Volume XXVII 2024
ISSUE no.2
MBNA Publishing House Constanta 2024

Scientific Bulletin of Naval Academy

SBNA PAPER • OPEN ACCESS

3D-FEM Modeling of a Parallelipipedic Crane for Head and Oblique Design Waves Testing

To cite this article: Elisabeta (Buzilă) Burlacu, Scientific Bulletin of Naval Academy, Vol. XXVII 2024, pg. 8-19.

Submitted: 15.03.2024
Revised: 20.06.2024
Accepted: 12.09.2024

Available online at www.anmb.ro

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-24-I2-001

3D-FEM MODELING OF A PARALLELIPIPEDIC CRANE FOR HEAD AND OBLIQUE DESIGN WAVES TESTING

Elisabeta (Buzilă) Burlacu

"Ovidius" University of Constanța,
Mamaia str., no. 124, Room S40, RO-900527, Constanța, România
Web: *www.imim.univ-ovidius.ro*
burlacu.elisabeta@gmail.com

Abstract. The FEM modeling is one of the most commonly used types of testing for structures in various industries. In the ship industry, FEM modeling and testing are required for any type of ship during the construction process, in the event of different changes occurring to the building strategy. In this case, the paper presents FEM modeling and preparation for a parallelepipedic crane barge with a length of 42.59 m, a breadth of 20.19 m, a depth of 1.50 m and a construction height of 3.5 m. The model will be tested for head (H.D.W.) and oblique (O.D.W.) design waves, also for buckling.

Keywords: 3D-FEM modelling, parallelepipedic crane barge, design waves testing

1. Purpose of the FEM analysis

The primary purpose of conducting Finite Element Method (FEM) analysis is to gain a comprehensive understanding of the structural behavior and performance of engineering components or systems. FEM provides a powerful numerical technique for simulating and predicting the response of structures under various loading conditions. Engineers and designers employ FEM to assess the integrity, strength, and stability of materials and structures, ensuring they meet safety and performance standards. By subjecting a virtual representation of a physical object to simulated conditions, FEM allows for the identification of potential weaknesses, optimal design modifications, and the exploration of alternative materials and configurations.

In addition to structural assessments, FEM analysis serves as a crucial tool in the optimization of designs, helping engineers refine and enhance their creations. The ability to simulate real-world conditions and evaluate the impact of different factors on the structural integrity empowers designers to iterate and fine-tune their models before physical prototypes are built. FEM aids in minimizing the need for costly and time-consuming trial and error approaches, ultimately leading to more efficient and robust engineering solutions. Whether applied in the aerospace, automotive, civil engineering, or other industries, FEM analysis plays a fundamental role in ensuring that structures not only meet safety standards but also perform optimally in their intended environments.

The FEM design waves testing for ships involves a comprehensive simulation and analysis process using Finite Element Method (FEM) to assess the structural response of marine vessels to various design wave conditions. This methodology is essential in the maritime industry, allowing engineers to evaluate the impact of head and oblique waves on the ship's structure during the design and construction phases. By subjecting a virtual model of the ship to simulated wave loading scenarios, FEM enables the identification of potential stress points, structural vulnerabilities, and areas requiring reinforcement. This rigorous testing ensures that the ship's design not only meets safety standards but also guarantees optimal performance and durability in diverse and challenging sea conditions, contributing to the overall reliability and seaworthiness of the marine vessel.

In this paper, some of these results are presented for a *floating crane hull*, in different heights of *head and follow design waves*. The numerical results are making possible to evaluate the several operations of floating crane by strength criteria for the first numerical study. Future studies will propose different cranes, with different loading cases for testing the admissible resistance in case of its exploitation.

2. Basic CAD/CAM model of the crane hull

In the initial phase, after reviewing the technical plan of the barge, we created its shell using the *Rhinoceros program*. This shell was then imported into the *Femap / NX Nastran program* [5]. We opted for this approach due to the fact that the Rhinoceros program is much more user-friendly for modeling complex surfaces, such as those presented in *Figure 6*, especially for the extreme corners of the barge.

Below we have the steps of creating the crane hull, the constraints and the loads for the studies in this paper.

2.1. Creating the CAD/CAM Model (Points, Curves, Surfaces layers)

According to the ***"Crane ponton construction plan"*** made by *BV Scheepswerf in '87* (figures 5, 7 and 8), that is used for operations in the Black Sea harbors of Constanța, in *table 1* are presented the main characteristics of the hull crane and the FEM model.

Table. 1. Main data of the crane

Length overall	$L_{OA}[mm]$	42590	Young module	$E\left[{}^{N}/_{mm^2}\right]$	$2.1 \cdot 10^5$
Design breath	$B[mm]$	20190	Poisson ratio	v	0.3
Design height	$D[mm]$	3650	Material density	$\rho_{A36}\left[{}^{kg}/_{mm^3}\right]$	$7.8 \cdot 10^{-6}$
Design draft	$T[mm]$	1500	Yielding stress	$R_{eH}\left[{}^{N}/_{mm^2}\right]$	285
Number of elements	Nr_{el}	231976	Gravity acc.	$g\left[{}^{m}/_{s^2}\right]$	9.81
Number of nodes	Nr_{nod}	220958	Element type 3D-FEM	thick shell and lumped mass	

The first step was to create the shell of the crane, in Rhinoceros and then import the surface in Femap [5]. All the elements and the main characteristics indicated in the construction plan wore modelled in Femap [5], with the procedures that are permitted to use in this software. So, in *table 2* are presented the properties and layer associativity of the model. For this specific crane hull there wore used a number of **8 properties** and **27 layers** according to table 2, also the thicknesses of the model are shown in *figures 1 to 4*. *Figure 9 to 11* presents some views over of the 3D-FEM model, which also presents the thickness of the elements distributed along the model, and the dimension of the elements across the crane structure.

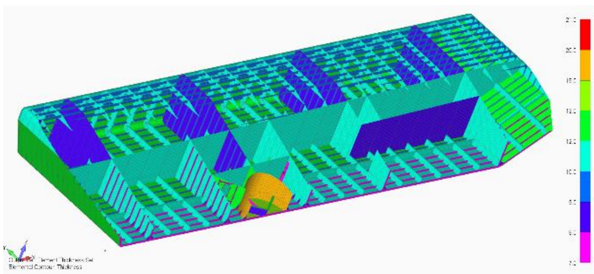


Fig. 1. View of thicknesses

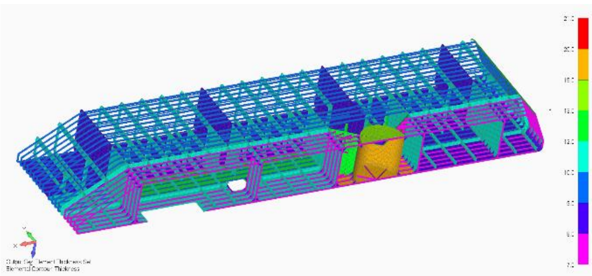


Fig. 2. View of thicknesses

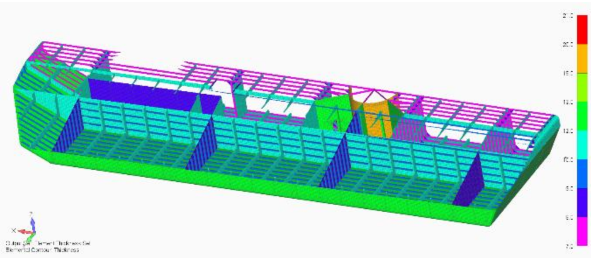


Fig. 3. View of thicknesses

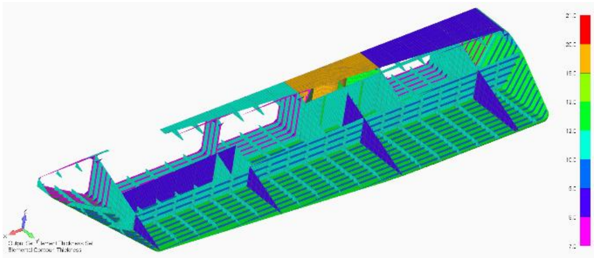


Fig. 4. View of thicknesses

Table. 2. Properties (8 properties) associated to Layers (27 layers)

