THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620, e-ISSN 2668-3156
DOI: https://doi.org/10.35219/AnnUgalShipBuilding/2023.46.09
2023

DETERMINATION OF STRESS STATE AND DEFORMATIONS THAT CAN APPEAR IN TRANSVERSAL FRAMES OF A 48309 DWT BULK CARRIER

Cosmin MOGUS
"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati, Domneasca Street, No. 47, 800008, Romania,
E-mail: moguscosmin@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 main objective of this study case is to analyze the response in terms of stress and deformation of transversal frames of a bulk carrier that occur due to the applied loads. This assessment helps ensure the safety of the vessel in terms of structural strength and resistance and to determine the best solutions to apply in unfavorable scenarios.

The analysis will be performed on a cargo tank, for three static stress cases, the vessel being placed on still water, on sagging and hogging.

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

1. RESEARCH PURPOSE

The entire research was possible using FEMAP/NX Nastran, which is a combination of two different software products. FEMAP is an interactive pre-processor that allows us to create and prepare the geometry that we want to analyze, generate the mash of elements, apply different constraints to the structure and to submit the model to different loads. The program provides a wide range of tools within a user-friendly interface.

NX Nastran is a software designed to perform numerical analyses using finite element methods. This program can solve a wide variety of analysis types (static, dynamic, frequency, stability, etc.)

To achieve the purpose of this study, three types of analysis will be conducted: the vessel being placed on still water, in lightship condition and wave crest condition.

2. MAIN DIMENSIONS

The main dimensions of the vessel can be found in Table 1, while the equilibrium parameters are listed in Table 2. As for the thickness of every element, they will correspond to the scantling calculations.

Table 1 Main ship characteristics (simplified structure)

L = 199.9 m	a_0 = 0.850 m	ρ = 7.8E-6 kg/mm^3
B = 32.2 m	3 x a_0 = 2.55 m	No. CR = 52015
D = 22.95 m	EL_{size} = 100 mm	No. SF = 22269
L_{mag} = 3 mag. x 28.050 m	E = 2.1E+5 N/mm^2	No. ND = 1720909
Z_{NN} = 7.877 m	ν = 0.3	No. EL = 1773140

Table 2 Equilibrium parameters

LOA = 199.9 m	XPP = 53.675 m	DPP = 11.25 m
HW = 9.750 m	HHC = 17.900 m	DPV = 11.25 m

3. MODEL GENERATION AND ANALYSIS

To reach the purpose of the study, we will need to go through the following steps: creating the geometry, generating the mash of elements, applying boundary conditions, and loading conditions. In the end we will be able to evaluate the obtained results.

3.1 Material characteristics

Steel is a widely used metallic material, mostly because of its excellent mechanical properties, one of the most important being its resistance to deformation. To measure the maximum stress a material can withstand before deforming, a value known as yield strength is used. This value can vary depending on the material composition, as well as the manufacturing process and any treatments applied.

For the following analyses, we will use high-strength steel with a yield strength of 355 MPa and a tensile strength range from between 470 to 630 MPa. This indicates that this type of steel is capable of withstanding high loads during its use. This type of material is chosen for applications that require increased strength and reduced weight. According to European Standard EN 10025-2, it is classified as S355.

This material has young modulus of E = 210 GPa, a Poisson ratio of ν = 0.3 (the ratio between transverse strain and longitudinal strain), and a density of 7800 kg/m^3.

3.2 Geometry and mesh

Since the global loads on a FEM model are dominant in the central areas of the ship, the model will be created for the length of 3 cargo holds, each 28.05 meters long, only in portside, the missing starboard side being corrected by the boundary conditions.

The structure (**Fig. 1)** is simplified by eliminating brackets and stiffening ribs, but also by simplifying the simple framing system, from bulb profiles (HP) to flat band profiles (FB).

