



Laboratório Nacional de Energia e Geologia, I.P.  
**UNIDADE DE ENERGIAS RENOVÁVEIS E INTEGRAÇÃO DE  
SISTEMAS DE ENERGIA**

## **Megafloat inicial state of Art**

**Relatório de Trabalho**

**Margarida Giestas**  
**LNEG, dezembro 2020**

## Conteúdo

<b>Conteúdo .....</b>	<b>2</b>
<b>1. State of Art .....</b>	<b>5</b>
<b>1.1 Introduction .....</b>	<b>5</b>
<b>1.2. Linear elastic material .....</b>	<b>6</b>
<b>1.3. Modal analysis .....</b>	<b>6</b>
<b>2.Is there any plate or shell element in ANSYS, to include transverse shear steel effect? .8</b>	
<b>3.How can one merge nodes on two co-planar faces in Ansys WB? .....</b>	<b>8</b>
<b>4.Analysis of Composite Plates by ANSYS .....</b>	<b>8</b>

# Summary

## WindFloat

### FCT Project submitted by Paulo Justino (2020)

#### Relevant Tasks for the author of this report

##### Task 1. Numerical modelling of structure elastodynamic response to loads

The object of this task is the numerical response of the overall platform modelling and computation of the elastic when subjected to known solicitations (pressure, point-loads or displacements) from the supporting cylinders and/or any other servicing situations.

###### Subtask 1.1 – Modal analysis of the platform

A computational model of the platform will be produced using a commercial finite element code (ANYSY). Due to the relative dimensions of the platform, the structure will be modelled with appropriate plate elements, material description and boundary condition applied in order to reproduce the real platform working condition.

###### Subtask 1.2 – Analysis of the platform with known applied solicitations

The previous platform model will now be analysed for known applied solicitations for different frequencies in order to model the wave motion influence. The model will be tested for several loading situations to assess its robustness.

The expected result from this task is to provide a computational model for the linear elastic analysis of the platform that can, in an automatic way, perform a modal analysis, perform analysis for several excitation loads, can provide results for displacements and stresses on the platform and can be easily integrated with the wave hydrodynamics module and wind load results of subtasks 2.2 and 3.2.

##### Task 3. Numerical modelling of wind-structure interaction

The object of this task is the numerical modelling and computation of the wind loads on the structure and the produced wind energy. accelerations that will influence the turbine's response (as it happens with WindFloat).

###### Subtask 3.1. Wind records

Since the platform will be designed to be deployed in water depths in the range 40-80m (a set of three locations will be used), records for wind velocities, average values, standard deviations, and

directions will be obtained for these three locations using historical long term observational wind data and simulated wind data driven by atmospheric mesoscale models already available at LNEG.

Subtask 3.2 Wind loads and wind turbine performance.

A CFD model that takes into consideration the presence of the wind turbine will be developed to compute the wind loads on the platform (LNEG already uses CFD for other applications. Stresses applied on the deck due to the presence of the wind turbine, its loads (namely aerodynamic steady flow loads), will be computed for a range of wind velocities and directions. A set of results, dependent on wind conditions already defined in subtask 3.1, will be used in Tasks 1. and 4. Also, the set of wind records together with a turbine power curve will allow to estimate the amount of power produced by the turbine for the three locations.

The expected results from this task are to provide, representative wind records for the possible platform locations; results from the numerical modelling and computation of the wind loads on the structure and produced wind energy. This task will interact with Task 1.

#### **Task 4. Optimization**

The integration of (i) wave hydrodynamic, (ii) elastic and (iii) OWC computing modules and payloads solicitations will be achieved in this task.

The three computing modules will be integrated into a single application having as input the incident wave description and the payloads on the platform deck including wind loads, and as output the platform motions and stresses, the drift forces, the OWC motions and the wave and wind energies production.

The resulting code will be run (i) for a small set of platform configurations (aircushion OWC size and array arrangement) and (ii) for a set of regular waves and several representative irregular wave sea states, as well as several aerodynamic loads on the turbine, for the three locations mentioned in Task 3. This will be done for optimization purposes having in view the mitigation of the hydro-elastic response and of the drift forces, and the estimate of the produced energy from the waves and wind. The expected result from this task is the selected configuration of the platform and its hydro-elastic response, as well as energy produced from the waves and drift forces, when subject to representative sea / wind states.

