THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2017

DETERMINATION OF DEFORMATIONS AND INDIRECT STRESS IN THE CENTRAL AREA BY FEM

Anişoara - 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

FEM models extended on a portion of the ship's length, with ship - wave balancing parameters calculated using the equivalent beam model, are suitable for the ship's body strength analysis in areas where global stresses are dominant, respectively for the central areas of cargo compartments. The purpose of this paper is to verify the induced stresses of a framework element from central area of the chemical tanker ship type. This static calculation was performed by the finite element method with Femap software as modeller and NX Nastran as solver.

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

1. INTRODUCTION

The main concern regarding the development with finite element is to generate a model providing the best possible results of the structural strength.

Sizing of the model was made in accordance with the rules of the classification company Germanischer Lloyd.

For modelling and analysis of the stresses around the relief cut-outs of a frame element, in this case of a floor, we used finite element software system FEMAP version 9.3.1.

The ship considered is a chemical tank type ship being designed for transporting chemicals.

1.1 Main dimensions and characteristics

Table 2 Balancing parameters in simplified 1D analysis

LOA=241358mm	Xpp=130000mm	Dpp=11620mm
Hwmax=10300mm	HHC=18000mm	Dpv=11620mm

Table 1 The main characteristics of the oil tanker (simplified structure)

L = 241358 mm	aL = 800 mm	E=2.1E+5 N/mm^2
B = 36000 mm	aF = 2400 mm	υ=0.3
D = 18000 mm	Znn = 8604 mm	ρ=7.7E-6 kg/mm^3
Lmag = 3magx19200 mm	Elsize = 200 mm	NDpp=392312
Xmpp = 130000 mm	Hwmax = 10300mm	NDpv=392311

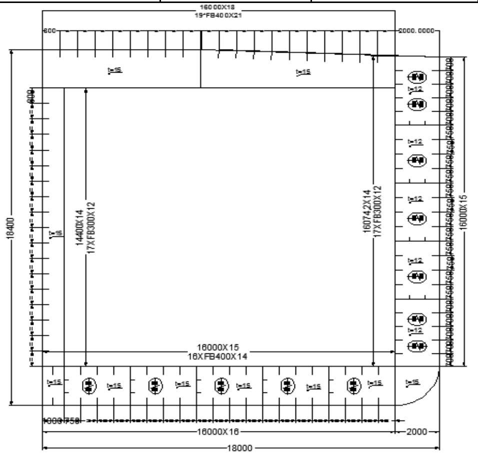

Fig. 1 Simplified midship section

1.2 Modelling the Structure

The hull structure considered in the 3-D hold model includes three cargo tanks of the parallel mid-body, as shown in Figure 1.

The global coordinate system of the finite element model is defined as follows:

X-axis: Longitudinal, positive from aft to fore;

Y-axis: Transverse (athwart ships), positive toward portside;

Z-axis: Vertical, positive upwards;

Origin: Base-line.

The following units are used for analysis:

- Length: millimetres (mm)
- Pressure: MPa (N/mm^2)
- Mass: kilogramme (kg)
- Stress: N/mm^2 (MPa)

2. MAKING THE 3D-CAD MODEL OF THE STRUCTURE IN THE CARGO AREA

2.1 Description of the model

Register conditions recommend, for the analysis with the finite element method of a ship structure, to consider two or three merchandise store rooms (cargo tanks) in order to reduce the influence of the edge conditions, throughout pressure distribution in the master section.

For this calculation we used a model extended on the length of three store rooms from the central area of the ship, so as to avoid the end effect and more precisely we have studied the central store room.

The elements of the plate were used for ground, double bottom, supports, and rod elements were used for simple framework.

2.2 Modelling with the FEMAP / NX Nastran software

FEM models extended on a portion of the ship's length, with ship - wave balancing parameters calculated using the equivalent beam model, are suitable for the ship's body strength analysis in areas where global stresses are dominant, respectively for the central areas of cargo compartments.