Crt. No.	Property Thickness [mm]	Layer
1.	7 mm	HP profile 160*7 (1 layer)
2.	8 mm	Main deck, long. BHD at 3307 from C.L., BHD at FR 2 – 8 – 14 – 20, Bracket, crane girders (5 layers)
3.	9 mm	L profile 125*75*9 (1 layer)
4.	10 mm	Main deck, Bottom shell, long. BHD at 5032 from C.L., L profile 410*110*10 – on bottom shell, L profile 365*110*10 – on shell side, main deck and BHD at 5032 from C.L., BHD at FR 2 – 6 – 10 – 14 – 20 (7 layers)
5.	12 mm	Main deck, Bottom shell, Side shell, Aft and Fore shell, sloped section and FR 8 at crane level, L profile 160*160*12, crane girder (9 layers)
6.	13 mm	Bottom shell at crane level (1 layer)
7.	15 mm	Main deck at crane level, crane column (2 layers)
8.	20 mm	Hole stiffener FB 80*20 at sloped section (1 layer)

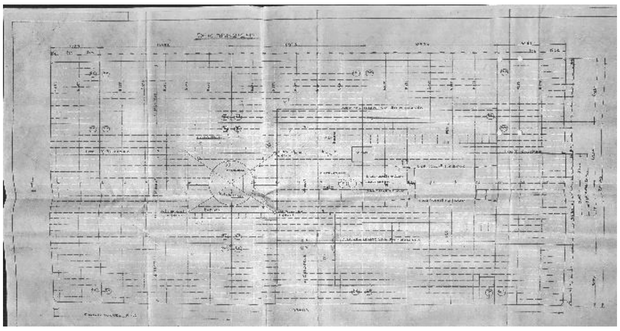


Fig. 5. Main deck of the "Crane ponton construction plan"

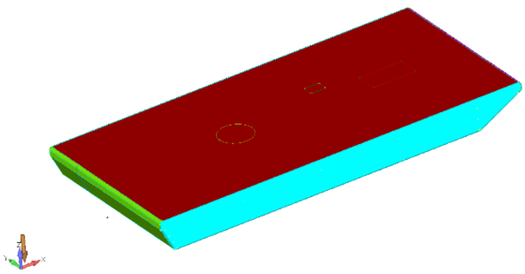


Fig. 6. Shell model imported in Femap of the crane ponton

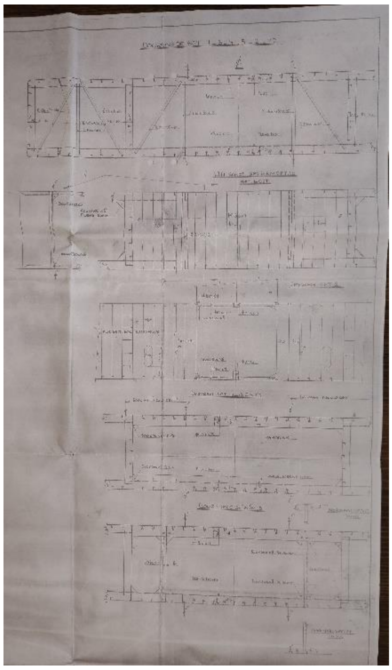


Fig. 7. Frames "Crane ponton construction plan"

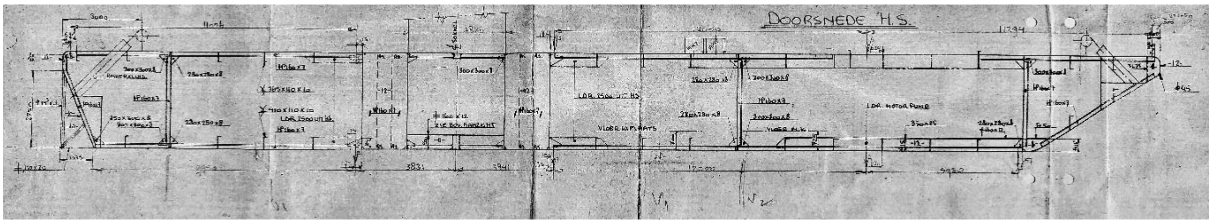


Fig. 8. C.L. section “Crane ponton construction plan”

2.2. Creating the FEM model (materials, properties and mesh elements and nodes)

2.2.1. Creating Materials. One of the most important aspects of the FEM analyses is the correct materials characteristics. In our case, the crane is made from steel materials with grade A36. For this material we have a Poison ratio of 0.3, and a Youngs Modulus $E = 2.1 \cdot 10^5 \,{}^{N}\!/_{mm^2}$, according to table 1. For the FEM model, we will use an isotropic material type. [1], [2], [3], [4]

2.2.2. Creating Properties. For the model, we used the properties presented in table 2.

2.2.3. Creating mesh. After all the surfaces are made and connected, using the Femap [5] procedure <u>Mesh on surface</u>, each surface is decisioned with the property that is assigned. So, in figures 9, 10 and 11.a., b. are presented the FEM model of the crane hull.

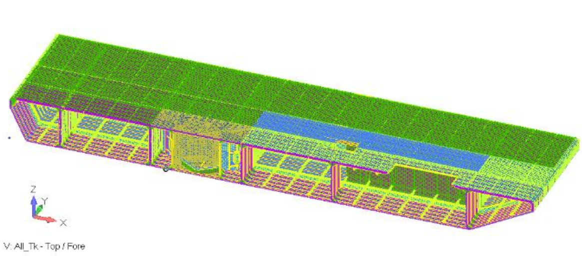


Fig. 9. Top – Fore View of the hole 3D-FEM crane hull model

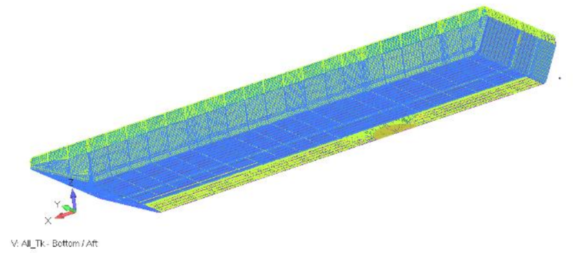


Fig. 10. Bottom – Aft View of the hole 3D-FEM crane hull model

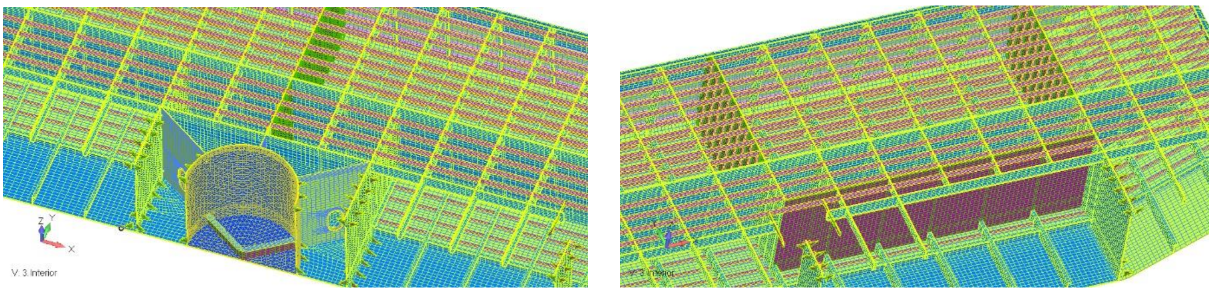


Fig. 11.a., b. View of the interior of the crane, a. crane column view, b. drink water tank, brand stop tank and water ballast tank

