The finite element method for calculating the marine structural design

A Ion¹ and I Ticiu²
¹ “Constanta Maritime University” – Romania, Department of Navigation
104 Mircea cel Batran Street, Constanta, 900663
² “Constanta Maritime University” – Romania, Department of Fundamental Sciences and Humanities 104 Mircea cel Batran Street, Constanta, 900663

E-mail: alina_ion@rocketmail.com

Abstract. The aim of this paper is to optimally design and dimension marine structures in order for them to fulfill both functional and safety requirements. A master level of structural mechanics is vital in order to check tests and analysis and to develop new structures. This study can improve the calculation and estimation of the effects of hydrodynamics and of other loads; movements, strains and internal forces in fixed and floating platforms and ships. The finite element method (FEM) ensures basic understanding of the finite element model as applied on static cases including beam and plate elements, experience with static analysis of marine structures like platforms and ships, along with the basic understanding of dynamic response of systems with one degree of freedom and simple continuous beams, and also how analysis models can be established for real structures by the use of generalized coordinates and superposition.

1. Introduction
The Finite element method (FEM) allows to numerically solve the most diverse problems of solid Mechanics (which also includes the Structural Mechanics of ships), but also from many other areas (Fluids, Thermodynamics, Electromagnetism, etc.). The FEM idea consists in modeling the field studied, through a number of elements with finite dimensions, connected to each other in a number of nodal points in which the solution is sought. The operation is called finite element mesh. In structures, the values of linear and, sometimes, angular displacements are determined in nodal points. The Finite element model must simulate as correct as possible the behavior of the real structure which is and remains continuous even after deformation and each infinitesimal element inside of it or on its boundary should be in equilibrium under the action of internal and external forces. Through the very idea of FEM, elements in which the actual structure is divided, are satisfying the continuity condition at nodes, and in the preparation process of the FEM equations system, the nodes balance is also achieved. Based on them, movements, specific strains and stresses at any point of any finite element are determined.

In principle, the finite element method has the following main steps: meshing the structure into finite elements; choosing the form functions; establishing the finite element equation and its specific matrices; assembling the finite elements and implementing boundary conditions; solving the system of equations; determining the stress and strain state in each finite element [9].
2. Structure mesh into finite elements
The structure mesh into finite elements represents the selection of the nodal points in which it is desired to obtain the finished solution and to set the affiliation of finite elements to the selected set of nodal points. This operation requires not only experience, but also some knowledge regarding the response of structures with certain geometries to various types of external actions (forces, imposed displacements, temperature variations). The structure mesh must, therefore, take into account geometry and how to load it, links, symmetry, possible proximity with infinite media, concentrations of power etc. Depending on the geometry of the meshed structure, elements can be mono-dimensional, two-dimensional or three-dimensional. Since most software dedicated to the finite element analysis has limited modeling capabilities, it is recommended to create geometry in dedicated CAD programs. Meshing is actually the process by which geometry representing the field with infinite degrees of freedom is replaced by a network with a finite number of elements to simulate as closely as possible the given geometry. The loading type and how it applies determine a specific behavior of the structure and, thus require a certain manner to mesh it. In points where external forces are applied, links or required movements, as well as in those where stiffness changes or characteristics and / or material changes occur, nodes are necessarily provided. The density of the nodes network determines the size of finite elements. Reducing these dimensions, thus increasing the number of nodes generally leads to increasing the accuracy of the calculation. In determining the network nodes, the following factors should be considered: achieving the optimum between the accuracy pursued, the computer performances and the analysis cost. The excessive increase in the number of nodes may not be the ideal solution. The tests demonstrated that there was a limited number of elements that if surpassed, the solution did not improve, in fact it could even get worse. ANSYS program has applications in almost all fields of engineering: structural, mechanical, electrical, electromagnetic, electronic, thermal, fluid and biomedical. Usually, solving in ANSYS includes: building the finite element model; applying geometric and mechanical boundary conditions (loads) and obtaining the solution; viewing results. Building the finite element model consumes the most time in an analysis. The element type determines the types of degrees of freedom, the element shape and the analysis size. The finite element model can be created in two ways: by modeling and direct engendering. By modeling, the analyst means creating geometry, which then the program automatically fills with nodes and elements; the model can be controlled by the user at the level of size and shape of the elements. The direct engendering means manually creating the finite element model; the user creates each node, and elements are after defined based on the nodes created. The analysis type is determined depending on the load conditions and on the response of the structure. Once the results determined, they can be viewed by using a post-processor [4].

