Zeszyty Naukowe Akademii Morskiej w Gdyni
Scientific Journal of Gdynia Maritime University

Nr 105/2018, 19–31
ISSN 2451-2486 (online)
ISSN 1644-1818 (printed)
DOI: 10.26408/105.02

Złożony/submitted: 19.04.2017
Zaakceptowany/accepted: 15.08.2017
Opublikowany/published: 29.09.2018

FINITE ELEMENT METHOD IN MODELING OF SHIP STRUCTURES PART II – PRACTICAL ANALYSIS EXAMPLE

METODA ELEMENTÓW SKOŃCZONYCH W MODELOWANIU KONSTRUKCJI OKRĘTOWYCH CZĘŚĆ II – PRZYKŁAD OBLICZEŃ PRAKTYCZNYCH

Do Van Doan[1], Adam Szeleziński[1a*], Lech Murawski[1b], Adam Muc[2]
[1] Uniwersytet Morski w Gdyni, Morska 81-87, 81–225 Gdynia, Wydział Mechaniczny, Katedra Podstaw Techniki, e-mail: a.szelezinski@wm.am.gdynia.pl
[a] ORCID 0000-0003-2842-0683
[b] ORCID 0000-0003-0089-5492
[2] Uniwersytet Morski w Gdyni, Morska 81-87, 81–225 Gdynia, Wydział Elektryczny, Katedra Automatyki Okrętowej, ORCID 0000-0002-9495-087X
* Adres do korespondencji/Corresponding author

Abstract: Part I of this paper presents basic knowledge about Finite Element Method including the modeling method of ship structures. Numerical modeling methods were also shortly described. A ship hull and the upper works are typical thin-walled structures. Modeling method of plates (typical 2-D elements) with stiffeners (1-D elements) is presented in details. In the part II of the article, practical example of Ro-Ro ship's deck analyses was performed using Patran-Nastran software (MSC Software). The most common and dangerous risks and errors occurring in the process of ship structure modeling were discussed.

Keywords: numerical methods, Finite Element Method, ship structure strength, ship structure vibrations, modeling methods.

Streszczenie: W pierwszej części pracy przedstawiono podstawową wiedzę o Metodzie Elementów Skończonych ze szczególnym uwzględnieniem metod modelowania konstrukcji okrętowych. Metody modelowania numerycznego zostały również krótko omówione. Kadłub statku oraz jego nadbudówka to typowa konstrukcja cienkościenna. Dokładnie przedstawiono metodę modelowania płyt (typowe elementy 2-D) wraz z usztywnieniami (elementy 1-D). W części drugiej artykułu zaprezentowano szereg analiz przeprowadzonych na przykładzie pokładu statku typu ro-ro, z wykorzystaniem oprogramowania Patran-Nastran (MSC *Software*). Omówiono najpopularniejsze i najgroźniejsze błędy występujące podczas modelowania konstrukcji okrętowych.

Słowa kluczowe: metody numeryczne, Metoda Elementów Skończonych, wytrzymałość konstrukcji okrętowych, drgania konstrukcji okrętowych, metody modelowania.

1. INTRODUCTION

Safety of sea navigation requires ship structure systems to be free from excessive stress and vibration levels [Murawski 2003]. Two main marine systems can be distinguished: a ship hull (with a superstructure and a main engine body) and a power transmission system (a crankshaft, a shaft line, a propeller). The operation of ships occurs often in extremely bad weather conditions. Marine structures are operating in more aggressive conditions than land-based constructions and the aerospace structures. Proper assessment of the ship technical condition in the critical environmental conditions is crucial from the perspective of safety of maritime navigation. Limitation of maritime disasters is of great economic importance and, more importantly, will reduce the environmental and human costs. Especially the propulsion system of the ship should be subject to significant assessment, because like in aviation, inoperative propulsion results in a very high probability of disaster in a stormy weather conditions.

International law states that each ship navigating in the sea has to fulfill regulations of one of the classification institutions. It is more vital that classification societies rules are based on wide knowledge collected over hundreds of years. Classification society rules are based on simplified, empirical equations. But not all problems can be solved using empirical rules or even differential equations [Murawski 2005]. Most problems with ship vibrations have to be analyzed with numerical calculations verified by measurements [Murawski and Charchalis 2014].

The Finite Element Method (FEM) is one of the best available approaches to the numerical analysis of continuum. It is currently the most popular technique and numerous commercial software packages are now available for its implementation. All classification societies present alternatives to their calculation methods, especially Finite Element Method. These more detailed analyses are usually more expensive, however, optimization is possible. The FEM consists in modeling the physical structure with a discrete mathematical model.

