FINITE ELEMENT METHOD
IN MODELING OF SHIP STRUCTURES
PART I – THEORETICAL BACKGROUND

METODA ELEMENTÓW SKOŃCZONYCH
W MODELOWANIU KONSTRUKCJI OKRĘTOWYCH
CZĘŚĆ I – PODSTAWY TEORETYCZNE

Do Van Doan[1], Adam Szeleziński[1*], Lech Murawski[1], Adam Muc[2]

[1] Akademia Morska w Gdyni, Morska 81-87, 81–225 Gdynia, Wydział Mechaniczny,
 Katedra Podstaw Techniki, e-mail: a.szelezinski@wm.am.gdynia.pl
[2] Akademia Morska w Gdyni, Wydział Elektryczny, Gdynia
* Corresponding author/Adres do korespondencji

Abstract: The 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 an upper works is typical thin-wallded structure. Modeling method of plates (typical 2-D elements) with stiffeners (1-D elements) is presented in details. In the part II of the article the practical example of Ro-Ro ship's deck analyses was performed with 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 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 2 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 sea going ship has to fulfill regulations of one of the classification institutions. More important is that classification societies rules are based on wide knowledge collected over hundreds years. Classification society's rules are based on simplified, empirical equations. But not all problems can be solved by empirical rules or even differential equations [Murawski 2005]. Most problems with ship vibrations have to be analysed by numerical calculations verified by measurements [Murawski and Charchalis 2014].

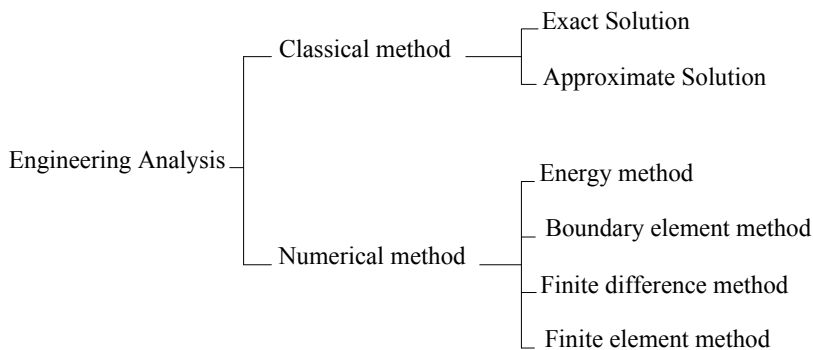The main types of problem solving methods for engineering are presented in Fig. 1.

Fig. 1. *The main types of problem solving methods for engineering*
Rys. 1. Podstawowe metody rozwiązywania problemów inżynierskich

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. NUMERICAL MODELING METHODS

Engineers and scientists, while researching, designing or manufacturing devices and systems have to model complex natural and technical phenomena. It is important to do them in physical structure and then convert them into simple mathematical models. Modeling or idealization of technical issues will make it easier to calculate, test and predict working conditions of the equipment. In order to do so, engineers and scientists must be able to describe and analyze objects and devices to predict their behavior which will allow them to see if they are consistent with the behaviors that the engineers, scientists desire. A mathematical model describes a system in a form that uses appropriate mathematical and language concepts to facilitate the process of solving technical and natural science problems. A model may help to explain a system, study the effects of different components, and make predictions about behavior.

Mathematical models are usually composed of relationships and variables. Relationships can be described by operators, such as algebraic operators, functions, differential operators etc. Variables are abstractions of system parameters of interest that can be quantified. There are currently several types of mathematical models that are widely used as follows: linear or nonlinear, static or dynamic, explicit or implicit, discrete or continuous, deterministic or probabilistic (stochastic), deductive, inductive, or floating.

To perform an engineering analysis, an engineer must determine such information as structural loads, geometry, support conditions, and materials properties. The results of such the analysis typically include support reactions, stresses and displacements. This information is then compared to criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior. There are two approaches to the analysis: classical method and numerical method but practically classical methods are limited to very simple problems. Marine structures are mostly modeled by numerical models, solved by numerical computations.

