

GLOBAL STRENGTH ANALYSIS USING 3D-FEM FULL EXTENDED SHIP MODELS, IN THE CASE OF EQUIVALENT QUASI-STATIC LOADS FROM OBLIQUE WAVES

Leonard Domnişoru

"Dunarea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domneasca Street, 800008, Romania,
E-mail: leonard.domnisoru@ugal.ro

ABSTRACT

This study focuses on a new numerical method developed for the ships' strength analysis in the case of quasi-static equivalent loads induced by oblique waves, on full 3D-FEM models. The ship-oblique wave equilibrium parameters are obtained by own program P_QSW, using an 1D ship model. The equivalent wave pressure load is applied on the 3D-FEM model by own subroutines and functions. The numerical tests are done on a 97m length offshore barge.

Keywords: ship 3D-FEM model, global strength, user subroutines and functions, oblique waves.

1. INTRODUCTION

Standard the global strength analysis of the ship hull structure is based on 1D-beam models [4],[7], 3D-FEM models partially extended over the ship length [5],[8] and 3D-FEM models one sided fully extended over the ship length [3], under head design waves. These approaches make possible the assessment of the ship's hull structure only for vertical in plane bending response.

In order to enhance the global strength analysis of the ship hull, the wave loads have to be considered for the generalised case of oblique equivalent waves. The advanced approach is based on 3D-FEM models fully extended over the ship length and both sides. The equilibrium parameters of the ship and oblique wave system are obtained by an 1D-beam model, based on a non-linear iterative procedure implemented in own program code P_QSW [4]. This approach makes possible the assessment of the global ship's strength for vertical and horizontal bending, with shearing and torsion, oblique wave loads.

In this study the user subroutines implemented into FEM program codes for applying the external equivalent hydrostatic oblique wave pressure over the 3D-FEM hull shell are presented. A numerical study case is considered for an offshore barge [4].

2. THEORETICAL BACKGROUND

2.1. 3D-FEM models by Femap/NX Nastran

(1) The 3D-CAD/FEM model development

Based on the ship offsetlines and the structural design project, using the CAD facilities of the Femap/NX Nastran [6], the 3D-CAD geometrical model of the ship hull, extended over the whole length and breadth of the ship, is obtained. Using the meshing techniques, the 3D-FEM model of the whole ship hull structure is obtained.

The 3D-FEM model is developed mainly based on the shell finite elements, with thick plate formulation (Mindlin) [1], [2], quad and triangle PLATE elements [6]. For the panels' ordinary stiffeners the BEAM

elements [1],[2] might be used. Some of the structural details are not included in the FEM model. Based on the structural response, where the extreme stress values are obtained, later detailed local 3D-FEM models have to be developed for the hot-spots stress domain to be obtained (not included in this study).

Using the same technique from cargo-holds or half sided 3D-FEM models [3], besides the own steel hull mass, the other onboard masses are added in the full 3D-FEM hull model. The onboard masses are considered to be with distributed or lumped mass formulation. The resulting total displacement Δ and gravity centre position (x_G, y_G, z_G) have to be according to the selected loading case. The FEM model masses are inducing the gravity loads (body loads) [6].

The ship model is considered mono-hull with CL symmetry plan, restriction coming up from the 1D-beam model [4] used for ship-wave equilibrium parameters computation.

(2) The 1D-beam equivalent hull model

The 1D-beam model [4] is developed for the equilibrium position of the ship and quasi-static oblique wave system computation.

It is very important to ensure the total compatibility between the 1D-beam and the 3D-FEM models, mass distribution and offset lines. Based on the 3D-FEM model, with user subroutines implemented into the FEM program, the mass diagram is obtained.

The ship - oblique wave equilibrium parameters by 1D-beam model approach is based on own non-linear iterative procedure, with program code P_QSW [4]. For a wave condition defined by h_w wave design height [10], ship-wave heading angle μ and quasi-static wave length λ , related to the ship's length L for maximum structural response [4],[7], the wave free-surface elevation is:

$$z_w(x,y) = d_m + (x-x_F) \cdot \theta + (y-y_F) \cdot tg(\varphi) \pm \frac{h_w}{2} \cos \left[\frac{2\pi}{\lambda} (x \cos \mu + y \sin \mu) \right] \quad (1)$$

$$x \in [0, L]; y \in [-B/2, +B/2]; \lambda = L \cos \mu \pm \text{sagging or hogging condition}$$

with the following 1D-beam model equilibrium parameters referring to the median plane of the quasi-static oblique wave: d_m vertical position (ship base plane reference); x_F, y_F centre plane position (ship CL and aft reference); θ, φ longitudinal and transversal trim (ship system reference).

The pressure from quasi-static oblique wave acting over the external shell of the 3D-FEM ship hull model has the expression:

$$p_w(x,y,z) = \rho \cdot g \cdot [z_w(x,y) - z]; x \in [0, L] y \in [-B/2, +B/2]; z \in [0, H] \quad (2)$$

where: L, B, H are the ship length, breadth and height; ρ is the water density; g is the gravity acceleration; $z=0$ at ship base plane.

(3) The 3D-FEM boundary conditions

The 3D-FEM model is fully extended from aft to fore and on both sides. Based on quasi-static oblique wave pressure (2),(1), with 1D-beam [4] parameters, the 3D-FEM model and wave are in equilibrium.