2.3. Preparing the model for analyses

2.3.1. Creating constraints or boundary conditions. After the hull model is finished and checked for coincident nodes and elements, the constraints and the loads are the next step for our purpose of the study. In this case, for the two types of waves, the boundary conditions are presented in *Table 3*, below. The “x” mark is the freedom degree that must be checked for the case scenarios. [4], [6], [8]

Table. 3. Boundary conditions used for the 3D-FEM model [4], [8]

Wave type	Boundary condition name and coordinates	X_{trans}	Y_{trans}	Z_{trans}	X_{rot}	Y_{rot}	Z_{rot}
Head / Follow EDW	N_{simm} (x from 0 to L_{OA}, y=0, z from 0 to D	-	x	-	x	-	x
	N_{Pv} (L_{OA}, 0, T=1,5)	-	x	x	x	-	x
	N_{Pp} (0, 0, T=1,5)	x	x	x	x	-	x
Oblique EDW	N_{Pv} (L_{OA}, 0, 0)	x	x	x	-	-	-
	N_{Pp-CL} (0, 0, 0)	-	x	-	-	-	-
	N_{Pp-Ps} (0, B/2, 0)	-	-	x	-	-	-
	N_{Pp-Sb} (0, -B/2, 0)	-	-	x	-	-	-

2.3.2. Creating loads. [2], [4], [6], [7], [8] The next step is to create the loads for the Gravity acceleration (acc. Step a), the hydrostatic pressure on the BHD of the tanks (acc. step b.) and the load acc. to the equivalent cvasi-static wave (acc step c.). Equation 1 [4], [8] was used for the hydrostatic pressure on the BHD of the tanks. Also, for the cvasi-static equivalent wave type, the formula used is presented in equation 2 [4], [8] for head ($\mu = 0^0$) and follow ($\mu = 180^0$)), and in equation 3 [4], [8] for oblique waves ($\mu = 0^0 \dots 360^0$).

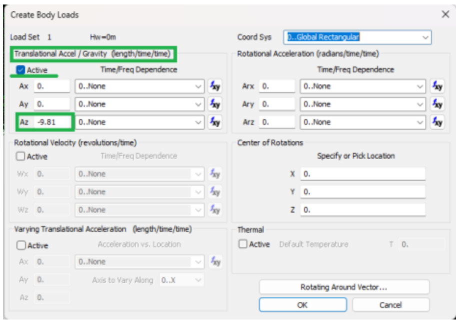


Fig. 12. Creating body load – Gravity acceleration

<u>*Step a.- Creating gravity acceleration*</u> *(figure 12.) > Model / Load / Body / Translational Accel / Gravity – Active -* $A_Z = -9.81$*. / OK*

<u>*Step b.- Creating the hydrostatic pressure on the BHD of the tanks*</u> *> Model / Load / Elemental (after this step we must select the elements that are used for the condition of pressure we create)*

In these cases, we need to define some variables, so that, the program, cand distribute a variable pressure according to the parameters marked with "!" in equation 1 [4], [8]. The pats to find the variable declaration command is *Tools / Variables* or *"Ctrl+L"*.

Step b.1. – Defining the variables – The variables defined are "$!\,H_{tank}$" for defining the reference height of the pressure to go up to, for the elements selected and "$!\,EL = ACTID(9)$ " the command of Femap to read the elements characteristics (ID, Layer, Color, Type, Property, Coordinates)

Step b.2. – Defining the hydrostatic pressure on the BHD of the tanks – for this case, after selecting the elements of the tank, the variable hydrostatic pressure according to equation 1 [4], [8] is defined. In our case of model, the density that is relevant for our tanks is $\rho = 0{,}9\, {}^{t}/_{m^3} = 0.9e-5$ for the ballast tanks, $\rho = 0{,}8\, {}^{t}/_{m^3} = 0.8e-5$ for the oil tank and $\rho = 0{,}748\, {}^{t}/_{m^3} = 0{,}748e-5$ for the fuel tank.

$$(\rho * (!\,H_{tank} - ZEL(!\,EL))/1000 \quad (1)$$

<u>*Step c.- Creating the cvasi – static equivalent wave pressure*</u> *> Model / Load / Elemental (after this step we must select the elements that are used for the condition of pressure we create)*

The waves for the scenario cases have heights from 0m – steal water up to 2,75m for the cases of head, follow and oblique waves. For this case also we have to declare some variables. They are presented below in step c.1. After the definition of these variables, we can define de variable pressure from the wave height scenario, using in the formula "+" if we are creating a sagging wave and "-" if we want to create a hogging wave. This is also regarded in the oblique wave type.

Step c.1. – Defining the variables – The variables defined are "$!\,EL = ACTID(9)$ " the command of Femap to read the elements characteristics (ID, Layer, Color, Type, Property, Coordinates), "$!\,H_{wave}$" the value of the wave height, "$!\,L_{OA}$" the length of the floating crane, "$!\,T_{Pp}$" and "$!\,T_{Pv}$" the drafts of the crane at aft and fore.

*Step c.2. – Defining the head / follow EDW (**fig. 13 to 16**)* – For the head / follow EDW we must each time when we define the load, to go back, and give the value "$!H_{wave}$" the value of the wave height. Equation 2 [4], [8] represents the formula which will define our wave. Figures 13 to 16 presents some of the wave's scenario.

$$max\big(0.000;\big(\rho_{whater}*g*\big(-ZEL(!EL)+!T_{Pp}+ \\ +\big(!T_{Pv}-!T_{Pp}\big)*\frac{XEL(!EL)}{!L_{OA}}\pm\frac{!H_{wave}}{2}*cos\left(2*180*\frac{XEL(!EL)}{!L_{OA}}\right)\bigg)\bigg) \quad (2)$$

Step c.3. – Defining the oblique EDW – For the oblique EDW, the variables that must be prepared for each case of study are the wave height - "$!H_{wave}$" and the ship – wave heading angle – "μ". For this case the model must be full extended all over length in both sides.

$$max\left(0.000*\left(\rho_{whater}*g*\left(-ZEL(!EL)+!T_{mid}+(XEL(!EL)-!x_F)*!\theta*\frac{180}{!\pi}+ \\ +(YEL(!EL)-!y_F)*tan\left(!\varphi*\frac{180}{!\pi}\right)\pm\frac{!H_{wave}}{2}* \\ *cos\left(XEL(!EL)*360*cos\frac{!\mu}{!\lambda}+XEL(!EL)*360*sin\frac{!\mu}{!\lambda}\right)\right.\right.\right. \quad (3)$$