Numerical computation, also called numerical analysis, is used for the problems that do not have closed-form solutions. In these cases we can only obtain numerical solutions. While the derivation of the governing equations for most practical problems is not too difficult, then solving it by exact analytical methods, in general case, is not feasible. In that case, the approximate methods of analysis provide alternative tools to solve the problem. Since the numerical computation is based on the approximate methods, it has a development history as long as that of the numerical methods.

A brief history of numerical methods is presented in Fig. 2 [Zienkiewicz and Taylor 2005]. Numerical methods consist of three main groups: the finite difference methods [Causon and Mingham 2010], the direct methods, and the variational methods. All these methods can be used to develop the FE model in the FEM. Among these methods, the finite difference method and variational methods such as the Rayleigh-Ritz and Galerkin are most often used in the literature. In the finite difference method [Richardson 1910], to solve the Partial Differential Equations (PDE's) of the problem, the derivatives of the unknown function are replaced by the differential quotients. The unknown function is expanded into a Taylor series. These differential quotients involve the values of the function at the preselected points of the domain. The resulting system of algebraic equations can be solved, after imposing the boundary conditions [Beer, Smith and Duenser 2008], for the values of the function at these preselected points. A variational equation may be solved using the direct method of the calculus of variations. In the direct method of the calculus of variations, the dependent variables are expressed as a set of trial functions multiplying parameters. This reduces a steady state problem to an algebraic process and a transient problem to a set of ordinary differential equations [Reddy 1993; Zienkiewicz and Taylor 2005].
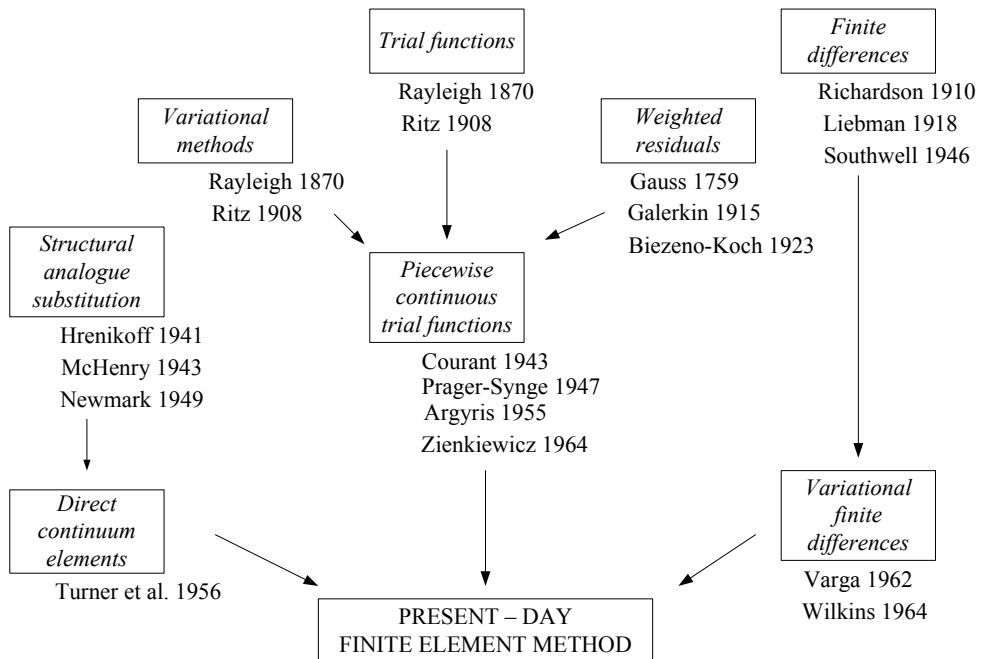

Fig. 2. *History of numerical methods development [Zienkiewicz and Taylor 2005]*
Rys. 2. Historia rozwoju metod numerycznych [Zienkiewicz i Taylor 2005]

From the practical point of view, the main disadvantage of the variational methods that prevents them from being competitive with traditional finite difference methods, is the difficulty encountered in selecting the approximation functions. Apart from the properties, the functions are required to satisfy, there exists no unique procedure for constructing them. The selection process becomes more difficult or even impossible when the domain is geometrically complex and/or the boundary conditions are complicated.

