Presentation of the FEM analysis methodology using MathCAD software

Juraj Králik, Jozef Bočkaj, Juraj Králik Jr.
Slovak University of Technology in Bratislava, Faculty of Civil Engineering
and Faculty of Architecture and Design
Vazovova 5, Bratislava 812 43
Slovakia

Received: August 12, 2022. Revised: September 15, 2023. Accepted: October 19, 2023. Published: December 14, 2023.

Abstract— This paper presents the authors' experience with teaching the finite element method (FEM) at a university using the MathCAD software. With the development of computational tools in the second half of the 20th century, there was also the development of computational methods aimed at algorithmizing engineering tasks based on FEM. This method is very effective for determining the forces and deformations of truss and frame structures under the influence of the external environment. Many processes in the automatic design system take place as if in a "black box", and the process of verifying the achieved results becomes the most important stage in the design activity. Without knowing the theoretical foundations of FEM, physical and mathematical modelling, verification procedures and methods, the structure design cannot be safe and reliable. In this paper, we present one of the possibilities, how a student can familiarize himself with the theoretical foundations of FEM and calculation procedures using the MathCAD software. MathCAD can be used as a vehicle to create, organize, and assemble all the pieces of the engineering calculations. MathCAD is open software for using input data from other programs.

Keywords— FEM, MathCAD, Statics, Truss structures, Frames, Reliability, Safety, Control.

I. INTRODUCTION

THIS paper presents our experience with teaching FEM at the university. The study, [1], in 1943, was the first who propose the finite element method as it is today. In his final analysis, Courant proposed to use the principle of stationary potential energy and interpolation functions of triangular subregions using polynomials. It was proved that continuum mechanics problems (one-, two- and three-dimensional problems) can be solved by applying a potential energy minimum or equivalent functionals. Further development of FEM, [2], [3], [4], [5], encountered difficulties related to the solution of large-scale systems of simultaneous equations at that time, so the development of FEM remained on hold until the development of electronic automatic computers and the creation of highly efficient programming languages was established.

Currently, there are several world-renowned programs in the field of FEM (ABAQUS, ADINA, ANSYS, ALGOR, CivilFEM, MARC, MSC PATRAN, NASTRAN, NISA II, COSMOS/M, SAP2000 and others).

The basic problem of using FEM for the safety design of building structures without sufficient theoretical knowledge and practical experience from the individual stages of creating a calculation model and the use of a calculation method in terms of knowledge of physical properties and mathematical formulations of static problems is that engineers in practice make tragic mistakes when designing safety structures.

In the past, students solved practical examples with FEM applications using calculators and paper. This methodology was very successful and led students to learn from the formulation of tasks, methods of checking the individual phases of the calculation, as well as methods of solving them.

Based on our pedagogical experience, [6], [7], confirmed by the opinions of colleagues from abroad [8], [9], engineering students must know the theoretical foundations of FEM, as well as go through the solution of simple applications to understand the essence of the problems in the application of FEM in solving engineering problems.

During doctoral studies, students must be able to develop FEM by developing new types of elements, new material models, more accurate and efficient calculation methods, and complex solutions that consider the interaction of different structures.

MathCAD is perfectly suited for engineering calculations, [10], [11], [12]. This software represents the balance of exploration, communication, and collaboration of engineering calculations with other software Fig. 1.

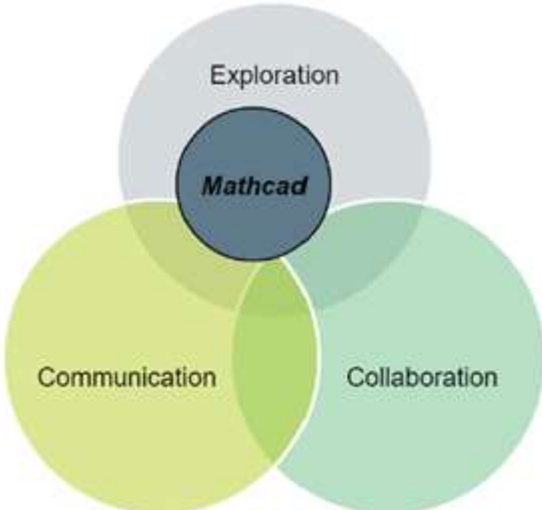

Fig. 1 MathCAD as the open software

The benefit of MathCAD is that your calculations are now electronic and can be archived, shared with coworkers, reused on other projects, and updated as variables change. Another advantage of MathCAD is that your results can be used in further calculations. If the variables for the first result change, then all the calculations based on that first result are also immediately updated. The use of the MathCAD system is of great importance in solving traditional engineering problems as well as mathematical programming problems. For that reason, there is a need to use modern information technologies in the educational process to generalize and improve traditional methods of solving problems in the field of engineering. This problem is reflected in this article.