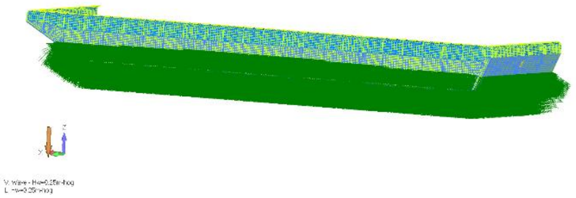


Fig. 13. Equivalent design wave pressure, hogging condition, $H_{wave}=0.25m$

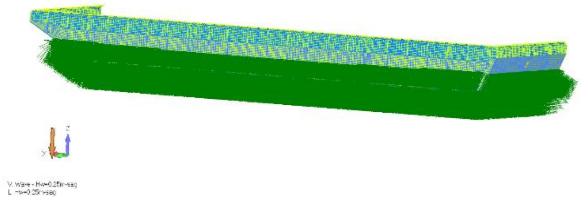


Fig. 14. Equivalent design wave pressure, sagging condition, $H_{wave}=0.25m$

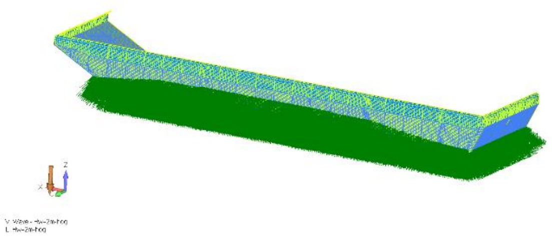


Fig. 15. Equivalent design wave pressure, hogging condition, $H_{wave}=2m$

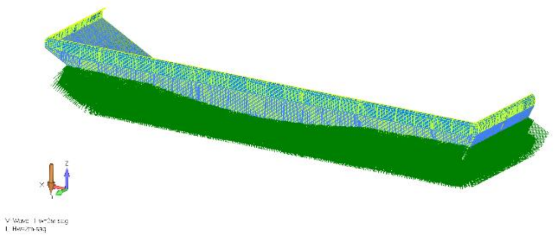


Fig. 16. Equivalent design wave pressure, sagging condition, $H_{wave}=2m$

2.4. Creating the Analyses

The analyses used are the static and buckling type analyses from the *Simcenter Nastran solver*. The analyses wore made on an AMD Ryzen 5 2600 Six-Core Processor, with 16GB DDR3 RAM @1600MHz and a 500GB SATA SSD, and the time for each case was 10 - 12 minutes for the static analyses and up to 20 - 25 minutes for the buckling analyze.

2.5. Results

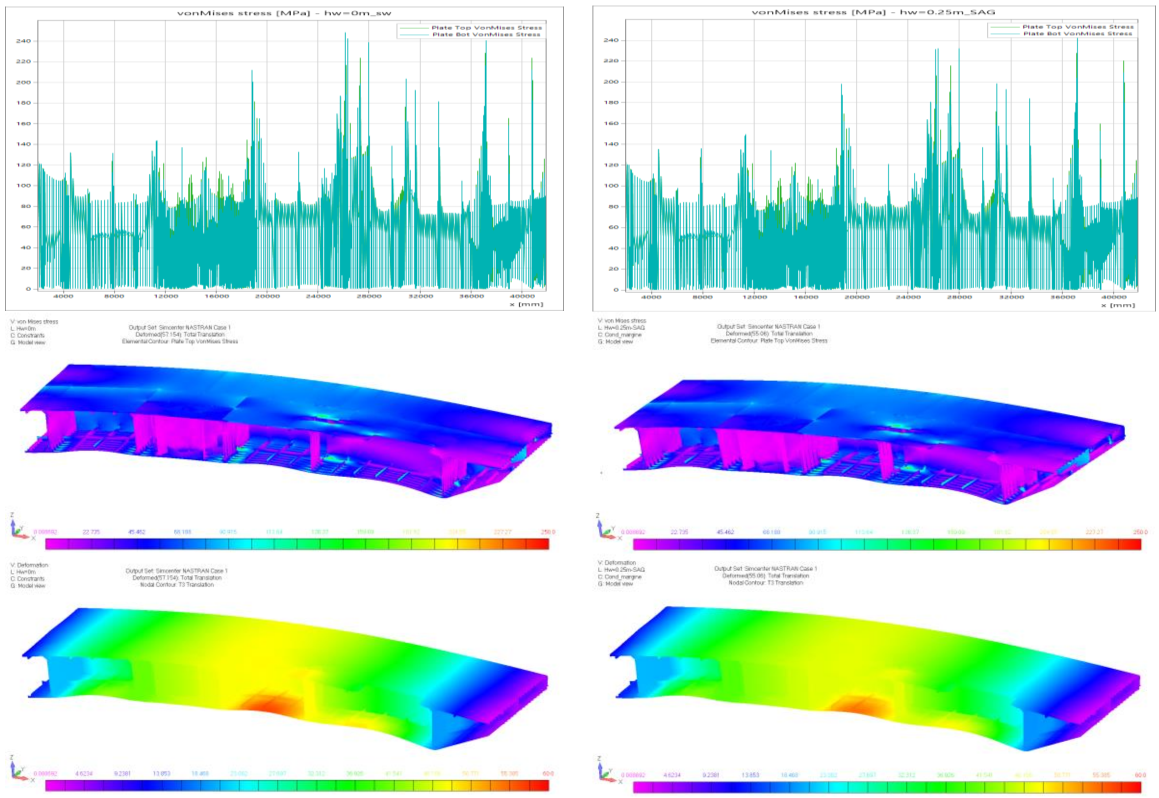
In this paper are presented the vertical deformations, the equivalent von Mises stress, the X normal stress and the eigen value for buckling for the 3D-FEM model of a crane hull, only for the head / follow EWD. These maximum values of the results are presented in *table 4*, for the vectors "4", "7020", "7033", "9020" and "9033". For some of these results *figures 17 to 25*, presents in "a." and "b" plot the von Mises stress for wave condition, in "c." plot it is presented the deformation of the 3D model along the length, in the "d.' plot the deformation at the wave condition, and in "e." plot normal X stress. For the buckling values, some of the representative figures are presented in *figures 27 to 32*.

Also, for the oblique waves, the studies will be mentioned after creating the model of the hull crane full extended over the dock length, in both sides.