The FEM overcomes the disadvantage of the traditional variational methods by providing a systematic procedure for the derivation of the approximation functions over subregions of the domain. The method is endowed with three basic features that account for its superiority over other competing methods:

1. A geometrically complex domain of the problem is represented as a collection of geometrically simple subdomains, called Finite Elements (FE).

2. Over each FE, the approximation functions are derived using the basic idea that any continuous function can be represented by a linear combination of algebraic polynomials.

3. Algebraic relations among the undetermined coefficients (i.e., nodal values) are obtained by satisfying the governing equations, often in a weighted-integral sense, over each element.

Thus, the FEM can be viewed, in particular, as an element-wise application of the Rayleigh-Ritz or weighted-residual methods, in which, the approximation functions are often taken to be algebraic polynomials, and the undetermined parameters represent the values of the solution at a finite number of nodes on the boundary and in the interior of the element. The approximation functions are derived using concepts from interpolation theory and are therefore called interpolation functions. The degree of the interpolation functions depends on the number of nodes in the element and the order of the differential equation being solved.

3. FINITE ELEMENT METHOD

The Finite Element Method is a numerical technique that gives approximate solutions to differential equations that model problems arising in physics and engineering. As in simple finite difference schemes, the finite element method requires a problem defined in geometrical space (or domain), to be subdivided into a finite number of smaller regions (a mesh).

In finite difference methods in the past, the mesh consisted of rows and columns of orthogonal lines (in computational space – a requirement now handled through coordinate transformations and unstructured mesh generators); in finite elements, each subdivision is unique and need not be orthogonal. For example, triangles or quadrilaterals can be used in two dimensions, and tetrahedra or hexahedra in three dimensions. Over each finite element, the unknown variables

(e.g. temperature, velocity etc.) are approximated using known functions; these functions can be linear or higher-order polynomial expansions in terms of the geometrical locations (nodes) used to define the finite element shape. In contrast to finite difference procedures (conventional finite difference procedures, as opposed to the finite volume method, which is integrated), the governing equations in the finite element method are integrated over each finite element and the contributions summed ("assembled") over the entire problem domain. As a consequence of this procedure, a set of finite linear equations is obtained in terms of the set of unknown parameters over the elements. Solutions of these equations are achieved using linear algebra techniques. Eight steps of the FEM calculation can be distinguished.

Step 1: Discretize and Select the Element Types. Step 1 involves dividing the body into an equivalent system of finite elements with associated nodes and choosing the most appropriate element type to model most closely the actual physical behavior.

An example of triangle mesh of the physical domain is presented in Fig. 3. The total number of elements used and their variation in size and type within a given body are primarily matters of engineering judgment. The elements must be made small enough to give usable results and yet large enough to reduce computational effort. Small elements (and possibly higher-order elements) are generally desirable where the results are changing rapidly, such as where changes in geometry occur; large elements can be used where results are relatively constant.
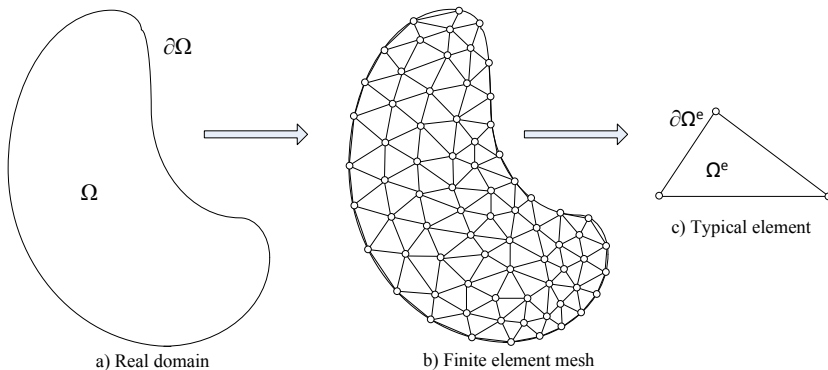

a) Real domain   b) Finite element mesh   c) Typical element

Fig. 3. *Triangular mesh of the domain of a 2D problem*
Rys. 3. Podział dwuwymiarowej domeny na trójkątne elementy skończone

