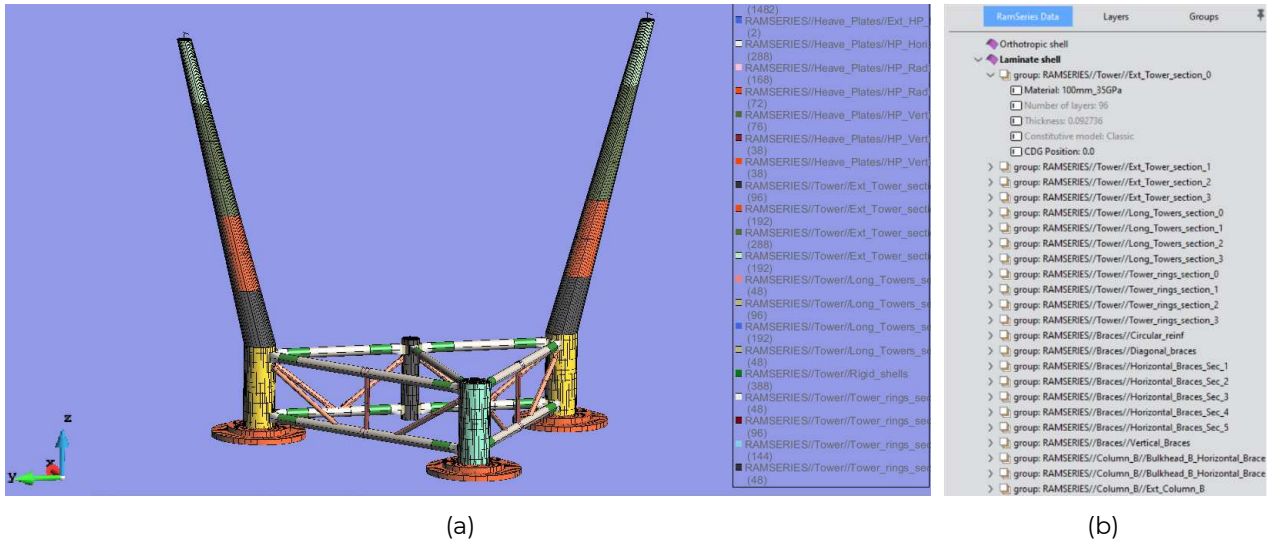


Figure 10. Materials assigned to FRP W2Power platform.

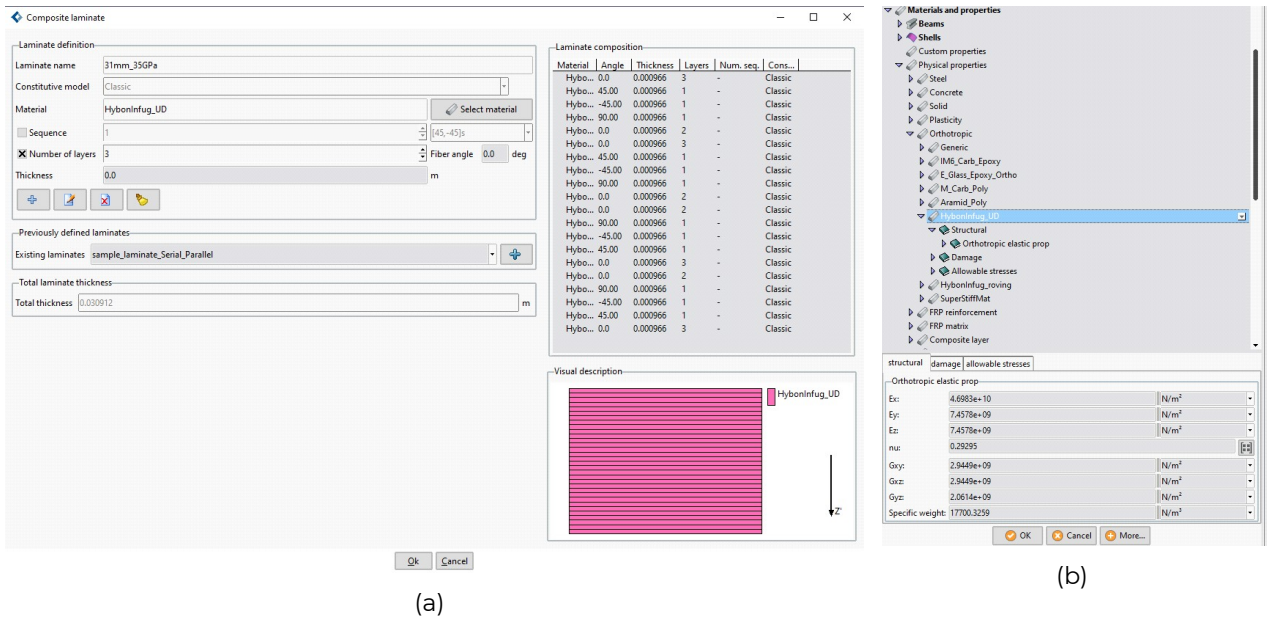


Figure 11. General stacking sequence example of the assigned FRP material to the W2Power platform.

In addition, beams of high stiffness material were defined on the top of the tower. Therefore, loads on rotor and nacelle can correctly be applied on the tower. The assignation and properties of this high stiffness material is shown on Figure 12 and Figure 13, according to NREL turbine.