The boundary conditions have only to eliminate the rigid body 6 DOF degrees of freedom of the structural model, having zero reaction forces and moments. According to the IACS-CSR Rules [10], the boundary conditions are applied at aft and fore ship's model nodes as in Fig.1 and Table 1.

Table 1. 3D-FEM model boundary conditions

Node	1- U_x	2- U_y	3- U_z	4- R_x	5- R_y	6- R_z
ND _{fore}	x	x	x	-	-	-
ND _{aft 1}	-	x	-	-	-	-
ND _{aft 2}	-	-	x	-	-	-
ND _{aft 3}	-	-	x	-	-	-

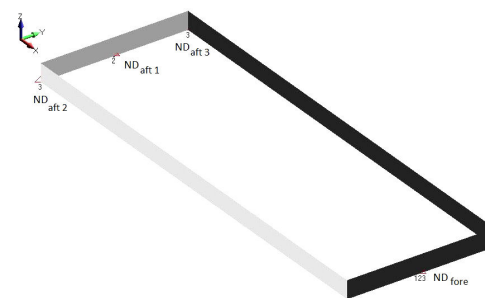


Fig.1. 3D-FEM model boundary conditions

(4) Quasi-static oblique wave pressure loads

Besides the gravity load (body) from all the model masses, on the 3D-FEM, the external load, induced by the oblique waves, has to be applied. The equivalent wave pressure is computed according to equation (2) for wave (1), with the ship-wave equilibrium parameters obtained by 1D-beam approach [4]. The external hull shell finite elements have to be checked and oriented so that the normal vector on face 1 is going from outside to inside the ship hull.

The external hull shell elements are selected and the following variables and function are used in order to apply the quasi-static equivalent oblique wave pressure.

- Tools / Define Variables (3)

Variable name: !HW
Value of Equations: < hw [m]>
Variable name: !LW
Value of Equations: < lambda [m] >
Variable name: !NIU
Value of Equations: < niu [deg] >
Variable name: !TETA
Value of Equations: < teta [rad]>
Variable name: !PHI
Value of Equations: < phi [rad]>
Variable name: !XF
Value of Equations: < xf [m]>
Variable name: !YF
Value of Equations: < yf [m]>
Variable name: !DM
Value of Equations: < dm [m]>
Variable name: !R
Value of Equations: < ro [t/m³] * g [m/s²] >
Variable name: !PI
Value of Equations: 3.141592654
Variable name: !EL
Value of Equations: ACTID(9)
Variable name: !A
Value of Equations: !HW/2
Variable name: !B
Value of Equations: 360*COS(!NIU)/!LW
Variable name: !C
Value of Equations: 360*SIN(!NIU)/!LW
Variable name: !D
Value of Equations: !TETA*180/!PI
Variable name: !E

Value of Equations: TAN(!PHI*180/!PI)

Variable name: !F

Value of Equations: !XF

Variable name: !G

Value of Equations: !YF

Variable name: !T

Value of Equations: !DM

Variable name: !H

Value of Equations: 0.0 {1.0E-14}

<OK>

{sagging +hw / hogging -hw} (4)

Model / Load / Elemental / Pressure

Direction: Normal to Element Face

Method: Variable

Advanced / Variable: EL

Value / Pressure: max (!H; (!R*(-ZEL(!EL)+
!T+(XEL(!EL)-!F)*!D+(YEL(!EL)-!G)*!E+
!A*COS((XEL(!EL)*!B+YEL(!EL)*!C))))

Selection Info / Face: 1

<OK>

In the *Analysis Set Manager* menu multiple cases can be defined, in order to run simultaneously the analyses for several loading cases corresponding to a set of wave heights. So several cases results will be in the same database and can be plotted on the same *XY chart*.

(5) The numerical results

The numerical results of the structural analysis include the deformations and stress distributions over the whole 3D-FEM. Based on the Femap/NX Nastran post-processing options [6] the stress charts (eq.von.Mises, normal or tangential) over the model length, at deck, bottom and side panels can be obtained.

View / Select / XY vs Position (5)

Position: X

Group / Select: <Elements Set>

Curve: <1>

Output Set: 1.NX Nastran Case 1

Output Vector: 7033: Plate Top VonMises Stress

.....
Curve: <9>

Output Set: 9.NX Nastran Case 9

Output Vector: 7033: Plate Top VonMises Stress

<OK>

<OK> List / Output / XY Plot

In order to improve the output stress diagrams and *XY charts plots* the following view setups can be applied:

```
Background (View) / Transparent / (6)
Top color 149 / Redraw (Window)
View Options
Label parameters / Use view Color / View color 0
Element / Use view color / View color 20
Load pressure / Use view color / View color 100
Filled Edges / No Draw Entities
<OK>
```

```
Background (View) / Solid / (7)
Top color 149 / Redraw (Window)
View Options / Post Processing
XY Titles / Top Centre / View color 0
XY Legend / ID Only / Centre Right
XY Axes Style/Axis color 100/ X Tics 11/ Y Tics 11
XY X Range / Grid / Max-Min
XY Y Range / Grid / Max-Min
XY Curve 1 / Points Only / Color <nr>
.....
XY Curve 9 / Points Only / Color <nr>
<OK>
```

Using *List / Output / XY Plot* menu, a list of the plotted results is obtained that can be exported into a *MS-Excel* file.

