THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XII - SHIPBUILDING. ISSN 1221-4620, e-ISSN 2668-3156
DOI: https://doi.org/10.35219/AnnUgalShipBuilding/2023.46.10
2023

ANALYSIS OF STRESS AND DEFORMATIONS IN THE TRANSVERSE WALLS OF A 50100 TDW TANKER

Nicusor-Danut ILIE
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail: ilie.nicusor.danut96@gmail.com

Anisoara - Gabriela CRISTEA
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail: anisoara.cristea@ugal.ro

ABSTRACT

The purpose of the stress and strain analysis study in the transverse walls of a 50100 tdw tanker is to assess the structural integrity of the vessel and identify potential problems or vulnerabilities as well as methods to improve the strength of the structure.

Keywords: finite element method, *hydrostatic water pressure, mechanical structural, stress calculation*

1. GENERAL CONCEPTS

The main objective of the study is to provide a detailed understanding of the ship's structural behavior under loading and operating conditions, such as the influence of fluctuations in the ship's loading level, wave motion and the action of external forces on the transverse walls.

Analysis of the stress and strain condition in transverse walls can help to optimize the design and construction of the tanker, identify the need for subsequent maintenance and repair, and make appropriate preventive maintenance decisions to prevent potential failures or structural problems.

For the analyzed model it was necessary to study 3 cases of static settlement of the ship (on calm water, on wave crest and on wave void) and to study the stresses arising from the deformations of the transverse walls through modifications of the mesh and of the modelling of the elements of the structure.

The study was conducted exclusively using FEMAP/NX Nastran software. FEMAP/NX Nastran is a simulation program that allows the creation of finite element analysis models for complex systems and displays solution results. This software allows components, assemblies, and systems to be modelled and their behavior to be determined in each given operating environment.

For the analyzed model, it was necessary to study 3 cases of static settlement of the ship (on calm water, on wave crest and on wave void) and to study the stresses arising from the deformations of the transverse walls through modifications of the mesh and of the modelling of the elements of the structure.

In conclusion, the purpose of this study is to assess and monitor the structural condition of the transverse walls of the 50100 tdw tanker to ensure the integrity and safety of the vessel during operation.

The structure was modeled in FEMAP with the dimensions shown in **Table 1.1**, based on the dimensions of the structural elements that make up the master sections, a section that was made using the POSEIDON

ND v.21.4 program package belonging to the DNV classification company.

Table 1.1 Dimensions and characteristics of the simplified structure

L_{pp} =174,000 [m]	L_{OA} =183 m	Elsize = 100 mm
B = 32.2 [m]	X_{pp} = 48.82 m	z_{NNpp} = 7.248 mm
D = 19.1 [m]	L_{mag} =24.9 [m]	EL No = 701997

where:
L_{pp} - length between perpendiculars;
B - breadth;
D - depth;
L_{OA} - length overall;
X_{pp} - abscissa of the aft extremity;
z_{NNpp} - The neutral axis coordinate of the cross-section at the aft end of the model;

2. MODEL GENERATION AND ANALYSIS

FEMAP/NX Nastran software was used to generate the FEM model.

The program consists of two parts:

✓ FEMAP is a pre- and post-processing software that allows engineers to create, model and analyze complex structures. It provides advanced geometric modelling and meshing (finite element generation) tools to create accurate models of the desired geometry. It also allows you to define material properties, loading conditions and desired analyses. Using it, users can generate and configure models for analysis with NX Nastran

✓ NX Nastran is a finite element method (FEM) solver used to solve structural, thermal, fluid dynamics and other simulation problems. NX Nastran can be used in conjunction with FEMAP or as an integrated module within the NX computer-aided design (CAD) environment. It offers a wide range of analytical capabilities, including linear and nonlinear analysis, static and dynamic analysis, frequency and thermal analysis, fatigue analysis, structural optimization, and more.