2. STATIC AND DYNAMIC ANALYSES OF SHIP DECK

The software to teach designing complex and large-scale thin-walled structures is shown by the authors. The bundle of MSC software, named Patran-Nastran, was presented. The software is based on Finite Element Method. The presentations are focused on basic analyses: linear static strength analysis, normal mode (eigenvectors) analysis and frequency response analysis (forced vibrations calculations). An example of RO-RO ship's deck analysis was presented. The deck was loaded by the wheel of a truck. Short description of modeling process, as well as results analyses, are described. In the authors' opinion, knowledge of that very specialized software is important not only for an engineer who is calculating. All mechanical engineers should have basic knowledge about calculations process because they will be recipients of the calculations performed during structure designing. Thin-walled structures are very useful for presenting software possibilities because during the structure modeling we have to choose types of FEM elements. Stiffeners may be modeled by 1-D elements (with offset) and the plates should be modeled by 2D elements. Differences between beam and rod 1D elements as well as shell, plate and membrane 2D elements should be stressed out during presentation.

The sketch of fragment of the ship structure (thin-walled) is presented in Fig. 1. The fragment consists of ship deck connected with ship side by beam knee and global and local stiffeners (beams). Our design focuses on the static and dynamic characteristics of deck section marked by the bold line. The analyzed model is limited to that section but the influence of other part of ship structure has to be taken into account in terms of special boundary conditions.

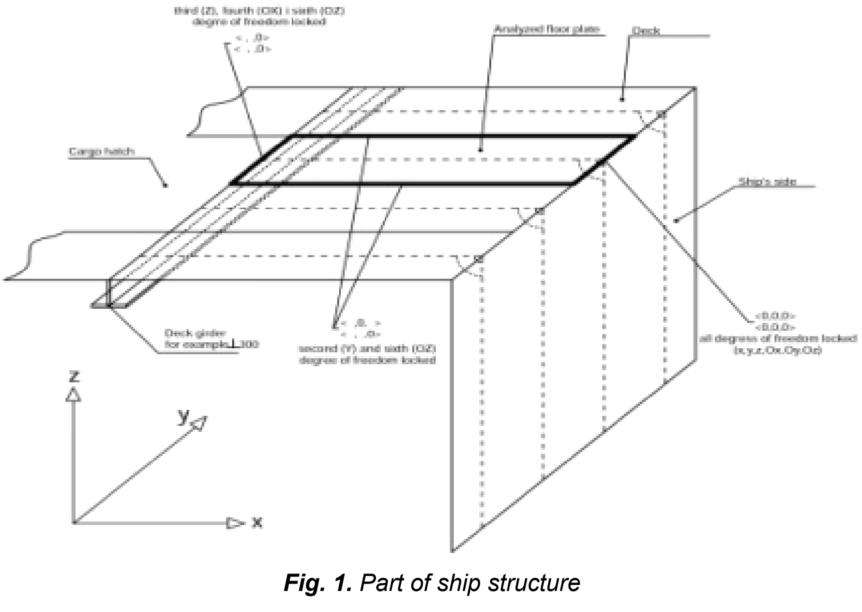

Fig. 1. *Part of ship structure*

Rys. 1. Fragment konstrukcji okrętowej

The drawing of analyzed deck section is presented in Fig. 2. During the analysis, the montage (geometrical) imperfection is taken into account. The imperfection is three-dimensional and it is determined by three dimensions: $\boldsymbol{h_{def}}$, $\boldsymbol{L_{def}}$, $\boldsymbol{S_{def}}$. The deck is loaded by the pressure coming from a truck wheel.

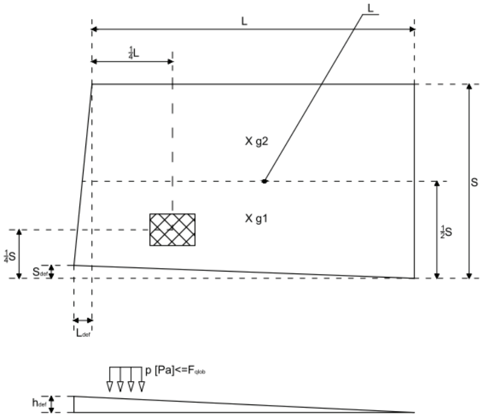

***Fig. 2.** Analyzed deck section*

Rys. 2. Analizowana płyta pokładu

The ship thin-walled structure will be analyzed by Patran-Nastran software. Patran is a pre- and post-processor, therefore, most job elements will be performed in that program. We cannot forget that FEM calculations are performed only by Nastran software. Patran is a helpful tool in case of model preparing and results visualization. Seven main tabs are distinguished in the program: ***Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results***.