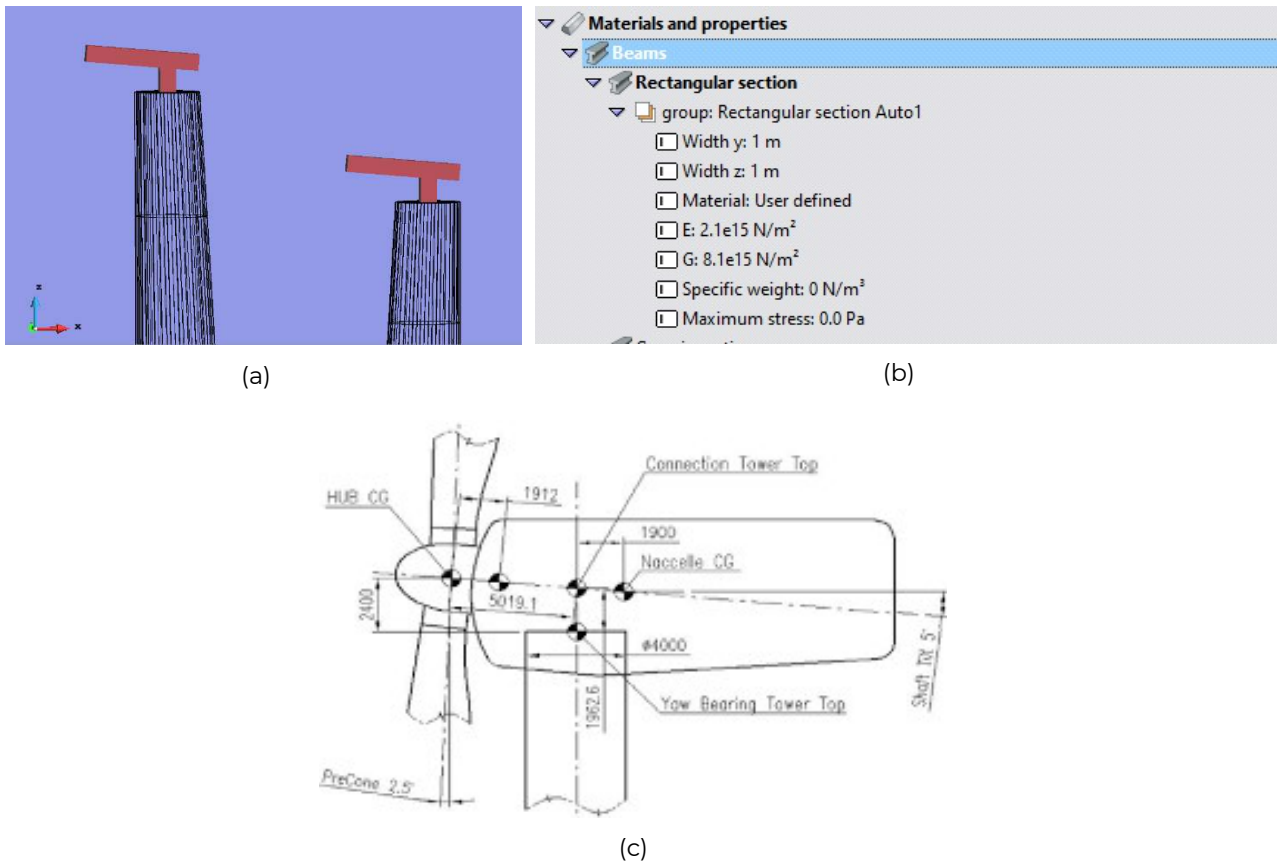


Figure 12. Beams of high stiffness material for taking into account the tower and blade loads. (a) CAE model. (b) Beams properties defined. (c) Diagram from NREL (Zhao, Yang, & He, 2012).

Besides, a high stiffness shell has been defined on the top of the tower, in order to properly transmit the loads from the turbine to the tower, as it is shown on the following figure.

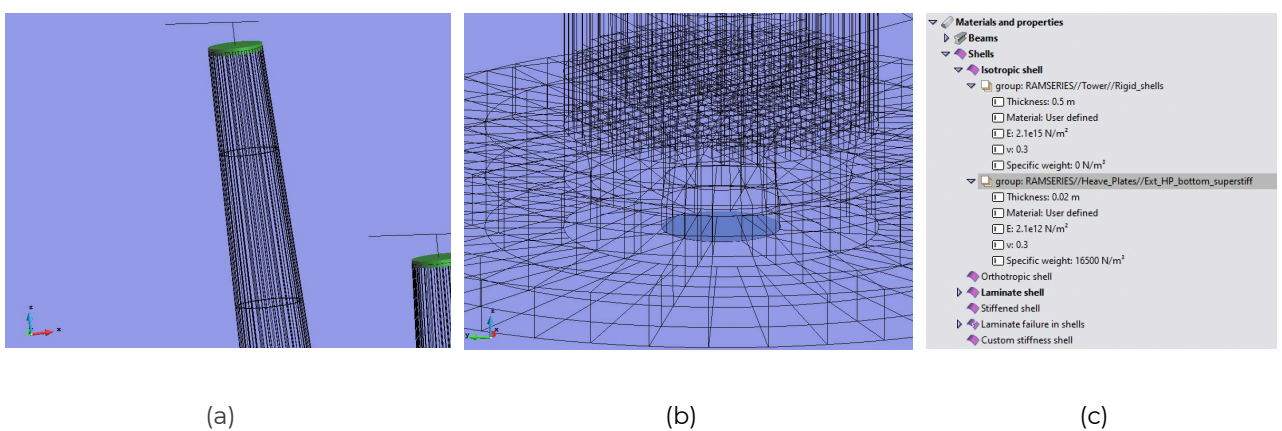


Figure 13. Materials of high stiffness. (a) Rigid shell on top of the tower, to support the nacelle and rotor loads. (b) Properties of the rigid materials. (c) Rigid material on the mooring connection, to avoid high stress concentration.

2.2.4. Loads assignment

The different design loads have been imposed according to the design assessment procedure. This assessment methodology is based on the standard of Bureau Veritas' Rule Note NI 572 DT R02 E.

The loads are detailed as follows.

2.2.4.1. Self-weight:

- o **Material:** Load corresponding to the self-weight defined for each structure part. It has been applied over the whole geometry.

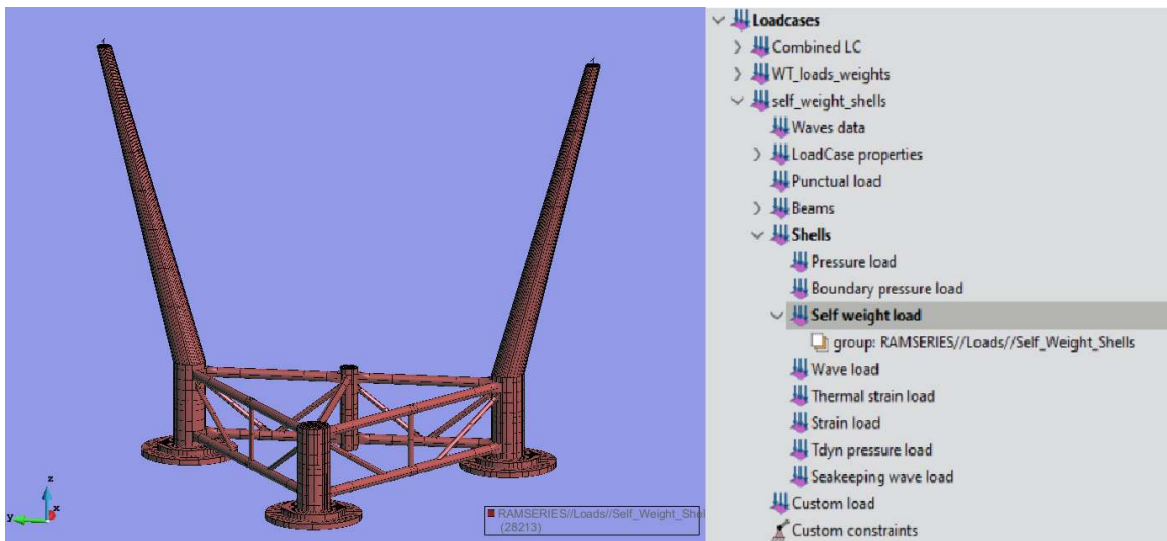


Figure 14: Self weight load assigned.

- o **Ballast:** The ballast is applied as a pressure load on the base of each platform tower. The load applied corresponds to the required ballast for obtaining the design trim and design draft of the platform. These loads are the following for each ballast tank:

Ballast tank	Ballast tanks A&C	Ballast tank B
Pressure applied (Pa)	2.382e+04	2.122e+04

In the next figure, the definition of the ballast load is shown:

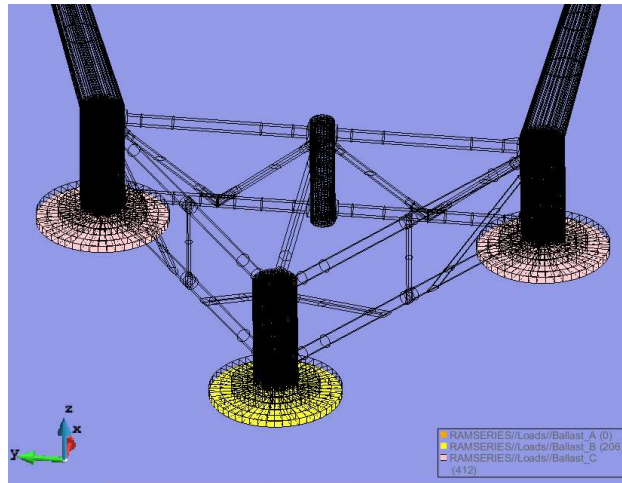


Figure 15. Ballast load definition.

