Debreceni Műszaki Közlemények 2014/1 (HU ISSN 2060-6869)

EGYOSZLOPOS MAGASRAKTÁRI FELRAKÓGÉPEK VÉGESELEM MODELLEZÉSE⊗

FINITE ELEMENT MODELING OF SINGLE-MAST STACKER CRANES

HAJDU Sándor

Főiskolai adjunktus
Debreceni Egyetem, Műszaki Kar
4028 Debrecen, Ótemető u. 2-4.
hajdusandor@eng.unideb.hu

Dr. GÁSPÁR Péter

az MTA doktora, Tudományos tanácsadó
MTA Számítástechnikai és Automatizálási Kutatóintézet
1111 Budapest, Kende u. 13-17.
gaspar.peter@sztaki.mta.hu

Kivonat: *A cikk egyoszlopos magasraktári felrakógépek végeselem modellezési lehetőségeivel foglalkozik. A külső gerjesztő hatások és tömegerők következtében az ilyen berendezések vázszerkezetében nem kívánatos oszloplengések alakulhatnak ki. Ezek a lengések csökkenthetik a berendezés pozícionálási pontosságát és stabilitását, valamint az egész anyagmozgató rendszer ciklusidejének növekedését is okozhatják. A bemutatott okok miatt szükséges ezen lengések részletes vizsgálata. Az elmúlt időszakban számos példa jelent meg a nemzetközi szakirodalmakban magasraktári felrakógépek dinamikai modellezésével kapcsolatban. Jelen cikkben az úgynevezett kétdimenziós gerendaelem (2D BEAM) tömeg –és merevségi mátrixai kerülnek levezetésre. Ennek az elemtípusnak a segítségével egy végeselem modellt állítunk össze egyoszlopos magasraktári felrakógépek dinamikai modellezése céljából. A dinamikai modell állapottér reprezentációja és átviteli függvénye is bemutatásra kerül.*

Kulcsszavak: *magasraktári felrakógép, végeselem módszer (VEM), végeselem analízis, állapottér reprezentáció, átviteli függvény*

Abstract: *This paper presents the finite element modeling possibilities of stacker cranes with single-mast structure. Because of the external excitation or inertial forces undesirable mast-vibrations may arise in the frame structure of stacker crane. These vibrations can reduce positioning accuracy and stability of the machine and increase the cycle time of the whole material handling system. Because of the reasons mentioned before it is necessary to model and investigate these vibrations. In the last few years several kinds of dynamical modeling methods of stacker cranes have been introduced in the literature. In this paper the derivation of mass and stiffness matrices for the so called two dimensional beam element (2D BEAM) is presented. By the help of this element type a finite element model of single-mast stacker cranes is constructed. The state space representation and transfer function of the model are also introduced.*

Keywords: *stacker carne, finite element method (FEM), finite element analysis (FEA), state space representation, transfer function*

1. INTRODUCTION

The advanced stacker cranes in automated storage/retrieval systems (AS/RS) have the requirement of

⊗ Szaklektorált cikk. Leadva:2014. június 26., Elfogadva: 2014. július 08.
Reviewed paper. Submitted: 26.06. 2014. Accepted: 08.07.2014.
Lektorálta: MANKOVITS Tamás/ Reviewed by Tamás MANKOVITS

fast working cycles and reliable, economical operation. Today these machines often dispose of 1500 kg pay-load capacity, 40-50 m lifting height, 250 m/min velocity and 2 m/s^2 acceleration in the direction of aisle with 90 m/min hoisting velocity and 0,5 m/s^2 hoisting acceleration. Therefore the dynamical loads on mast structure of stacker cranes are very high, while the stiffness of these structures due to the dead-weight reduction is relatively low. Thus undesirable mast-vibration may arise in the frame structure during operation. The high amplitude mast-vibration reduces stability and positioning accuracy of the stacker crane and in extreme case it may damage the structure.

Practically the mast structure has two fundamental configurations: the so-called single-mast and twin-mast structures. In our work we analyze the single-mast structures since this configuration is more responsive to dynamical excitations. A schematic drawing of single-mast stacker crane with its main components is shown in Figure 1.