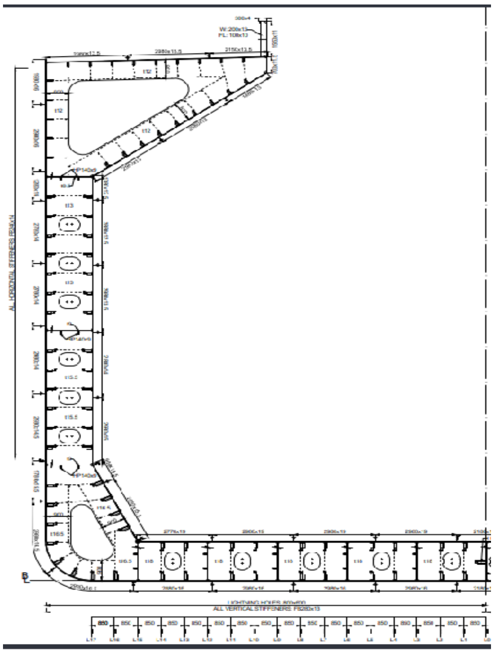

Fig. 1. Cross-section area at the amidships section

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

Most of the elements are quadrilaterals, 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.

Following the making of the 3D-CAD model, 1720909 points, 52015 curves and 22269 surfaces resulted (**Fig. 2**).

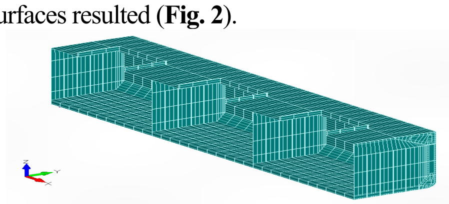

Fig. 2 3D - CAD model of the cargo tank

In the next step, the 3D-CAD model is discretized to generate the 3D-FEM model. The element surface varies from 150 mm^2 to

200 mm^2 depending on the distances between structure members. The discretization process was made manually, resulting in a total of 1.720.909 nodes and 1.773.140 elements (**Fig. 3**).

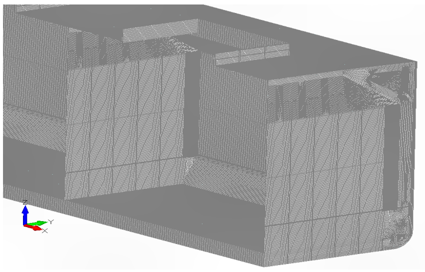

Fig. 3 3D - FEM model of the cargo tank

4. EDGE CONDITIONS

In order to effectively control the behavior of interactions and load transfer between elements, master nodes (also known as 'spider nodes') will be applied at both ends of the model. Constraints will be imposed on the connection node according to Table 3.

In addition to the master nodes, to simulate the symmetry of the ship, constraints will be applied along the central support, at the intersection of the transverse walls with the diametral plane, and at the intersection of the main deck and the hatch coaming with the diametral plane, also highlighted in Table 3.

Table 3 Boundary conditions for the 3D - FEM model

Boundary condition	Blocked degrees of freedom					
	T_x	Ty	T_z	Rx	Ry	Rz
Symmetry in the diametral plane PD	-	x	-	x	-	x
Stern master node NDpp	x	x	x	x	-	x
Stern master node NDpv	-	x	x	x	-	x

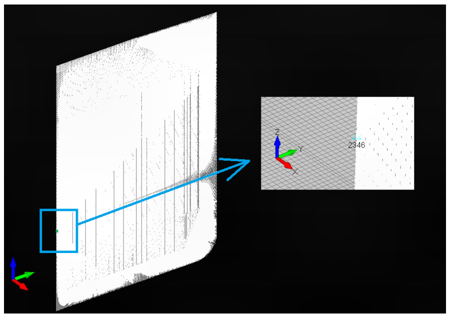

Fig. 4 Spider node and Boundary conditions (Fore)

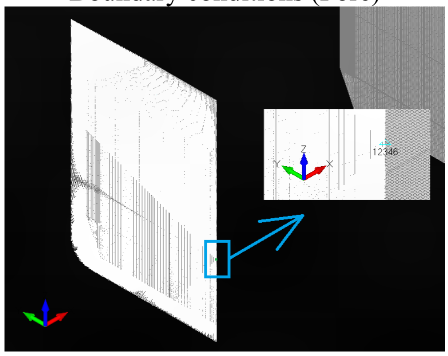

Fig. 5 Spider node and Boundary conditions (Aft)

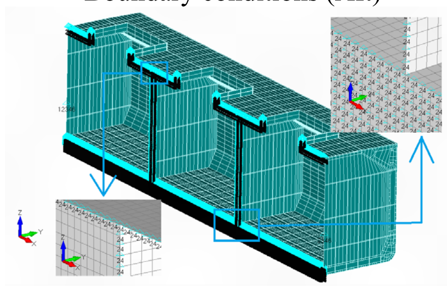

Fig. 6 Boundary conditions at intersection with diametral plane

5. 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 (**Fig. 7**):

• Gravitational load given by the net weight of the structural elements of the vessel: g = 9.81 m/s^2, ρ = 7.8 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 (ρ = 0.9 t/m^3) [N/mm^2], for a reference quota HHC - 17. 9 m.