3. THE FEATURES OF THE MATERIAL

The material used must meet the requirements for structural strength, corrosion resistance, durability, and other specific characteristics of the ship and its application. It must comply with the standards and classification society requirements.

The material used is high-strength steel with the following characteristics:

✓ Yield strength: σ_c =315 MPa;
✓ Tensile strength: σ_r = 440 MPa;
✓ Longitudinal elastic modulus: E=2.1*10^5 MPa;
✓ Poisson's ratio coefficient: ν = 0,3;
✓ Material density: ρ= 7800 kg/m^3.

4. GEOMETRY AND MESH

Based on the 3D-CAD model, a 3D-FEM model (Figure 1.2) will be generated using the plate and membrane elements of the FEM program, and stress concentrations in all structural elements will be determined. To highlight voltage concentrators in all structural elements, it is necessary to use the membrane and plating elements implemented in the FEM program.

The plate type elements (PLATE - Mindlin) implemented in the FEM program were used in the FEM model.

Most of the elements are quadrilateral, but if they cannot be used, triangular elements are used.

The variation of the shapes and sizes of the elements in the FEM model occurs due to the different sizes of the longitudinal profiles and their positions.

To analyze the strength of the ship's hull, the central area of the cargo compartments will be modeled, as it is the most heavily stressed area.

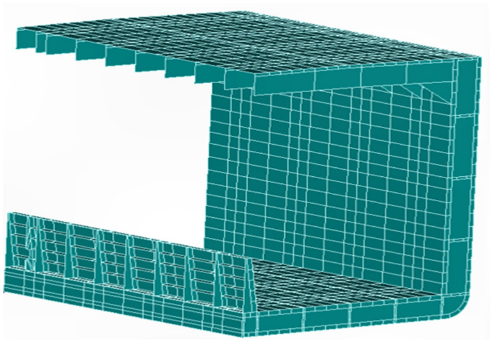

Fig.1.1 The extended 3D-CAD structure

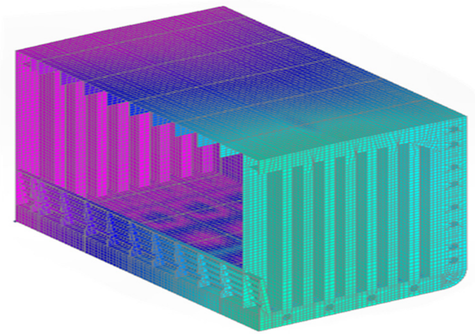

Fig.1.2 3D - FEM

5. EDGE CONDITIONS

Boundary conditions are the conditions that must be applied to the geometric boundaries of a structure to subject the model to real loading conditions and obtain a response (**Fig. 1.5**).

The edge conditions applied on the FEM model have the role of simulating the existence of the real structure of the vessel in the aft, in the fore and in the opposite board of the shaped warehouse.

These edge conditions used for the study of the model are shown in **Table 1.2**.

Table 1.2 Boundary conditions for the 3D - FEM model

Boundary condition	Blocked degrees of freedom					
	T_x	T_y	T_z	R_x	R_y	R_z
Symmetry in the diametrical plane	-	x	-	x	-	x
NDpp - master node located aft of the model	x	x	x	x	-	x
NDpv - master node located in the fore of the model	-	x	x	x	-	x

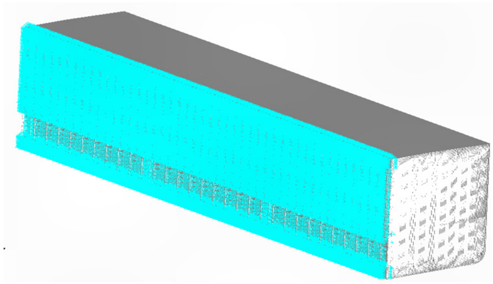

Fig. 1.5 3D MODEL - Margin conditions

6. LOADS APPLIED UPON THE MODEL

A) The case of static placement on still water

The FEM model is subjected to the following types of loads:

• Gravitational load given by the net weight of the structural elements of the vessel: $g = 9.81\ m/s^2$, $\rho = 0.9\ t/m^3$ and other components on board of the vessel around the modeled cargo tank.