An example of modeled ship hull is presented in Fig. 3. In the figure all tabs are visible.

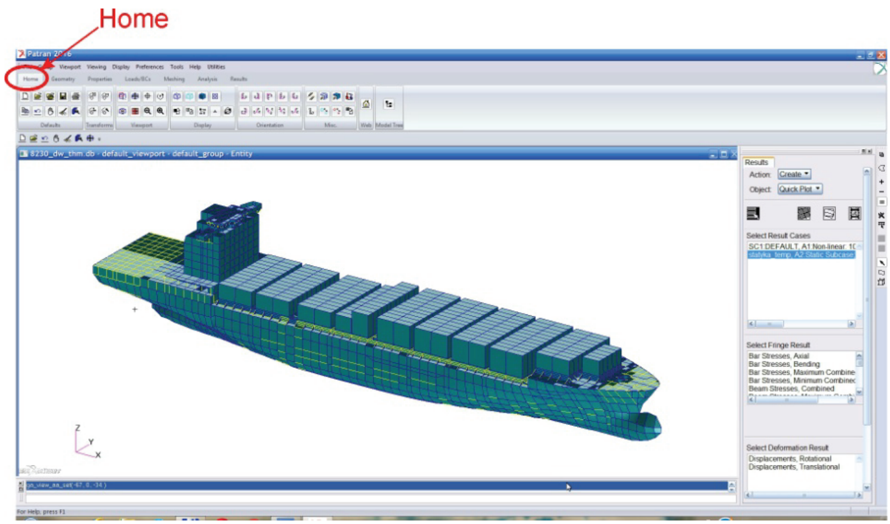

***Fig. 3.** General view of Patran software with main tabs*

Rys. 3. Ogólny widok programu Patran z głównymi zakładkami

The ***home*** tab (opened in Fig. 3) is destined mainly for visualization changes. That tab does not have influence on model properties. During FEM analysis, first of all, the shape of calculated structure has to be determined. The ***Geometry*** tab is designed for that step of modeling process.

Modeled shape of analyzed deck section is presented in Fig. 4.

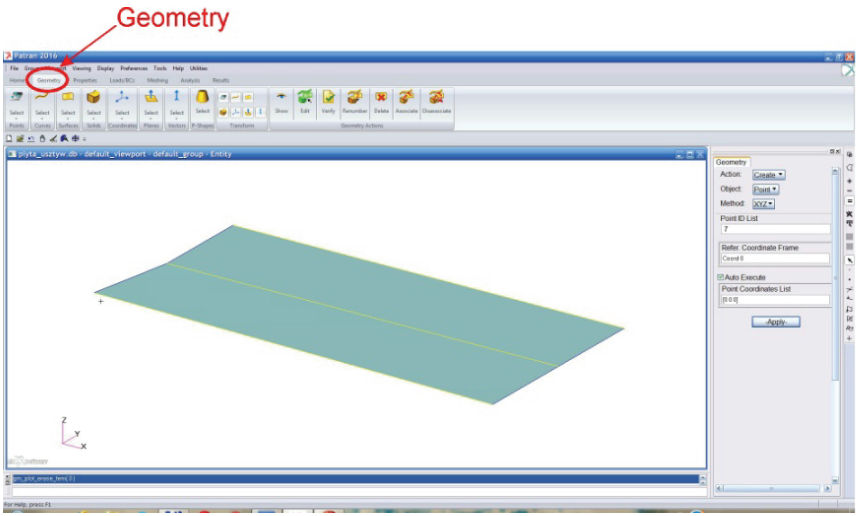

Fig. 4. *The model shape definiction – Geometry tab*

Rys. 4. Zdefiniowany kształt modelu – zakładka *Geometry*

In the next step, the material properties have to be defined in the ***Properties*** tab. Definition of steel material properties is presented in Fig. 5. Obviously, more than one material can be determined. In our case (linear static and dynamic calculations) only Young modulus, Poisson ratio and material density must be defined. The other properties are calculated by the software or omitted (e.g. thermal expansion coefficient is important only for thermal analyses).

In the same tab (***Properties***) the properties of the finite elements can be defined. In our case, the plate thicknesses and beam properties (e.g. cross section, moments of inertia) should be determined. Previously defined materials must be assigned to each structures element. In Fig. 6 an example of beam properties definition is presented. During beam's (1-D FE) properties definition, ***offset*** value of each beam ends should be determined. Without those values, the main inertia axis of beam is placed in the same plane as plate elements (2-D FE). That kind of action means poor modeling of connection between stiffener (modeling of beam) and the deck plate - the beam is welded inside the plate.