The choice of elements used in a finite element analysis depends on the physical makeup of the body under actual loading conditions and on how close to the actual behavior the analyst wants the results to be. This choice concerning the appropriateness of one-, two-, or three-dimensional consideration is necessary. Moreover, the choice of the most appropriate element for a particular problem is one of the major tasks that must be carried out by the designer/analyst.

Elements that are commonly employed in practice are shown in Fig. 4. The primary line elements consist of a bar (or truss) and/or beam elements. The first one has only tensile strength characteristic; the second one has full beam characteristics (including bending and torsional strength). These elements are often used to model trusses and frame structures. The simplest line element (called a linear element) has two nodes – one at each end. Higher-order elements having three or more nodes (called quadratic, cubic, etc. elements) also exist.The basic two-dimensional elements are loaded by forces in their own plane (plane stress or plane strain conditions). They are triangular or quadrilateral elements. The simplest two-dimensional elements have corner nodes only (linear elements) with straight sides or boundaries, although there are also higher-order elements, typically with mid-side nodes (called quadratic elements) and curved sides. The elements can have variable thicknesses throughout or be constant. The most common three-dimensional elements are tetrahedral and hexahedral (or brick) elements; they are used when it becomes necessary to perform a three-dimensional stress analysis. The basic three-dimensional elements have corner nodes only and straight sides, whereas higher-order elements with mid-edge nodes (and possible midface nodes) have curved surfaces on their sides.
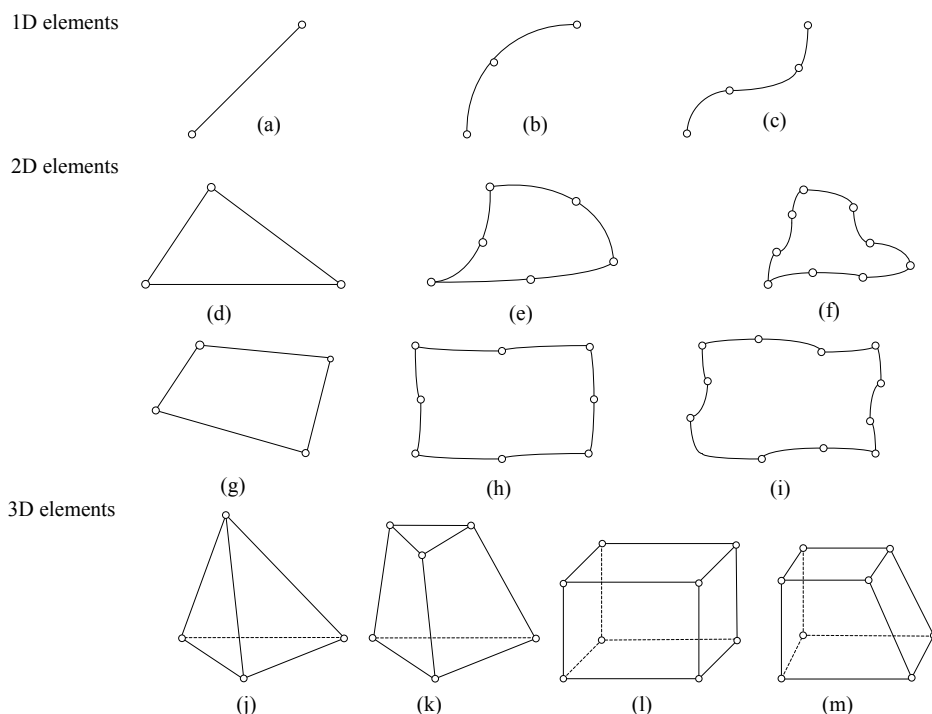

Fig. 4. *Typical elements used in the FE analysis*
Rys. 4. Typowe elementy skończone wykorzystywane w metodzie elementów skończonych

In geometry, shape, as well as the type of the element to be selected, depends not only on the shape and type of the domain of the problem but also on the problem to be analyzed and the degree of accuracy desired. Whereas the number and the location of the nodes in an element depend on the geometry of the element, on the degree of the polynomial approximation, and on the integral form of the equations.