- **Rotor and nacelle:** These elements have been defined as punctual loads equivalent to their mass, according to NREL (Jonkman, Butterfield, Musial, & Scott, 2009). The corresponding mass of the nacelle is 240000 kg, and 110000 kg for the rotor.

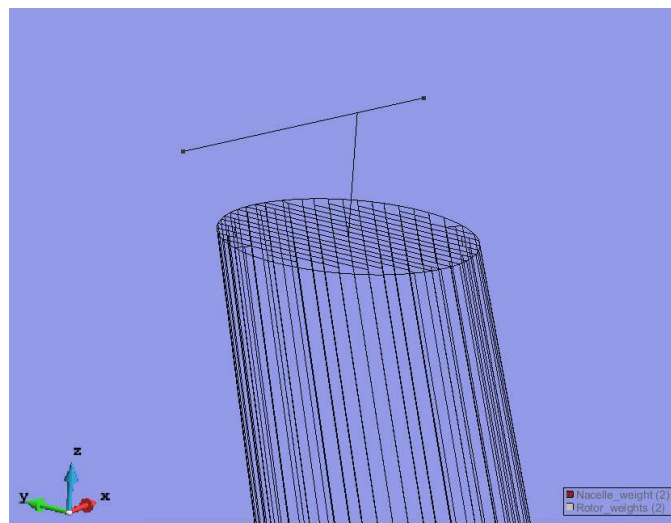


Figure 16. Rotor and nacelle load application.

2.2.4.2. Hydrostatic loads:

The applied hydrostatic load corresponds to the design draft given by the manufacturer. Note that the actual condition is applied to all the external hull surfaces of the platform, even above the draft reference, in order to account for the movements taking place in the dynamic analysis.

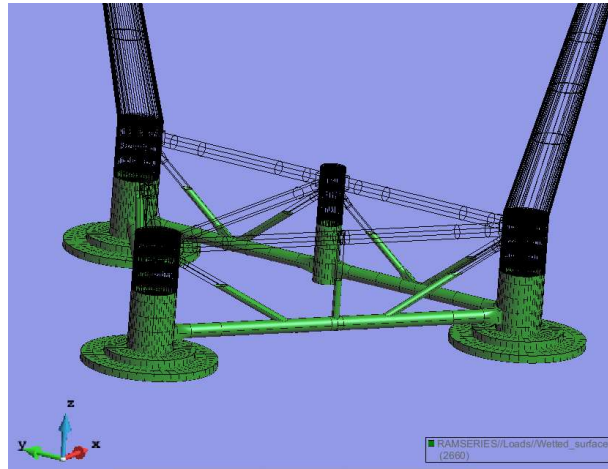


Figure 17. Hydrostatic load.

2.2.4.3. Wave loads:

The applied wave load is obtained from CompassIS' Tdyn-SeaFEM solver. The results of the hydrodynamic pressures are imported to the structural analysis. These loads are applied to the same surfaces as the hydrostatic load (same condition group is used).

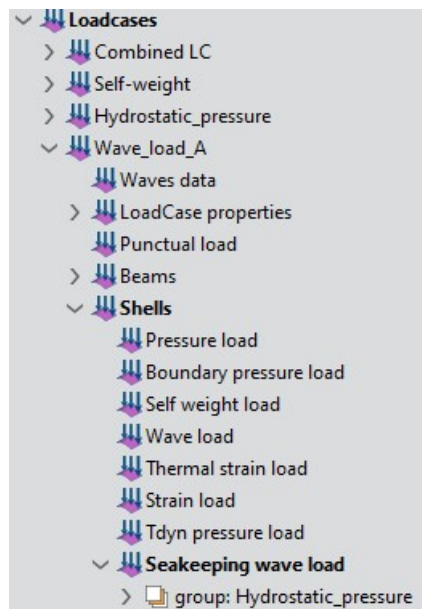


Figure 18. Wave load definition.

2.2.4.4. Wind loads:

The wind loads applied are the thrust on the rotor and the wind pressure on the tower. Thrust is applied by coupling Tdyn-Ramsereis-SeaFEM with OpenFAST, which is applied on the point corresponding to the rotor,

while the wind pressure is assigned over the towers' surface as a pressure load in function of the height, as it is defined in the BV Rules.

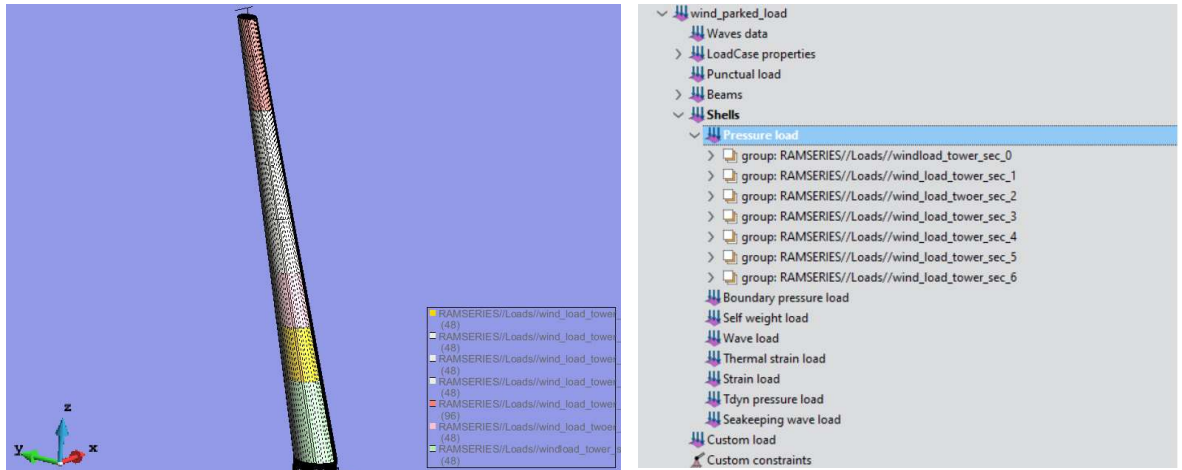


Figure 19. Wind loads.

2.2.4.5. Mooring load:

The pretension of the mooring system is applied on the platform, as it is defined on the design assessment methodology. Its value is 810 tn, and it is assigned as a punctual load in the vertical direction.

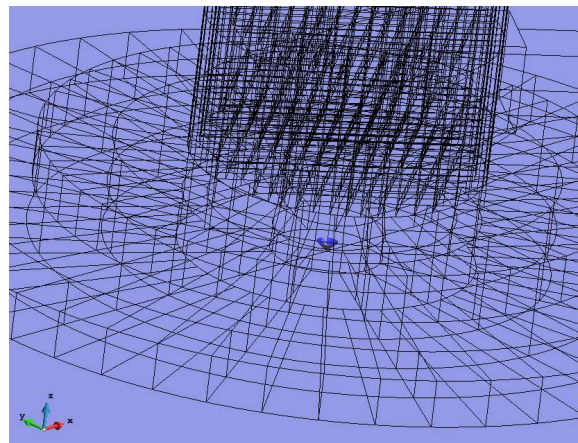


Figure 20. Mooring load.

2.2.5. Conditions assigned

The two types of conditions applied are dynamic conditions and boundary conditions. The first one is referred to masses and moment of inertias of the masses, to take into account inertial forces in the dynamic analysis. The second one is referred to restrictions and constraints.