The range of tasks performed using this is very wide, and their use in many ways helps students to work actively and smoothly, increasing the efficiency of the learning process and the quality of education. The peculiarity of this software is that they have the following capabilities, and their implementation is as follows:

1. perform digital calculations,
2. perform symbolic (analytical) calculations and changes,
3. create various graphs,
4. creation of documents using new multimedia tools, including hypertext and hypermedia links,
5. integration with other software tools.

MathCAD software offers us an effective tool for solving applications in FEM, [6]. The software allows matrix operations, as well as programming tools to define various mathematical operations in the creation of the stiffness matrix, and the load vector, as well as in the arrangement of the analysis results, including the graphical interpretation of the results of the deformations of the structure.

This article presents our experience, [5], [6], [7], in teaching FEM to doctoral students using the MathCAD software.

II. Theoretical Basis of FEM

The Finite Element Method (FEM), [2], is one of the most effective variational methods for solving continuum mechanics problems, as well as gases and liquids and other potential problems (electromagnetism, acoustics, heat, ...). Its essence is in the division of the construction, or of a continuous body on a set of finite elements, connected to each other at dividing nodes. Such a discrete system must satisfy the conditions of continuity and equilibrium at the dividing nodes.

The calculation process in FEM can be divided into five phases:

1. Discretization of the structure to a finite number of elements
2. Approximation of deformation or force quantities for each element separately,
3. Integration of finite elements into a whole while preserving the conditions of continuity of deformations,
4. Energy minimization - solution of conditional equations and determination of unknown nodal parameters,
5. Determination of unknown elements - calculation of internal forces on individual elements.

The finite element method is based on the variational principle of minimum potential energy or the theorem on the virtual work of forces on displacements at the nodes of a discretized body. The total virtual work of the forces on the given system is defined in the form:

$$\delta\Pi = \int_V \{\delta\varepsilon\}^T \{\sigma\} dV - \int_V \{\delta u\}^T \{b\} dV - \int_S \{\delta u\}^T \{p\} dS , \qquad (1)$$

where $\{\delta\varepsilon\}$ is the virtual strain vector, $\{\sigma\}$ is the stress vector, $\{\delta u\}$ is the virtual displacement vector, $\{b\}$ is the volume force vector, $\{p\}$ and is the vector of surface loads.

The displacement vector is approximated by polynomial functions of the form:

$$\{u\} = [\Phi]\{\alpha\} , \qquad (2)$$

where $[\Phi]$ is a polynomial matrix and $\{\alpha\}$ is a polynomial coefficient vector.

$$P_n(x) = \sum_{k=1}^{n+1} \alpha_k x^{k-1} . \qquad (3)$$

The vector of unknown parameters of the degrees of freedom (DOF) $\{r\}$ is determined in the nodes of the element depending on the selected polynomial after substituting the specific coordinates of the nodes of the element into the matrix of the polynomial in (2). Therefore,

$$\{r\} = [A]\{\alpha\} \;=>\; \{\alpha\} = \lfloor A^{-1} \rfloor \{r\} . \qquad (4)$$

The Table I presents the approximation functions of 1D elements.