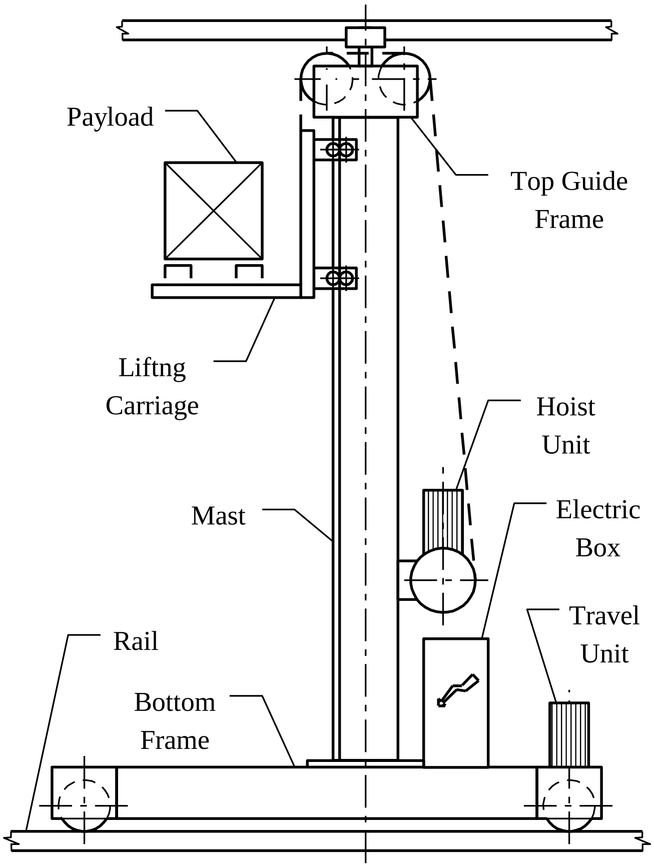

Figure 1. Single-mast stacker crane

Estimation of structural vibrations during design period of stacker cranes, dynamical investigation of an existing structure as well as reduction of these effects requires dynamical modeling of the frame structure. In our paper we introduce a finite element model for modeling dynamical behavior of single-mast stacker cranes. The finite element modeling (FE modeling) is a widely used modeling technique of engineering structures and it has extensive international [4-8] as well as Hungarian [9-14] literature. However in the literature of stacker cranes only a few examples can be found for FE modeling of these machines. In [1] the author introduces a beam model for calculating static deformations of a single-mast stacker crane. Schiller investigates in his work [2] a 3D beam model for dynamical modeling and structural optimization of stacker cranes. Kühn applies in his thesis [3] FEM for determination the dynamical behavior of load handling system with telescopic fork during hoisting movement.

This paper presents the dynamical modeling capabilities of single-mast stacker crane structures based on FE modeling techniques. The main features of 2D beam elements and determination of mass and stiffness matrices of these elements are also introduced. As an example, in our paper we introduce a finite element model (FE model) with 2D beam elements for dynamical modeling mast-vibrations of single-mast stacker cranes. Finally the main features of our model are presented. The most important parameters of the investigated stacker crane are shown in Table 1.

Denomination	Denotation	Value
Payload:	m_p	1200 kg
Mass of lifting carriage:	m_{lc}	410 kg
Mass of hoist unit:	m_{hd}	470 kg
Mass of top guide frame:	m_{tf}	70 kg
Mass of bottom frame:	m_{sb}	2418 kg
Mass of entire mast:	m_{sm}	8148 kg
Lifted load position:	h_h	1-44 m
Length of sections (lifted load is in uppermost position):	l_1	2,9 m
	l_2	3 m
	l_3	3,5 m
	l_4	11,5 m
	l_5	29 m
	l_6	1 m
Cross-sectional areas:	A_1; A_2	0,03900 m^2
	A_3; A_4	0,02058 m^2
	A_5; A_6	0,01518 m^2
Second moments of areas:	I_{z1}; I_{z2}	0,00152 m^4
	I_{z3}; I_{z4}	0,00177 m^4
	I_{z5}; I_{z6}	0,00106 m^4

Table 1. Main parameters of investigated stacker crane

2. BASIC FORMULATION OF FEM

In the theory of elasticity FEM has two fundamental forms: the so-called flexibility or force method and the stiffness or displacement method. In practice the displacement method is more frequently used. More detailed information about displacement method can be found in references [4-14]. This method is based on the principle of minimum potential energy, which states: for conservative systems, of all the kinematically admissible u displacement fields the actual displacement field (which satisfies the equilibrium conditions) is the one that minimizes the potential energy function. Kinematically admissible displacement field is the one that satisfies the boundary conditions and compatibility conditions (strain-displacement equations). Thus the basic equation of displacement method is:

$$\frac{\partial \Pi}{\partial u} = 0 \ , \tag{1}$$

where Π is the potential energy of the system and u is the exact solution of elasticity problem presented above.

In FEM the investigated elastic continuum is represented by separating the continuum into a number of finite elements. The elements are interconnected at a number of discrete points called nodes. The U nodal displacement vector is the basic unknown of the problem, which is the approximation of the exact u solution. Thus the basic equation of FEM is:

$$\frac{\partial \Pi}{\partial U} = 0 \ . \tag{2}$$

The potential energy of the system is the sum of the Π_s strain energy and the Π_w work potential.

$$\Pi = \Pi_s + \Pi_w \quad . \tag{3}$$

The strain energy is calculated by means of the ε normal strain and σ normal stress:

$$\Pi_s = \frac{1}{2}\int_{(V)} \left(\varepsilon^T \sigma\right) dV \quad . \tag{4}$$

The work potential is the sum of works done by external nodal (F_n), surface (P) and body (Q) forces (these works are assumed to be negative).

$$\Pi_w = -U^T F_n - \int_{(A)} \left(u^T P\right) dA - \int_{(V)} \left(u^T Q\right) dV \quad . \tag{5}$$

In case of dynamic analysis the $-\rho\ddot{u}$ inertial force (d'Alambert force) also must be taken into account. This force can be assumed as the part of body forces. By means of this force the augmented form of work potential is:

$$\Pi_w = -U^T F_n - \int_{(A)} \left(u^T P\right) dA - \int_{(V)} \left(u^T Q\right) dV + \int_{(V)} \left(u^T \rho\ddot{u}\right) dV \ . \tag{6}$$

The constitutive law (stress-strain relationship) with the material matrix D in general form is:

$$\sigma = D\varepsilon \quad . \tag{7}$$

The compatibility equation is:

$$\varepsilon = L_d u \quad , \tag{8}$$

where L_d is a differential operator depends on the actual problem. Substituting (7) and (8) into (4) the potential energy of system is:

$$\Pi = \frac{1}{2}\int_{(V)} \left(u^T L_d^T D L_d u\right) dV - U^T F_n - \int_{(A)} \left(u^T P\right) dA - \int_{(V)} \left(u^T Q\right) dV + \int_{(V)} \left(u^T \rho\ddot{u}\right) dV \ . \tag{9}$$

In FEM the real u displacement field is approximated by the following equation.