• The load given by the cargo is idealized on the double bottom shell, double board, longitudinal and transversal walls, as hydrostatic pressure in the cargo ($\rho = 1.05\ t/m^3$) [N/mm^2], for a reference quota HHC (D=19100 mm).

The hydrostatic pressure is given by relation (1) where:

$$p=\rho g z\ [kN/m^2] \tag{1}$$

where:
ρ - density of transported goods [t/m^3];
g - gravitational acceleration [m/s^2];
z - vertical distance to the highest point the goods reach inside the warehouse [m].

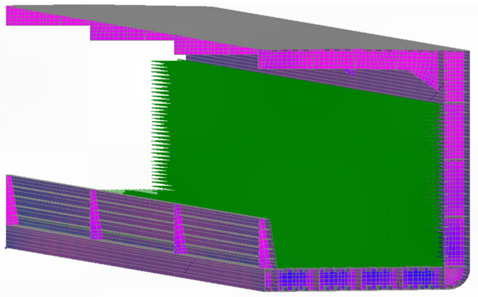

Fig. 1.6.1 Hydrostatic cargo pressure distribution

• The load from still water:

The load given by the sea water in which the hull of the vessel is immersed, idealized on the outer shell, as the hydrostatic pressure in the water (ρ = 1,025 t/m^3), for a full load draught of T = 14500 mm (**Fig. 1.7**).

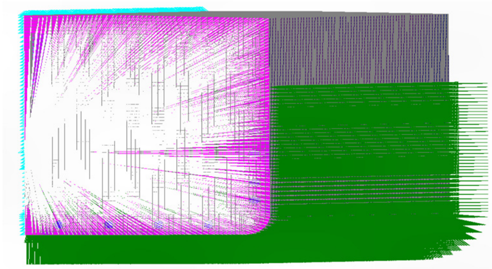

Fig. 1.6.2 Hydrostatic pressure generated by the still water (ρ_{water} = 1,025 t/m^3)

B) The case of static wave settlement

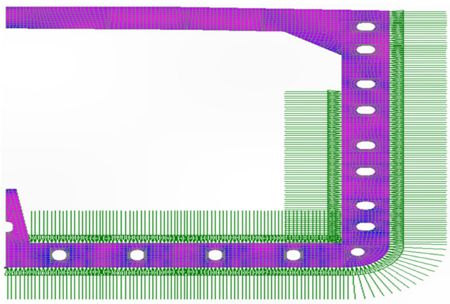

Fig. 1.6.3 Hydrostatic pressure distribution of cargo and water on the inner and outer shell of the model - Hogging analysis

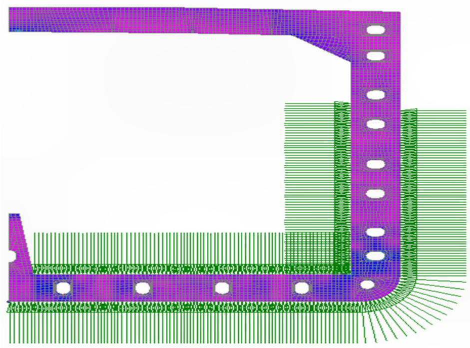

Fig. 1.6.4 Hydrostatic pressure distribution of cargo and water on the inner and outer shell of the model - Sagging analysis

For the study of the static settlement of the ship on the wave, the 3D-FEM model is subjected to similar loads in terms of gravity loading and cargo loading. Modifications will occur in the case of wave pressure loading.

7. RESULTS OBTAINED FOLLOWING FEM ANALYSIS

The analyses (two types of elements were taken into account, i.e. QUAD and TRI elements) revealed maximum stresses in the stiffening elements of the structure and deformations of the hull and transverse walls.