The stiffener bad modeling (without offset) and good modeling (with offset) method are presented in Fig. 7.

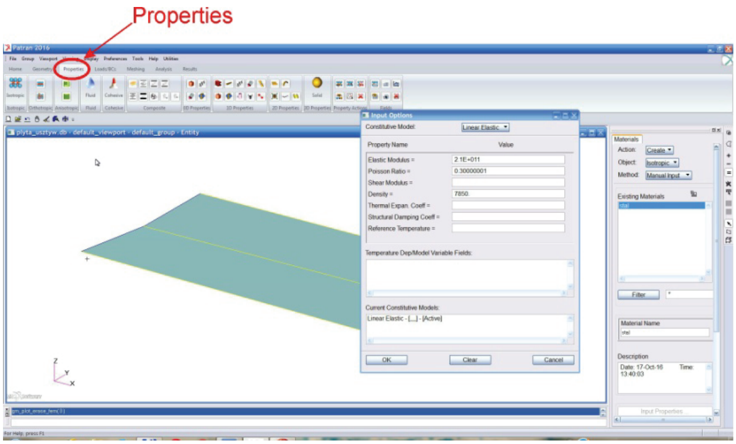

***Fig. 5.** Definition of material properties – Properties tab*

Rys. 5. Definiowanie własności materiałowych – zakładka *Properties*

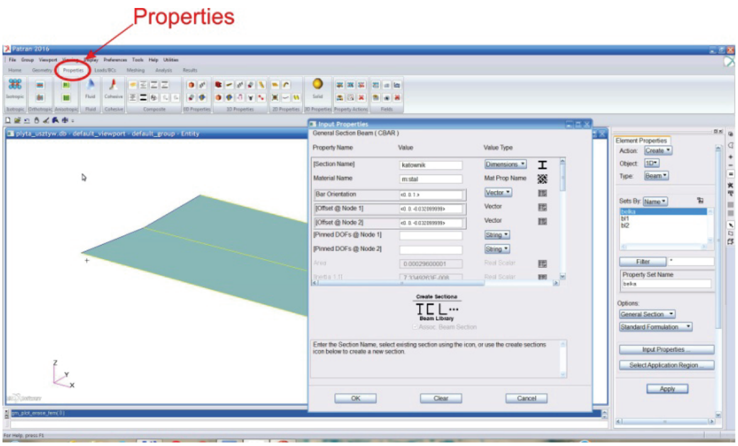

***Fig. 6.** Definition of FE elements properties – Properties tab*

Rys. 6. Definiowanie właściwości elementów skończonych – zakładka *Properties*

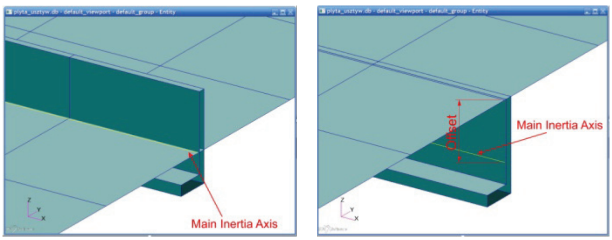

***Fig. 7.** The beam modeled without (left side – wrong) and with offset (right side – good)*

Rys. 7. Belka zamodelowana bez offsetu (lewa strona – niepoprawna) oraz z offsetem (prawa strona – poprawna)

Properties, loads and boundary conditions can be assigned to geometry or directly to finite elements. In the data for Nastran (calculating software) only finite elements exist; the geometrical definition (structure shape) is only for our modeling facilitation. All values assigning to geometry are going automatically to finite elements during their definitions. In our example, the properties are assigned to geometry but the loads (wheel pressure in separate place of deck) and boundary conditions will be assigned to finite elements. Therefore, firstly, the finite elements should be defined using the ***Meshing*** tab (presented in Fig. 8). The user must remember that finite elements should be defined on each surface and line (where stiffener is modeled) separately. After FE definition, the ***equivalence*** function should be activated. This function checks whole model and finds doubled (two nodes located in the same place but belonging to different FE e.g. beam and plate) nodes of elements and after that deleted unnecessary nodes. Without that function, the deck plate and stiffener will be not connected (welded).