Usually, the body of merchant ships has full forms in the central area so that, for the model to be analyzed, it is enough to know the form of the midship section, with vertical borders and flat bottom.

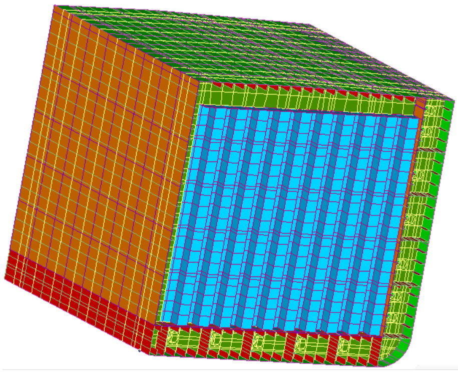

Fig. 2 3D - CAD hold

2.3 Defining material properties

The material used for this structural analysis is high strength steel whose stress at which the flow starts $\sigma_c = 355$ MPa and stress at which breakage starts is 500 MPa, the modulus of longitudinal elasticity (Young's modulus) is $E = 210\ GPa$ and Poisson's ratio is $\nu = 0.3$. Material density $\rho = 7850\ kg/m^3$ has been defined.

2.4 Making the 3D-FEM model of the structure in the cargo area

Modelling the structure has been continued with Femap software, but it is not wrong to use another software and then import the model. Based on 3D-CAD model, the 3D-FEM model is generated. In order to highlight stress concentrators in all structural elements, it is necessary to use the membrane and plate elements implemented in the FEM program.

The structure was divided into elements of approximately 200 mm^2, which are different in some cases. Meshing was performed manually with quad elements.

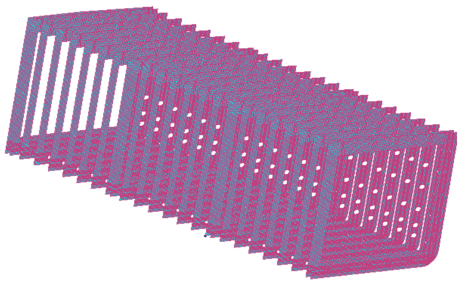

Fig. 3 Frames on the length of the holds - FEM

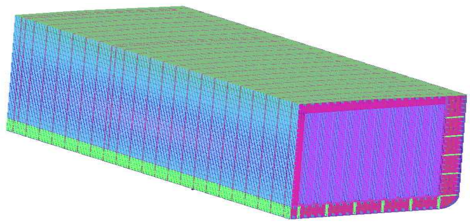

Fig. 4 The complete structure on the length of the holds 3D - FEM

2.5 Defining boundary conditions for the 3D - FEM model in the cargo area

The boundary conditions of the 3D - FEM model partially extended along the length of the ship are summarized in the following table.

Since the loads of the 3D - FEM model are in equivalent quasi-static encountering waves or in still water, model stresses are only vertical, symmetrical in relation to the fore-and-aft line of the ship (PD).

The symmetry condition in the fore-and-aft line, with the following degrees of freedom blocked:

- displacement in the Uy transverse direction, rotation along the longitudinal axis Rx and rotation along the vertical axis Rz;
- boundary condition at the aft extremity of the model.

Table 4 Boundary conditions for the 3D - FEM model [3]

Boundary condition	Blocked degrees of freedom					
	Ux	Uy	Uz	Rx	Ry	Rz
Symmetry in the diametric plane PD	-	x	-	x	-	x
Stern master node NDpp	x	x	x	x	-	x
Stern master node NDpv	-	x	x	x	-	x

At the stern of the ship model, for the NDpp node the following degrees of freedom will be blocked: all three displacement components, Ux, Uy, Uz and Rx, Rz rotations.

The stern control (master) point NDpp is defined with the coordinates xpp = xmpp; ypp = 0; zpp = zNNpp where zNNpp is the quota of the neutral axis in the cross section at the aft end of the model, in our case zNNpp = 8600 mm. Using RIGID elements, all degrees of freedom of the nodes in the stern section xmpp are coordinated through the degrees of freedom of the stern master node NDpp.