The results of the analysis include information on:

✓ Stress distribution - allows us to observe areas of high stresses or stress concentrations that could lead to structural weakening or cracking;

✓ Maximum stresses - maximum stresses give us information about the structure, the loads it is subjected to and the limits within which they fall;

✓ Deformations on the structure - through interpretation we can assess how the structure responds to loading and identify areas of excessive deformation;

✓ Safety factors of the structure - based on the results obtained from the finite element analysis and with the help of the acquired knowledge we can assess the structural safety of the transverse walls and hull, therefore if the stresses and strains are within acceptable limits, the structure is considered safe. If the stresses or deformations exceed the acceptable limits corrective measures can be applied to bring them within the required limits.

Next, the 3D-CAD geometry is discretized to obtain a 3D-FEM finite element model. A comparative study of four types of QUAD and TRI discretization is followed.

For a better observation of the results obtained from the analysis carried out on the two types of transverse plate floor, on the two types of mesh and on the different levels of discretization, the values of the stresses have been centralized in the following tables:

Table 1.3 The results of the centralized stresses obtained from the analysis of the 3 cargo tanks

		Transverse plate floor with cutouts				Watertight transverse plate floors				UM.
Stress von Misses	Quad mesh	83	166	207	332	83	166	207	332	[mm]
		210.78	157.14	160.32	250.85	211.16	170.25	160.81	251.08	[MPa]
	Tri mesh	83	166	207	332	83	166	207	332	[mm]
		209.81	127.4	143.03	280.49	208.46	127.62	143.23	280.46	[MPa]

Table 1.4 Centralized deformation results obtained from the analysis of the 3 cargo tanks

		Transverse plate floor with cut-outs				Watertight transverse plate floors				UM.
Translation	Quad mesh	83	166	207	332	83	166	207	332	[mm]
		25.755	14.048	21.793	26.71	25.588	21.682	21.711	26.675	[mm]
	Tri mesh	83	166	207	332	83	166	207	332	[mm]
		25.814	21.545	21.5	26.672	25.612	21.467	21.43	26.57	[mm]

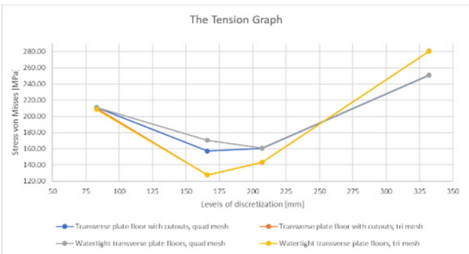

Fig. 1.7.1 Graph of the voltages resulting from the analysis of the 3 cargo tanks

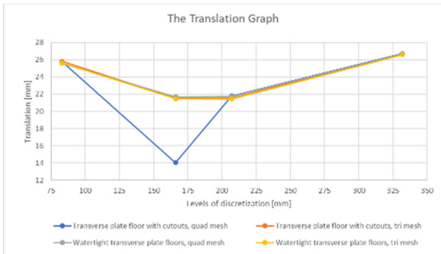

Fig. 1.7.2 Graph of translations resulting from the analysis of the 3 cargo tanks

Tabel 1.5 The results of the stresses obtained from the analysis of the 4 transversal bulkheads

		Transverse plate floor with cutouts				Watertight transverse plate floors				UM.
Stress von Misses	Quad mesh	83	166	207	332	83	166	207	332	[mm]
		93.122	45.019	63.536	44.741	90.619	64.493	63.255	43.395	[MPa]
	Tri mesh	83	166	207	332	83	166	207	332	[mm]
		105.84	70.852	64.328	49.438	103.01	69.259	63.319	48.47	[MPa]

Tabel 1.6 The results of the deformations obtained from the analysis of the 4 transversal bulkheads