2.2.5.1. Dynamic conditions:

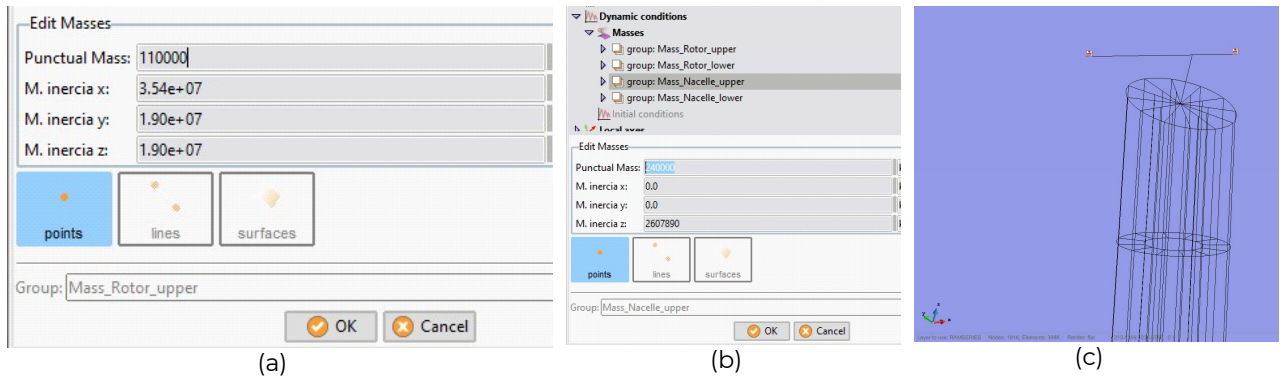


Figure 21. Dynamic conditions assigned. (a) Rotor mass. (b) Nacelle mass. (c) Assignment of masses.

2.2.5.2. Constraints

Elastic constraints have been applied in order to introduce the restoration effect of the mooring. The stiffness of the spring ($K_{mooring}$) is provided by linearizing the catenary equation of the mooring lines.

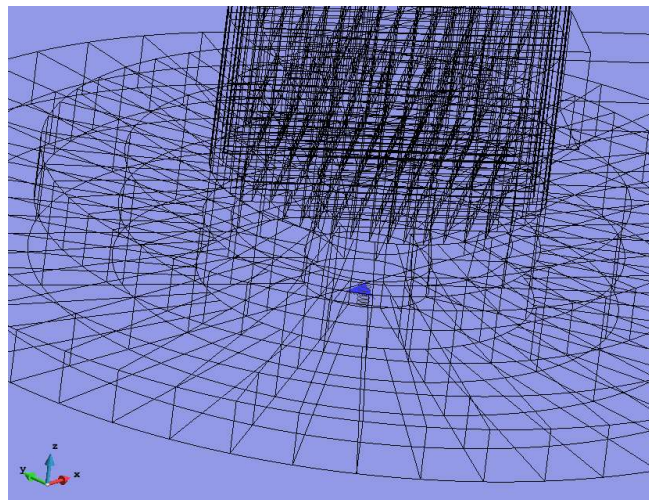


Figure 22. Elastic boundary conditions.

The elastic constraint acts on displacements, although the rotations are not restricted. The values of the equivalent linear stiffness are shown in the next table.

Axe	X	Y	Z
Stiffness [N/m]	3.0e+05	3.0e+05	8.2507e+04

2.3. FEM mesh generation and refinement for FEA

Different successively refined meshes have been generated, in order to provide consistency to the FEM solution. The meshes used 3-noded (linear) triangle elements with drilling-rotation formulation and line elements for beams. Unstructured meshes were used.

Static analyses have been performed with different meshes, taking the Von Misses stress on the material as a reference for comparing and checking the solution convergence. The meshes have been refined where highest stresses on the FRP platform were expected. The element size on the platform has been kept as a reasonably size to not distort the performance of the tower-platform. Once a first mesh was defined, the procedure was to scale up the mesh size, obtaining three meshes of different size. Table 2 shows the results regarding the different meshes used. Finally, size 2 was used for meshing the FEM model.

Table 2. Results from different meshes used.

	Minimum size of elements	Von Missess (Mpa)	Difference (%)
Mesh 1	0.50	1471	-
Mesh 2	0.25	1299	13%
Mesh 3	0.125	1273	2%

In the next figure, the comparison of different element size studied can be seen.

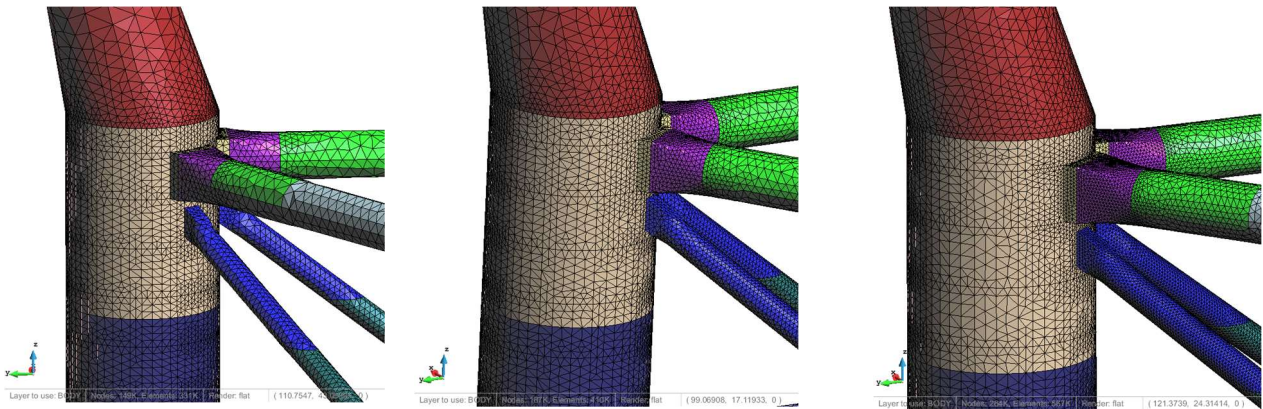


Figure 23: Size element comparison between meshes studied: (a) Mesh 1. (b) Mesh 2. (c) Mesh 3.

In the following pictures, the final mesh used for the whole platform-tower is shown in different details of the model.

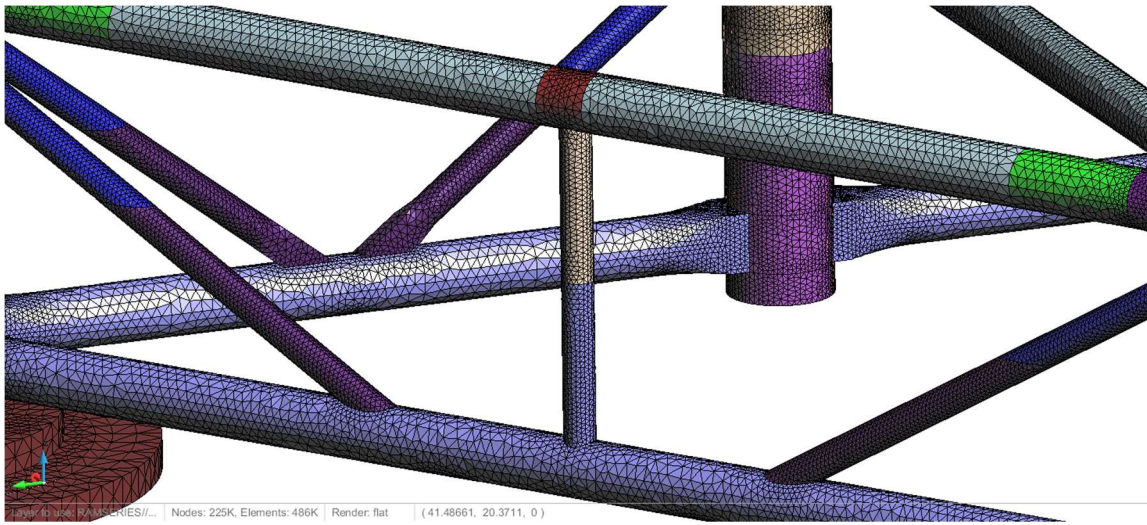


Figure 24. Global mesh view of braces.