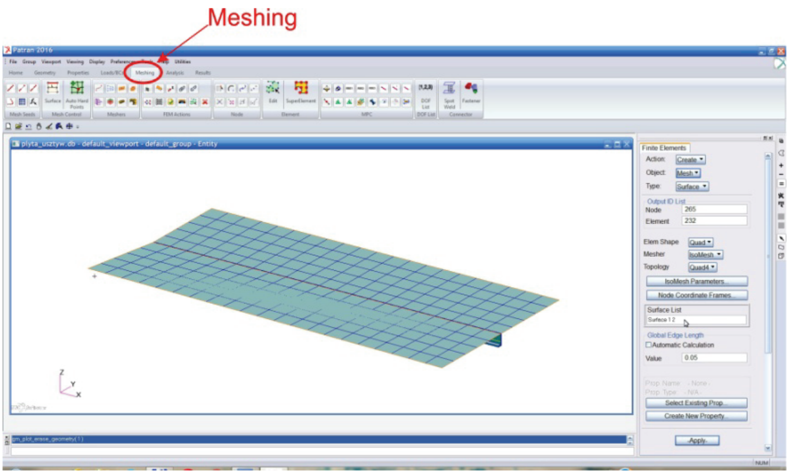

Fig. 8. *The meshing procedure – defining the finite elements*

Rys. 8. Procedura *meshing* – definiowanie elementów skończonych

If finite elements are defined, the boundary conditions can be determined with BCs function. The ***Loads/BCs*** tab is presented in Fig. 9. The boundary conditions model interaction between analyzed deck section and other ship structure. For instance, connection between deck and ship side (stiffened by a beam knee) is very strong, therefore, all degrees of freedom are locked by enforcing all displacements (three translations and three rotations) to equal zero. While the influence of other parts of deck is much weaker – the only displacement along "y" axis should be locked.

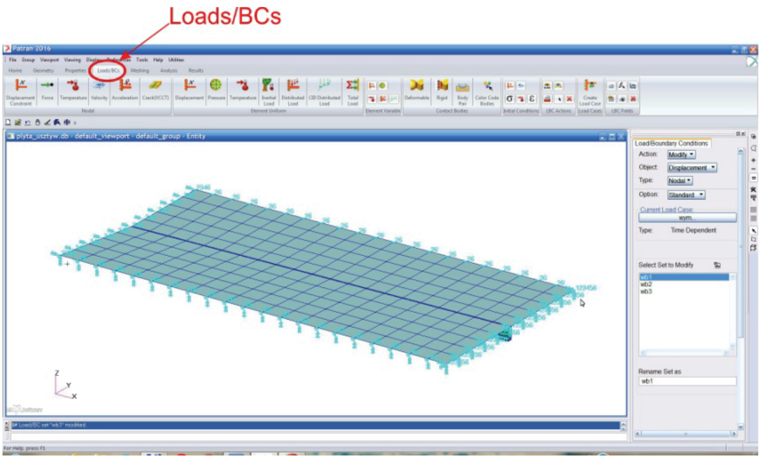

Fig. 9. *Defining the boundary conditions*

Rys. 9. Definiowanie warunków brzegowych

In the same tab (***Loads/BCs***), the loads in our example coming from a track wheel, can be modeled. Loads may be modeled by force, pressure, temperature (thermal expansion coefficients have to be modeled in the material properties), inertial load (gravity), crack and even velocity or acceleration (for dynamic analyses). In analyzed example, the track wheel can be modeled by force or by pressure. Pressure is a better method because real wheel is connected to deck not only in one point. What is more, force acts only at one node of FE and therefore, locally strain-stress distribution reaches very high, unrealistic level.

Modeled load of track wheel is presented in Fig. 10.