This task will be very time consuming due to the nature and amount of the proposed computations.

# 1. State of Art

## 1.1 Introduction

Ocean space utilization has been a key issue for Japan. Until the 20th century, reclamation of shallow waters has been the only technology available expand human activities onto the sea. Architects had proposed the concept of ocean space utilization using a floating structure. Shipbuilding technology had attracted the attention of Japanese architects in the late 1950s, and there was movement in architecture and urban design to utilize ocean space and expand human habitat onto the ocean. The floating city project started at the University of Hawaii in 1971. The Okinawa International Ocean Exhibition was held in Japan in 1975. Aquapolis, was constructed for this exhibition as a large semi-submersible unit of a floating city. In the 1970s and 1980s, concepts and proposals of floating cities were published. A floating airport was proposed for both phase 1 and phase 2 construction of Kansai International Airport in 1973 and 1994.

### ***Research of TRAM ( Technological Research Association of Megafloat)***

The research of TRAM may be catergorized into six research areas that follow the flow of the realization process of Megafloat. Fig. 1 shows the process of realization of Megafloat and research areas worked on by TRAM. The research areas are design, government approval, construction and functionality, and environment. Design-related research The dynamic behavior of the Megafloat is characterized by its hydroelasticity. A group of analysis programs were developed with various complexities.

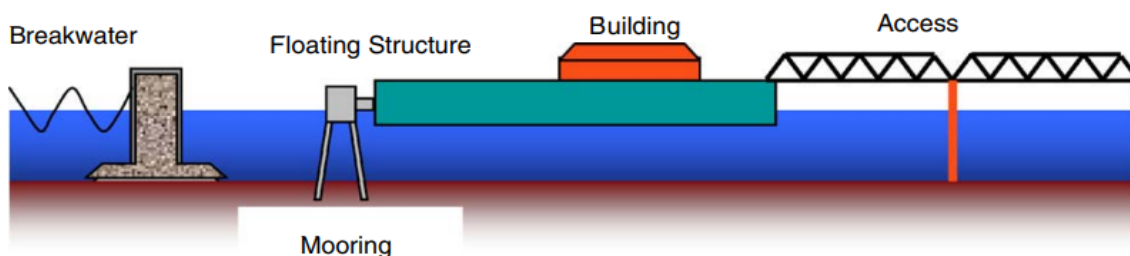


Fig .1 Concept of a Megafloat

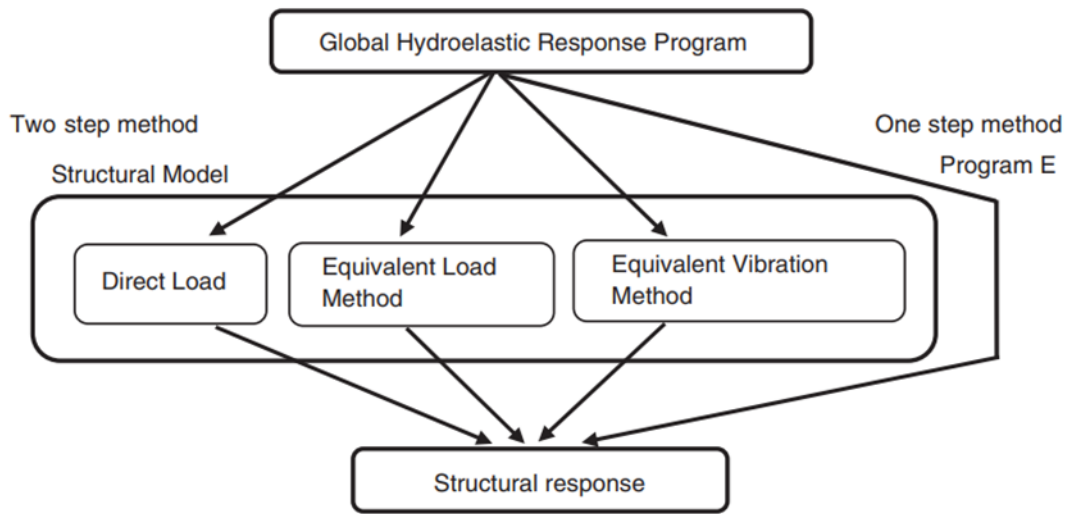


Fig2. Analysis program and procedure for the structural analysis

## 1.2. Linear elastic material

A **linear elastic** material is a mathematical model used to analyze the deformation of solid bodies. It is useful to compute the relation between the forces applied on the object and the corresponding change in shape. In other terms, it relates the stresses and the strains in the material.