$$u \approx NU \quad , \tag{10}$$

where N is a matrix of special interpolation functions called shape functions or base functions (in most cases polynomials). With this approximation:

$$\begin{aligned} \Pi = {} & \frac{1}{2} U^T \int_{(V)} \left(N^T L_d^T D L_d N\right) dV * U - U^T F_n - U^T \int_{(A)} \left(N^T P\right) dA - \\ & - U^T \int_{(V)} \left(N^T Q\right) dV + U^T \int_{(V)} \left(\rho N^T N\right) dV * \ddot{U} \end{aligned} \quad . \tag{11}$$

Thus the basic equation of FEM applying the $L_d N = B$ denotation is:

$$\frac{\partial \Pi}{\partial U} = \int_{(V)} \left(B^T D B\right) dV * U + \int_{(V)} \left(\rho N^T N\right) dV * \ddot{U} - F_n - \int_{(A)} \left(N^T P\right) dA - \int_{(V)} \left(N^T Q\right) dV = 0 \ . \tag{12}$$

The first integral in equation (12) is the so-called element stiffness matrix (S_e), the second one is the element mass matrix (M_e) and the other three terms are the external forces reduced into nodes (F_e). Thus the dynamic equation of motion for the investigated element is:

$$M_e \ddot{U} + S_e U = F_e \quad . \tag{13}$$

3. INTRODUCTION OF THE LINE ELEMENTS OF FEM

In our work we use line elements to model the dynamical behavior of single-mast stacker cranes, see in Figure 2. It means that the approximated differential equation of these elements has one independent spatial variable (i.e. it is ordinary differential equation). In our model the transversal displacements are approximated by the expressions of so-called bending beam elements, while the longitudinal displacements are approximated by truss elements.

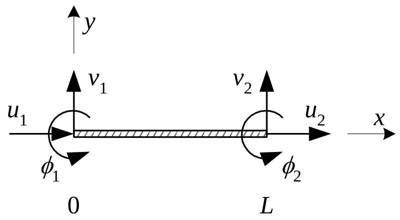

Figure 2. Line element

3.1. *Bending beam elements*

The displacement state of bending beam is represented by the $v(x)$ transversal deflection function of the beam. This deflection function is approximated by a $p(x)$ single-variable polynomial, which order is equal to the order of base functions. In the first step we determine the compatibility equation. From strength of materials it is known that: $\sigma_x = \frac{M(x)}{I_z} y$ and $\sigma_x = E\varepsilon_x$. Thus: $\varepsilon_x = \frac{M(x)}{I_z E} y$. From the Euler-Bernoulli beam theory: $\frac{\partial^2 v(x)}{\partial x^2} = -\frac{M(x)}{I_z E}$. With the last two equations the compatibility equation in this case is:

$$\varepsilon_x = -y \frac{\partial^2 v(x)}{\partial x^2} \ . \tag{14}$$

Thus the differential operator of the bending beam problem is:

$$L_d = -y \frac{\partial^2}{\partial x^2} \quad . \tag{15}$$

The operator presented here prescribes second order differential operation. As can be seen in expression (11) the L_d operator acts on the N matrix of base functions. This enables us to determine the main properties of base functions (base polynomials). Let us assume that the L_d operator is nth-order differential operator. If the nth (or previous) derivation performed on the base functions (polynomials) of N matrix results zero, then the stiffness matrix will singular. In this case the fundamental equation of static finite element analysis is unsolvable. The degree of polynomials in the N matrix (the order of approximation) therefore must be at least n.

In our investigations we apply line elements with two nodes at its endpoints and cubic approximation polynomials (see in [12]). Applying cubic polynomials means that we have to determine four independent parameters during determination of the polynomials. At the same time this specifies also the so-called fitting order of elements if the number of nodes is fixed. In case of two node elements for determination of the four independent parameters we have to specify at connection points of elements not only the continuity of approximation functions but the continuity of its derivatives (C^1-continuous fitting). Therefore in the vector U beside nodal displacements the nodal angular deflections also appear:

$$U = \begin{bmatrix} v_1 \\ \phi_1 \\ v_2 \\ \phi_2 \end{bmatrix} \quad . \tag{16}$$

The interpolation polynomials and their matrices are as follows (the detailed derivation of these polynomials see in [12]):

$$N_v(x) = \left[N_1(x) \;\; N_2(x) \;\; N_3(x) \;\; N_4(x) \right] \; . \tag{17}$$

$$\begin{aligned} N_1(x) &= 1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3} \; ; \quad & N_2(x) &= x - \frac{2x^2}{L} + \frac{x^3}{L^2} \; ; \\ N_3(x) &= 3\frac{x^2}{L^2} - 2\frac{x^3}{L^3} \; ; \quad & N_4(x) &= -\frac{x^2}{L} + \frac{x^3}{L^2} \quad . \end{aligned} \tag{18}$$

Because of the one dimensional state of stress the material matrix is simplified as:

$$D = E \quad . \tag{19}$$

Using the expression (15) the B matrix in the element stiffness matrix is:

$$B = L_d N = \left(-y \frac{\partial^2}{\partial x^2} \right) N_v \; . \tag{20}$$

Thus:

$$B^T D B = y^2 E \left(\left(\frac{\partial^2 N_v}{\partial x^2} \right)^T \left(\frac{\partial^2 N_v}{\partial x^2} \right) \right) \; . \tag{21}$$