For selecting different FEM model parts, in order to assess the stress level, besides the *layers* technique, the following command sequence can also be applied:

```
View / Select / Model Data (8)
Load Set
Active → Select <load set nr & name>
Group
None → Select <group elements nr & name>
```

2.2. 3D-FEM models by SolidWorks Cosmos/M

Using the SolidWorks Cosmos/M [9] program for global strength analysis, based on fully extended 3D-FEM ship hull model, the analysis includes the same steps as for Femap/NX Nastran [6] (section 2.1). The changes involve the syntax of the user-subroutines and user-functions adapted to the programming language of the SolidWorks Cosmos/M [9] program. The 3D-FEM can be

developed with SHELL3T, SHELL4T finite elements, thick plate and membrane elements, triangle or quad, and the ordinary stiffeners with BEAM3D elements [1],[2],[9].

After selecting the external hull shell elements, the following user-subroutines for quasi-static equivalent oblique wave pressure definition are used.

```
File: press_oblique_wave.geo (9)
C* **OBLIQUE WAVES_PRESS [kN,t,m]
ACEL,0,0,-9.81
C* - Hw hogg, +Hw sagg
parassign, Hw, real, <hw [m]>
parassign, NIU, real, <niu [deg]>
parassign, DM, real, <dm [m]>
parassign, LL, real, <L [m]>
parassign, XF, real, <xf [m]>
parassign, YF, real, <yf [m]>
parassign, TETA, real, <teta [rad]>
parassign, PHI, real, <phi [rad]>
parassign, RO, real, <ro [t/m3]>
CALLMACRO, IN_EL_P
CALLMACRO, P_WAVE, HW, NIU, DM, LL,
XF, TETA, YF, PHI, RO
A_STATIC, G, 0, 0, 1E-006, 1E+010, 0, 0, 0, 0,
0, 0, 0, 2, 0, 0, 0, 0
R_STATIC
```

```
File: geomacro.geo (10)
$macro, IN_EL_P
INITSEL, EL, 1, 1
ESELPROP, RC, 9, 9, 1, 1
ESELPROP, RC, 14, 14, 1, 1
ESELPROP, RC, 19, 19, 1, 1
$ENDM
$macro, P_WAVE, HW, NIU, DM, LL, XF, TETA,
YF, PHI, RO
parassign, csid, int, 0
parassign, i, int, 0
#loop LB1 ELMAX
parassign, i, int, i+1
#if (exist(EL|i) && listsel(EL|i))
parassign, fnum, int, 0
parassign, x, real, XELF(i|fnum|csid)
parassign, y, real, YELF(i|fnum|csid)
parassign, z, real, ZELF(i|fnum|csid)
parassign, CC, real, x*COS(NIU*PI/180)
+y*SIN(NIU*PI/180)
```

```

parassign, LW,real,LL*COS(NIU*PI/180)
parassign, Tief,real,DM+(x-XF)*TETA
parassign,Tief,real,Tief+(y-YF)*TAN(PHI)
parassign,Tief,real,Tief+HW/2*COS(2*PI*CC/LW)
parassign, pval, real,(Tief-z)*RO*9.81
parassign, fnum, int,5
#if (pval>0)
    PEL, i, pval, fnum,i,1,4
#endif
#endif
#label LB1
$ENDM

```

The numerical results deformations and stress distributions over the 3D-FEM model can be plotted based on the menu *Results / Plot*. Using the following commands sequence, the *XY charts* for the stress components can be exported and plotted into a *MS-Excel* file.

```

Edit / List / Nodes (11)
{the nodes for the selected panel}
Results / List / Stress Component
Load case number: <1..9>
Stress flag: Node
Set number: SX/ SY/ SZ/ TXY/ TXZ/ TYZ
Set number: P1/ P2/ P3/ VON/ INT/ ERR
<OK>

```

3. THE 3D-FEM MODEL OF AN OFFSHORE BARGE

For the numerical global strength analysis, an offshore barge hull structure is considered with the characteristics in Table 2 [4]. The barge hull is prismatic with rectangular transversal sections and with equivalent uniform mass per unit length. The 3D-FEM model is developed with quadratic and triangle shell elements.

Based on 1D-beam model with own program code P_QSW [4], the equilibrium barge-wave parameters from Table 3 are obtained. The wave height is $h_w=0, 1, 2, 3, 4$ m (sagging and hogging) and the heading angle is $\mu=0, 15, 30, 45, 60, 75$ deg. Due to the prismatic barge shape, the equilibrium parameters remain unchanged for $h_w \leq 4$ m, having vertical sides all around the floating body and the wave elevation is up to the barge height $H=4$ m.

Figs.2.a-d present the 3D-CAD barge hull model by Femap/NX Nastran (FNN) [6].

Figs.3.a-e present the 3D-FEM barge hull model by Femap/NX Nastran (FNN) [6].

Figs.4.a-e present 3D-FEM model by SolidWorks Cosmos/M (SWC) [9] which has been imported using the *NAS2COS* module [9] for transfer between the two FEM codes.