- boundary condition at the fore extremity of the model.

To be defined as in the above case for the stern.

At the bow of the model, the following degrees of freedom are blocked for the NDpv node: Uy, Uz displacements and Rx, Rz rotations.

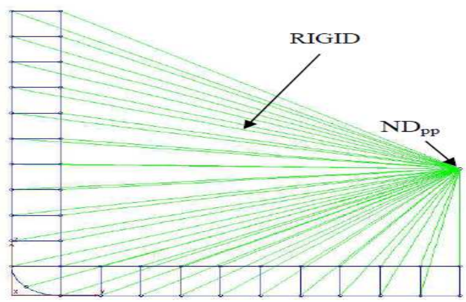

Fig. 5 Boundary conditions at the stern of the 3D-FEM ship model [3]

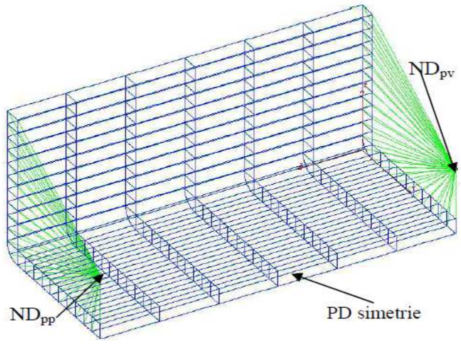

Fig. 6 Boundary conditions at the stern, the bow and the diametrical plane of the 3D-FEM model [3]

2.6 Defining loads for the 3D - FEM model in the cargo area

The 3D - FEM model corresponding to the centre of cargo compartments is subjected to the following types of stresses:

- gravitational load, from the net weight of the structural elements of the ship (g = 9.81 m/s^2, ρ_{steel} =7.85 t/m^3) and other components of the ship in the cargo area;
- stress from cargo, idealized on the shell of the double bottom and double board, as hydrostatic pressure in the cargo (0.9 t/m^3), for a HHC reference quota (D = 18000 mm).

2.6.1 The static pressure (Figure 7)

The static pressure of the merchandise, P_{in-tk}, is:

$$P_{in-tk} = \rho g z_{tk}, \text{[kN/m}^2\text{]} \qquad (1)$$

where:

z_{tk} - vertical distance the highest point of the merchandise in the tank, [m];

ρ - density of the merchandise in the tank, [t/m^3];

$g = 9{,}81 m/s^2$ - gravitational acceleration.

- stress from the bending moments on the stern master nodes NDpp, NDpv bow nodes, loading from the general stress of the beam, values being extracted from the Poseidon software, depending on the height of the quasi-static equivalent wave hw, according to the sign convention in the following table.

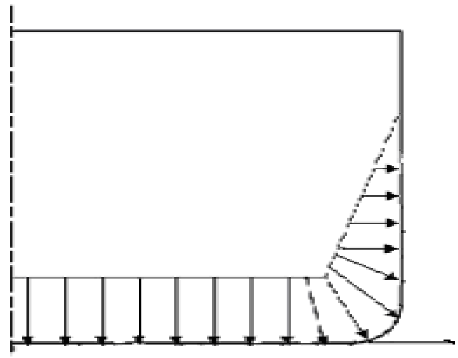

Fig. 7 Hydrostatic cargo pressure distribution

Table 4 Bending moments on master nodes for the 3D-FEM model

Wave condition	sag[kNm]	hog[kNm]
Value	2750266	2565622
Stern master node NDpp	+Mypp	-Mypp
Stern master node NDpv	-Mypv	+Mypv

- Stress from the quasi-static equivalent wave of encounter, with Smith correction, with equivalent hydrostatic pressure [N/mm^2], with elongation relative to the basic plan of the ship from (2) relation [3], taking into account the balancing parameters calculated based on the beam pattern h_w = 10300 mm.