## 1.3. Modal analysis

**Modal analysis** is a technique to study the dynamic characteristics of a structure under vibrational excitation. Natural frequencies, mode shapes and mode vectors of a structure can be determined using **modal analysis**. ... A graphical variation of number of modes vs the frequency can also be obtained from **ANSYS** Workbench.

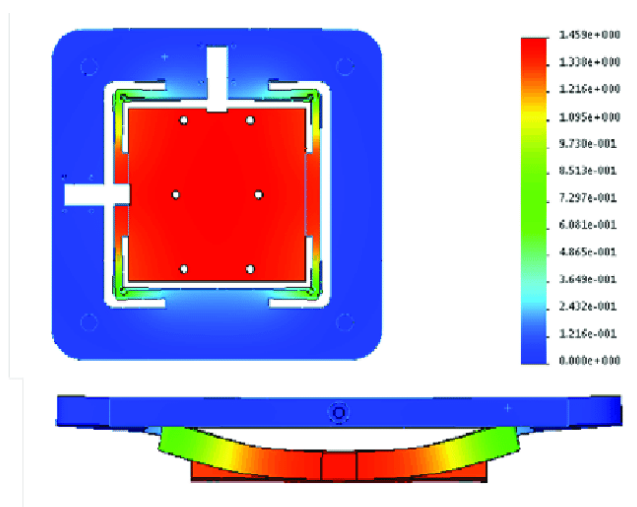
We often hear the terms “modal analysis,” “eigenvalues,” “eigenmodes” or “eigenfrequencies”, and by the time one comprehends a single terminology, a new terminology emerges. People always prefer to make things sound more complicated than they need to be.

More often these terms develop out of necessity, but many times it's to express things with mathematical accuracy. Let's explain the difference between quasi-static, dynamic, and modal analysis.

The most common type of analysis is quasi-static analysis, where the load is applied at a very slow rate so that the acceleration is negligible (or almost zero). Dynamic analysis is where the effects of acceleration cannot be ignored. Both types provide a one-to-one relationship between a particular input (for example, a force applied on a system) to its system response (for example, a displacement of the system due to its load).

In contrast to quasi-static and dynamic, modal analysis provides an overview of the limits of the response of a system. For example, for a particular input (like an applied load of certain amplitude and frequency), what are the limits of the system's response (for example, when and what is the maximum displacement).

Every object has an internal frequency (or resonant frequency) at which the object can naturally vibrate. It is also the frequency where the object will allow a transfer of energy from one form to another with minimal loss—here it is from vibrational to kinetic. As the frequency increases towards the “resonant frequency,” the amplitude of response asymptotically increases to infinity. In other words, the result of modal analysis are these frequencies at which the amplitude increases to infinity



(a ) Mode shape at the first natural frequency (545Hz)

## **2. Is there any plate or shell element in ANSYS, to include transverse shear steel effect?**

Effect of horizontal steel is incorporated using layering technique. Is there any standard package plate/Shell element that includes the stiffness contribution from transverse stirrups? Or can anyone suggest how that effect can be accommodated in plate/shell elements?

That is really a challenge!! I think it does not exist a plate/shell element with the effect of transverse stirrup (I know many researches are working to find a way to include stirrups in fiber-beam based elements). I suppose it could be included only at the material level (concrete law) by using a classical confinement rule such the one proposed by Mander. Or, on the other side, to create a solid/brick model with embedded reinforcement.

## **3. How can one merge nodes on two co-planar faces in Ansys WB?**

I have a steel structure and the welds need to be merged in share topology in SpaceClaim in order for the elements of both parts to match.

But the contact between beam must no be merged in shared topology so that I can make a contact frictionless in Mechanical.

I can only manage to either separate all bodies so that they all make contacts (but mesh does not match) or merge all bodies so that mesh matches (but no contact surfaces are created and I cannot make the beam-beam contact frictionless).

