# Static analysis of the hull plate using the finite element method

**A Ion**[1]

[1]Constanta Maritime University – Romania, Department of Navigation
104, Mircea cel Batran Street, Constanta

E-mail: alina_ion@rocketmail.com

**Abstract.** This paper aims at presenting the static analysis for two levels of a container ship's construction as follows: the first level is at the girder / hull plate and the second level is conducted at the entire strength hull of the vessel. This article will describe the work for the static analysis of a hull plate. We shall use the software package ANSYS Mechanical 14.5. The program is run on a computer with four Intel Xeon X5260 CPU processors at 3.33 GHz, 32 GB memory installed. In terms of software, the shared memory parallel version of ANSYS refers to running ANSYS across multiple cores on a SMP system. The distributed memory parallel version of ANSYS (Distributed ANSYS) refers to running ANSYS across multiple processors on SMP systems or DMP systems.

## 1. Introduction

This paper aims at presenting the static analysis for two levels of a container ship's construction as follows: the first level is at the girder / hull plate and the second level is conducted at the entire strength hull of the vessel. This article will describe the work for the static analysis of a hull plate. Static analysis will be conducted for two constructive levels of a container ship: at the panel / plate level of the hull and at the level of the entire ship.

Until the 1970s, the design of ship structures was done with predilection (sometimes even exclusively) according to Rules of the Classification and Shipbuilding Companies (Books). The disadvantages and risks of designing based only on algorithms of the classification companies have revealed , since the 1980s , the need for design and structural analysis of the hull, on rational basis, by widely using the structural theories specific to the mechanics of the continuous media and strength of materials , vibrations and elastic stability, fatigue and fracture mechanics, as well as computers, as tools for implementation of analysis methods, data management, creation of mathematical patterns and numerical calculations.

Because of these facts, the issues of design and resistance calculation of the ship structures always remain important and current, of great interest to ship owners and classification and shipbuilding companies and stuck to the attention of engineers and researchers dealing with design, construction and shipping issues. The computer-aided rational design has as starting point, the goal that the ship must fulfill and takes into account all aspects of performance and structure, achieving precise estimations regarding the behavior of the hull. The designed ships must accomplish their mission in the safest way possible and under economic conditions. Forces (or loads) acting on the hull come either from the interaction with the gravitational field (force of gravity) and the environment (forces created under water and wind pressure) or are generated by the ship in its movement. These loads determine the emergence of solicitations in the hull and in the structures within it [10, 11].

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 MEF 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 shifts 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 continue 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. Static analysis will be conducted for two constructive levels of a container ship: at the level of the panel / plate of the hull and at the level of the entire ship. The material is considered uniform and homogeneous. The pressure is constant on all surfaces where it is applied. Taking advantage of the linearity of the finite element model, a pressure of 1 *Pa* is applied and results can be scaled as needed. The way in which the effects of pressure for different values can be determined, is described in the defining procedure of the analysis and evaluation of results. The operation method for the static analysis of a hull panel is described. The Mechanical software from the ANSYS 14.5 package was used. The program was run on a computer with 4 CPU Intel Xeon X5260 at 3.33 GHz, 32 GB installed memory.

ANSYS Mechanical software is a graphical interface for preprocessing, solver and post-processing mechanical problems. The solver used is Mechanical APDL. The validity of the program results will be verified in the verification book which contains mechanical problems with given solutions. In this case, I will apply this software for static analysis of the effect of wave pressure on the hull. All static analysis will follow the procedures described and applied at this point.

## 2. General information
The general information concerning the materials properties is presented on the figure 1.



**Figure 1**. Material properties.

## 3. Application of material properties
ANSYS Mechanical software has a large library of linear and nonlinear materials, which optionally can be edited. Furthermore, a material can be created on the basis of properties given as known.

For the static analysis presented below, ordinary steel was used as material and its mechanical properties are presented in figure 1, as shown in ANSYS library according to ASME BPVC standard (American Society of Mechanical Engineers, Boiler and Pressure Vessels Code).

In the static analysis, only the longitudinal elastic modulus (Youngh's Modulus) and the transverse contraction coefficient (Poisson's ratio) are of great interest.

## 4. Creating geometry

The CAD software from the ANSYS package is Design Modeler. It has most of the capabilities of any CAD software. Therefore, it can be used to create a parametric geometry. The drawback is that it is not used in the design, which makes the preparation of geometry for analysis to be an operation even more complicated than it seems at first sight. There are two methods commonly used in the industry. The first method involves saving geometry in a neutral format and the import in Design Modeler. Through international standards, more neutral extensions have been established: igs, stp, step, x_t. However, geometry still has to be changed, after import, in order to be used in the analysis. The operations of geometry simplification are time consuming and if several variations of the same geometry are running, these operations must be repeated at every new import of geometry. Therefore, for projects involving many tests run on the same geometry with small changes, the procedure of export / import of neutral files is not advantageous. In this case, the graphical interface version can be used; this procedure involves creating geometry in dedicated CAD software and importing geometry in the existing format based on an interface. In this way, geometry changes only in the native CAD program, which requires that the FEA analyst would know the program [6, 7].
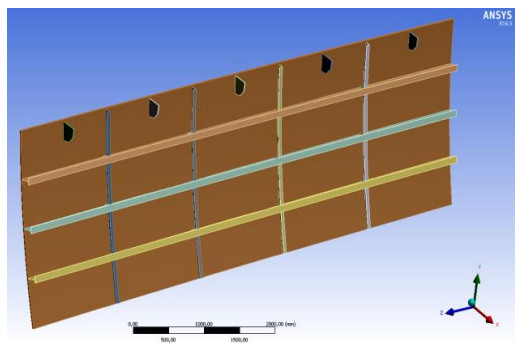