		Transverse plate floor with cutouts				Watertight transverse plate floors				UM.
Translation	Quad mesh	83	166	207	332	83	166	207	332	[mm]
		18.913	8.6736	15.787	11.893	18.86	15.73	15.792	11.903	[mm]
	Tri mesh	83	166	207	332	83	166	207	332	[mm]
		19.009	15.661	15.717	15.861	18.943	15.664	15.72	15.869	[mm]

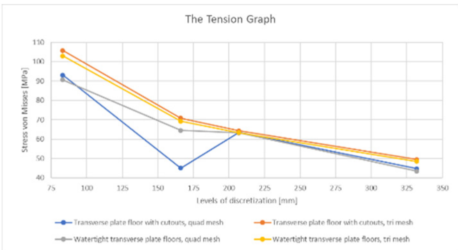

Fig. 1.7.3 Graph of the voltages resulting from the analysis of the 4 transversal bulkheads

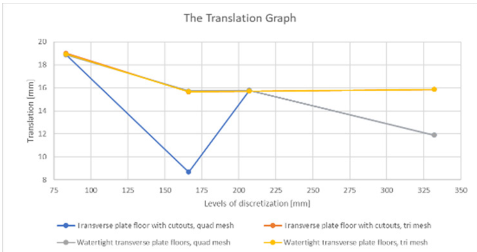

Fig. 1.7.4 Graph of translations resulting from the analysis of the 4 transversal bulkheads