With the equations presented before the stiffness matrix of C^1-continous bending beam element can be expressed as:

$$S_e = \int_{(V)} (B^T DB)dV = E \int_{(A)} y^2 dA \int_{(L)} \left(\frac{\partial^2 N_v}{\partial x^2} \right)^T \left(\frac{\partial^2 N_v}{\partial x^2} \right) dx = I_z E \int_{(L)} \left(\frac{\partial^2 N_v}{\partial x^2} \right)^T \left(\frac{\partial^2 N_v}{\partial x^2} \right) dx \ . \tag{22}$$

$$S_e = \frac{I_z E}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} . \tag{23}$$

The mass matrix of this kind of element is:

$$M_e = \rho A \int_{(L)} \left(N_v{}^T (x) N_v (x) \right) dx \ . \tag{24}$$

$$M_e = \rho A \begin{bmatrix} \frac{13L}{35} & \frac{11L^2}{210} & \frac{9L}{70} & -\frac{13L^2}{420} \\ \frac{11L^2}{210} & \frac{L^3}{105} & \frac{13L^2}{420} & -\frac{L^3}{140} \\ \frac{9L}{70} & \frac{13L^2}{420} & \frac{13L}{35} & -\frac{11L^2}{210} \\ -\frac{13L^2}{420} & -\frac{L^3}{140} & -\frac{11L^2}{210} & \frac{L^3}{105} \end{bmatrix} . \tag{25}$$

3.2. *Truss elements*

The displacement state of truss elements is represented by the $u(x)$ elongation function. This elongation function here is also approximated by a $p(x)$ single-variable polynomial, which order is equal to the order of base functions. In the first step we determine the compatibility equation and its differential operator from the definition of elongation per unit length.

$$\varepsilon_x = \frac{\partial u(x)}{\partial x} \ . \tag{26}$$

$$L_d = \frac{\partial}{\partial x} \ . \tag{27}$$

As can be seen in this case application of linear interpolation polynomials is suitable for the approximation of displacement field. The fitting order of this element is C^0-continous. The matrix of interpolation polynomials in this case is as follows (see in [12]):

$$N_u(x) = \left[1 - \frac{x}{L} \quad \frac{x}{L} \right] \ . \tag{28}$$

The material matrix is the same as in expression (19) due to the one dimensional state of stress. Thus determination of the stiffness and mass matrices of truss element can be performed as follows:

$$B = L_d N_u(x) = \left[-\frac{1}{L} \quad \frac{1}{L} \right] = \frac{1}{L} [-1 \ \ 1] \ . \tag{29}$$

$$B^T DB = \frac{1}{L^2}\begin{bmatrix} -1 \\ 1 \end{bmatrix} E\begin{bmatrix} -1 & 1 \end{bmatrix} = \frac{E}{L^2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad . \tag{30}$$

$$S_e = \int_{(V)} (B^T DB)\,dV = A\int_{(L)} (B^T DB)\,dx = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad . \tag{31}$$

$$M_e = \rho A \int_{(L)} \left(N_u^{\,T}(x) N_u(x) \right) dx = \rho A L \begin{bmatrix} \frac{1}{3} & \frac{1}{6} \\ \frac{1}{6} & \frac{1}{3} \end{bmatrix} \quad . \tag{32}$$

3.3. *Two dimensional beam elements (2D BEAM)*

By means of the results presented in previous sections the stiffness and mass matrices of 2D beam element can be constructed. As mentioned before in this line element the transversal displacements are approximated by C^1-continous interpolation polynomials, while the longitudinal displacements are approximated by C^0-continous interpolation polynomials. The nodal displacement vector (nodal generalized coordinate vector) can be as follows:

$$U = \begin{bmatrix} u_1 \\ v_1 \\ \phi_1 \\ u_2 \\ v_2 \\ \phi_2 \end{bmatrix} \quad . \tag{33}$$

Taking the order of coordinates in the vector above into account the element stiffness and mass matrices are:

$$S_e = \frac{I_z E}{L^3}\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 12 & 6L & 0 & -12 & 6L \\ 0 & 6L & 4L^2 & 0 & -6L & 2L^2 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & -12 & -6L & 0 & 12 & -6L \\ 0 & 6L & 2L^2 & 0 & -6L & 4L^2 \end{bmatrix} + \frac{AE}{L}\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \quad . \tag{34}$$

$$M_e = \rho A \begin{bmatrix} \frac{L}{3} & 0 & 0 & \frac{L}{6} & 0 & 0 \\ 0 & \frac{13L}{35} & \frac{11L^2}{210} & 0 & \frac{9L}{70} & -\frac{13L^2}{420} \\ 0 & \frac{11L^2}{210} & \frac{L^3}{105} & 0 & \frac{13L^2}{420} & -\frac{L^3}{140} \\ \frac{L}{6} & 0 & 0 & \frac{L}{3} & 0 & 0 \\ 0 & \frac{9L}{70} & \frac{13L^2}{420} & 0 & \frac{13L}{35} & -\frac{11L^2}{210} \\ 0 & -\frac{13L^2}{420} & -\frac{L^3}{140} & 0 & -\frac{11L^2}{210} & \frac{L^3}{105} \end{bmatrix} . \tag{35}$$

4. COORDINATE TRANSFORMATION AND ELEMENT ASSEMBLY

In the previous section results of investigation i.e. the stiffness or mass matrices of elements and force vectors correspond to the local coordinate systems of each element. To determine the global matrices and vectors of the complete frame structure, a common global coordinate system must be established for all structural elements. The choice of this coordinate system can be arbitrary.

Before the element assembly (merge of elements) the matrices and vectors of each element must be transformed into common global coordinate system. Thus we need a transformation method between the local and global coordinate systems. In Figure 3. a beam element is shown with its local and global nodal displacements. Let us denote the nodal generalized coordinate vector in the local system by U as well as in the global system by $\overline{U}$.