TABLE I. APPROXIMATION FUNCTIONS OF 1D ELEMENTS

1D elements	Approximation
•——————•	$u = \alpha_1 + \alpha_2 x$
•———•———•	$u = \alpha_1 + \alpha_2 x + \alpha_3 x^2$

After substituting (3) into (2) we obtain the shape matrix of deformation parameters [*N*] in the form of

$$\{u\} = [\Phi]\{\alpha\} = [\Phi]\lfloor A^{-1} \rfloor\{r\} = [N]\{r\} . \quad (5)$$

On the finite element, we generally define a displacement vector depending on the vector of initial displacements and displacements from a given load in the form of

$$\{u\} = \{u_o\} + [N]\{r\} . \quad (6)$$

Then we express the strain vector $\{\varepsilon\}$ in the form of

$$\{\varepsilon\} = [\partial]^T \{u\} - \{\varepsilon_o\} = \{\varepsilon_u\} - \{\varepsilon_o\} + [\partial]^T [N]\{r\},$$

$$\{\varepsilon\} = \{\varepsilon_u\} - \{\varepsilon_o\} + [B]\{r\}, \quad (7)$$

where $\{\varepsilon_u\}$ is the strain vector from the initial displacements, and $[B]$ is the strain-displacement matrix. The initial strains $\{\varepsilon_o\}$ usually correspond to the effects of body loading by temperature, creep, or shrinkage of the structural material.

The stress vector $\{\sigma\}$ is obtained from the material equations,

$$\{\sigma\} = \{\sigma_0\} + [D]\{\varepsilon\} = \{\sigma_0\} + [D]\left([B]\{r\} + \{\varepsilon_u\} - \{\varepsilon_0\}\right), \quad (8)$$

where $\{\sigma_0\}$ is the initial stress vector.

After substituting relations, (5), (7), and (8), into equations (1), we get the total virtual work of the forces in the form of

$$\delta\pi = \{\delta r\}^T \Bigg(\int_V [B]^T [D][B] dV \{r\} + \int_V [B]^T \{\sigma_o\} dV + \int_V [B]^T [D]\{\varepsilon_u\} dV - \int_V [B]^T [D]\{\varepsilon_o\} dV - \int_V [N]^T \{b\} dV - \int_S [\tilde{N}]^T \{p\} dS) \Bigg). \quad (9)$$

Based on the virtual work theorem, the expression in relation (9), for the equilibrium force system acting on a body with volume *V* and area *S* must be equal to zero. Then, we have algebraic equations in the form of

$$\int_V [B]^T [D][B] dV \{r\} + \int_V [B]^T \{\sigma_o\} dV + \int_V [B]^T [D]\{\varepsilon_u\} dV - \int_V [B]^T [D]\{\varepsilon_o\} dV - \int_V [N]^T \{b\} dV - \int_S [\tilde{N}]^T \{p\} dS = \{0\}, \quad (10)$$

where the matrix $[\tilde{N}]$ is equal to the matrix $[N]$ only on the surface *S*.

Relations (10) correspond to the equilibrium forces in the nodes of the elements,

$$[K]\{r\} = \{F\} = \{F_o\} + \{F_\sigma\} + \{F_u\} + \{F_\varepsilon\} + \{F_b\} + \{F_p\}, \quad (11)$$

where $[K]$ is the stiffness matrix of the element, $\{r\}$ is a vector of nodal deformation parameters of the element, $\{F\}$ is the vector of generalized forces at the nodes, $\{F_o\}$ and is the vector of singular forces at the nodes. Therefore,

$$[K] = \int_V [B]^T [D][B] dV , \quad \{F_\sigma\} = -\int_V [B]^T \{\sigma_0\} dV ,$$

$$\{F_u\} = -\int_V [B]^T [D]\{\varepsilon_u\} dV , \ \{F_\varepsilon\} = \int_V [B]^T [D]\{\varepsilon_0\} dV ,$$

$$\{F_b\} = \int_V [N]^T \{b\} dV , \qquad \{F_p\} = \int_S [\tilde{N}]^T \{p\} dS . \quad (12)$$

The relations in (12) are defined in the local coordinate system. It is necessary to transform forces and displacements from the local to the global coordinate system.

We define vectors $\{r\}_{glob}$ and $\{F\}_{glob}$ in the global coordinate system as follows:

$$\{r\} = [T]\{r\}_{glob} \quad \text{and} \quad \{F\} = [T]\{F\}_{glob} . \quad (13)$$

By substituting relations (II.12) into (II.10) we get

$$[K][T]\{r\}_{glob} = [T]\{F\}_{glob} , \quad (14)$$

and after multiplying the equations from the left by the matrix, we have:

$$[T]^T [K][T]\{r\}_{glob} = [K]_{glob} \{r\}_{glob} = \{F\}_{glob} , \quad (15)$$

where $[T]^T [T] = [1]$ is in the orthogonal coordinate system.