3. Solutions and experimental tests on the dynamic analysis of a hull through FEM
Vibrations are dynamic phenomena, most often unwanted, manifested in almost all fields. In principle, the mechanical vibration occurs after an exchange of energy between two components of a system: one that develops kinetic energy and another that stores potential energy. The energy is transferred from one component to another, thereby generating swinging motion around a position considered of equilibrium. If the movement stops after a while, it is said that the vibration is damped, i.e. the system loses energy through various forms: friction, viscous or structural damping. Being an undesirable phenomenon, it occurs most often accidentally or it is uncontrolled. It may happen that after the removal of a system from the balance, the disturbance would cease. That is the case of free vibrations. But if disturbances tend to repeat or last a longer period of time, it is said that vibrations are maintained. The characteristics of disturbances transmitted to the system are generating deterministic or random vibrations [8].

3.1. Frequencies and natural vibration forms of a hull
For the modal analysis of the hull, a finite element model is created, according to the study. Since the vessel is floating in a continuous medium, in the modal analysis, the vessel does not require additional
links, i.e. the modal analysis will be of free-free type. For such an analysis, the first six vibration modes come from the rigid solid movement; they can be easily identified by frequencies that are very small; theoretically, frequencies of rigid solid movements are void, but due to errors inherent in the solver, they will have values of MHz order. In order to estimate the influence of different stiffeners on frequencies and vibration modes, three constructive variants of hull will be analyzed: without stiffeners, with transverse stiffeners and with transverse and longitudinal stiffeners. For each constructive type studied, frequencies and associated modes were extracted. In the study of wave action, it is not needed to study modes with frequencies higher than 5 Hz, but as good engineering practice, between 100 and 200 modes are extracted; this requirement arises because of multiple excitation sources that may arise in the operation of a ship [5].

3.2. Idealized hull without stiffeners

Figure 1. CAD model of the idealized hull without stiffeners – isometric view from outside.

Figure 2. CAD model of the idealized hull without stiffeners- isometric view with transparency.

Figure 3. Mesh preparation and the finite element model for the hull without stiffeners.

Figure 4. The first 100 natural frequencies of the idealized hull without stiffeners.

Figure 4 shows the first vibration mode of the hull without stiffeners corresponding to the minimum frequency of 2.142 Hz [1].

Figure 5. The vibration mode of the hull without stiffeners at the min frequency of 2.142 Hz.

Figure 6. Vibration mode of the hull without stiffeners – frequency of 6.1305 Hz.
3.3. Hull with transverse stiffeners

For the hull with transverse stiffeners, the mesh preparation and the final model can be seen in figures 7a and b.

![Figure 7](image1)

**Figure 7.** Mesh preparation (a, b) and the finite element model (c) for the hull with transverse stiffeners: stiffening structure.

The frequencies of the first 100 vibration modes of the hull transversely stiffened are shown in figure 8. Figure 9 and 10 show the first form of vibration of the hull transversely stiffened corresponding to the minimum frequency of 1.965 Hz and 11.209 Hz at the frequency of 11.209 Hz and the frequency of 11.209 Hz [2].

![Figure 8](image2)

**Figure 8.** The first 100 natural frequencies of the hull transversely stiffened.

![Figure 9](image3)

**Figure 9.** The vibration form of the hull transversely stiffened at the minimum frequency 1.965 Hz.

![Figure 10](image4)

**Figure 10.** Vibration mode of the hull transversely stiffened at frequency 11.209 Hz.
3.4. *Hull with transverse and longitudinal stiffeners*

For the hull with transverse and longitudinal stiffeners, the mesh preparation and the final finite element model can be seen in figures 11a and b, and figure 12.