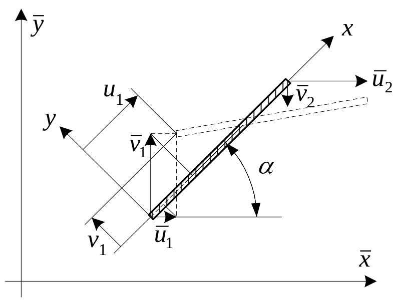

Figure 3. Beam element in local and global coordinate systems

By means of Figure 3. the desired coordinate transformation can be expressed as follows:

$$\begin{bmatrix} u_1(t) \\ v_1(t) \\ \phi_1(t) \\ u_2(t) \\ v_2(t) \\ \phi_2(t) \end{bmatrix} = \begin{bmatrix} cos\alpha & sin\alpha & 0 & 0 & 0 & 0 \\ -sin\alpha & cos\alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & cos\alpha & sin\alpha & 0 \\ 0 & 0 & 0 & -sin\alpha & cos\alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \bar{u}_1(t) \\ \bar{v}_1(t) \\ \bar{\phi}_1(t) \\ \bar{u}_2(t) \\ \bar{v}_2(t) \\ \bar{\phi}_2(t) \end{bmatrix} . \tag{36}$$

$$U(t) = \Gamma_r \overline{U}(t)$$

In global coordinate system let us denote the element stiffness and mass matrices by $\overline{K}_e$ and $\overline{M}_e$ as well as the force vector by $\overline{F}_e$. With the transformation matrix Γ_r the desired operation can be performed as presented in the following expressions.

$$\overline{M}_e = \Gamma_r^{\,T} M_e \Gamma_r \quad . \tag{37}$$

$$\bar{S}_e = \Gamma_r^{\,T} S_e \Gamma_r \quad . \tag{38}$$

$$\overline{F}_e = \Gamma_r^{\,T} F_e \quad . \tag{39}$$

Thus the dynamic equation of motion for the transformed element is:

$$\overline{M}_e \ddot{\overline{U}} + \bar{S}_e \overline{U} = \overline{F}_e \quad . \tag{40}$$

After transformation of element stiffness and mass matrices into a common global coordinate system the assembly of element matrices must be performed in order to determine the global matrices of the whole system. The scheme for the assembly of global system matrices from element matrices is shown in Figure 4. As can be seen in the figure first the element matrices must be positioned in the global system matrix. After that the overlapping parts of element matrices must be added. A detailed derivation of element assembly can be found in [4].

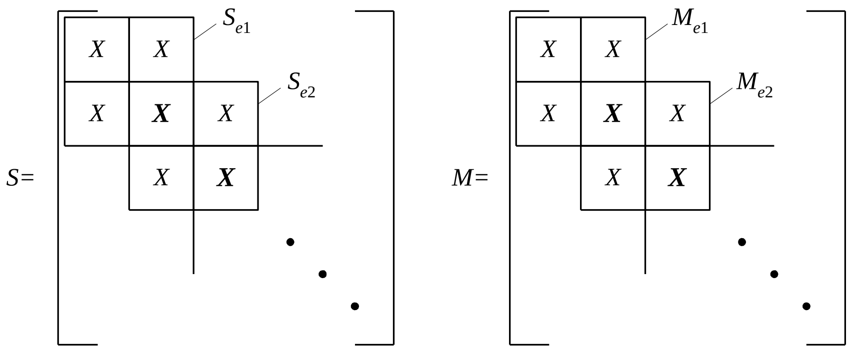

Figure 4. Element assembly

5. FE MODELS OF SINGLE MAST STACKER CRANES

With the 2D beam elements introduced in the previous sections FE models are defined for dynamical modeling of single mast stacker cranes shown in Figure 5. For construction and investigation of these models numerical algorithms, functions of Matlab software are applied, more information about Matlab can be found in [15]. In the following figure every sections of mast and bottom frame with its length and the number of elements (in curly brackets) as well as the lumped masses are shown. In these models we take the upper and lower lifted load positions into consideration. Some of the nodes between elements are denoted by solid dots.