The total vector of deformation parameters in the nodes is defined on the entire structure. Based on the continuity of the elements in the division nodes, we define the relationship between the vector $\{r^e\}_{glob}$ and $\{F^e\}_{glob}$ the element "*e*" and the total vector $\{r_{tot}\}$ and $\{F_{tot}\}$ as follows

$$\{r^e\}_{glob} = [L^e]\{r_{tot}\} \quad \text{and} \quad \{F^e\}_{glob} = [L^e]\{F_{tot}\} , \quad (16)$$

where the localization matrix $\lfloor L^e \rfloor$ is defined by values equal to 1 or 0 as follows

$$\{r^e\}_{glob} = [L^e]\{r_{tot}\} \quad \text{for} \quad L^e_{mi} = \begin{matrix} 1 & \forall r^e_m \equiv r^{tot}_i \\ 0 & \forall r^e_m \neq r^{tot}_i \end{matrix} . \quad (17)$$

If the structure consists of *n*-elements, then the total virtual work of the forces is equal to the algebraic sum of the virtual work of the forces on individual elements,

$$\delta\pi = \sum_{e=1}^{n} \delta\pi^e = \{\delta r_{tot}\}^T \left(\sum_{e=1}^{n} [L^e]^T [K^e]_{glob} [L^e]\{r_{tot}\} - \sum_{e=1}^{n} [L^e]\{F\}_{tot} \right) = 0. \quad (18)$$

From there, we get the overall equations for the state of the balance of forces on the given structure,

$$[K_{tot}]\{r_{tot}\} = \{F_{tot}\} , \quad (19)$$

where the elements of the overall stiffness matrix of the structure are obtained from the stiffness matrices of the individual elements

$$K^{tot}_{ij} = \sum_{e=1}^{n} K^e_{km} L^e_{ki} L^e_{mj} \quad \text{for} \quad L^e_{ki} = \begin{matrix} 1 & \forall r^e_k \equiv r^{tot}_i \\ 0 & \forall r^e_k \neq r^{tot}_i \end{matrix} ,$$

$$\text{and} \ L^e_{mj} = \begin{matrix} 1 & \forall r^e_m \equiv r^{tot}_j \\ 0 & \forall r^e_m \neq r^{tot}_j \end{matrix} , \quad (20)$$

and the elements of the total vector of generalized forces in the form

$$F^{tot}_i = \sum_{e=1}^{n} F^e_k L^e_{ki} . \quad (21)$$

In the case of linear analysis of structures with elastic materials, the solution of the displacement degrees of freedom is based on the relation (19) in the form:

DOI: 10.46300/9109.2023.17.11 Volume 17, 2023

$$\{r_{tot}\} = [K_{tot}]^{1}\{F_{tot}\}. \quad (22)$$

Next, the vector $\{r^e\}$ on the element "*e*" is calculated from relations (16) and (13). Strain and stress vectors are calculated from relations (7) and (8).

III. Truss Structures

Truss constructions consist of straight rods connected to each other by joint connections. The structural element is stressed in tension-compression or buckling. For the case of tensile-compressive stress, we define the following fields as single-element vectors,

$$\{u\} = u, \quad \{\varepsilon\} = \varepsilon_x, \quad \{\sigma\} = \sigma_x. \quad (23)$$

The strain is defined as follows,

$$\varepsilon_x = \frac{\partial u}{\partial x} = u_{,x}. \quad (24)$$

The stress vector is defined as follows,

$$\{\sigma\} = [D].\{\varepsilon\}, \text{ for } [D] = E. \quad (25)$$

The displacement vector is approximated by a linear function,

$$\{u\} = u = \alpha_1 + \alpha_2 x = [\Phi]\{\alpha\}, \quad (26)$$

where $[\Phi] = [1 \quad x]$ and $\{\alpha\}^T = \{\alpha_1 \quad \alpha_2\}$.

The vector of DOF's parameters $\{r\} = \{u_i, u_j\}^T$ is defined,

$$\{r\} = \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{bmatrix} 1 & x_i \\ 1 & x_j \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = [A]\{\alpha\}, \quad (27)$$

The polynomial coefficient vector is

$$\{\alpha\} = \begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{1}{(x_j - x_i)} \begin{bmatrix} x_j & -x_i \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \frac{1}{l} \begin{bmatrix} l & 0 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \left[A^{-1}\right] \quad (28)$$

for $x_i = 0$ and $x_j = l$ in the interval $0 \le x \le l$.