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

$$p=\rho gz \ [\mathrm{kN/m^2}] \qquad (1)$$

where:
ρ – density of transported goods [t/m^3];
g – gravitational acceleration [m/s²];
z – vertical distance to the highest point the goods reach inside the warehouse [m].

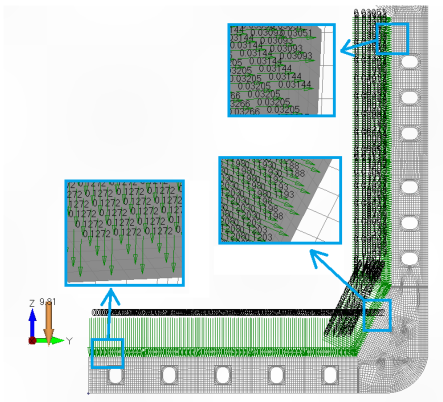

Fig. 7 Hydrostatic cargo pressure distribution

 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 = 11.25 m (**Fig. 8**).

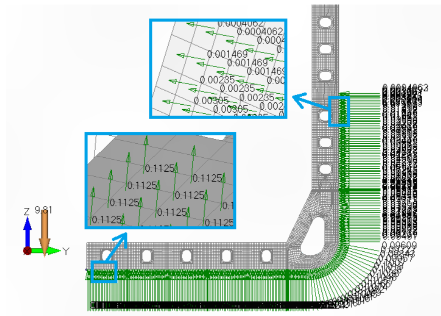

Fig. 8 Hydrostatic pressure generated by the still water (ρ_{water} = 1,025 t/m^3)

B. The case of static placement on the wave

In the case of static vessel placement on the wave, the 3D-FEM model shall be subjected to the following types of loads:

• gravitational loading (same as in the case of placement on still water);

• the load given by the goods (the same as in the case of placement on still water);

• the load in the equivalent quasi-static meeting wave, with Smith correction, with the equivalent hydrostatic pressure [N/mm^2], with the elongation relative to the basic plane of the vessel from the relation (2), considering the balancing parameters, calculated on the basis equivalent beam model, depending on the wave height h_w.

$$\begin{gathered}\zeta_w(x) = d_{pp} + (d_{pv} - d_{pp})\frac{x}{L} \pm \frac{h_w}{2}\cdot\cos\left(\frac{2\pi x}{L}\right) \\ x \in [0,L],\ \text{respectiv}\ x \in \left[x_{mpp}, x_{mpv}\right] \\ p_w(x) = \rho\cdot g\cdot \zeta_w(x),\ \rho = 1{,}025\,\mathrm{t/m^3} \\ h_w = 1.25\sqrt{\lambda}\end{gathered} \qquad (2)$$

Where λ, the length of the wave, is considered equal to Lvessel, the length of the vessel, to consider the most unfavorable case.

Following the calculation, the height of the wave used for the analysis on the sagging and hogging will be 7.3 meters (**Fig. 8** and **Fig. 9**).

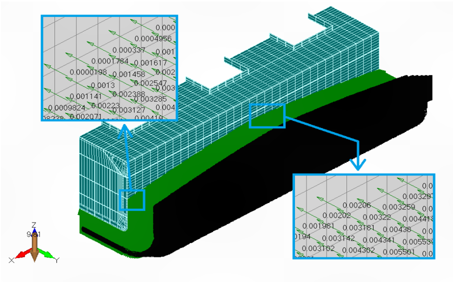

Fig. 8 Hydrostatic pressure – Hogging

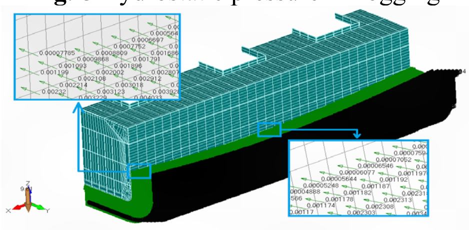

Fig. 9 Hydrostatic pressure – Sagging

6. RESULTS OBTAINED FOLLOWING FEM ANALYSIS

Following FEM analysis, the maximum allowable stresses that appear on the structure will be checked.