Table 2. Offshore barge characteristics [4]

$L = 97$ m	$x_G = 48.5$ m	$\rho_m = 7.7$ t/m ³
$B = 33$ m	$y_G = 0$ m	No.PT=2648
$H = 4$ m	$z_G = 4.3$ m	No.CR=2496
$T = 2$ m	$h_w = 0, 1, 2, 3, 4$ m	No.SF=636
$\Delta = 6562.05$ t	$\mu = 0 \div 75$ deg	No.EL=239361
$v = 0$ m/s	$\delta\mu = 15$ deg.	No.ND=308625
$\rho = 1.025$ t/m ³	$E = 2.1e+11$ N/m ²	ELsize=0.1÷0.3m
$g = 9.81$ m/s ²	$\nu = 0.3$	No.prop.=17

Table 3. Equilibrium parameters barge-wave [4]

μ [deg]	λ [m]	eq. parameters:
0	97.00	$h_w = 0, 1, 2, 3, 4$ m
15	93.69	$x_F = 48.50$ m
30	84.00	$y_F = 0.00$ m
45	68.59	$d_m = 2.00$ m
60	48.50	$\theta = 0.00$ rad
75	25.11	$\varphi = 0.00$ rad

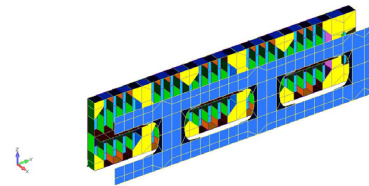


Fig.2.a. 3D-CAD, FNN, primitives objects

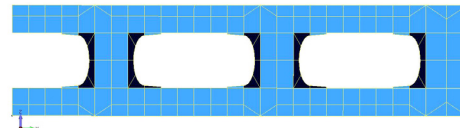


Fig.2.b. 3D-CAD, FNN, primitives for strengthened frame

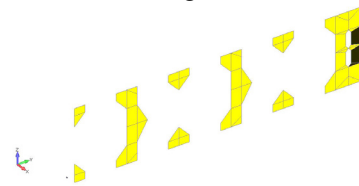


Fig.2.c. 3D-CAD, FNN, primitives for between frames

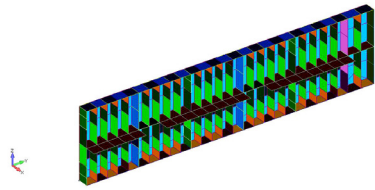


Fig.2.d. 3D-CAD, primitives LBH & TBH

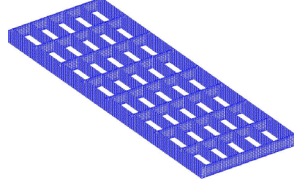


Fig.3.a. 3D-FEM, long. & transversal shells

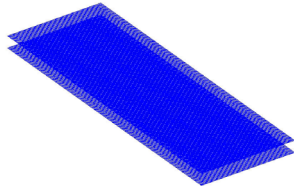


Fig.3.b. 3D-FEM, deck & bottom shells

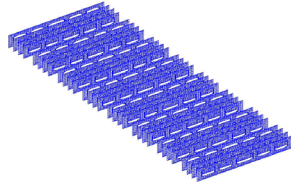


Fig.3.c. 3D-FEM, FNN, strengthened frames

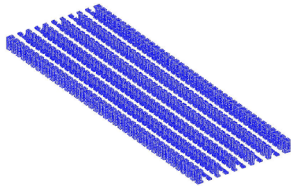


Fig.3.d. 3D-FEM, FNN, between frames

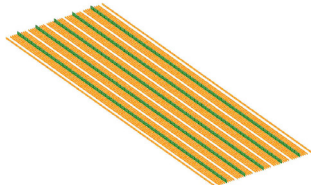


Fig.3.e. 3D-FEM, FNN, longitudinal stiffeners

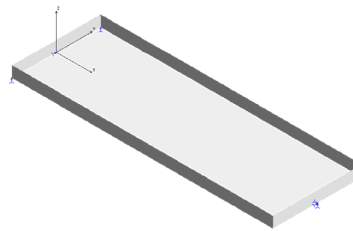


Fig.4.a. 3D-FEM, SWCM boundary conditions

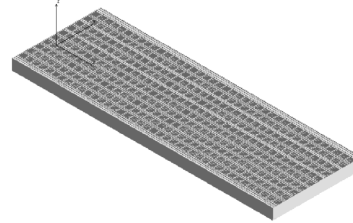


Fig.4.b. 3D-FEM - SW Cosmos/M model, (SWCM) longitudinal and transversal elements

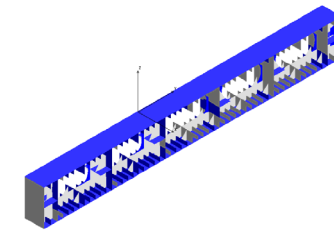


Fig.4.c. 3D-FEM - SW Cosmos/M model, (SWCM) frame details

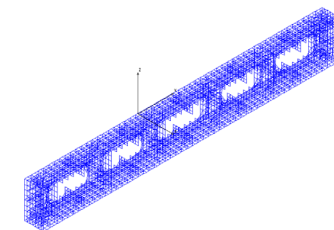


Fig.4.d. 3D-FEM - SW Cosmos/M model, (SWCM) frame details

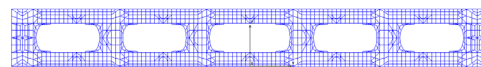


Fig.4.e. 3D-FEM - SW Cosmos/M model, (SWCM) frame details

4. NUMERICAL RESULTS

The results based on 3D-FEM models by Femap/NX Nastran [6] (section 2.1) are presented in the following figures: wave pressure, eq.von.Mises stress, deck, bottom and side panel stress distributions.

