Mechanical simulation approach in underwater high pressure acting on ships hull

I.C. Scurtu, M. Cucu, A. Sturzu
Mircea cel Batran Naval Academy, 1 Fulgerului, RO900218, Constanta, Romania

ABSTRACT The industry of shipbuilding is a segment that faces many problems caused by flexible environment. We will try to determine the stresses and strain caused by an underwater explosion, which acts on a ship’s hull. The numerical simulation for ship plate is done with commercial software ANSYS using water pressure from other measured sources.

1. INTRODUCTION
In the past few years, structural analysis has emerged for checking the current eligible low-cost techniques, including shipbuilding. In this article we will analyze the high pressure caused by an underwater explosion, near the ship’s hull. In the past wars, especially world wars, mines were used at large scale, and had done a lot of destruction against military ships. The purpose of these analysis is to find out how we can improve the ship’s hull, so that he can resist against the high pressure, without changing her handling or displacement. To perform the test it was used Ansys Static Structural and the method of calculation was „Finite Element Method”

2. PROBLEM DESCRIPTION
The problem we choose is focused on finding the area on ships hull that is affected from the high pressure water caused by an underwater mine explosion. Magneto-acoustic mine has a maxim pressure of 5 bars, and the chosen ship will pass at 400 m from mine, when it explodes. The pressure near the ships hull in the moment of explosion is 3 bars.

Figure 1 Ship plate
3. NUMERICAL INVESTIGATION

Methods for solving problems by FEM

The main stages of the application of finite element calculation are:
1. The finite element discretized structure in which only the nodes to be interconnected, whose movements are chosen as unknowns.
2. Choose interpolation functions and is calculated for each element matrix B.
3. Determine the matrices of rigidity and stiffness matrix elements of the mesh.
4. Determine the nodal forces vector, write the system of equations, and determine the unknowns.
5. Calculate movements of the points of interest on any item and then tensions.
6. Given the tensions and movements in the required check points of strength and stiffness conditions.

The results of the simulation depend on the fineness of the plate and how it is constructed. It depends by the zone of the ship where de high pressure acts and the quantity of the explosives that it is used. Construction subjected to analysis is modeling in CAD environment, ANSYS Static Structural, meeting the specifications 0.6 m thickness, 48.554 length, 25.805 width and the yield strength of 445 N/mm²

4. INPUT DATA FOR MESHING AND SOLUTION

For the simulation we will use a simple piece of ships hull, but with all pieces of structure so that we can have our simulation more real.

![Figure 2 Ships Plate](image)

Shockwave speed is calculated as follows:

\[ D = \sqrt{\frac{p - p_0}{1 - \frac{\rho}{\rho_0}}} \]
Before being analyzed according to the FEM shaft is meshed in elements by ANSYS meshing procedure according to figure below.

![Meshing connections inside solver](image)

**Figure 4** Meshing connections inside solver

5. NUMERICAL RESULTS AND DISCUSSION
The stress study on surface was done on the table with an equivalent high pressure force uniformly distributed.

![Pressure-Specific volumes rapport](image)

**Figure 3** Sea water adiabatic dynamics

![Naval plate](image)

**Figure 5** Naval plate

![Total Deformation](image)

**Figure 6** Total Deformation
Figure 5 shows that in this moment is the beginning of the explosion, and now the forces that act on ships hull are minimum.

In figure 6 we can observe the areas where the high pressure acts with big intensity, we can notice that these areas are between the pieces of structure that sustain the ships plate, where are just the pieces of steel plate. For now the high pressure does not affect the steel beams of the ships.

![Figure 7 Elastic Strain](image7.png) ![Figure 8 Strain Energy](image8.png)

In figure 7 the elastic strain of the plate are represented and we can identify that the biggest value are found between the ships element of structure which sustain the plate. The elastic strain produce by the high pressure can tore apart the pieces of steel. In figure 8 is presented the Strain Energy reach huge value between 5 and 571 J. This energy had gathered in ships plate and reduce the efficiency of the protection of the ships. Using this Ansys simulation model, engineers will fast if it will tear apart or not.

6. CONCLUSIONS

The simulation we go through has reviewed the influence of the high pressure caused by an underwater explosion on a ship’s hull. The modern technology, especially military technology, can use this simulation for making a stronger ship’s hull that can resist against an underwater explosion coming from a mine. The deformation simulation in ANSYS Static Structural shows that the most affected part of a plate is the region between the ships structure (Figure 6). We hope that this work will help the Shipbuilders for understanding the effects of a underwater explosion acting on ship plates, will improve the protection of the ships and will also provide protection for the sailors.

REFERENCES

[1] Buletinul forțelor navale nr. 14/2011- modele ale undelor de soc pentru studiul efectului exploziei submarine asupra bordajului navei- gheorghe ichimoaiei
[2] Scurtu I. C., Oncică V., Garcia D. I., Babiuc B., Stress and strain analysis of heave plates, Constanța Maritime University Annals, Year XIV, Vol. 22, ISSN 1582-3601, Constanța 2015.
[3] Cole, R., Underwater explosion, Princeton University Press, 1948.
[4] Clark, C., The Management of Merchant Ship Stability, Trim and Strength, Nautical Institute, Londra, 2002.
[5] Maier V., Mechanics and construction of ship (Mecanica și construcția navei), Vol. I Statica navei, Editura Tehnică, București, 1985.
[6] Radoiu, V.B., Pintilie, A., Bormambet, M., Numerical analysis of stress and strain field in gas pipelines, 2015.Annals of "Dunarea de Jos" University of Galati, Fascicle XII, Welding Equipment and Technology