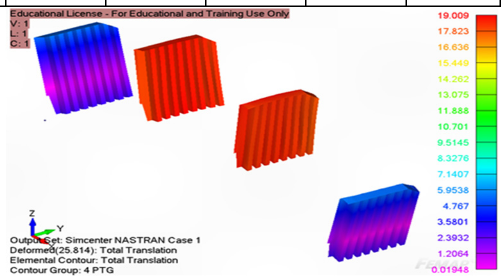

Fig. 1.7.5 The result of the deformations of the transversal bulkheads

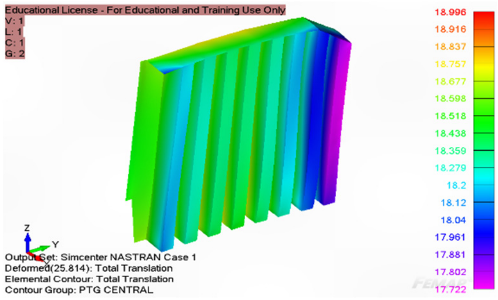

Fig. 1.7.6 The result of the deformation of the transverse bulkhead

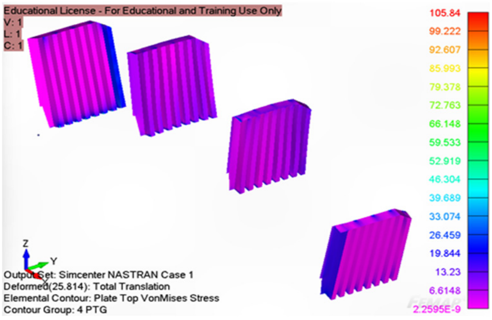

Fig. 1.7.7 Result of stresses in transversal bulkheads

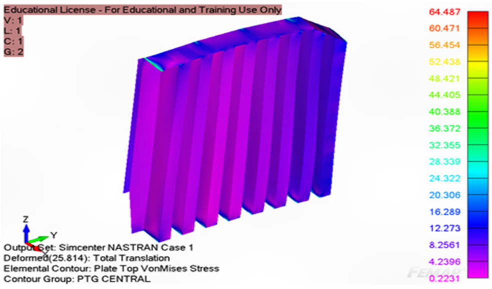

Fig. 1.7.8 The resultant of the stresses in the transverse bulkhead

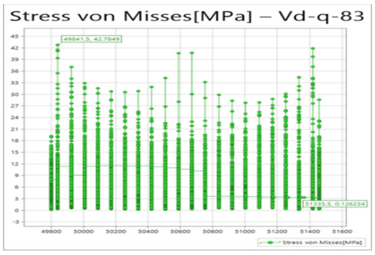

Fig. 1.7.9 Graph of the central wall tensions - model with cutout - quad mesh 83

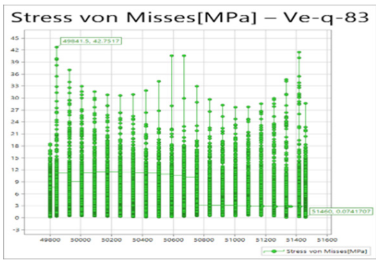

Fig. 1.7.10 Graph of the central wall tensions - watertight model - quad mesh 83

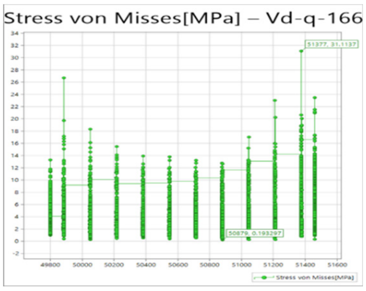

Fig. 1.7.11 Graph of the central wall tensions - model with cutout - quad mesh 166

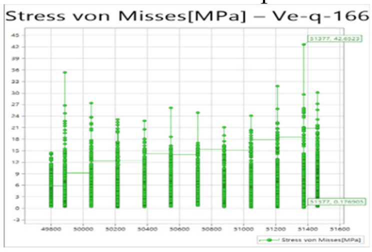

Fig. 1.7.12 Graph of the central wall tensions - watertight model - quad mesh 166

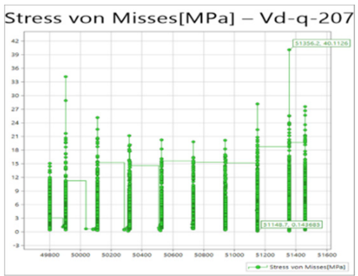

Fig. 1.7.13 Graph of the central wall tensions - model with cutout - quad mesh 207

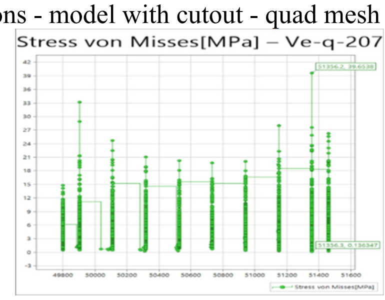

Fig. 1.7.14 Graph of the central wall tensions - watertight model - quad mesh 207

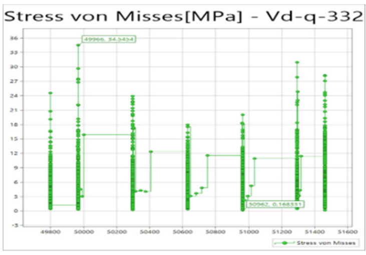

Fig. 1.7.15 Graph of the central wall tensions - model with cutout - quad mesh 332

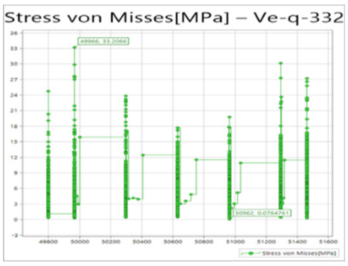

Fig. 1.7.16 Graph of the central wall tensions - watertight model - quad mesh 332