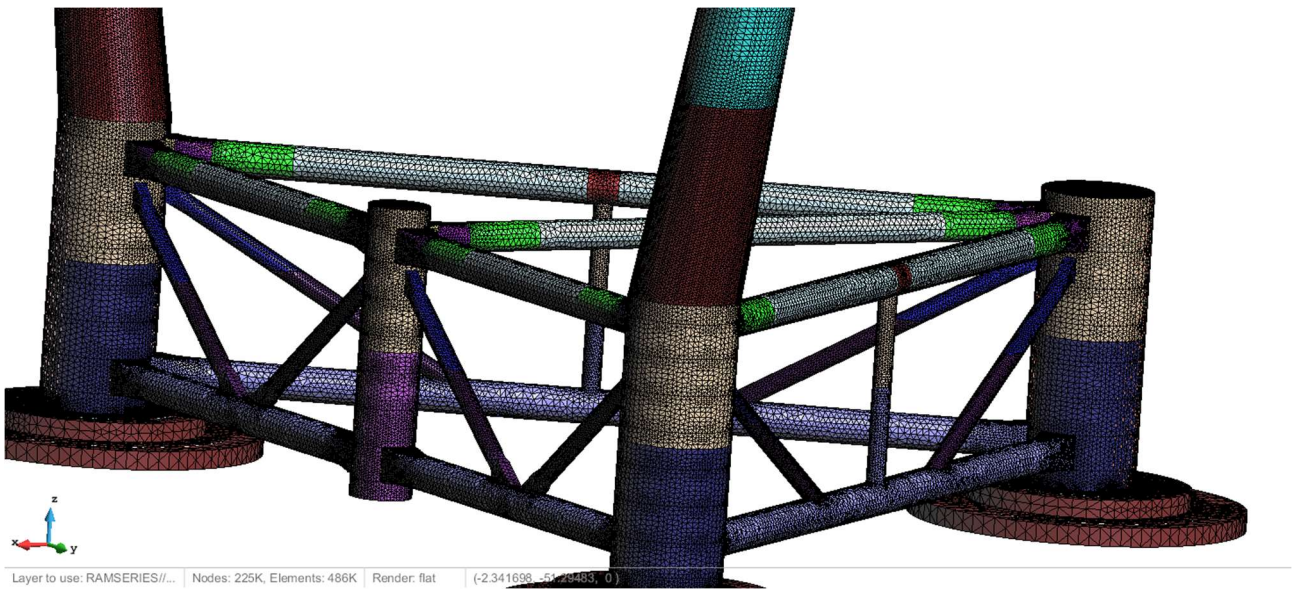


Figure 25. Global mesh view of the platform.

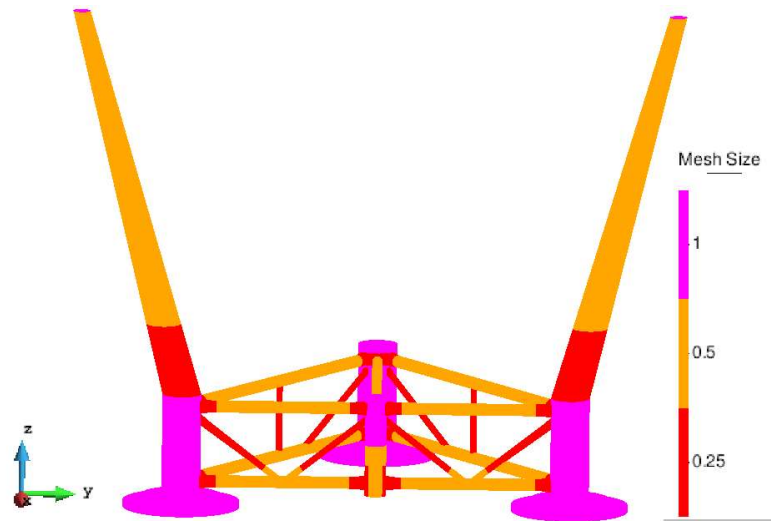


Figure 26. Global mesh size of the W2Power platform.

3. TIDETEC TURBINE HOUSING MODEL

The CAD geometry has been selected mainly considering the delimitations of the turnable area and the manufacture procedure (winding wire). From the original metallic geometry, a study with FRP materials with the same geometry took place in order to prove, with a static and modal analysis, that the new piece can withstand the loads. Afterwards, the external shape was redesign in order to be feasible with the manufacturability method selected.

As it is shown in the figures below, the shape is more of an oval sphere made of 4 parts; main body (1), where the shafts are attached, a cover (2) and two identical fairings (3 & 4).

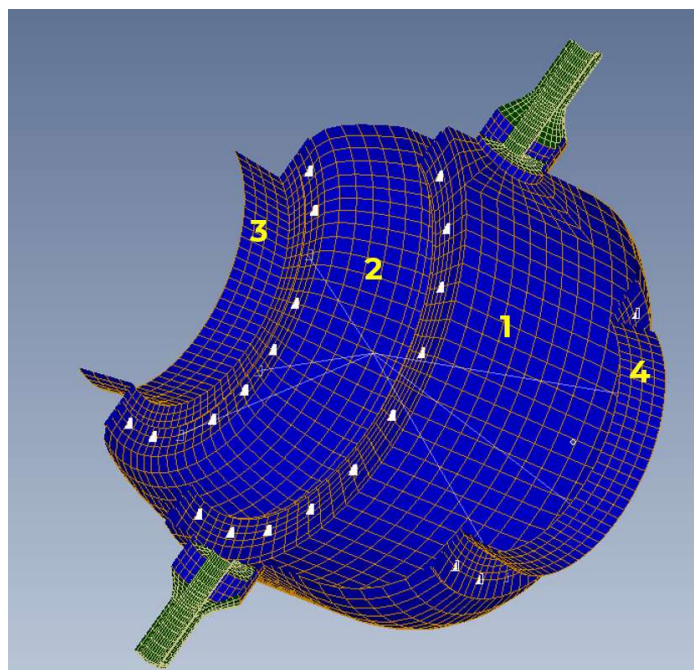


Figure 27. The main 4 parts of TIDETEC turbine housing model.