Table. 4. Von Mises stress $\sigma_{vonMises}$, normal X stress and vertical deformation w for different wave height for head / follow EWD

Case	$h_w[m]$	$w[mm]$	Normal X [MPa] – Top Plate	Normal X [MPa] – Bottom Plate	$\sigma_{vonMises}[MPa]$ – Top Plate	$\sigma_{vonMises}[MPa]$ – Bottom Plate	Buckling factor [-]
Steal water	0	57.14	248.320	191.574	234.592	243.495	1.507753
Hogging	0.25	59.24	247.806	202.900	261.362	265.227	1.500905
	0.5	61.347	260.939	214.166	279.251	282.148	1.507958
	0.75	63.462	273.987	225.372	297.157	299.081	1.506484
	1	65.586	286.957	236.521	315.082	316.025	1.536334
	1.25	67.719	299.847	247.614	333.021	332.978	1.500498
	1.5	69.860	312.656	258.648	350.972	349.940	1.504111
	1.75	72.011	325.385	269.626	368.935	366.908	1.500947
	2	74.171	338.034	280.546	386.909	383.884	1.52438
	2.25	76.34	350.608	291.412	404.891	400.864	1.513
	2.5	78.519	363.100	302.220	422.882	417.850	1.546126
	2.75	80.708	375.515	312.974	440.880	434.840	1.509988
Sagging	0.25	55.05	221.307	180.200	227.644	243.363	1.533174
	0.5	52.962	207.953	168.773	230.300	246.153	1.503545
	0.75	50.883	194.523	161.399	232.980	248.935	1.566359
	1	48.808	181.033	158.958	235.681	251.712	1.524487
	1.25	46.739	167.482	156.556	238.405	254.488	1.505279
	1.5	44.673	153.885	154.181	241.149	257.266	1.509903
	1.75	42.957	143.581	151.838	243.914	260.049	1.523277
	2	42.179	142.028	149.526	246.700	262.842	1.506338
	2.25	41.401	143.717	148.792	249.502	265.646	1.556993
	2.5	40.623	145.407	150.553	252.323	268.467	1.516148
	2.75	39.846	147.096	152.314	255.160	271.305	1.517907

a.
b.
c.



d.

e.

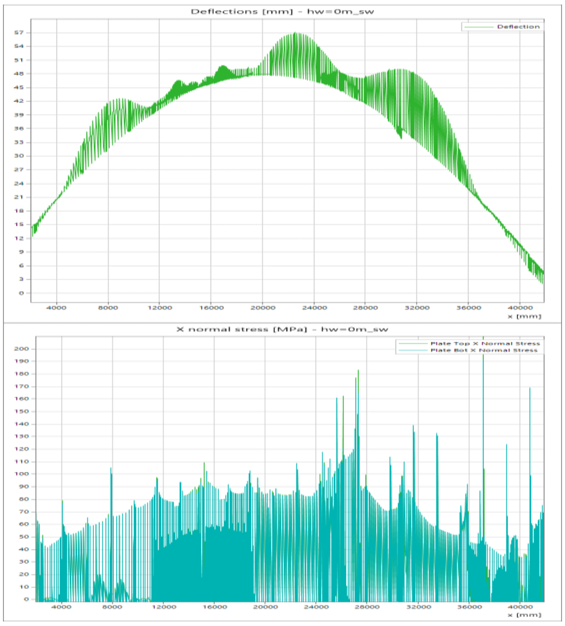


Fig. 17. Wave height case Hw=0m

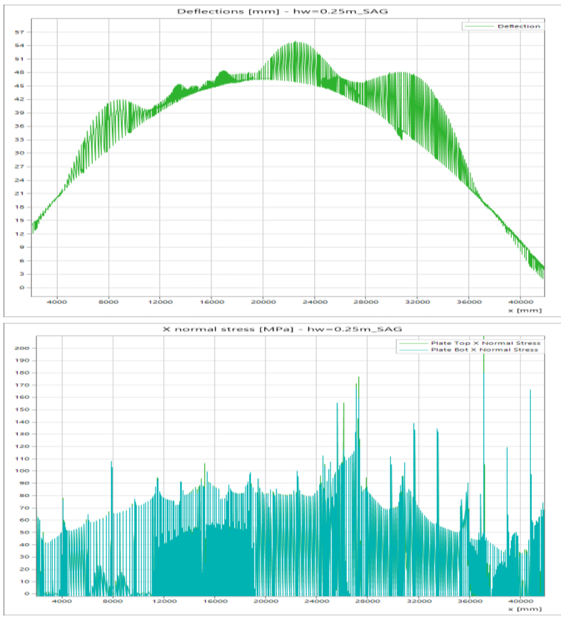


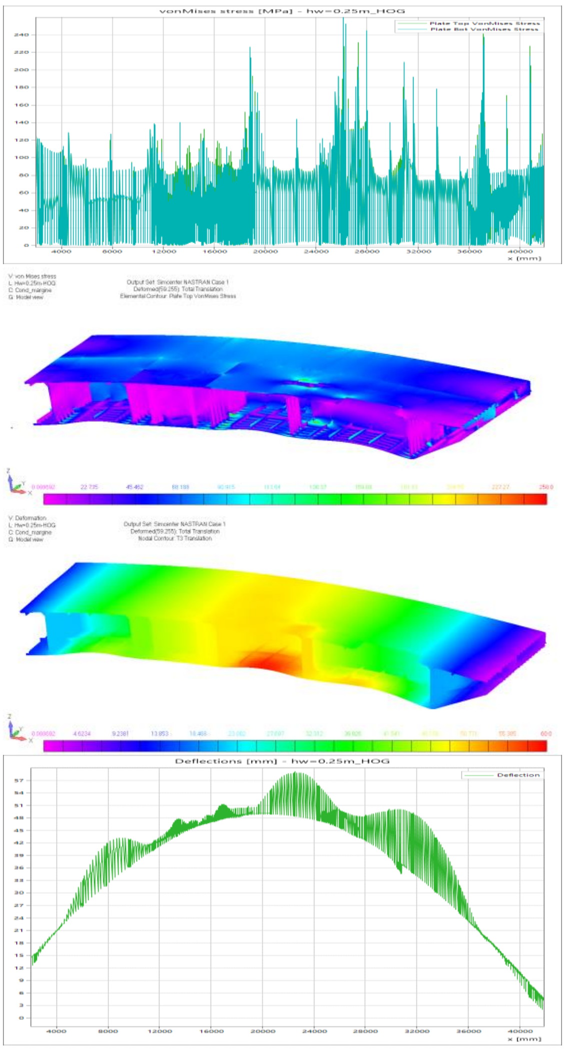
Fig. 18. Wave height case Hw=0.25m - SAG

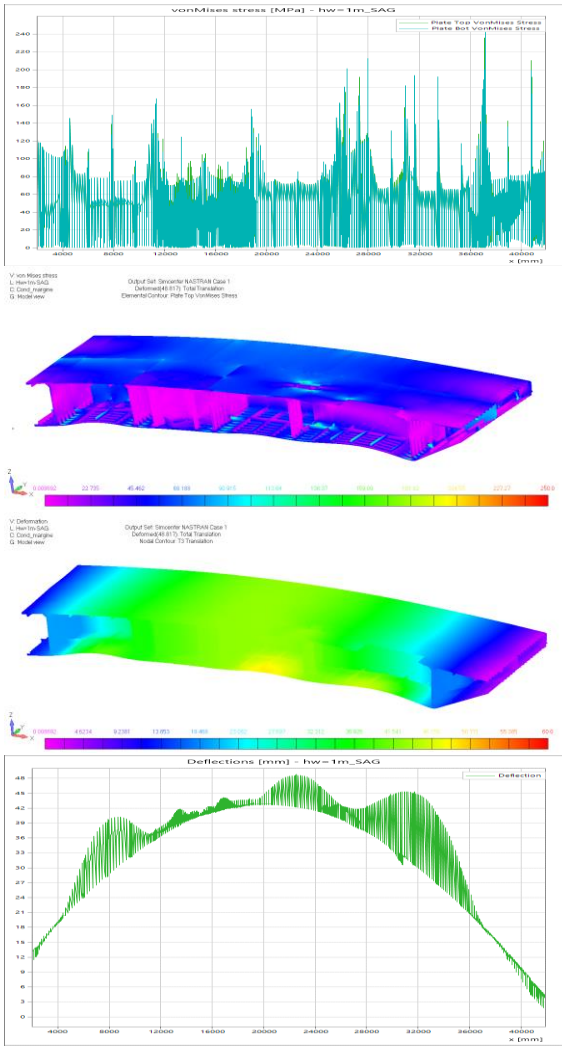
a.

b.

c.

d.





e.