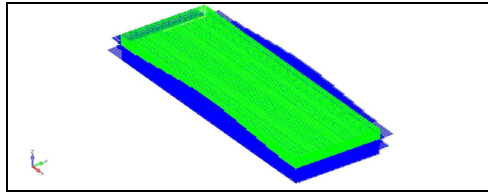


Fig.5.a. FNN, $h_w=4m$, $\mu=0$ deg, hogg. (00H) - wave pressure distribution

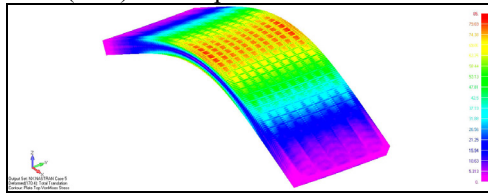


Fig.5.b. FNN, $h_w=4m$, $\mu=0$ deg, hogg. (00H) - eq.von.Mises stress [N/mm²]

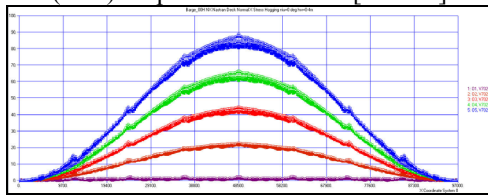


Fig.5.c. FNN, $h_w=0-4m$, $\mu=0$ deg, hogg. (00H) - deck normal stress [N/mm²]

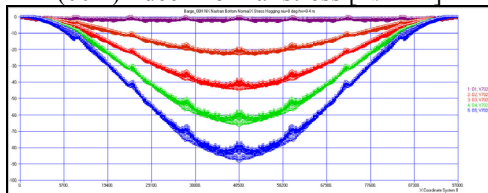


Fig.5.d. FNN, $h_w=0-4m$, $\mu=0$ deg, hogg. (00H) - bottom normal stress [N/mm²]

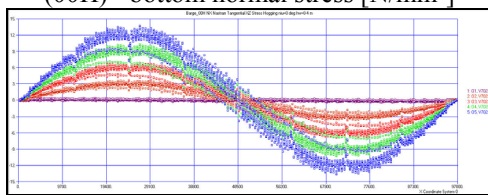


Fig.5.e. FNN, $h_w=0-4m$, $\mu=0$ deg, hogg. (00H) - side tangential stress [N/mm²]

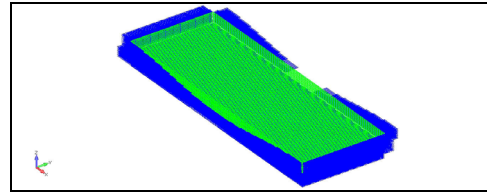


Fig.6.a. FNN, $h_w=4m$, $\mu=0$ deg, sagg. (00S) - wave pressure distribution

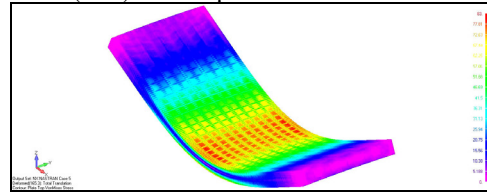


Fig.6.b. FNN, $h_w=4m$, $\mu=0$ deg, sagg. (00S) - eq.von.Mises stress [N/mm²]

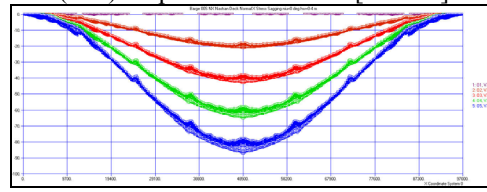


Fig.6.c. FNN, $h_w=0-4m$, $\mu=0$ deg, sagg. (00S) - deck normal stress [N/mm²]

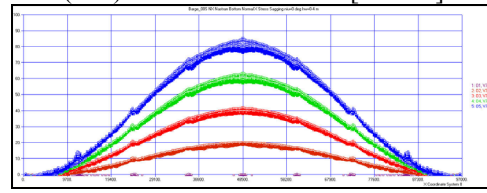


Fig.6.d. FNN, $h_w=0-4m$, $\mu=0$ deg, sagg. (00S) - bottom normal stress [N/mm²]

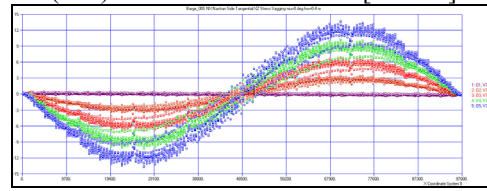


Fig.6.e. FNN, $h_w=0-4m$, $\mu=0$ deg, sagg. (00S) - side tangential stress [N/mm²]