3.1. CAD model(geometry) for FEA

The dimensions of the original metal turbine provided by TIDETEC are:

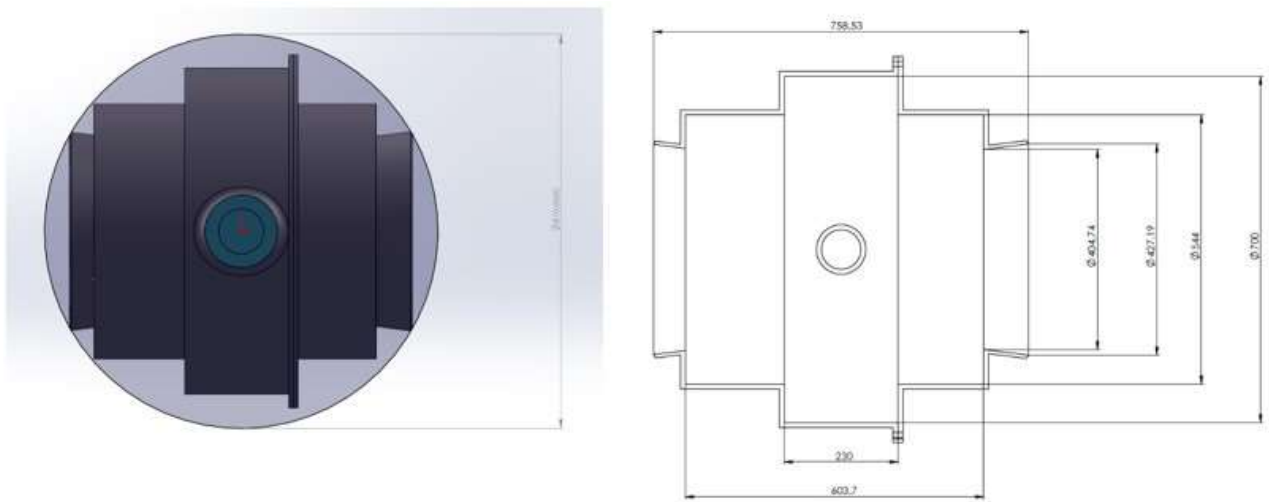


Figure 28. TIDETEC steel housing main dimensions.

From the operation and performance constraints, the geometry of the component, evolved as described in WP4, resulting in the shape and dimensions shown in the following figure: By similarity, this configuration has also taken as basis for manufacturing technologies demonstrator.

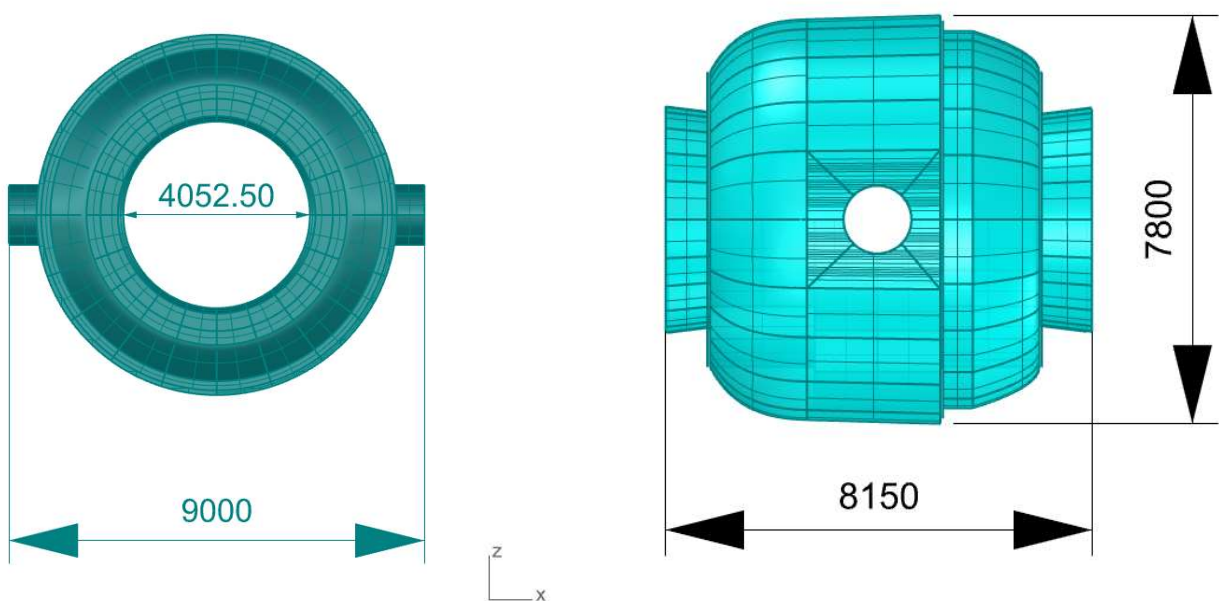


Figure 29. Main dimensions of the redesign housing.

A uniform mesh has been carried out on the generated surfaces, since the stress state of the overall component is well captured by this type of elements, especially when using 2D laminate elements as in this case. The following figure shows the discretization used.

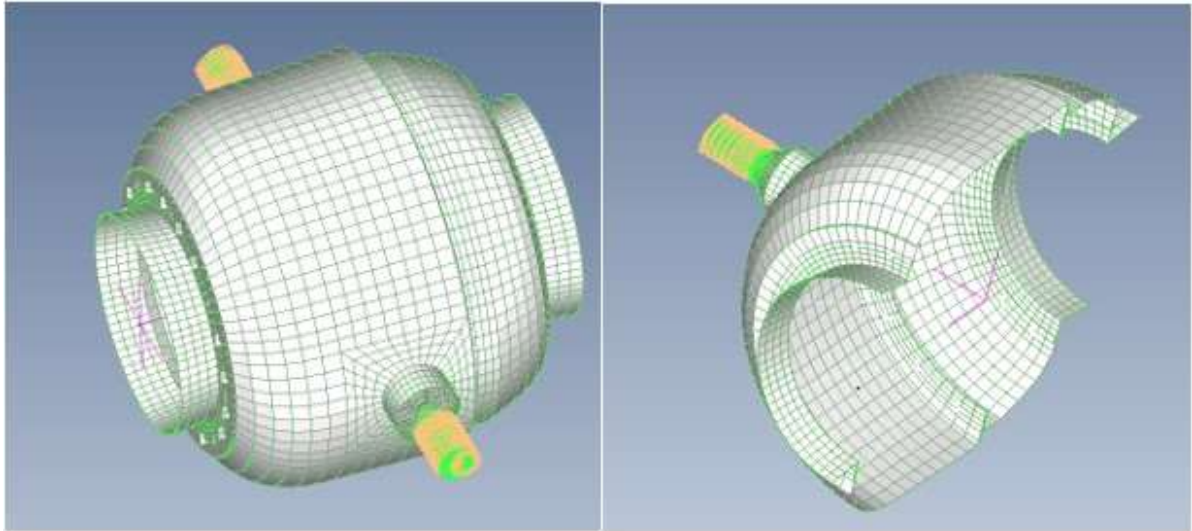


Figure 30. Finite Element Model of the redesign FRP housing.

3.2. Conditions assigned

The FRP system in use for the alternative turret design is based on two possible combinations as detailed in Table 2, "Elium 191 O/SA with Hybon 2026", and "T700 with Infugreen 810", which were kindly suggested by the research institute INEGI. For the time being, a coupon testing is under progress with the aim to have a clearer idea of the material properties/performance of both FRP systems as well as to conclude which configuration is more friendly manufacturer. For structural analysis purposes, TSI has selected the first architecture "glass fibre-reinforced composite" as baseline for the turret design.

Table 3. FRP materials proposed for the redesign of the turret.

	Architecture 1: GFRP		Architecture 2: CFRP	
	Resin	Fiber	Resin	Fiber
ID	Elium 191	Hybon 2026	Infugreen 810	T700
Manufacturer	Arkema	Nippon Electric	Sicomina	Toray

The material constants used to simulate the GFRP laminate are detailed in Table 4, where TS stands for tensile strength, E is the young modulus, ν is the poisson's ratio, ρ is the density of the materials, and G is the shear modulus, ξ is the elongation max, and C is the compressive strength. Further information about the materials of the turret design can be found on the data sheet of the glass fibre "Hybon 2026" and the resin "Elium 191" on annexes 7.1 and 7.2. It should be noted that the values of the shear modulus, Poisson's ratio

for the resin and glass fibre are not given in the data sheet and therefore, it has been extracted from typical values for similar materials (Defense, 1999).