**Figure 2.** The structure of the steel plate reinforced with three longitudinal and four transverse beams.
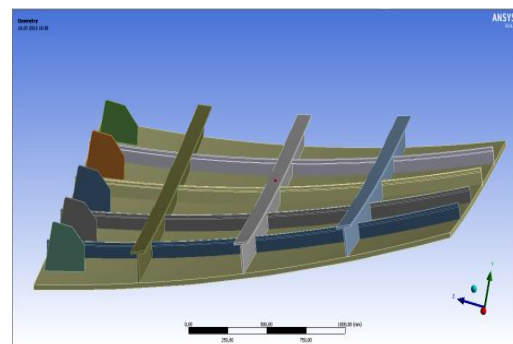


**Figure 3.** Detail elements of the structure studied- section of longitudinal beams.

In the current issue, which analyzes the hull, geometry was created in a dedicated CAD soft. The hull is steel plate with double curvature (cover), reinforced with longitudinal and transverse bars (figure 2, figure 3). In figure 4 you can see some details of joints and sections of reinforced bars that are welded to the plate throughout their length [4, 5].
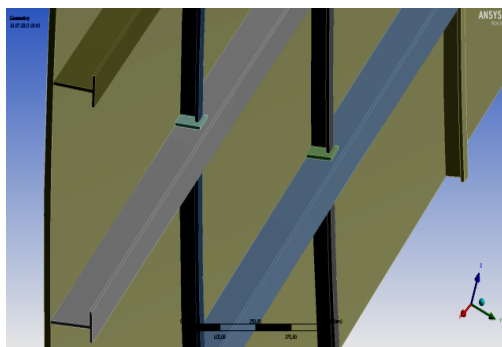


**Figure 4.** Detail elements of the structure studied joining elements between the two types of stiffeners.

## 5. Achieving the finite element model

At this stage, geometry is replaced by a network of elements. There are several possibilities for the creation of network elements. ANSYS offers several kinds of elements, as shown in a previous

chapter. When using the Mechanical software, the element type is selected automatically depending on the settings entered by the user and depending on geometry. Depending on these settings, the type of items can be controlled; for example, by opting to keep the middle edge nodes (*Element Midside Nodes = Kept*), the program will choose a parabolic element. Since it is a solid geometry type, the program will choose a SOLID187type element. The meshing obtained is shown in figure 5 [1, 4, 5].
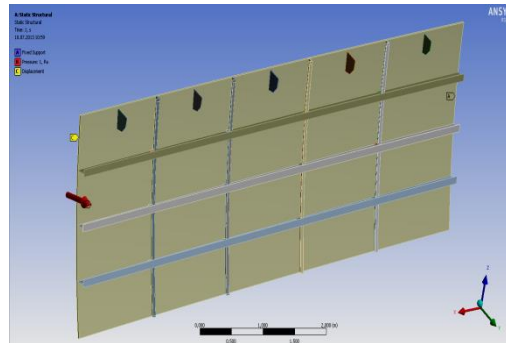


**Figure 5.** Mesh of the studied structure.

## 6. Setting the boundary conditions and obtaining the solution

Boundary conditions are affecting directly and essentially the results of the analysis. Although the time required for setting boundary conditions is much lower than that used in meshing, setting boundary conditions is particularly important. Taking into consideration the arrangement of the plate, symmetry conditions for faces at the ends of longitudinal bars were considered; they blocked movements after Z; one side is blocked by all degrees of freedom in order to eliminate the possibility of a mechanism that would lead to a divergent solution; the boundary conditions applied can be seen in figure 6 [1].
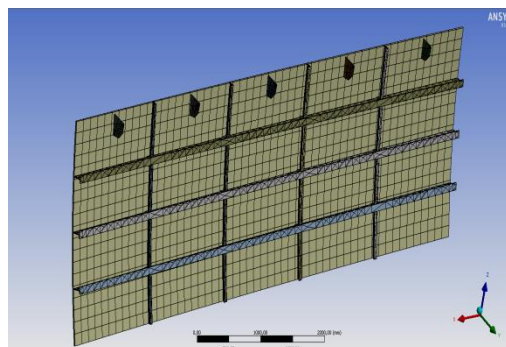