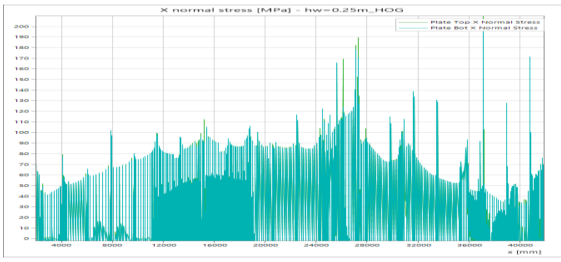


Fig. 19. Wave height case Hw=0.25m - HOG

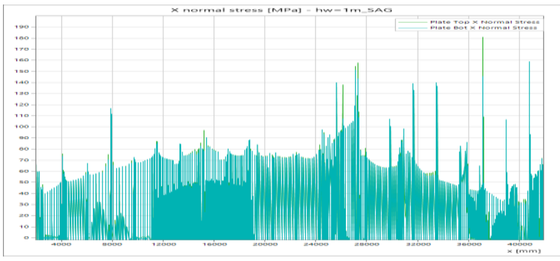


Fig. 20. Wave height case Hw=1m - SAG

a.

b.

c.

d.

e.

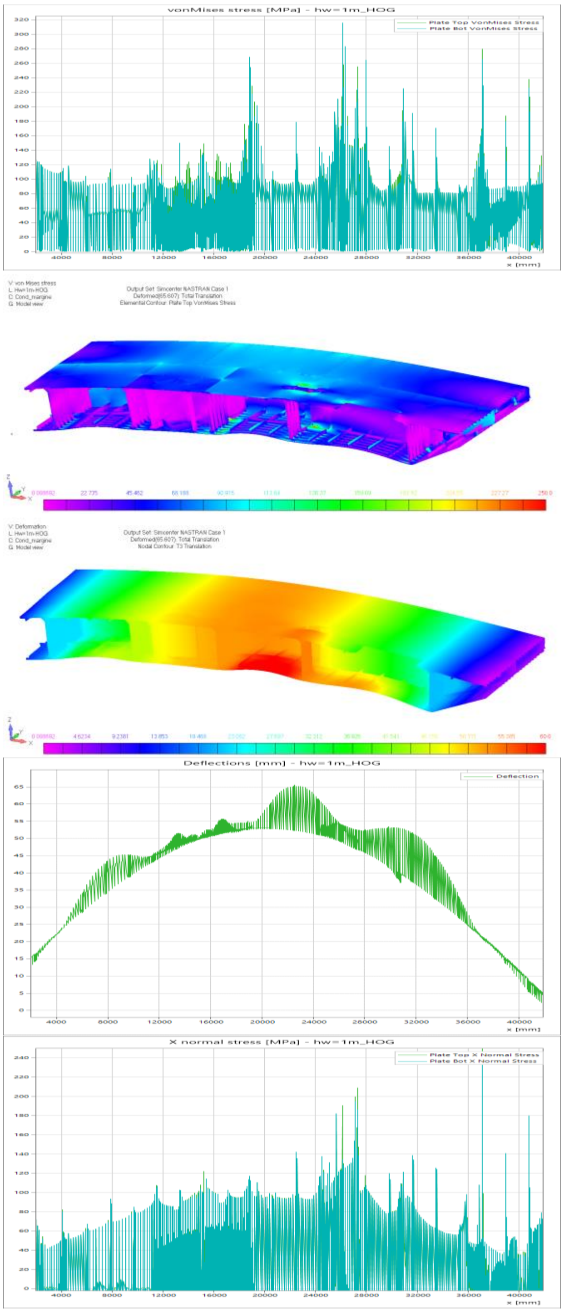


Fig. 21. Wave height case Hw=1m - HOG

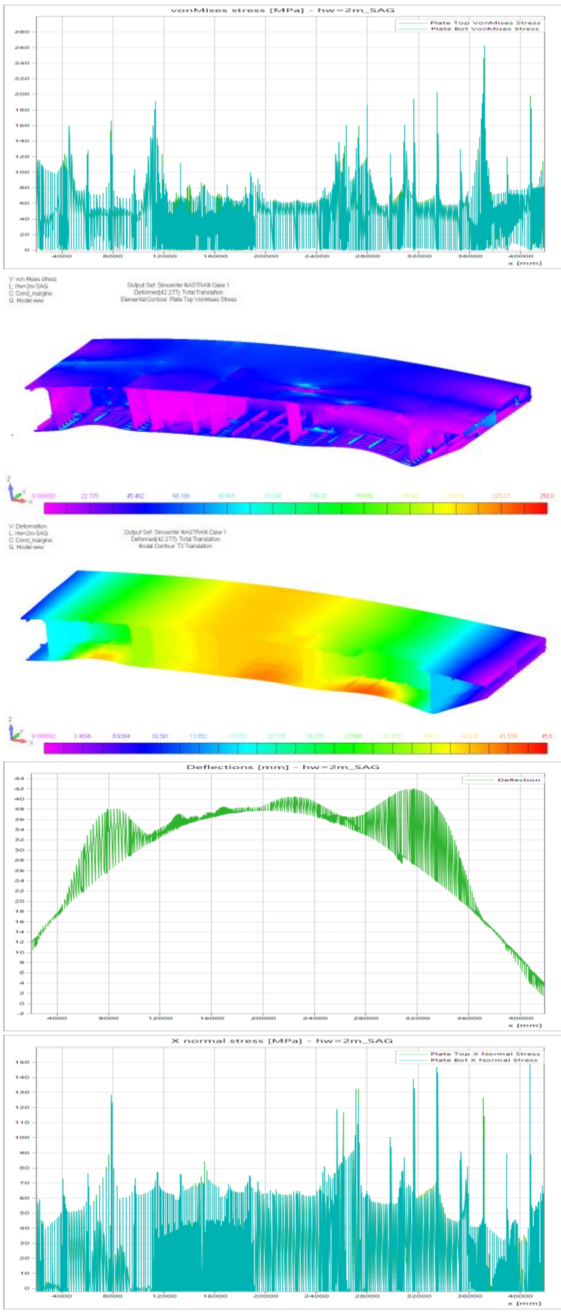


Fig. 22. Wave height case Hw=2m SAG

a.

b.

c.

d.

e.