The properties of the epoxy-based laminates are calculated according to the rule of mixture as indicated in reference (Mallick, 1997). The mass density of the ply fibre-based epoxy interfaces is composed of a 54% glass fibre and 46% of resin is 1.81 g/cm³. The estimated bundle diameter or ply thickness for each layer of the FE model is 1.5 mm. This value is suitable only for the roving format, in which the material is presented in bundles. For the present application, a thickness of 0.254 mm for each ply is selected for the turret design.

Table 4. Properties of epoxy-based resin and glass fibre from Arkema and Nippon.

	Units	Abbreviation	Resin	Glass
Tensile Strength	MPa	TS	66	2000
Young modulus	MPa	E	3170	78500
Poisson's Ratio	DL	v	0.39	0.22
Mass Density	g/cm ³	p	1.01	2.5
Shear Modulus	MPa	G	1140	33000
Elongation max	ξ	%	2.8	
Compressive Strength	MPa	C		4500

To set up the simulation, there are some boundary conditions and assumptions considered. The boundary conditions include the self-weight of the structural components (turret, generator, turbine), the radial force, the total axial force, and the total drag force. The axial force, radial force, and drag forces are applied only when the turret is in an operational mode. For maintenance mode, there is no water flow through the turbine, so only the self-weight is considered.

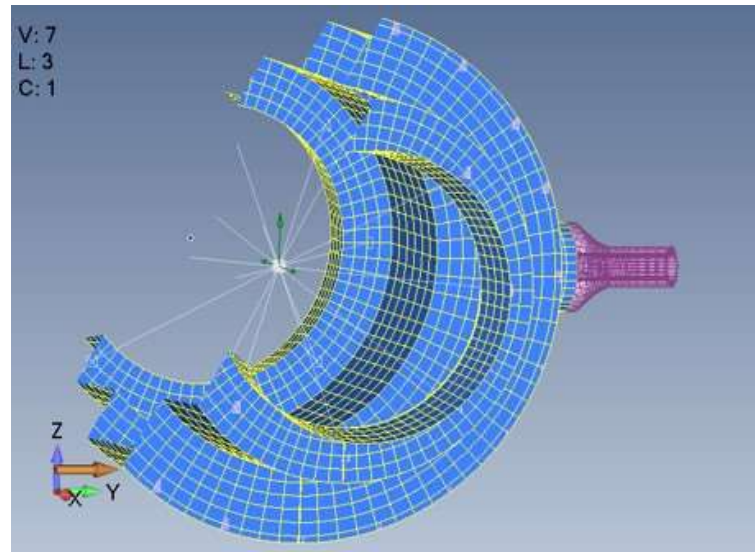


Figure 31 Turret loads applied on FE model

All loads are applied at the centre of mass of the turret/turbine/generator. This corresponds just to the left of the centre of the turret, under the centre flange of the turret. The loads are connected to the bolt holes where the turbine attaches to the turret. The turbine and generator are considered rigid bodies because significant deformation is not expected with steel materials. All the forces on the system will be applied as static loads. These conditions will be applied for all three scenarios. Figure 31 shows how the remote load (represented as a dot) connects to the eight bolt holes on each side of the turret.

There are three scenarios of the turret considered for the analyses, maintenance mode, runaway turbinning and normal operation pumping. Table 5 includes all the loads applied for the three scenarios.

Table 5. Loads on turret for the three scenarios.

	Maintenance Mode	Runaway Turbinning	Normal Operation Pumping
Self-weight	3737.61 N	3737.61 N	3737.61 N
Total Axial Force	-	11380.40 N	-6036.63 N
Radial Force	-	600 N	-500 N
Total Drag Force	-	11481.80 N	5189.75 N

The connections between the different parts of the turret, main body, aft cover and flanges are modelled by means of rigid connections (RBE2). The connection between the turbine and the turret has been made coincident with some internal interfaces, namely the forward and aft fairings with the main body.

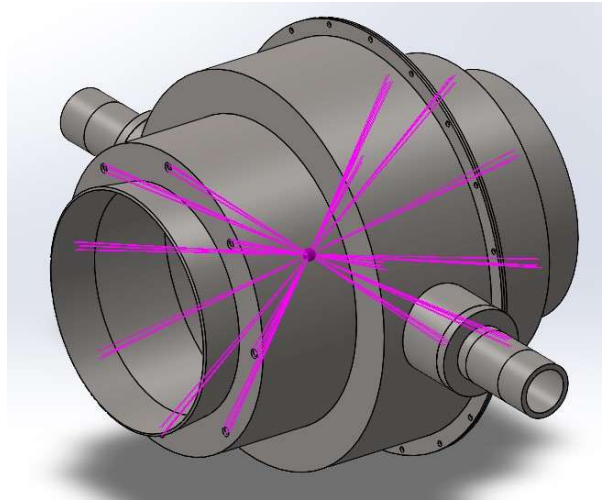


Figure 32. Turbine and turret connection by means of rigid elements.

3.3. FEM mesh generation and refinement for FEA

The finite element model is discretized with 4576 linear shell elements “CQUAD” for all the surface elements in GFRP. The shaft elements (isotropic Steel material) are modelled with 4096 linear “HEXA” elements. 29 rigid elements are used for the simulation of the bolted joints and connections of the turbine. The total number of elements of FE model is 8702. The laminate material is oriented with its 0° reference in axial direction.

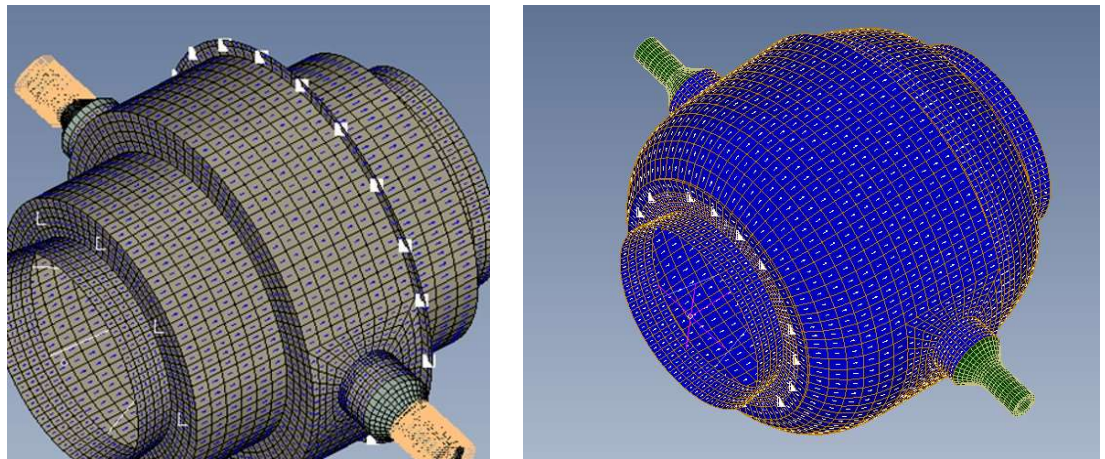


Figure 33. Initial FRP housing design.

The final figures of the model after the redesign process are presented in the following table

Table 6 model summary for the redesign FRP housing

MODEL SUMMARY	
NUMBER OF GRID POINTS	10271
NUMBER OF CHEXA ELEMENTS	3840
NUMBER OF CONM2 ELEMENTS	1
NUMBER OF CQUAD4 ELEMENTS	9728

NUMBER OF RBE2 ELEMENTS	53
--------------------------------	-----------

The quality of the mesh is good as shown in the following figure.