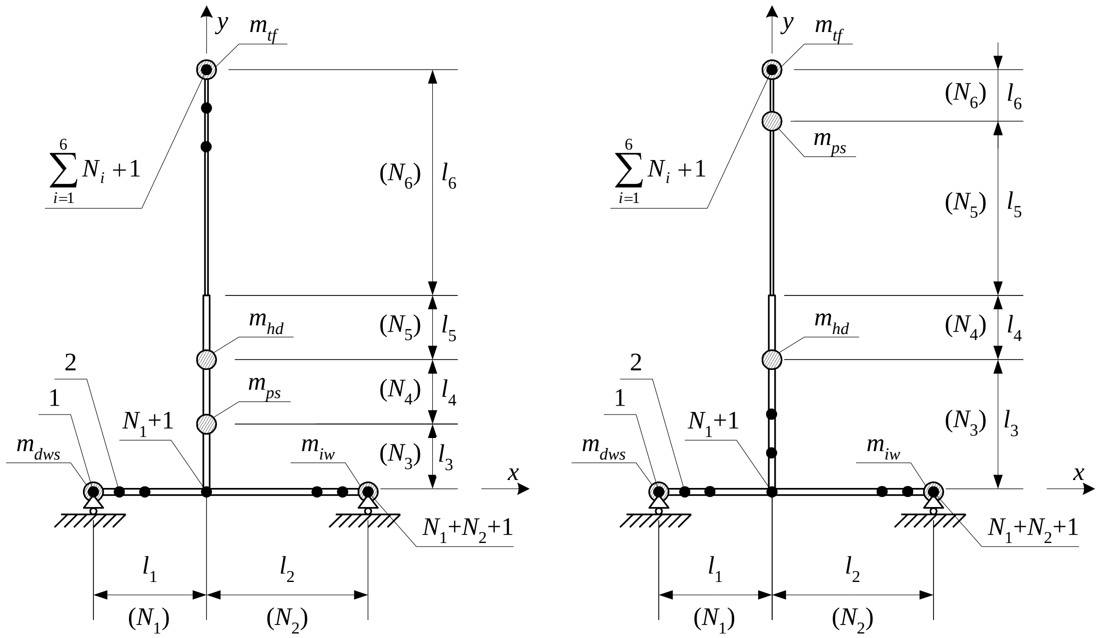

Figure 5. FE models of single-mast stacker cranes

The number of elements for both models is:

$$N_e = \sum_{i=1}^{6} N_i \quad . \tag{41}$$

With the previous equation the number of nodes and the degrees of freedom are:

$$N_c = \sum_{i=1}^{6} N_i + 1 = N_e + 1 \quad , \tag{42}$$

$$N_{DOF} = 3\left(\sum_{i=1}^{6} N_i + 1\right) = 3N_c \quad . \tag{43}$$

After coordinate transformation and element assembly for the final form of dynamic equation of motion the determination of constraints (boundary conditions) is also necessary. These constraints prevent the vertical movement in the endpoints of bottom frame (see in Figure 5.). In case of fixed boundary conditions in the global system matrices the rows and columns corresponding to fixed degree of freedom have to be deleted since the actual displacement in these directions is zero. Thus the degree of freedom of constrained model equals to $N_d = 3N_c - 2$ and the global vector of generalized coordinates is:

$$q = \begin{bmatrix} u_1 & \phi_1 & u_2 & v_2 & \phi_2 & \cdots & u_{N_c} & v_{N_c} & \phi_{N_c} \end{bmatrix}^T \quad . \tag{44}$$

The dynamic equation of motion for the whole constrained system is:

$$M\ddot{q} + Sq = F \quad . \tag{45}$$

In the first part of the model investigations the analysis of free vibrations is carried out. In this case the external excitation forces are zero, thus the basic equation of motion is:

$$M\ddot{q} + Sq = 0 \quad . \tag{46}$$

$$\left(S - \alpha^2 M\right)\psi = 0 \ . \tag{47}$$

Analytical solution of equation (46) leads to the (47) generalized eigenvalue problem. The number of eigenvalues of this problem equals to the degree of freedom of system (46). The α_j^2 eigenvalues are the squares of natural frequencies of the dynamic system. Since our investigated model is a free model, i.e. it has rigid body motion facility (unconstrained degree of freedom), thus the smallest α_0^2 eigenvalue equals to zero. The corresponding eigenvectors are also known as the mode shapes of the dynamic system. The first three natural frequencies in case of upper and lower lifted load position are shown in Table 2.

Upper lifted load position:	Lower lifted load position:
$\alpha_1 = 3{,}3661\ {}^{rad}\!/_{s}$	$\alpha_1 = 2{,}5684\ {}^{rad}\!/_{s}$
$\alpha_2 = 16{,}836\ {}^{rad}\!/_{s}$	$\alpha_2 = 15{,}044\ {}^{rad}\!/_{s}$
$\alpha_3 = 43{,}527\ {}^{rad}\!/_{s}$	$\alpha_3 = 40{,}439\ {}^{rad}\!/_{s}$

Table 2. Natural frequencies

The first four mode shapes in case of upper and lower lifted load position are shown in next Figures.

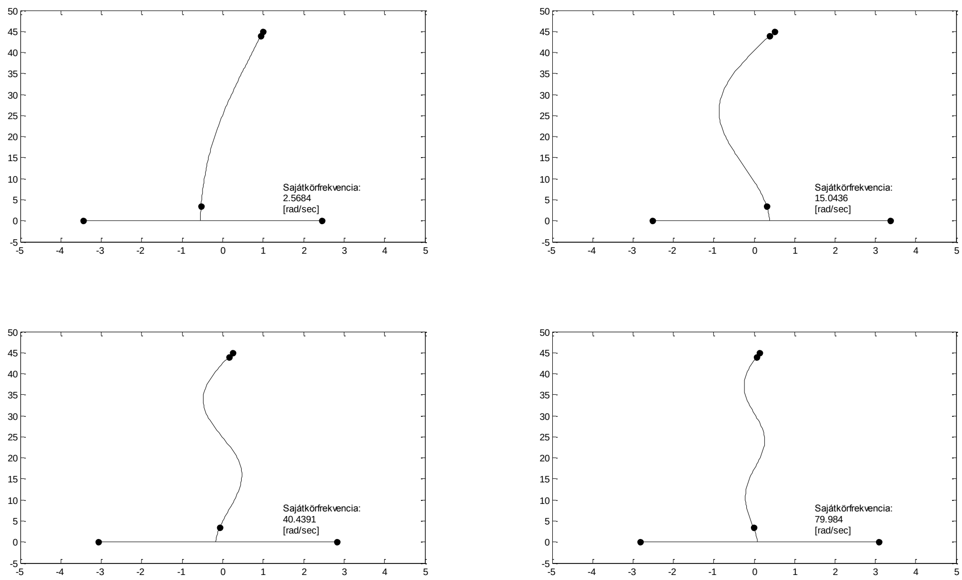

Figure 6. Mode shapes (upper lifted load position)