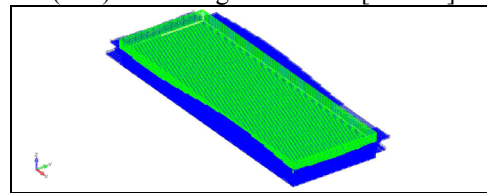


Fig.7.a. FNN, $h_w=4m$, $\mu=45$ deg, hogg. (45H) - wave pressure distribution

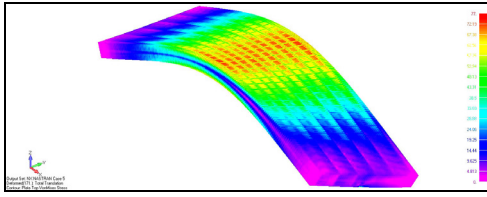


Fig.7.b. FNN, $h_w=4m$, $\mu=45$ deg, hogg. (45H) - eq.von.Mises stress [N/mm²]

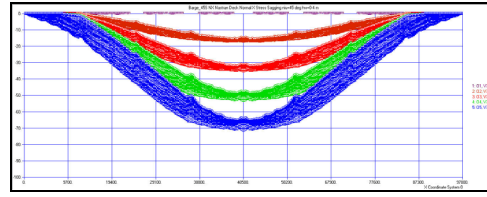


Fig.8.c. FNN, $h_w=0-4m$, $\mu=45$ deg, sagg. (45S) - deck normal stress [N/mm²]

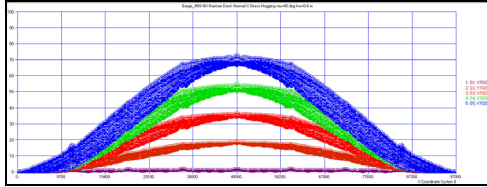


Fig.7.c. FNN, $h_w=0-4m$, $\mu=45$ deg, hogg. (45H) - deck normal stress [N/mm²]

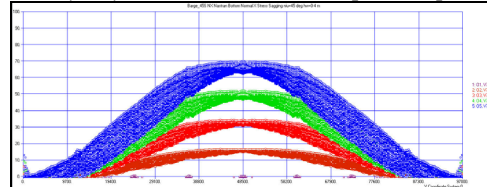


Fig.8.d. FNN, $h_w=0-4m$, $\mu=45$ deg, sagg. (45S) - bottom normal stress [N/mm²]

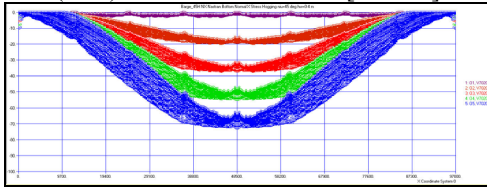


Fig.7.d. FNN, $h_w=0-4m$, $\mu=45$ deg, hogg. (45H) - bottom normal stress [N/mm²]

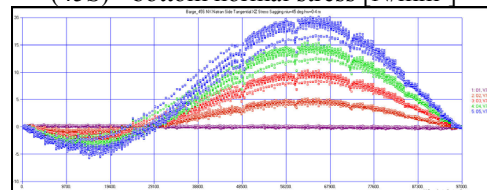


Fig.8.e. FNN, $h_w=0-4m$, $\mu=45$ deg, sagg. (45S) - side tangential stress [N/mm²]

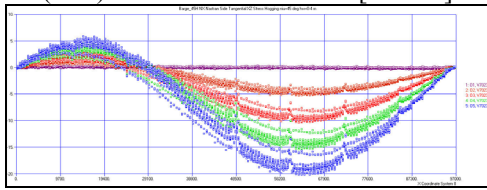


Fig.7.e. FNN, $h_w=0-4m$, $\mu=45$ deg, hogg. (45H) - side tangential stress [N/mm²]

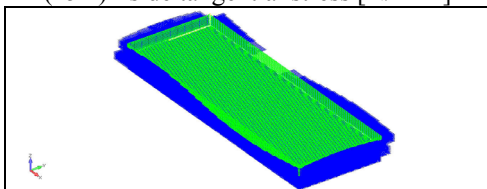


Fig.8.a. FNN, $h_w=4m$, $\mu=45$ deg, sagg. (45S) - wave pressure distribution

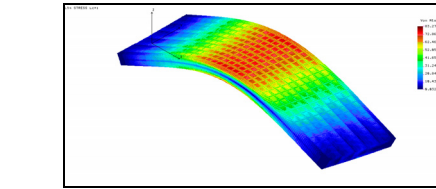


Fig.9.a. SWCM, $h_w=4m$, $\mu=30$ deg, hogg. (30H4) - eq.von.Mises stress [N/mm²]

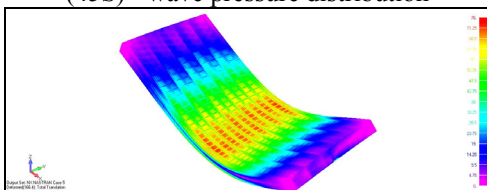


Fig.8.b FNN, $h_w=4m$, $\mu=45$ deg, sagg. (45S) - eq.von.Mises stress [N/mm²]

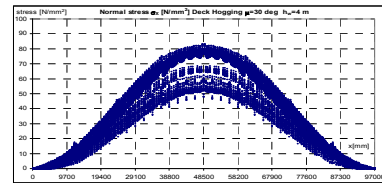


Fig.9.b. SWCM, $h_w=4m$, $\mu=30$ deg, hogg. (30H4) - deck normal stress [N/mm²]

The results based on 3D-FEM models by SolidWorks Cosmos/M [9] (section 2.2) are presented in the next figures: wave pressure, eq.von.Mises stress, deck, bottom and side panel stress distributions.