The stresses will be compared with the maximum values accepted by the IMO, by the Common Structural Rules Convention, for vessels with $L \geq 150$ m and for vessels with $L < 150$ m will be compared with the values imposed by the used classification company.

For the studied vessel the rules of the classification company DNV (**Table 4**) will be used.

Table 4 Permissible coarse mesh yield utilisation λ_{yperm}[5]

Structural member	*Acceptance criteria*	*Load components*[2)]	λ_{yperm}
Plating of all longitudinal hull girder structural members, primary supporting structural members and bulkheads. Dummy rod of corrugated bulkhead. Face plate of primary supporting members modelled using shell or rod elements.	AC-I	S	0.8[(3)]
	AC-II	S + D	1.0
	AC-III[(1)]	A, T	1.00
Corrugation of corrugated bulkheads under lateral pressure from liquid loads, for shell elements only. For corrugation angle between 45° and 55° the reduction in λ_{yperm} as given in Ch.3 Sec.6 [6.1.1] applies.	AC-I	S	0.72[(4)]
	AC-II	S + D	0.90
	AC-III[(1)]	A, T	0.90

1) For members of the collision bulkhead, AC-I shall be used.
2) See Ch.1 Sec.2 [4.2].
3) λ_{yperm} = 0.85 when hull girder permissible loads for harbour operations or special operations are applied.
4) λ_{yperm} = 0.77 when hull girder permissible loads for harbour operations or special operations are applied.

The allowable stresses for the plates and profiles used in the structure will be calculated by multiplying the flow limit of the material to $\lambda_{yperm} = 0.8$.

Following the calculations, the medium stresses on an element for steel with flow limit 235 N/mm^2 should not exceed the value of 188 N/mm^2.

A. The case of static placement on still water

For the interpretation of the analyses, the maximum stresses in the structure and the medium stresses on the element will be taken into account (**Fig. 10 and Fig. 11**).

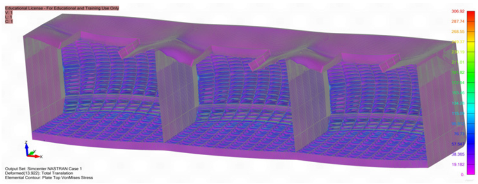

Fig. 10 Von Mises stresses on the complete model – still water condition

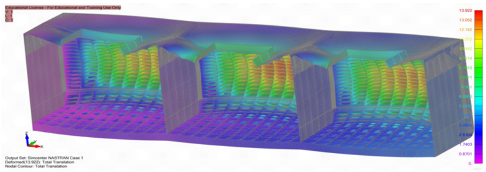

Fig. 11 Displacements in the complete model - still water condition

B. The case of static placement on the wave

B1. Von Mises stresses on the complete model – Sagging

For the interpretation of the analyses, the maximum stresses in the structure and the medium stresses on the element will be considered (**Fig. 12 and Fig. 13**).

B2. Von Mises stresses on the complete model – Hogging

For the interpretation of the analyses, the maximum stresses in the structure and the medium stresses on the element will be taken into account (**Fig. 14 and Fig. 15**).

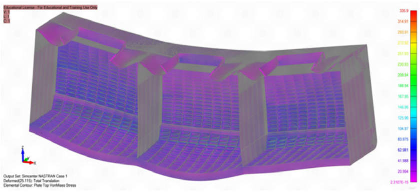

Fig. 12 Von Mises stresses on the complete model – Sagging

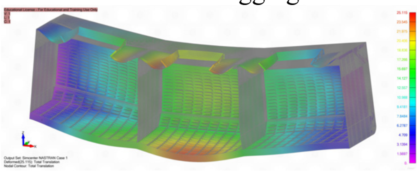

Fig. 13 Displacements in the complete model - Sagging

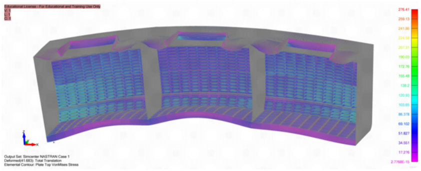

Fig. 14 Von Mises stresses on the complete model – Hogging

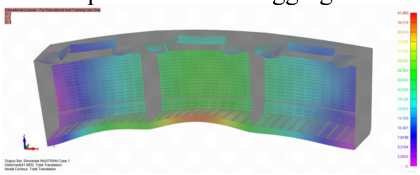

Fig. 15 Displacements in the complete model - Hogging

7. CONCLUSIONS

As it was seen in Figure 17, the maximum stress in the case of still water case is equal to 306.92 N/mm² and is located at the intersection of the reinforced frame of the bilge tank with the transversal frames.