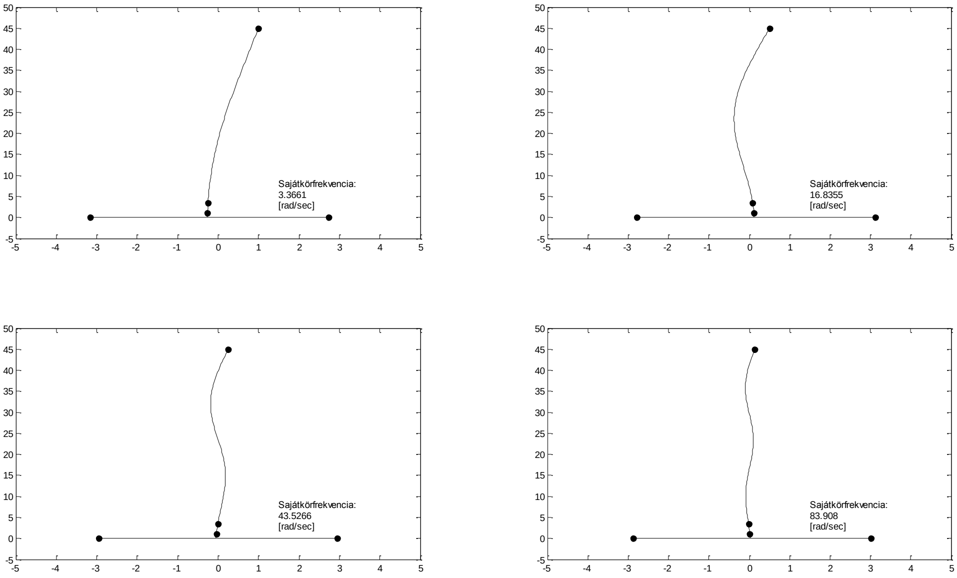

Figure 7. Mode shapes (lower lifted load position)

6. STATE SPACE REPRESENTATION AND TRANSFER FUNCTION OF MODELS

For the investigation of excited vibrations of the flexible structure it is necessary to express the motion equations in state space representation and to derive the transfer function of excited system. The matrix motion equation of structures subject to external excitation forces is as shown in equation (45). In this section we investigate a single-input and single-output (SISO) system. The input signal of our model is the external force acting in the direction of q_1 generalized coordinate. Let's denote the input signal by $F_1=u$ and the degree of freedom of the constrained model by N_d. The output signal of our model is the vertical position of mast-tip i.e. the value of generalized coordinate q_{N_d} .

Let us introduce the so called state vector and its derivative respectively:

$$x = \begin{bmatrix} \dot{q} \\ q \end{bmatrix}; \qquad \dot{x} = \begin{bmatrix} \ddot{q} \\ \dot{q} \end{bmatrix} \quad . \tag{48}$$

Generally the state space representation can be expressed in the following form:

$$\dot{x} = Ax + bu \ . \tag{49}$$

$$y = c^T x \ , \tag{50}$$

where A, b, c^T are the matrix and vectors of the system, u is the input and y is the output of the system. By means of (48-50), the matrix and vectors of state space representation of the system can be expressed as follows:

$$A = \begin{bmatrix} 0_{N_d} & -M^{-1}S \\ I_{N_d} & 0_{N_d} \end{bmatrix} \quad . \tag{51}$$

$$b = \begin{bmatrix} M^{-1} \\ 0_{N_d} \end{bmatrix} \begin{bmatrix} 1 \\ 0 \\ \vdots \\ 0 \end{bmatrix}_{N_d} \quad . \tag{52}$$

$$c^T = \begin{bmatrix} 0 & \cdots & 0 & 1 \end{bmatrix}_{2N_d} \quad , \tag{53}$$

where 0_{N_d} is a zero matrix and I_{N_d} is an identity matrix with the corresponding size.

The transfer function of the model is determined by Laplace-transform of the state space representation. If we denote the Laplace operator by *s*, then the transfer function of the model can be expressed by the matrix and vectors of state space representation as follows.

$$G(s) = c^T (sI - A)^{-1} b \quad . \tag{54}$$

With the substitution $s=i\omega$ into transfer function we get the $G(i\omega)$ frequency response function (FRF) of the system. The magnitude of FRF as a function of angular frequency, i.e. the Bode magnitude diagram is shown in the following Figures.

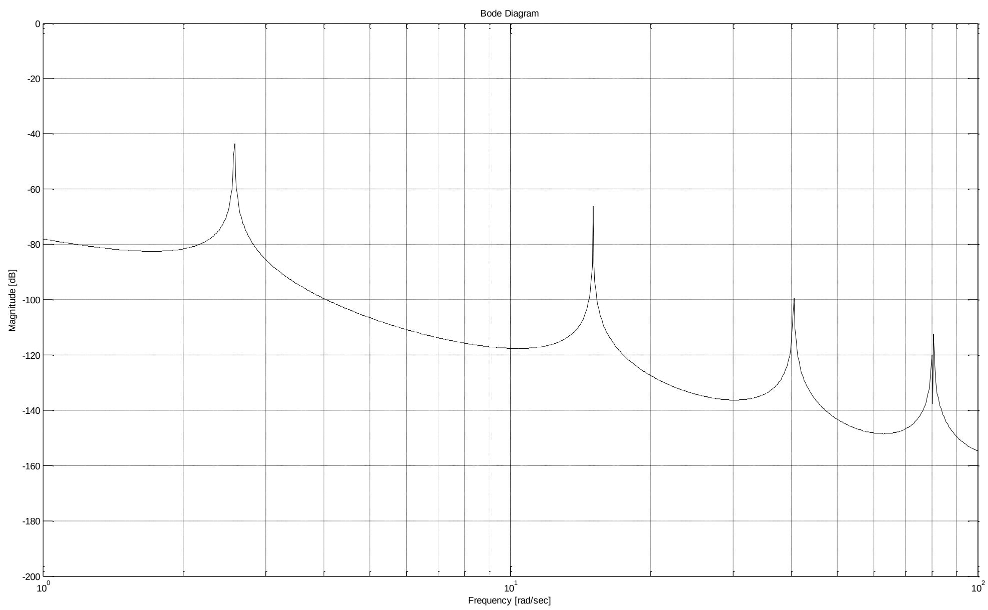

Figure 8. Bode diagram (upper lifted load position)