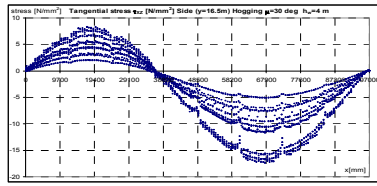


Fig.9.c. SWCM, $h_w=4m$, $\mu=30$ deg, hogg. (30H4) - side tangential stress [N/mm²]

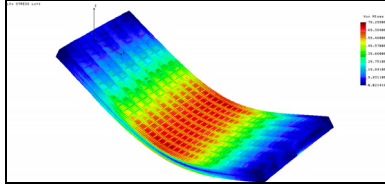


Fig.10.a. SWCM, $h_w=4m$, $\mu=30$ deg, sagg. (30S4) - eq.von.Mises stress [N/mm²]

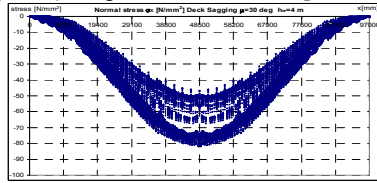


Fig.10.b. SWCM, $h_w=4m$, $\mu=30$ deg, sagg. (30S4) - deck normal stress [N/mm²]

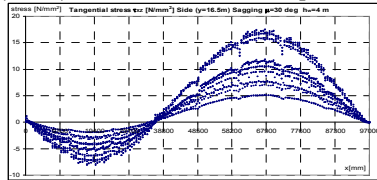


Fig.10.c. SWCM, $h_w=4m$, $\mu=30$ deg, sagg. (30S4) - side tangential stress [N/mm²]

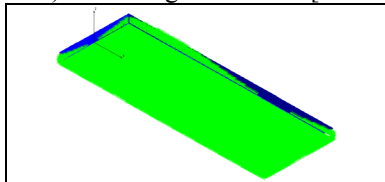


Fig.11.a. SWCM, $h_w=4m$, $\mu=60$ deg, hogg. (60H4) - wave pressure distribution

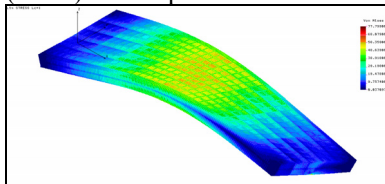


Fig.11.b. SWCM, $h_w=4m$, $\mu=60$ deg, hogg. (60H4) - eq.von.Mises stress [N/mm²]

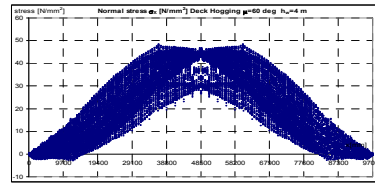


Fig.11.c. SWCM, $h_w=4m$, $\mu=60$ deg, hogg. (60H4) - deck normal stress [N/mm²]

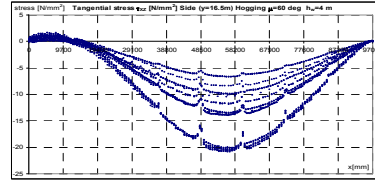


Fig.11.d. SWCM, $h_w=4m$, $\mu=60$ deg, hogg. (60H4) - side tangential stress [N/mm²]

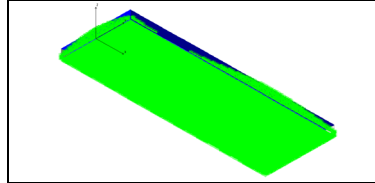


Fig.12.a. SWCM, $h_w=4m$, $\mu=60$ deg, sagg. (60S4) - wave pressure distribution

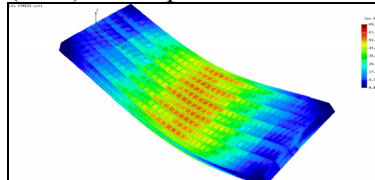


Fig.12.b. SWCM, $h_w=4m$, $\mu=60$ deg, sagg. (60S4) - eq.von.Mises stress [N/mm²]

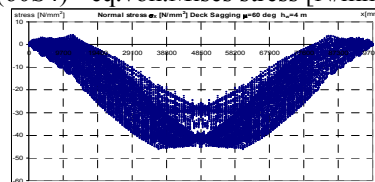


Fig.12.c. SWCM, $h_w=4m$, $\mu=60$ deg, sagg. (60S4) - deck normal stress [N/mm²]

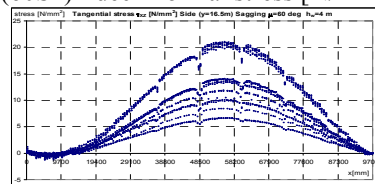


Fig.12.d. SWCM, $h_w=4m$, $\mu=60$ deg, sagg. (60S4) - side tangential stress [N/mm²]

5. CONCLUSIONS

Based on the numerical results from section 4 for the offshore barge [4] from section 3, in Tables 4 & 5 the maximum stress results.