Other significant values are found in the areas of technological cutouts in the bulkheads, at 173.78 N/mm², or around the technological cutout in the reinforced frame of the bilge tank, at 287.74 N/mm². As for displacements, the highest values are recorded around the double-side shell.

Therefore, we can see that in no stress concentration area, the values exceed the yield strength limit of the material used, which is 355 N/mm².

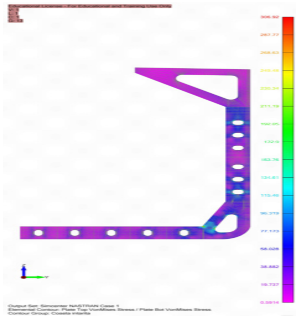

Fig. 16 Stresses in the reinforced frame - still water condition

In the case of lightship condition (Fig. 17), the maximum stress increases by approximately 30 N/mm², reaching the maximum value of 335.9 N/mm². This value is encountered in the same location as in the still water case.

Other increases in stresses can be found around the technological cutouts of the bulkheads and the double bottom frames, at 154.21 N/mm² and 212.15 N/mm². Additionally, around the technological cutout in the reinforced frame of the bilge tank, the maximum stress reaches 314.9 N/mm².

Therefore, it is considered that the maximum stress value does not exceed the yield strength limit of the material, but it comes significantly close to it.

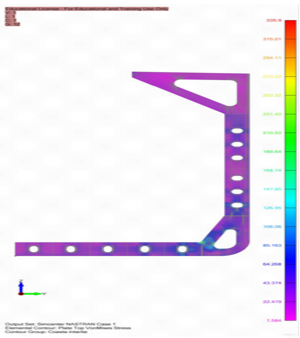

Fig. 17 Stresses in the reinforced frame - Sagging

Regarding the wave crest condition (Fig. 18), the maximum stress is no longer located along the reinforced frame, but at the intersection between the main deck and the hatch coaming, with a value of 276.41 N/mm².

Analyzing the stress values specifically in the reinforced frame, the highest value is recorded around the technological cutout in the reinforced frame of the bilge tank, at 185.35 N/mm², followed by 142.5 N/mm² in the bulkhead area, and 141.7 N/mm² in the double bottom frames area.

Therefore, the yield strength limit of the material used is not exceeded by any stress appearing in the entire model.

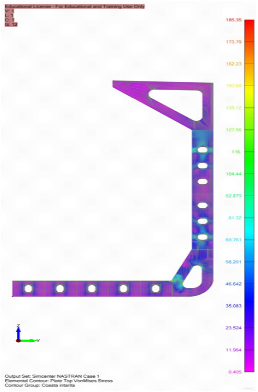

Fig. 18 Stresses in the reinforced frame - Hogging

As a result, based on the three analyses, it can be concluded that the structure is resistant to all acting forces. The yield strength is not exceeded in any of the analyzed cases. However, if an increase in strength is desired (considering the significant proximity of the maximum stress obtained in the lightship condition analysis to the yield strength limit), one could consider using a higher-grade steel, such as A420, which has a yield strength of 420 N/mm².

Therefore, the results can be used for structural strength optimization. However, it is important to consistently consider that the analyzed structure is simplified, as it lacks stiffening elements (such as brackets, stiffening plates, or stiffening ribs) which would normally play a significant role in maintaining structural strength.

8. REFERENCES

[1] Cristea A.G., *Contribuții privind optimizarea structurilor de navă*, Teză de doctorat, Galați, 2014

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

[3] Tărăbuși Ana-Maria, Cristea Anișoara-Gabriela, *The influence of plating with flat bars around technological cutouts on the stress state of a 37500 tdw chemical tanker,* The Annals of "Dunarea de Jos" University of Galati, 2023

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

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

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

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

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

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

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

Paper received on November 6th, 2023