Step 2: Select a Displacement Function. Step 2 involves choosing a displacement function within each element. The function is defined within the element using the nodal values of the element. The linear (Fig. 4: *a, d, g, j*), quadratic (Fig. 4: *b, e, h*), and cubic polynomials (Fig. 4: *c, f, i*) are frequently used functions because they are simple to work with in finite element formulation. Trigonometric series can also be used. The simplest – linear elements are the most popular. Higher order elements are used only in special cases; e.g. when the strain-stress field is highly variable (fatigue analysis near scores).

Step 3: Define the Strain/Displacement and Stress/Strain Relationships. Strain/displacement and stress/strain relationships are necessary for deriving the equations for each finite element. In the case of one-dimensional deformation, say, in the x direction, we have strain ε_x related to displacement u described by equation 1.

$$\varepsilon_x = \frac{du}{dx} \tag{1}$$

The equation (1) applies to small strains. In addition, the stresses must be related to the strains through the stress/strain law, generally called the constitutive law. The ability to define the material behavior accurately is most important in obtaining acceptable results. The simplest of stress/strain laws, Hooke's law, which is often used in stress analysis, is given by equation 2.

$$\sigma_x = E\varepsilon_x \tag{2}$$

where:
    σ_x – stress in the *x* direction,
    E – modulus of elasticity.

Step 4: Derive the Element Stiffness Matrix and Equations. Derive the equations within each element. Initially, the development of element stiffness matrices and element equations was based on the concept of stiffness influence coefficients, which presupposes a background in structural analysis. Three general methods can be distinguished: direct equilibrium or stiffness method (the most common), work or energy method and method of weighted residuals.

In the direct equilibrium or stiffness method, the stiffness matrix and element equations relating nodal forces to nodal displacements are obtained using force equilibrium conditions for a basic element, along with force/deformation relationships. This method is most easily adaptable to line or one-dimensional elements respectively.

In terms of work or energy methods, the stiffness matrix and equations developing, is much easier for two- and three-dimensional elements [Oden and Ripperger 1981]. The principle of virtual work (using virtual displacements), the principle of minimum potential energy, and Castigliano's theorem are methods frequently used for the purpose of derivation of element equations.

The principle of virtual work is applicable for any material behavior, whereas the principle of minimum potential energy and Castigliano's theorem are applicable only to elastic materials. Furthermore, the principle of virtual work can be used even when a potential function does not exist. However, all three principles yield identical element equations for linear-elastic materials; thus which method to use for this kind of material in structural analysis is largely a matter of convenience and simplicity.

The methods of weighted residuals are useful for developing the element equations; particularly popular is Galerkin's method. These methods yield the same results as the energy methods wherever the energy methods are applicable. They are especially useful when a functional such as potential energy is not readily available. The weighted residual methods allow the finite element method to be applied directly to any differential equation.

Galerkin's method, along with the collocation, the least squares, and the subdomain weighted residual methods will be used to derive the bar element equations and the beam element equations and to solve the combined heat-conduction, convection, mass transport problem. For more information on the use of the methods of weighted residuals, see reference [Finlayson 1972], for additional applications to the finite element method, consult references [Cook, et al. 2002].