**Figure 1.** Boundary conditions applied.

For this analysis, it was considered that the plate studied is horizontal and totally submerged in water. Therefore, constant pressure of *1 Pa* is applied to one of the outer faces of the plate as shown in figure 7. Given the linearity of the phenomenon, with results for a load of 1 *Pa*, results for any value of load pressure can be obtained. In other words, if it is desired to determine the state of stress and strain for a load of 1 *MPa,* it is sufficient to multiply by 106 the values obtained for a load of 1 *Pa*. The operation of solving matrix equations resulted from mathematical modeling is started by pressing the *SOLVE* button. The characteristics of how to solve the problem can be changed by the user. For this problem, options recommended by the program were used [1, 4].
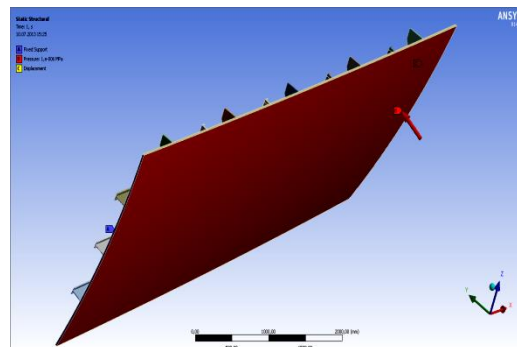
**Figure 7.** Loading structure with uniform pressure.

## 7. Viewing the results

The diagram of total displacements and stresses is of great interest in the present static analysis. For the state of stresses, the chart of von Mises equivalent stresses is the most appropriate. Results of the analysis can be seen in the following figures.
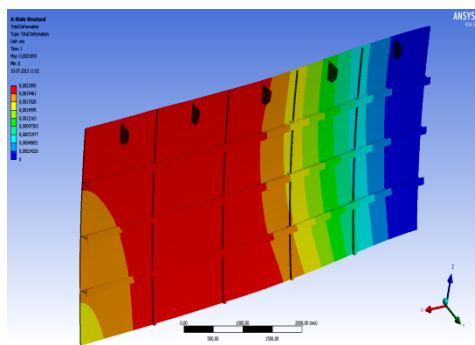


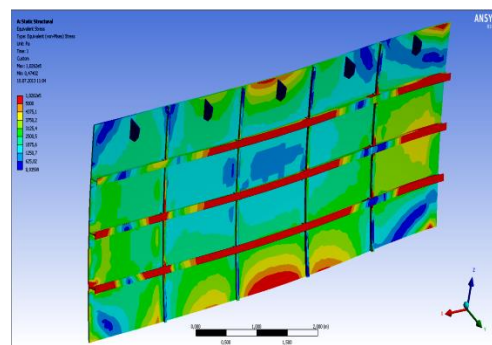**Figure 8.** Distribution of total movements



**Figure 9.** The von Mises equivalent stress status

The maximum equivalent stress must not exceed the yield strength of the material, which is of 250 *MPa*. As it can be seen in the chart, tensions are far below the limit value. If it is desired to determine the charging voltages for a 1*MPa*, which is equivalent to 10 bar, it is enough to multiply the previously obtained values to106. In figure 10 the chart of stresses in the plate, in the case of loading it with a pressure of 1*MPa* is represented. As it can be seen, the values are now expressed in *MPa*, which confirms the linearity of the results. In figure 10 the dark areas are those where the stress exceeds the yield limit, i.e. the material fails [1, 4, 5].
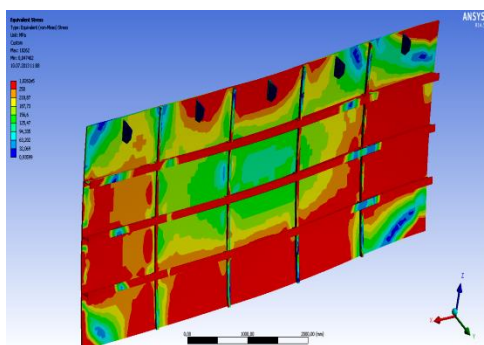


**Figure 10.** Von Mises equivalent stresses for a 1*Mpa* pressure load

## 8. Conclusions

For safety of the hull, the loading of 1 *MPa* is dangerous. By using this procedure, the limit pressure to withstand the hull can be determined. In principle, for values of stresses lower than the flow limit, in order to determine the actual safety coefficient, the yield stress is divided to the maximum stress from the diagram. This actual coefficient is then compared to the admissible coefficient. Due to the development of information technology, strong and effective methods of numerical analysis have been developed. In particular, 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 [1], [9].

## 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]　ANSYS 1997 *Structural Aanalysis Guide* ANSYS, INC Canonsburg, PA

[6]　Avrashi J and Cook R D 1993 New Error Estimation for C Eigenproblems in Finite Element Analysis *Engineering Computations* **10(3)** pp 243-256

[7]　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

[8]　Barkanov E 2001Introduction to finite element method (Institute of Materials and Structures, Riga)

[9]　Bathe K J 1982 Finite element procedures in engineering analysis (New Jersey: Prentice-Hall)

[10]　Zienkiewicz O C and Taylor R L 2000 *The finite element method* **I** (Butterworth - Heinemann, Oxford)

[11]　Zienkiewicz O C and Taylor R L 2005 *The Finite Element Method for Solid and Structural Mechanics* Sixth Edition