## **4. Analysis of Composite Plates by ANSYS**

ANSYS, Inc is a computer-aided engineering software provider. Their products include general-purpose FEA packages for numerically solving a wide variety of mechanical problems. For structural analysis, ANSYS Mechanical can solve linear or nonlinear static or dynamic problems. ANSYS FEA software is used throughout 5 industry to enable engineers to optimize their product designs. ANSYS capability and functionality to model composite structures is given below. In general, ANSYS can model composite structures by solid element, solid shell element and shell element. ANSYS provides ANSYS Composite PrepPost (ACP) to facilitate building finite element models (FEMs) and access results. So we

will first introduce solid and shell elements, and then ACP. For solid elements, ANSYS provides 8-node structural solid SOLID185 and 20- node structural solid SOLID186. Both of them are based on 3D Cauchy continuum model with three degrees of freedom at each node: translations in the nodal x, y, and z directions. SOLID185 is a linear element whereas SOLID186 is a quadratic serendipity element.

However, real composite plates could have very complex definitions that include numerous layers, materials, thicknesses and layup orientation. Without the proper FEA tools to set up a model, one may spend too much time before one can actually get meaningful results.

Even if the solution is done, it is very tedious to extract all the results needed such as the stress/strain field and failure index within every layer. To handle this problem, ANSYS Composite PrepPost (ACP) is designed to perform preprocessing (build model) and postprocessing (extract solution) easier. ACP has a pre- and post

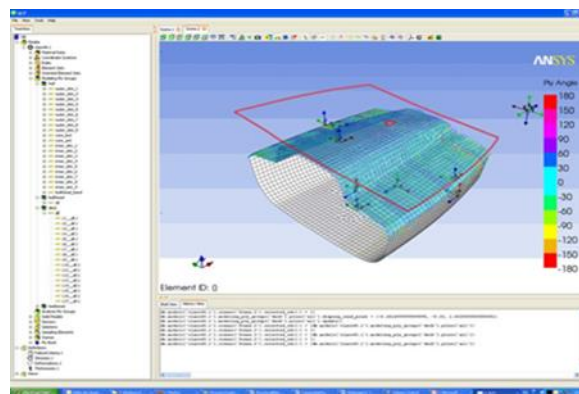


Fig.4. ANSYS Composite PrepPost processing mode.

In the pre-processing mode (Fig.4), all composite definitions can be created and are mapped to the geometry (FE mesh). In the post-processing mode, after a completed solution and the import of the result files, post-processing results (failure, safety, strains and stresses) can be evaluated and visualized.

This is important to make the distinction between structures and materials, which are often used interchangeably in literature. A structure is a solid body made of one or more materials with clearly defined interactions with its external environment through boundary conditions,

applied loads, and environment effects like temperature and moisture. It can be modeled using one or more typical components as shown in Fig. 4. If all three dimensions of a component are of similar size, it can be called a 3D structure; if one dimension is much smaller than the other two dimensions, it can be called a plate/shell; if two dimensions are much smaller than the third dimension, it can be called a beam.

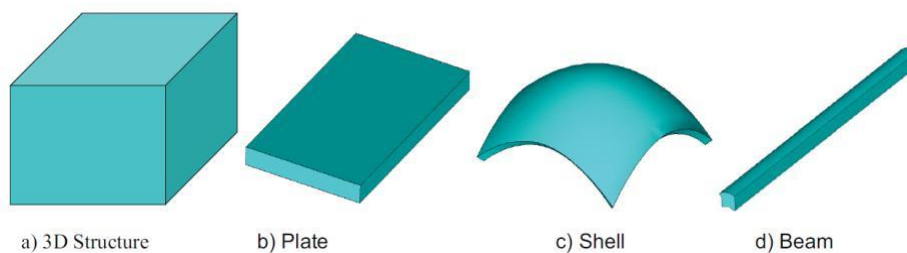


Fig. 5 Typical structure componentes

Many models have been developed for these structural components.

The classical ones are Cauchy elasticity for 3D structures, Kirchho -Love model for plates/shells, and Euler-Bernoulli model for beams. 3D structures can be considered as 3D models because the unknown functions are three-variable functions with three coordinates as independent variables.

Plates/shells can be considered as 2D models because the unknowns functions are two-variable functions with two in-plane coordinates as independent variables. Beams can be considered as one-dimensional (1D) models because the unknowns functions are one-variable functions with the axial coordinate as the independent variable. To this end, beams/plates/shells can be collectively called dimensionally reducible structures as the models of these structural components are reduced from the original 3D model.

*These are some lines about the proposed project.  
They are based in some literature, blogs and Researchgate literature.*