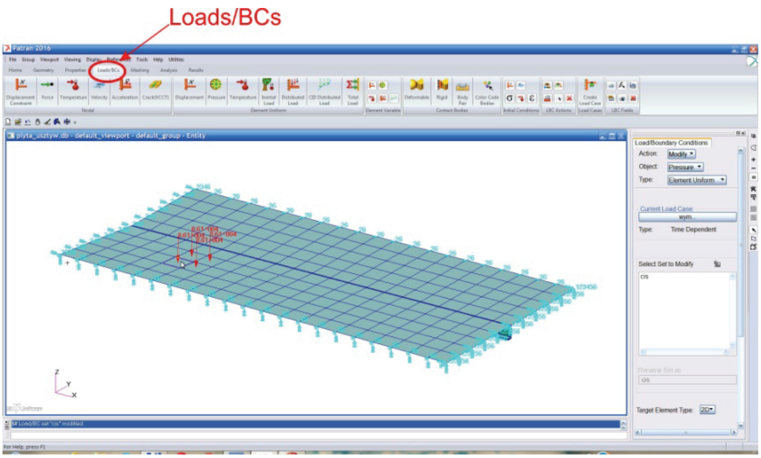

Fig. 10. *Defining the deck load by pressure*

Rys. 10. Definiowanie obciążenia pokładu ciśnieniem

After defining finite elements, all properties, loads and boundary conditions, the model of ship deck is finished. Now, the data for calculating program (Nastran) should be prepared. Data preparation process can be done in the ***Analysis*** tab. Several different kinds of calculations can be performed. In the presented example three different analyses were performed: static, normal modes and frequency response. Calculations should be prepared by separate software Nastran by activating it with prepared data. The same tab (***Analysis***) is used for loading calculation results into postprocessor - Patran. There is a possibility to analyze the results by reading all the numbers but it is highly inefficient. Nearly always, the results are analyzed with the Patran software.

The results can be analyzed using the ***Results*** tab. Displacements and Von-Misses stresses distribution, coming from first, ***static analysis***, are presented in Fig. 11 and 12. It is obvious that maximal displacements are located around load. Maximal stress level is located in the similar area but modeled montage imperfection is also a source of higher stress level.

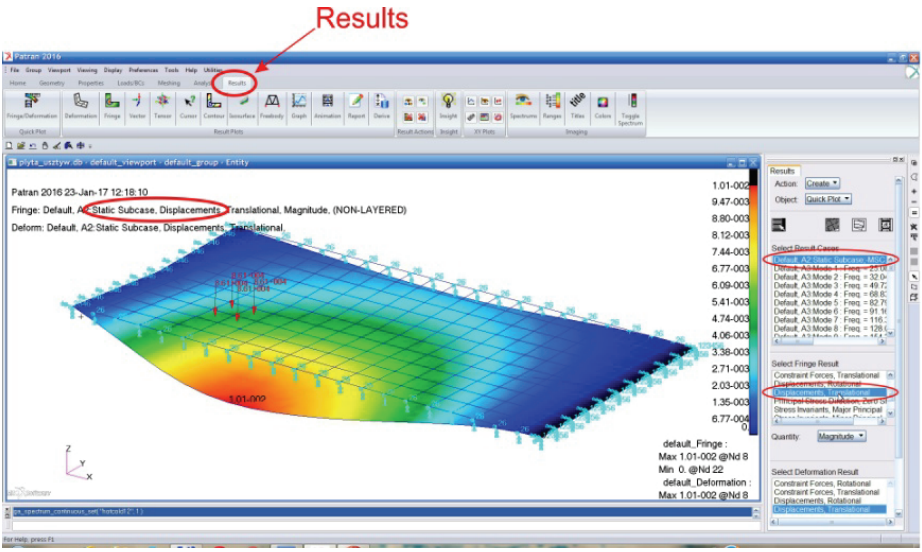

Fig. 11. *Static deformation of ship deck*

Rys. 11. Statyczna deformacja pokładu statku

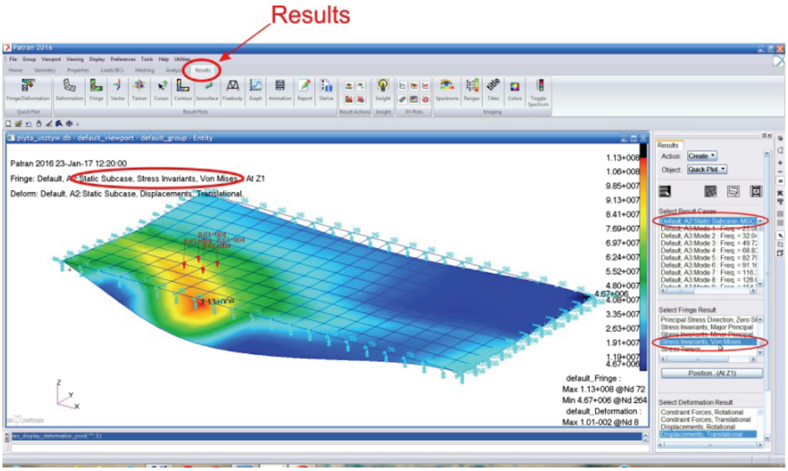

Fig. 12. *Static Von-Misses stresses distribution of ship deck*

Rys. 12. Rozkład statycznych naprężeń Von-Misses pokładu statku

Not only static behaviour of ship structure should be analyzed. Usually, ship vibrations are also very dangerous. First of all, a designer should check frequencies and modes of natural vibrations. Calculation of ***Natural modes*** (eigenvectors) should be also performed to check model consistency. But the first target of that kind of analysis is resonance hazard check. Usually, first ten natural modes are sufficient but for natural vibrations of ship hull at least 30 natural modes should be determined. First (f_1 = 25.1 Hz) and second (f_1 = 32.0 Hz) natural mode of ship deck is presented in Fig. 13 and 14.