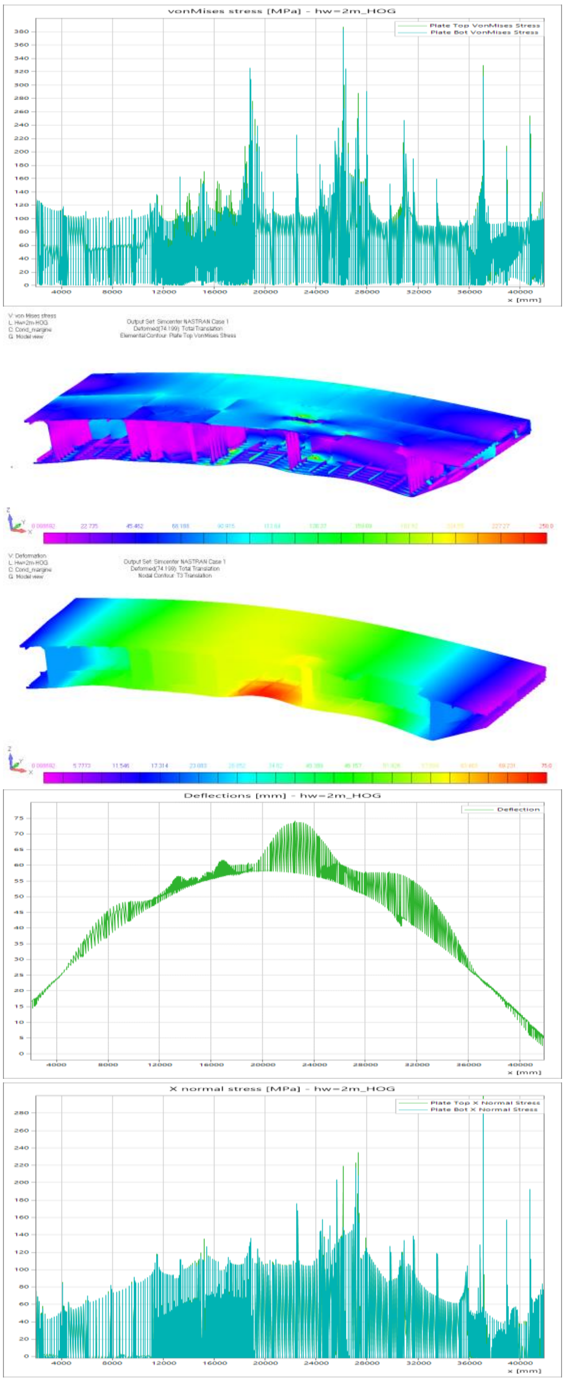


Fig. 23. Wave height case Hw=2m - HOG

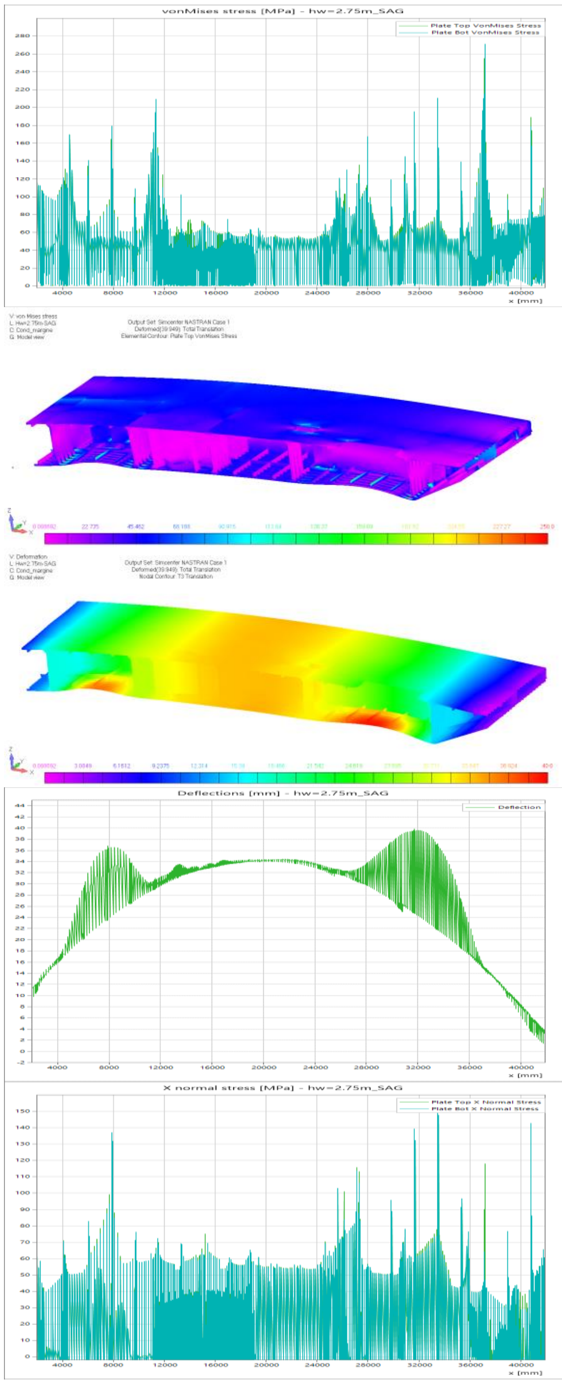
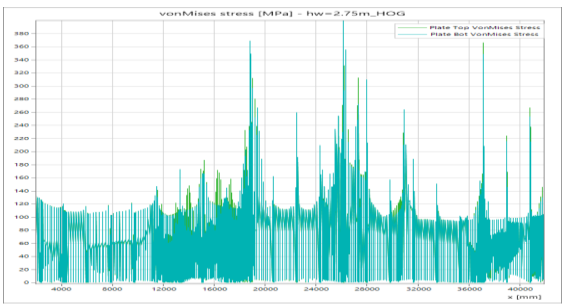


Fig. 24. Wave height case Hw=2.75 SAG

a.



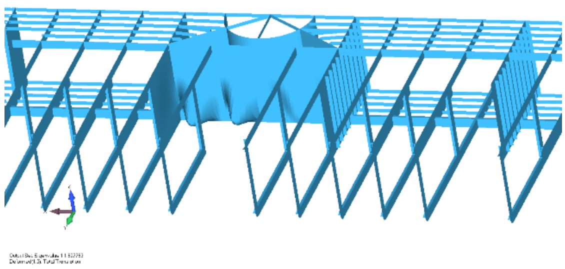


Fig. 26. Buckling - Wave height case Hw=0m

b.

c.

d.

e.