**Figure 11.** Virtual topologies created to assimilate small surfaces (a, b).

**Figure 12.** Mesh preparation and the finite element model for the hull transversely and longitudinally stiffened: finite element model.

The frequencies of the first 100 vibration modes of the hull transversely stiffened are shown in figure 13, figures 14 and 15 presents the first form of vibration of the hull transversely and longitudinally stiffened at the minimum frequency of 2.057 Hz and at the frequency of 12.435 Hz.

**Figure 13.** The first 100 natural frequencies of the hull transversely and longitudinally stiffened.

**Figure 14.** Form of vibration of the hull transversely and longitudinally stiffened at the minimum frequency of 2.057 Hz.

**Figure 15.** Form of vibration of the hull transversely and longitudinally stiffened at a frequency of 12.435 Hz.

It is noted that, for all three constructive configurations, the frequencies of the first vibration modes
are higher than the frequencies of waves. The results in tables 1, 2, 3 show that, with the stiffening of the structure, frequencies of the homologous vibration modes are growing.

4. Conclusions

Data from these plots are rather qualitative than quantitative; even though the modal forms are represented as displacements, they are not real, being associated only to the mode of deformation of the structure at the appropriate frequency. A modal form of a structure is unique, but vibration amplitudes after this modal form are not unique, being directly proportional to the energy of the excitation source, in our case the waves [5].

The estimation of external forces induced by waves and stresses occurring in the structure of the ship due to them is a very complex issue and its solution requires knowledge of several areas, such as the probability theory, the theory of random processes, the statistical theory of extreme values, the analysis of experimental data, naval hydrodynamics, numerical analysis. Since the estimation of wave characteristics, the deduction of the spectral density function from experimental data is equivalent to the total characterization of the sea surface, the knowledge of wave spectra for different situations and geographical areas is particularly important for the structural analysis of ships’ behavior at real sea, the prediction of characteristic extreme values is necessary. Static analyzes performed for three different structural models of a container port hull (ideal without stiffeners, with transverse stiffeners and transverse and longitudinal stiffeners) submitted to loads, come from the pressure distribution in calm water, allow to determine the influence of stiffeners and loading manner on the stress state.

The introduction of the finite element method (FEM) has enabled new approaches to complex problems of structural analysis. By using this method, more accurate resistance calculations and evaluation of criteria for failure of structures can be achieved. FEM also allows the iterative optimization of structural dimensions to fulfill all requirements [7].

This article is the result of the project “Increasing quality in marine higher education institutions by improving the teaching syllabus according to International Convention STCW (Standards of Training, Certification and Watchkeeping for Seafarers) with Manila amendments”. This project in co-funded by European Social Fund through The Sectorial Operational Programme for Human Resources Development 2007-2013, coordinated by Constanta Maritime University. The content of this scientific article does not reflect the official opinion of the European Union. Responsibility for the information and views expressed in the article lies entirely with the authors.

References

[1] Ion A 2014 Research on the calculation of a container ship structural resistance to wave oscillations PhD Thesis (Constanta Maritime University)
[2] Ion A, Bocanete P and Ticu R I 2013 The modern design of naval structures Constanta Maritime University Annals 20
[3] Adegeest L J M, Braathen A and Vada T 1998 Evaluation of methods for estimation of extreme nonlinear ship response based on numerical simulations and model tests Proc. 22nd Symp. on Naval Hydrodynamics (Washington DC: National Academy Press) pp 70–84
[4] ANSYS Mechanical User Guide Release 14.5, 2012.
[5] Avrashi J and Cook R D 1993 New Error Estimation for C Eigenproblems in Finite Element Analysis Engineering Computations 10(3) pp 243-256
[6] Baarholm G S and Moan T 2002 Efficient estimation of extreme long-term stresses by considering a combination of longitudinal bending stresses J. Marine Science and Technology 6(3) pp 122–134
[7] Bathe K J 1982 Finite element procedures in engineering analysis (New Jersey: Prentice-Hall)
[8] Beards C F 1995 Engineering vibration analysis with application to control systems (London: Arrowsmith Ltd)
[9] Zienkiewicz O C and Taylor R L 2005 The Finite Element Method for Solid and Structural Mechanics Sixth Edition