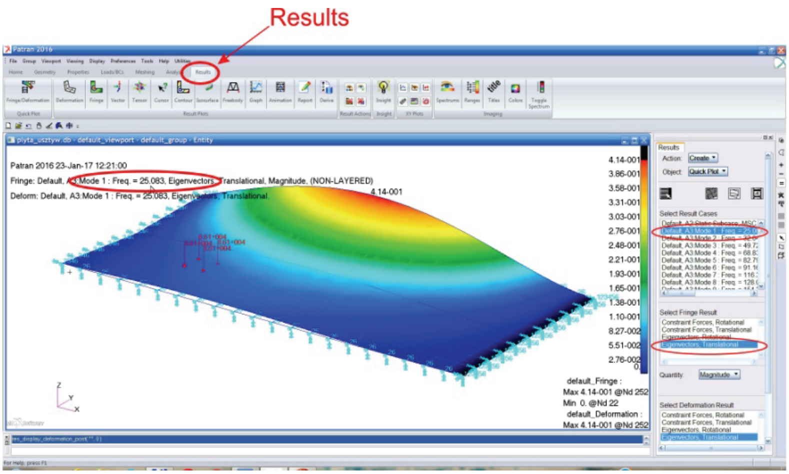

Fig. 13. *First natural mode of ship deck*

Rys. 13. Pierwsza postać drgań własnych pokładu statku

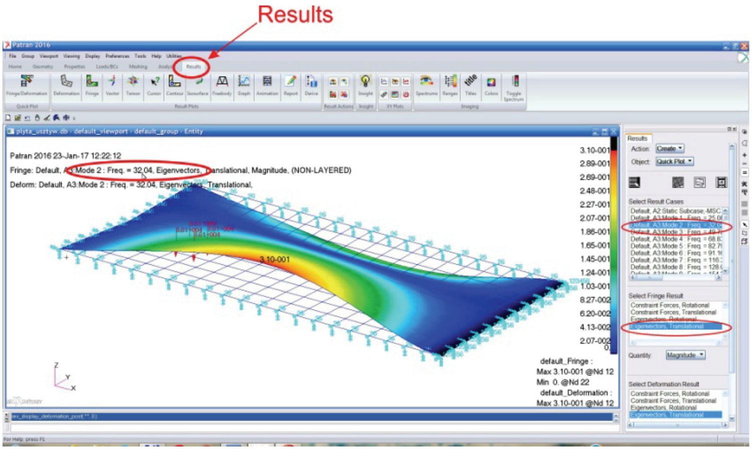

Fig. 14. *Second natural mode of ship deck*

Rys. 14. Druga postać drgań własnych pokładu statku

After natural modes calculations, the forced vibrations analysis should be performed. There are several methods of forced vibrations calculations. Most often, the ***frequency response*** (with superposition method) analysis are the most suitable for steady-state vibrations. The frequency of dynamic load (in our case 10% of static load) is changed step by step. Therefore, hundreds of calculations can be performed. After that, for exact node (or element) the resonance curve (vibration amplitude in function of loading frequency) can be drawn.

The resonance curves for displacements on the deck edge (close to loading) and middle point of the deck are presented in Fig. 15.

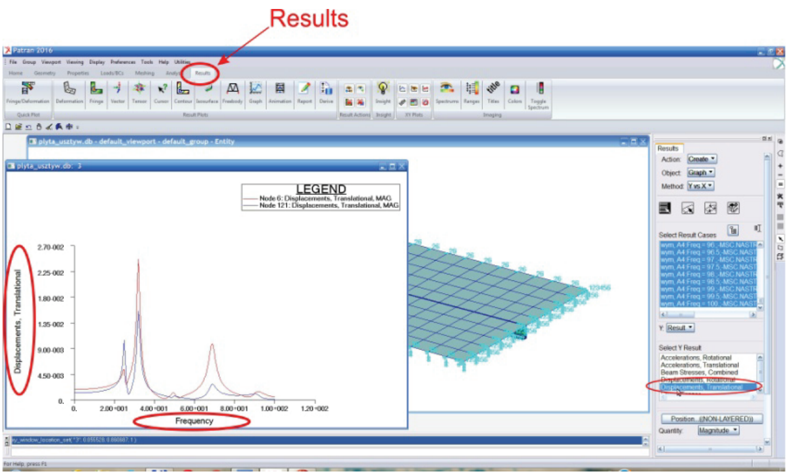

Fig. 15. *Resonance curves determined for deck displacements*

Rys. 15. Krzywa rezonansowa wyznaczona dla odkształceń pokładu

The same curves for Von-Misses stresses are presented in Fig. 16. The resonance amplitudes may be observed for frequencies equal to natural frequencies. First three-four modes are enforced by given load.