$$\zeta_w(x) = d_{pp} + \left(d_{pv} - d_{pp}\right)\frac{x}{L} \pm \frac{h_w}{2}\cos\left(\frac{2\pi x}{L}\right) \quad x \in \left[x_{pp}, x_{pv}\right] \qquad (2)$$

2.6.2 The static pressure of the merchandise (Figure 8)

The static pressure of the merchandise, P_{hys}, is:

$$P_{hys} = \rho_{sw} g (T_{LC} - z), \text{[kN/m}^2\text{]} \qquad (3)$$

where:

z - the vertical coordinate of loading point in [m] and it should not be higher than T_{LC} ;

$\rho_{sw} = 1{,}025 t/m^3$ - density of sea water;

T_{LC} - draft in m;

$g = 9{,}81 m/s^2$ - gravitational acceleration.

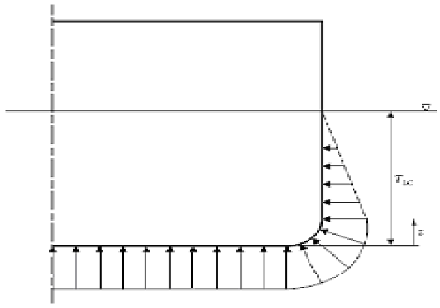

Fig. 8 Hydrostatic water pressure distribution

3. ANALYSIS OF RESULTS

3.1 The case of stress in the structure on wave hollow (sagging)

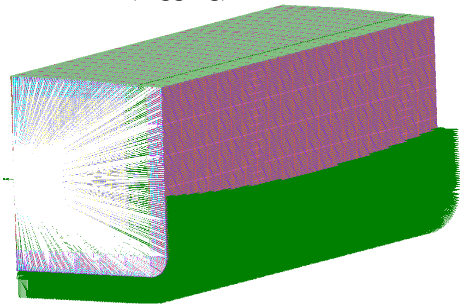

Fig. 9 The condition of structure stress given by water pressure, wave hollow case - FEM

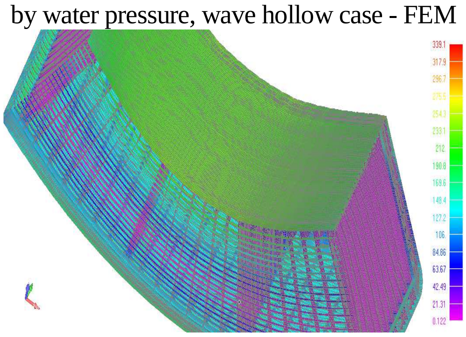

Fig. 10 Wave hollow case, Von Mises stress - FEM

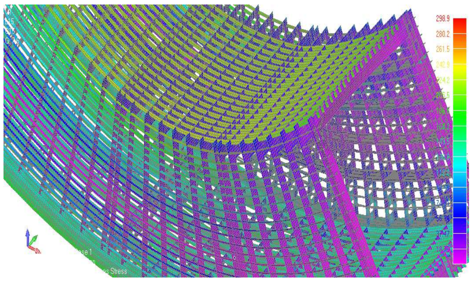

Fig. 11 Transverse and longitudinal framing on the length of the holds, wave hollow case, Von Mises stress - FEM