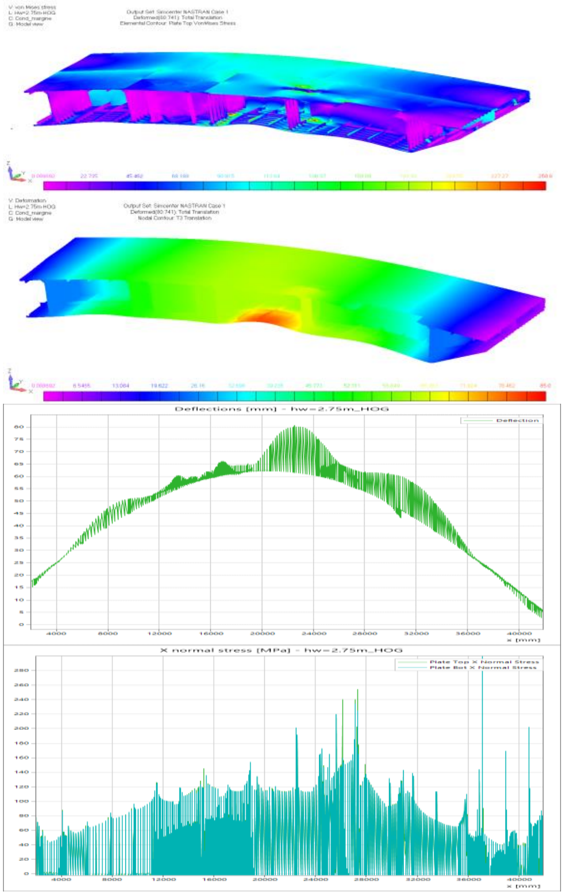


Fig. 25. Wave height case Hw=2.75m - HOG

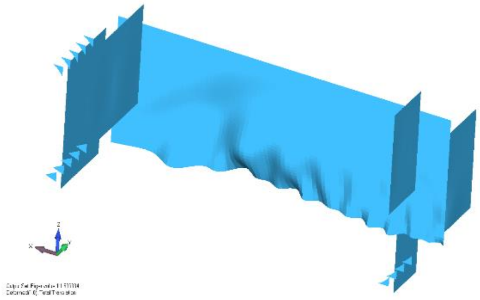


Fig. 27. Buckling - Wave height case Hw=1m-HOG

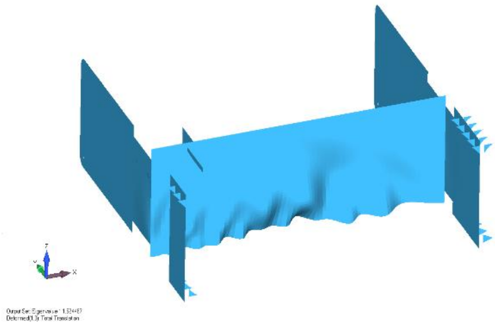


Fig. 28. Fig. 27. Buckling - Wave height case Hw=1m-SAG

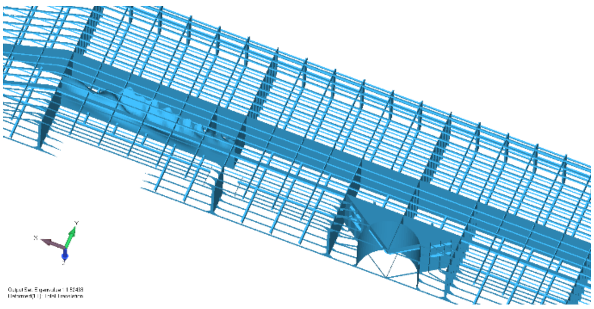


Fig. 29. Buckling - Wave height case Hw=2m-HOG

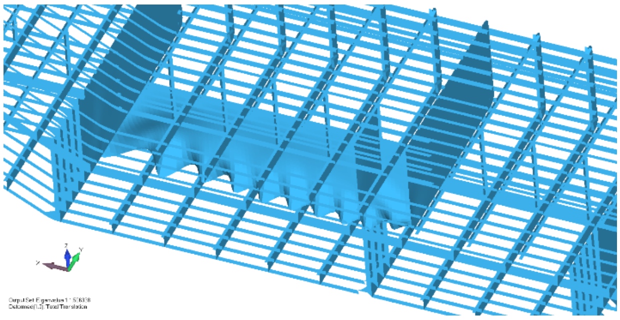


Fig. 30. Buckling - Wave height case Hw=2m-SAG

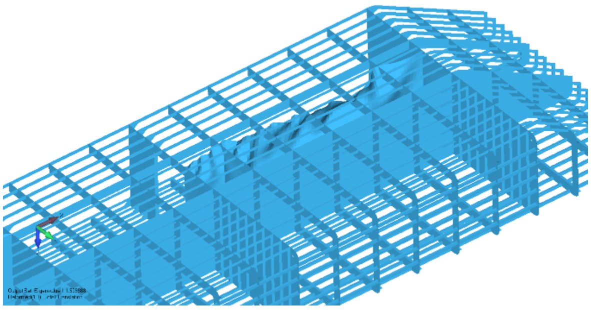


Fig. 31. Buckling - Wave height case Hw=2.75m-HOG

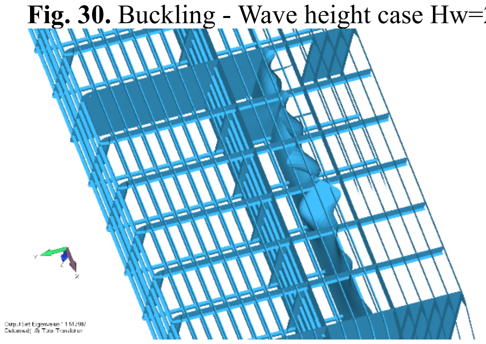


Fig. 32. Buckling - Wave height case Hw=2.75m-SAG

3. Conclusions

In conclusion the strength analyses result of the crane hull without the forces involved from the crane cases scenarios, by the theoretical and numerical approach for head equivalent design wave loads, are synthesized in table 4, some of these cases presented in figures 17 to 32, gives us the next conclusions:

- A 3D-FEM model, full extended overall length, in one side for the crane hull has been developed (figures 1 – 4, 9 - 11), by Femap/NX Nastran [5], involving approximatively 9 million of degrees of freedom;
- The results for the head and follow wave scenarios, in which the von Misses stress has maximum values are presented in table 3, in which after the 1D beam calculations will show the worse scenario in the cases tested. The single limit that we can conclude from the results, is the one for the admissible yielding stress in with the sagging wave do not had limits, but for the hogging wave type, the maximum height is of 0.5m. Acc to these conclusions, some brackets and flat bars will be added in the hot spot areas.
- The result for the head and follow wave scenarios, in which the deformation of the 3D-FEM model has maximum values will also be tested for the admissible deflection, and then the case of 3D model will be mentioned.

Further studies for this floating crane model will be in accordance to obtain the limits criteria in cases of follow waves, because the hull is not symmetrical aft with fore by the midship transverse section. Also, other studies will be made for obtaining the limit criteria for oblique static design waves and dynamic irregular waves.

To obtain the admissible values (ultimate strength, admissible vertical bending moment and vertical shear force, vertical deflection and admissible displacement) an equivalent beam model will be made.

References:

[1] Eyres D J, Bruce G J, 2012, *Ship Construction* (Boston: Butterworth - Heinemann)

[2] Biran A, 2003, *Ship Hydrostatics and Stability* (Oxford: Butterworth - Heinemann)

[3] Hughes O F, 2000, *Ship Structural Design. A Rationally Based, Computer-Aided Optimization Approach* (New Jersey: The Society of Naval Architects and Marine Engineering)

[4] **Burlacu, E.**, Domnișoru L., Hadăr A., Roșca I. C., Mocanu C. I., 2020, *Contribuții privind analiza structurii unui doc plutitor la solicitări extreme*, Editura Fundației Universitare ”Dunărea de Jos”, Galați, ISBN 978-973-627-630-9

[5] FNN 2021 *Femap/NX Nastran Users' Manual* (Siemens PLM Software Inc.)

[6] ISSC, 2018, *International Ships and Offshore Structures Congress. Environment. Loads. Quasi-static response. Ultimate strength. Dynamic response. Design principles and criteria*, IOS Press, Amsterdam.

[7] Yao T., Fujikubo M, 2016, *Buckling and ultimate strength of ship and ship-like floating structures*, Butterworth – Heinemann, Oxford.

[8] Domnișoru L., 2017, *Special chapters on ship structure analysis. Aplications.* Editura Fundației Universitare ”Dunărea de Jos”, Galați, ISBN 978-973-627-589-0