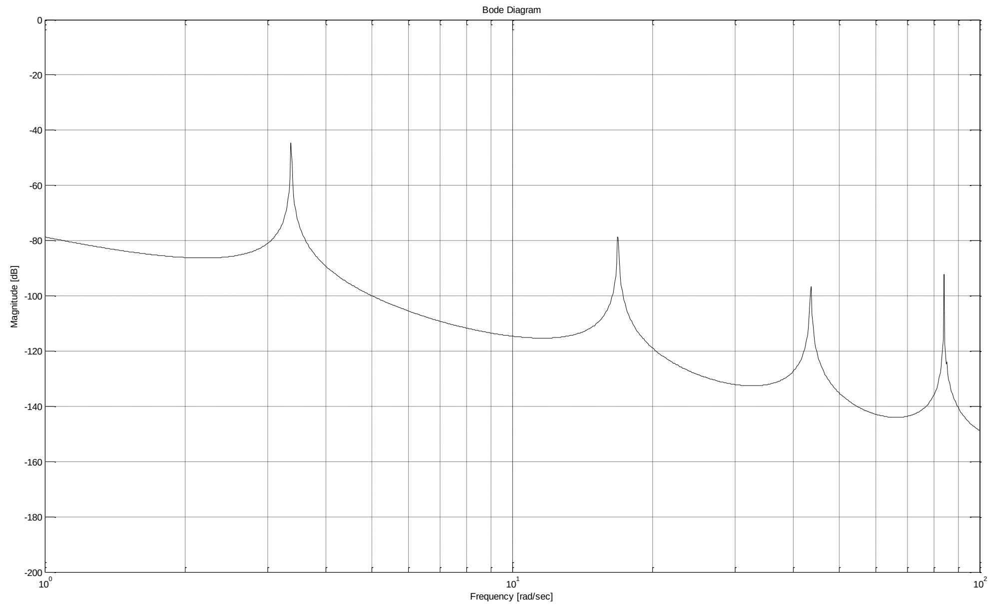

Figure 9. Bode diagram (lower lifted load position)

7. SUMMARY

In our paper we presented a modeling technique based on finite element modeling approach. After the introduction of the basic formulation of finite element method the main properties of line elements were presented. With this kind of element a simple two dimensional finite element model was generated to investigate the dynamical behavior of single mast stacker cranes. Beside the natural frequencies and mode shapes of this model the Bode-diagram of frequency response function was also provided. These investigations can be performed with various lifted load positions. The modeling technique presented here can be useful during the design period of stacker cranes as well as in investigation of existing structures.

8. REFERENCES

[1] **BOPP, W.,** Untersuchung der statischen und dynamischen Positionsgenauigkeit von Einmast-Regalbediengeräten, Dissertation, Institut für Fördertechnik Karlsruhe, 1993

[2] **SCHILLER, M.,** Beanspruchungsermittlung und Optimierung der Tragwerksstruktur von Regalbediengeräten, Dissertation, Universität Stuttgart, 2001

[3] **KÜHN, I.,** Untersuchung der Vertikalschwingungen von Regalbediengeräten, Dissertation, Institut für Fördertechnik Karlsruhe, 2001

[4] **JUANG, J.-N., PHAN, M. Q.,** Identification and control of mechanical systems, Cambridge University Press, Cambridge, 2001, pp. 80-116.

[5] **HUTTON, D. V.,** Fundamentals of finite element analysis, McGraw-Hill, 2004

[6] **MOAVENI, S.,** Finite element analysis, Prentice-Hall, 1999

[7] **LOGAN, D. L.,** A first course in the finite element method, Nelson, 2007

[8] **BATHE, K. J., WILSON, E. L.,** Numerical Methods in Finite Element Analysis, Prentice-Hall, 1976

[9] **POPPER, GY.,** A végeselem-módszer matematikai alapjai; Műszaki Könyvkiadó; Budapest; 1985

[10] **PÁCZELT, I.,** A végeselem-módszer alapjai, Miskolci Egyetemi Kiadó, Miskolc, 1993

[11] **FODOR, T., ORBÁN, F., SAJTOS, I.,** Mechanika – Végeselem-módszer – Elmélet és alkalmazás, Szaktudás Kiadó Ház, Budapest, 2005

[12] **KURUTZNÉ, K. M., SCHARLE, P.,** A végeselem-módszer egyszerű elemei és

elemcsaládjai, Műszaki Könyvkiadó, Budapest, 1985
[13] **PÁCZELT, I., HERPAI, B.,** A végeselem-módszer alkalmazása rúdszerkezetekre, Műszaki Könyvkiadó, Budapest, 1987
[14] **PÁCZELT, I.,** A végeselem-módszer lineáris rúdelemei, Miskolci Egyetemi Kiadó, Miskolc, 1993
[15] **BIRAN, A., BREINER, M.,** Matlab for Engineers, Addison-Wesley Publishing Company, 1995