3.1.1 Results for the shell (sagging)

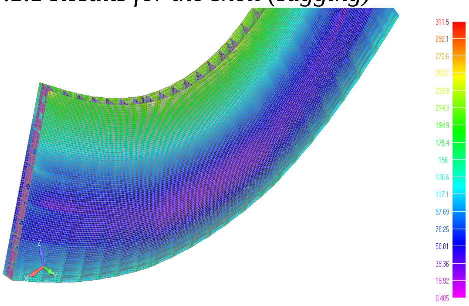

Fig. 12 Results for the double bottom, wave hollow case, Von Mises stress – FEM

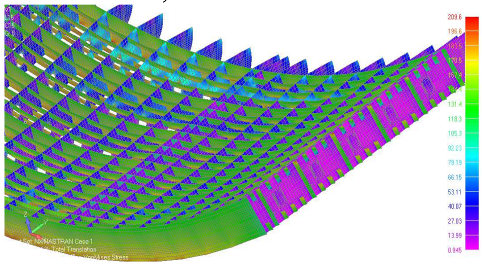

Fig. 13 Results for the inner bottom, wave hollow case, Von Mises stress – FEM

3.1.2 Results for the deck (sagging)

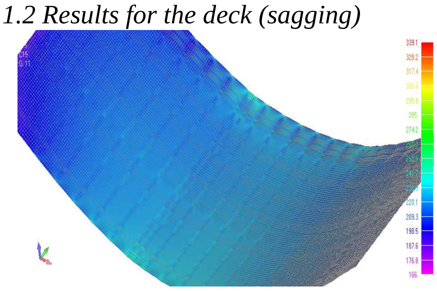

Fig. 14 Results for the deck, wave hollow case, Von Mises stress – FEM

In this case, the deck is subjected to compression, and stresses fall within 200 - 250 N/mm^2, as one can see, in areas without concentrators.

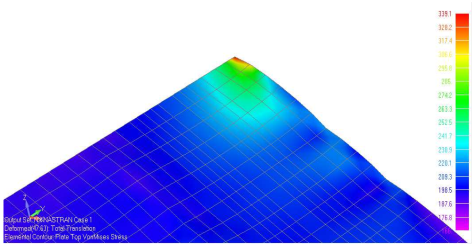

Fig. 15 Results for the deck, wave hollow case, Von Mises stress – FEM

It can be seen in the picture that there are also high stress concentrators that reach 330 N/mm^2, due to non-continuity of the structure.

3.2 Stress in the structure on wave crest (hogging)

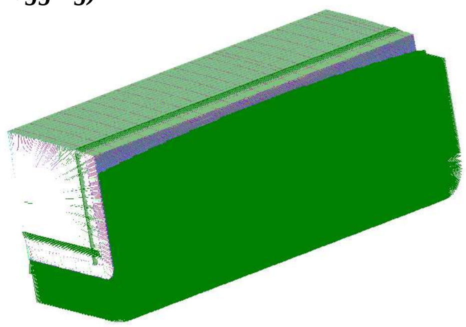

Fig. 16 The condition of structure stress given by water pressure, wave crest case – FEM

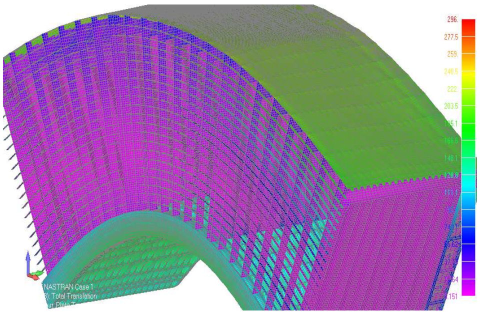

Fig. 17 Wave crest case, Von Mises stress - FEM

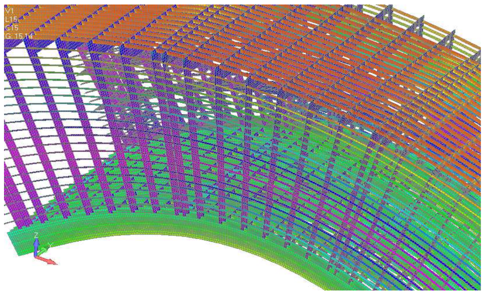

Fig. 18 Longitudinal and transverse framing on the length of the holds, wave crest case, Von Mises stress - FEM