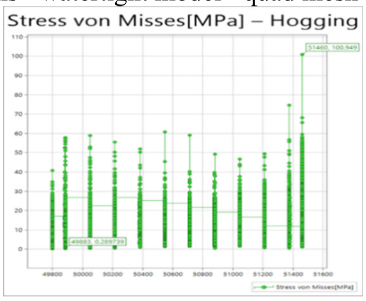

Fig. 1.7.17 Graph of the central wall tensions - model with cutout - Hogging

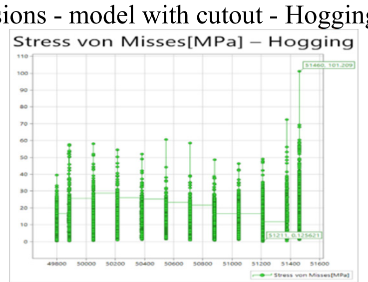

Fig. 1.7.18 Graph of the central wall tensions - watertight model - Hogging

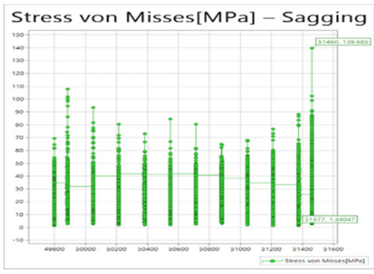

Fig. 1.7.19 Graph of the central wall tensions - model with cutout - Sagging

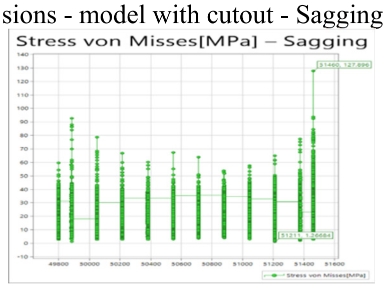

Fig. 1.7.20 Graph of the central wall tensions - watertight model - Sagging

8. CONCLUSIONS

The objective of the work was to analyze the stresses in the transverse walls of an oil tanker in the central area.

The structural analysis was carried out using the finite element method, using the Femap software, with the educational license, the analysis being static.

The model was built with 4 degrees of discretization: fine, intermediate, medium, and coarse in order to have the highest accuracy on deformations, displacements and stresses occurring on the transverse walls and hull. We also aimed at observing structural differences according to the geometry performed for the analysis thus, this was carried out on structures that had watertight transverse plate floors but also transverse plate floor.

At the same time, for better accuracy, the analysis was performed with finite elements of membrane and plate with quadrilateral and triangular geometric shape.

In the 4 discretization cases, an increase in stresses is observed for the triangular type elements only for the highest discretization of 332, this is visible for both plate floors, watertight and with cutouts, models due to a lower accuracy compared to the quadrilateral elements.

The 3D-FEM model was subjected to various equivalent analyses such as sagging, hogging and calm water cases according to the reference wave height.

In order to avoid maximum stresses and to ensure proper structural behavior, several strategies can be applied, such as design optimization, use of appropriate materials, mesh refinement, and careful analysis of loads and loading conditions.

In conclusion, the analysis of stresses in the transverse walls of an oil tanker using the finite element method is an essential process for assessing and improving the structural safety of the vessel, thus ensuring its correct and reliable operation in harsh marine environments.

9. REFERENCES

[1] Cook, R. D., *Finite Element Modeling for Stress Analysis*, Wiley, New York, 1995.

[2] Cristea A.G., *Contributions on the optimization of ship structures*, PhD thesis, Galati, 2014

[3] Jonathan Whiteley, *Finite Element Methods*, Ed. Springer International Publishing AG, 2007

[4] **x x x** – Siemens PLM Software Inc, "FNN, Femap/NX Nastran users' manual," 2020

[5] **x x x** – DNV GL, Part 3, Chapter 7, Section 3, 2021

[6] **x x x** – DNV, Poseidon User Manual, v.21.4

[7] **x x x** – DNV rules for classification, Ships (RU-SHIP), https://www.dnvgl.com/rules-standards/

[8] **x x x** – "*Common Structural Rules for Bulk Carriers and Oil Tankers*", 1 Jan. 2020

[9] **x x x** – http://www.iacs.org.uk/publications/common-structural-rules/csr-for-bulk-carriers-and-oil-tankers/

[10] **x x x** - http://www.imo.org/en/OurWork/Safety/Cargoes/CargoesInBulk/Pages/IBC-Code.aspx

[11] **x x x** – FEMAP 12.0.1a / NX NASTRAN User's Guide, UGS Corporation / Siemens PLM Software Inc., 2021

Paper received on November 8th 2023