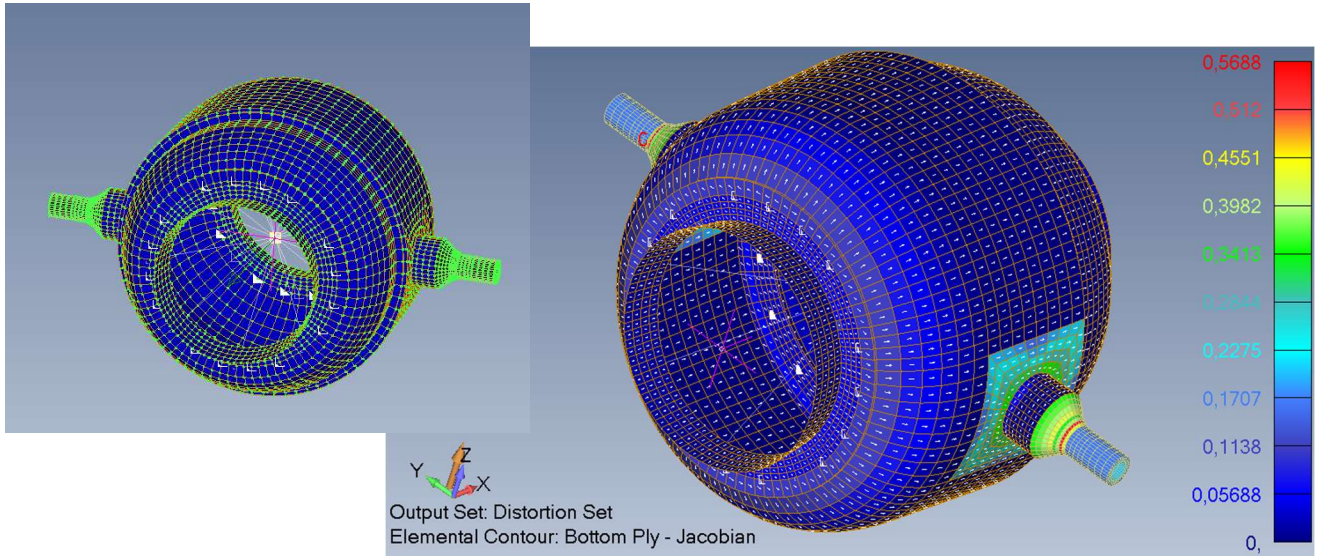


Figure 34. Final FEM of the redesign FRP housing. Mesh quality indication.

The joints between the FRP shells and metal supports, modelled by HEXA elements, are achieved by means of shared nodes as shown in the following figures.

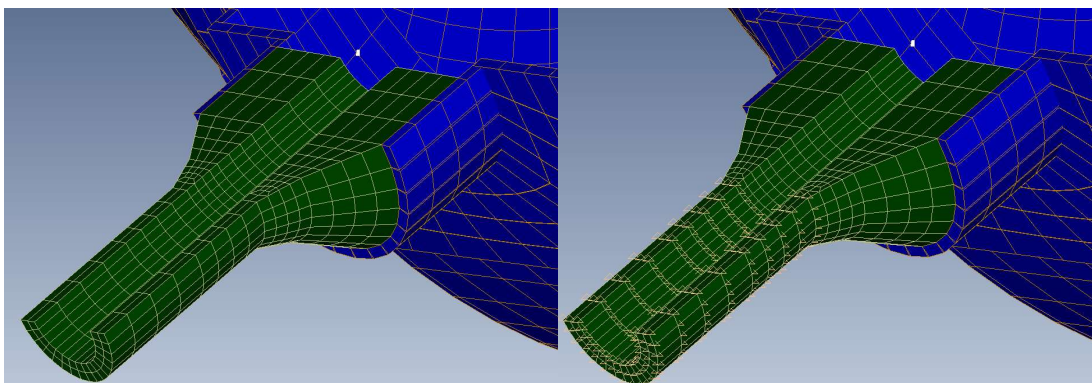


Figure 35. Metal shaft with (right) and without (left) constraints.

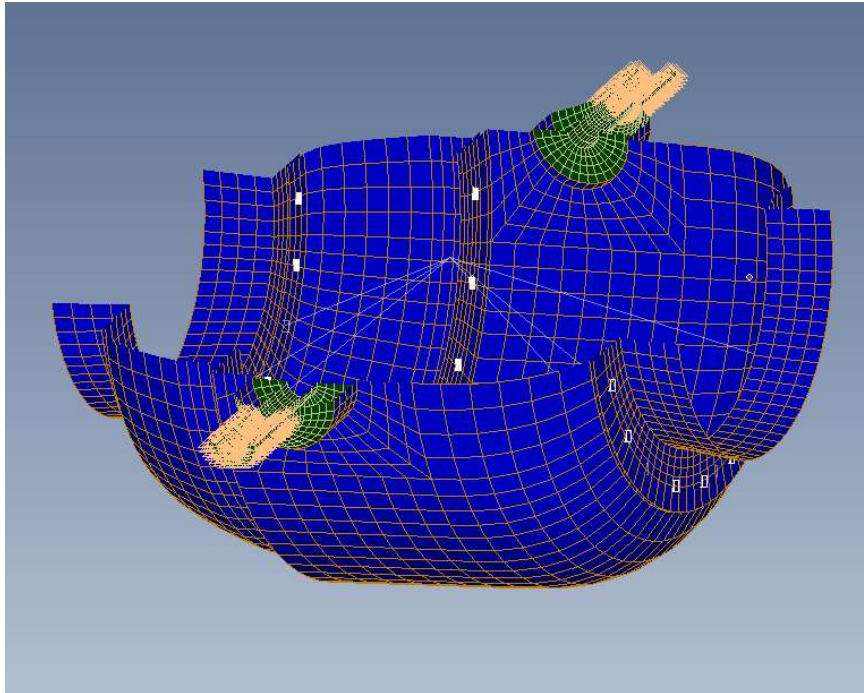


Figure 36. The overall FEM, with loads, constrains and mesh.

4. CONCLUSIONS

Two complete 3D CAD/CAE models of Offshore Wind and Tidal Stream Platforms (OWTPs) have been conducted. Both models include all the structural details required to assess and analyse their structural behaviour, subjected to realistic loads and conditions.

The CAD models describe the geometry of each device. More specifically, the W2Power model describes the platform, tower and braces of the floating offshore wind turbine. On the other hand, the TIDETEC turbine model encompasses the FRP housing of it.

The materials used on each CAE model are described according to the design work conducted on WP4. Boundary conditions have been applied according to the loading scenarios of each platform, as well as realistic loading conditions.

Finally, mesh convergence studies have been conducted in both models, in order to avoid meshing dependence of the results, although assuming an affordable computational cost.

To sum up, both CAD/CAE models are ready to be used by the engineering teams in order to assess and study their structural performance.

5. REFERENCES

- Defense, U. D. (1999). *Composite Materials Handbook-MIL 17, Volume III: Materials Usage, Design, and Analysis*. Washington: CRC Press, Taylor & Francis.
- Jonkman, J., Butterfield, S., Musial, W., & Scott, G. (2009). *Definition of a 5-MW Reference Wind Turbine for Offshore System Development*. National Renewable Energy Laboratory (NREL).
- Mallick, P. (1997). *Composites Engineering Handbook*. Boca Raton: CRC Press, Taylor & Francis.
- Zhao, Y., Yang, J., & He, Y. (2012). Preliminary Design of a Multi-Column TLP Foundation for a 5-MW Offshore Wind Turbine. *Energies*, 3874-3891.