Step 5: Assemble the Element Equations to Obtain the Global or Total Equations and Introduce Boundary Conditions. In this step, the individual element nodal equilibrium equations are generated. After that, the characteristic matrix of finite elements is assembled into the global nodal equilibrium equations. Another more direct method of superposition (called the direct stiffness method), whose base is nodal force equilibrium, can be used to obtain the global equations for the whole structure. The implication of the direct stiffness method is the concept of continuity, or compatibility, which requires that the structure remains together. The finally assembled, global equation for dynamic problems is written in matrix form and presented in equation 3.

$$M\ddot{u} + C\dot{u} + Ku = F \tag{3}$$

where:
    F – force vector,
    K – stiffness matrix,
    u – displacement vector,
    C – damping matrix,
    M – mass matrix.

For most problems, the global characteristic matrixes (stiffness, damping, mass) are square and symmetric. The displacement vector is the vector of known and unknown structure nodal degrees of freedom or generalized displacements. It can be shown that at this stage, that the global stiffness matrix is a singular matrix because its determinant is equal to zero. To remove this singularity problem, we must invoke certain boundary conditions (or constraints or supports) so that the structure remain in place instead of moving as a rigid body. Further details and methods of invoking boundary conditions are given in subsequent chapters. At this time, it is sufficient to note that invoking boundary or support conditions results in modification of the global equation 3. The applied known loads have been accounted for in the global force matrix.

Step 6: Solve for the Unknown Degrees of Freedom. The equation (3) can be simplified to static strength analyses by expunction of elements with acceleration vector (mass matrix) and speed vector (damping matrix). Simplified equation modified by the boundary conditions (limits of some of the displacements), is a set of simultaneous algebraic equations that can be written in expanded matrix form as equation 4.

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} K_{12} \cdots K_{1n} \\ K_{21} K_{22} \cdots K_{2n} \\ \cdots \cdots \cdots \cdots \\ \cdots \cdots \cdots \cdots \\ \cdots \cdots \cdots \cdots \\ K_{n1} K_{n2} \cdots K_{nn} \end{bmatrix} \cdot \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ \vdots \\ u_n \end{Bmatrix} \tag{4}$$

In the equation 4 ***n*** is the structure total number of unknown nodal degrees of freedom. These equations can be solved for the ***u_i*** by using an elimination method (such as Gauss's method) or an iterative method (such as the Gauss–Seidel method). The ***u_i*** are called the primary unknowns because they are the first quantities determined using the stiffness (or displacement) finite element method.

Step 7: Solve for the Element Strains and Stresses. For the structural stress-analysis problem, important secondary quantities of strain and stress (or moment and shear force) can be obtained because they can be directly expressed in terms of the displacements determined in step 6. Typical relationships between strain and displacement and between stress and strain (such as equation 1, 2) presented in step 3 can be used.

Step 8: Interpretation of the Results. The final goal is to interpret and analyze the results for use in the design/analysis process. Determination of locations in the structure where large deformations and large stresses occur is generally important in making design/analysis decisions. Postprocessor computer programs help the user to interpret the results by displaying them in graphical form.

4. CONCLUSIONS

All engineers should be knowledgeable about numerical methods. There are engineers specialized in numerical analyses but also designers can have the ability to support their drafts with calculations. Strengths and vibrations analyses are a special part of numerical calculations. But also, engineers working with machine exploitation should have knowledge about numerical calculations. Usually, they receive several documents with applied procedures as well as with numerical analyses with practical conclusions (e.g. barred speed range for marine propulsion system caused by torsional vibration).

Among numerical methods, Finite Element Method is a leader. It is a good, valuable and reliable method, having several implementations. The Finite Element Method has been applied to numerous problems, both structural and nonstructural. This method has a number of advantages that have made it very popular. The FEM of structural analysis enables the designer to detect stress, vibration, thermal problems, fluid flow, electromagnetic potentials etc. during the design process and to evaluate design changes before constructing possible prototype. Thus, confidence in the acceptability of the prototype is enhanced. Moreover, if used properly, the method can reduce the number of prototypes that need to be built.

However, 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.

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*, Wiley, New York.

Finlayson, BA., 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*, 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 of London, Series A, vol. 210, s. 307–357.

Zienkiewicz, O.C., Taylor, R.L., 2005, *The Finite Element Method,* vol. 1: *The Basis*, Butterworth-Heinemann, Oxford.