Table 4. Maximum stresses [N/mm²] based on 3D-FEM model by Femap/NX Nastran [6]

FNN	$h_w=4m$	Deck(z=4m)		Bottom(z=0m)		Side(y=16.5m)			
		μ	H - S	σ_{vonM}	σ_x	σ_{vonM}	σ_x	σ_{vonM}	τ_{xz}
00H	0	hogg		89.48	89.85	88.63	-89.75	89.48	12.84
00S		sagg		87.95	-88.51	85.89	86.39	87.95	12.72
15H	15	hogg		88.55	88.76	87.70	-88.64	88.55	-15.12
15S		sagg		87.01	-87.39	85.00	85.31	87.01	15.10
30H	30	hogg		85.06	84.58	84.24	-84.42	85.06	-17.40
30S		sagg		83.50	-83.12	81.64	81.18	83.50	17.48
45H	45	hogg		77.03	74.31	75.81	-74.12	76.55	-19.80
45S		sagg		74.94	-72.69	73.42	71.04	74.94	19.92
60H	60	hogg		64.16	48.68	62.59	-50.49	54.61	-20.86
60S		sagg		62.79	-46.50	60.94	47.38	53.04	20.86
75H	75	hogg		22.92	-18.97	24.62	17.85	19.53	-7.75
75S		sagg		22.96	20.82	24.97	-20.57	21.27	7.88

Table 5. Maximum stresses [N/mm²] based on 3D-FEM model by SolidWorks Cosmos/M [9]

SWC	$h_w=4m$	Deck(z=4m)		Bottom(z=0m)		Side(y=16.5m)			
		μ	H - S	σ_{vonM}	σ_x	σ_{vonM}	σ_x	σ_{vonM}	τ_{xz}
00H	0	hogg		85.01	88.72	82.20	-87.54	84.13	12.85
00S		sagg		83.59	-87.33	79.73	84.55	82.76	12.77
15H	15	hogg		84.32	87.46	81.11	-86.34	83.19	-15.07
15S		sagg		82.85	-86.04	78.58	83.38	81.80	15.19
30H	30	hogg		82.01	83.12	77.92	-82.19	79.64	-17.35
30S		sagg		79.85	-81.61	75.39	79.20	78.19	17.56
45H	45	hogg		77.90	72.69	70.81	-72.39	70.96	-19.75
45S		sagg		75.65	-71.02	67.66	69.59	69.40	20.03
60H	60	hogg		64.94	48.33	53.26	-48.22	49.34	-20.85
60S		sagg		63.39	-45.80	50.26	45.52	47.00	21.04
75H	75	hogg		22.91	-18.58	21.38	17.20	18.12	-7.72
75S		sagg		22.67	20.38	20.63	-20.47	19.77	7.94

The conclusions of the analysis are:

1. Based on the 1D-beam model [4], the ship-wave equilibrium parameters are obtained. Using section 2 theoretical approach, the wave pressure is applied on 3D-FEM models. The 3D-FEM models result as being well balanced, ensuring a realistic structural response (section 4).
2. The offshore barge structure is made of standard steel grade type A [10], $R_{eH}=235$

($\sigma_{adm}=175$ N/mm², $\tau_{adm}=110$ N/mm²). The yielding stress limit criterion is satisfied (Tables 4,5).

3. At oblique waves, due to the changes of the wave length and the relative position ship-wave, the maximum stress values have significant variations function of the heading angle $\mu=0-75$ deg (section 4, Tables 4,5).

4. The barge stress hot-spots are not significant, having a uniform and robust structure.

5. Differences occur between the two 3D-FEM models stress results (Tables 4,5), due to FEM implemented code procedures for stress analysis and elements formulation [6],[9].

6. In the further studies, the theoretical approach from section 2, with ship - oblique wave equilibrium parameters [4], will be tested on more complex ship hull structures.

Acknowledgements

In this study the program codes and the numerical models were developed in the frame of UDJG Galati Research Centre of the Faculty of Naval Architecture.

REFERENCES

- [1]. **Bathe, K.J.**, "Finite Elementen Methoden", Springer Verlag, Berlin, 1990.
- [2]. **Domnisoru, L.**, "The finite element method applied in shipbuilding", Technical Publishing House, Bucharest, 2001.
- [3]. **Domnisoru, L., Dumitru, D., Ioan, A.**, "Numerical methods for hull structure strengths analysis and ships service life evaluation, for a LPG carrier", OMAE 15-20 June 2008, Estoril: 509-518, 2008.
- [4]. **Domnisoru, L.**, "Numerical approach for global ship strengths analysis based on 1D-beam model under oblique equivalent quasi-static wave loads", The Annals of "Dunarea de Jos" University of Galati, Fascicle XI, Shipbuilding, Galati, 2015 (in press).
- [5]. **Eyres, D.J.**, "Ship construction", Butterworth Heinemann, Boston, 2006.
- [6]. **FNN**, Femap NX Nastran Program, 2007.
- [7]. **Hughes, O.F.**, "Ship structural design. A rationally based, computer-aided optimization approach", The Society of Naval Architects and Marine Engineering, New Jersey, 2000.
- [8]. **Lehmann, E.**, "Guidelines for strength analyses of ship structures with the finite element method", Germanischer Lloyd's, Hamburg, 2000.
- [9]. **SWCM**, SolidWorks Cosmos/M Program, 2007.
- [10]. **xxx**, ABS, BV, DNV-GL, IACS-CSR, Classification Societies Structural Rules, 2010-2015.

Paper received on November 30th, 2015