3.2.1 Results for the shell (hogging)

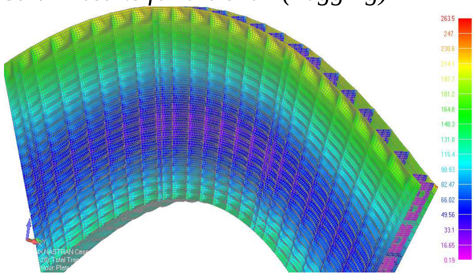

Fig. 19 Results for the shell, wave crest case, Von Mises stress – FEM

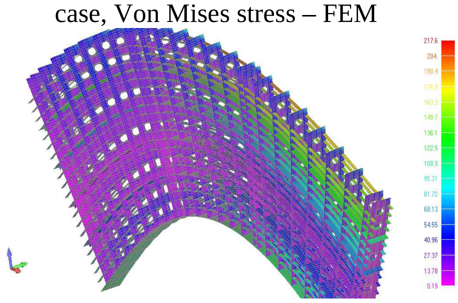

Fig. 20 Results for the shell, wave crest case, Von Mises stress – FEM

3.2.1 Results for the deck(hogging)

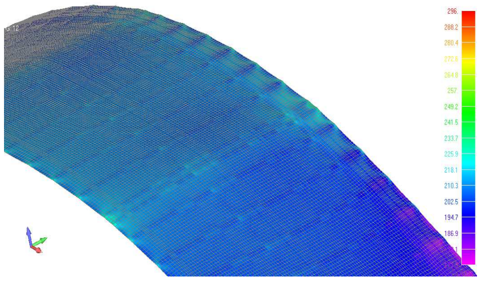

Fig. 21 Results for the deck, wave crest case, Von Mises stress – FEM

The deck is subject to elongation, and stresses fall as one can see within 180 - 220 N/mm^2 in areas without concentrators.

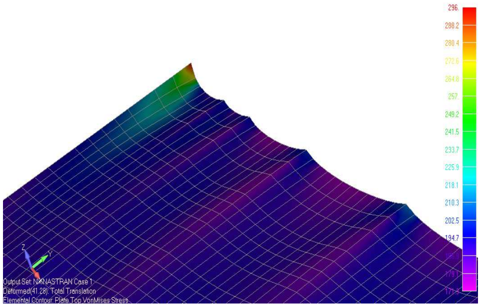

Fig. 22 Results for the deck, wave crest case, Von Mises stress – FEM

It can be seen in the picture that there are also high stress concentrators that reach 290 N/mm^2, due to non-continuity of the structure.

Stresses from the deck in the case of wave crest, due to deck stretching not to its compression, are lower than in the wave hollow case.

4. CONCLUSIONS ON DEFORMATIONS AND INDIRECT STRESS OF THE STRUCTURE

Analyzing the previous results, the following conclusions can be drawn:

- We can observe that the greatest stresses are found in the most stressed components of the structure, namely deck plating and bottom plating, due to their compression and elongation, depending on the stress case.
- Also, we can observe that tensions at the deck, in both cases of stress, are greater than the stresses of the lower part of the structure, namely, ship bottom. That is why it is recommended to increase the thickness of the sheets and of the structural elements in the upper part of the ship.
- The results obtained using this method can be used to optimize naval structures in terms of structural resistance.

Acknowledgements

The research was supported by the Research Centre of the Naval Architecture Faculty, in "Dunarea de Jos" University of Galati, which is greatly acknowledged.

5. REFERENCES

[1]. **Cristea A.G.**, "Determination of the induced stresses at the level of a floor using the finite element method", Editura Nautica, Constanţa, 2010.

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

[3]. **Domnişoru L.,** *"Metoda elementului finit în construcţii navale". Aplicaţii*, Editura Evrika, Brăila, 2003

[4]. **Domnişoru L., Găvan E., Popovici O.,** "*Analiza structurilor navale prin metoda elementului finit"*, Editura Didactică şi Pedagogică, Bucureşti, 2005

[5]. **Domnişoru L.,** "*Analiza structurilor navale prin metoda elementului finit. Aplicatii numerice"*, Editura Fundației Universitare "Dunărea de Jos", Galați, 2009

[6]. **x x x** – FEMAP 9.3.1 / NX NASTRAN 5.0 User's Guide, UGS Corporation / Siemens PLM Software Inc., 2007

Paper received on December 20th, 2017