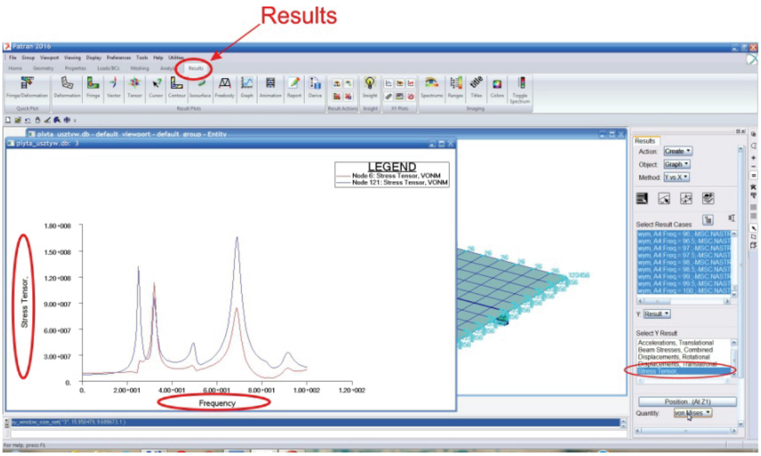

Fig. 16. *Resonance curves determined for deck displacements*

Rys. 16. Krzywa rezonansowa wyznaczona dla odkształceń pokładu

3. CONCLUSIONS

The engineers should remember that FEM analyses work only as the modeling method of the real world. Each model has got limitations. If we use linear strain-stress theory to model vibrations of the machine placed on rubber pads at hot temperature (strong nonlinear material), we receive proper results in terms of FEM theory (computers are usually unerring) but these results are completely wrong from the practical point of view. Basic knowledge about Finite Element Method is crucial for modern engineers. The most common mistakes made by users are as follows:

- wrong type of elements, e.g. shell elements are used where solid elements are needed or membrane elements are used rather than plate elements;
- distorted elements in comparison to real domain shapes;
- support is insufficient to prevent all rigid-body motions;
- unit selection is inconsistent and inaccurate;
- too large stiffness differences leading to numerical difficulties;
- connections between 1-D and 2-D elements (and also 2-D and 3-D) should be modeled carefully (e.g. with offset values).

REFERENCES

Beer, G., Smith, I., Duenser, C., 2008, *The Boundary Element Method with Programming*, Springer-Verlag, Wien.

Causon, D.M., Mingham, C.G., 2010, *Introductory Finite Difference Methods for PDEs*, Publishing ApS.

Cook, R.D., Malkus, D.S., Plesha, M.E., Witt, R.J., 2002, *Concepts and Applications of Finite Element Analysis*, 4^{th} ed., Wiley, New York.

Finlayson, B.A., 1972, *The Method of Weighted Residuals and Variational Principles*, Academic Press, New York.

Murawski, L., 2003, *Static and Dynamic Analyses of Marine Propulsion Systems,* Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa.

Murawski, L., 2005, *Shaft Line Alignment Analysis Taking Ship Construction Flexibility and Deformations into Consideration*, Marine Structures, vol. 18, s. 62–84.

Murawski, L., Charchalis, A., 2014, *Simplified Method of Torsional Vibration Calculation of Marine Power Transmission System*, Marine Structures, vol. 39, s. 335–349.

Oden, J.T., Ripperger, E.A., 1981, *Mechanics of Elastic Structures*, 2^{nd} ed., McGraw-Hill, New York.

Reddy, J.N., 1993, *Introduction to the Finite Element Method*, McGraw-Hill, Inc.

Richardson, L.F, 1910, *The Approximate Arithmetical Solution by Finite Differences of Physical Problems*, Philosophical Transactions of the Royal Society.

Zienkiewicz, O.C., Taylor, R.L., 2005, *The Finite Element Method, Vol. 1: The Basis*, 6^{th} ed., Butterworth-Heinemann, Oxford.