After substituting (28) for (26) we have

$$\{u\} = [\Phi]\{\alpha\} = [\Phi]\lfloor A^{-1} \rfloor\{r\} = [N]\{r\}, \quad (29)$$

where $[N] = \lfloor N_i, \quad N_j \rfloor$, $N_i = \frac{l-x}{l} = \frac{1-\xi}{2}$, $N_j = \frac{x}{l} = \frac{1+\xi}{2}$.

and $N_i + N_j = 1$ on the element in Fig. 2.

The deformation matrix [B] is obtained from (7)

$$[B] = [-1/l; 1/l]^T. \quad (30)$$

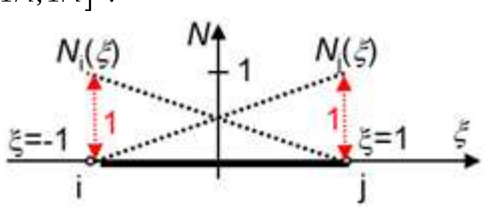

Fig. 2 Shape functions on an element

The stiffness matrix of the element is given by (25), (30) and (12)

$$[K] = \frac{ES}{l} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (31)$$

The transformation of a local vector $\{r\}$ to a global vector $\{r\}_{glob}$ is defined in (13). Therefore, we have

- 2D structure

$$[T] = \begin{bmatrix} \cos(\alpha) & \sin(\alpha) & 0 & 0 \\ 0 & 0 & \cos(\alpha) & \sin(\alpha) \end{bmatrix} \text{ for 2D} \quad (32)$$

$$\cos(\alpha) = \frac{x_j - x_i}{l}, \qquad \sin(\alpha) = \frac{y_j - y_i}{l}.$$

- 3D structure

$$[T] = \begin{bmatrix} \cos(\alpha) & \cos(\beta) & \cos(\gamma) & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos(\alpha) & \cos(\beta) & \cos(\gamma) \end{bmatrix}$$

$$\cos(\alpha) = \frac{x_j - x_i}{l}, \ \cos(\beta) = \frac{y_j - y_i}{l}, \ \cos(\gamma) = \frac{z_j - z_i}{l},$$

where the angles *α, β, γ* are between the local and global axes $\bar{x}x$, $\bar{y}y$, $\bar{z}z$.

IV. Frame structures

Frame structures are discretized by beam elements. These elements are defined based on Euler-Bernoulli or Timoshenko's theory. Euler-Bernoulli's hypothesis assumes the preservation of the angle of the normal section after beam deformation. Timoshenko's theory assumes the occurrence of shear deformations and thus the normal section on the axis of the bar does not maintain its normal direction but rotates.

In the case of 2D stressing of the beam for tension-compression, bending, and shear, we have:

$$\{u\} = \{u, v, \theta_z\}^T, \ \{\varepsilon\} = \{\varepsilon_x, \gamma_{xy}\}^T, \ \{\sigma\} = \{\sigma_x, \tau_{xy}\}^T \quad (33)$$

Based on the Euler-Bernoulli hypothesis (the normal to the axis of the beam before deformation is normal after deformation), we have

$$u(x,y) = u_s(x) - y.\theta_z(x), \ v(x,y) = v_s(x), \quad (34)$$

where $u_s(x), v_s(x)$ are the displacement of the beam axis, $\theta_z(x)$ is the rotation of the beam section. Next, we have

$$\theta_z(x) = \frac{dv_s(x)}{dx}. \quad (35)$$

In the case of the Timoshenko hypothesis, the pseudo rotation of the normal $\bar{\gamma}_y(x)$ is defined from the strain-displacement relation as follows:

$$\gamma_{xy}(x,y) = \frac{\partial u(x,y)}{\partial y} + \frac{\partial v(x,y)}{\partial x}. \quad (36)$$

After we substitute the (2) to (4), we have:

$$\theta_z(x) = \frac{dv_s(x)}{dx} - \bar{\gamma}_y(x), \quad (37)$$

where $\bar{\gamma}_y(x)$ is the constant shear strain in the *z* coordinate.

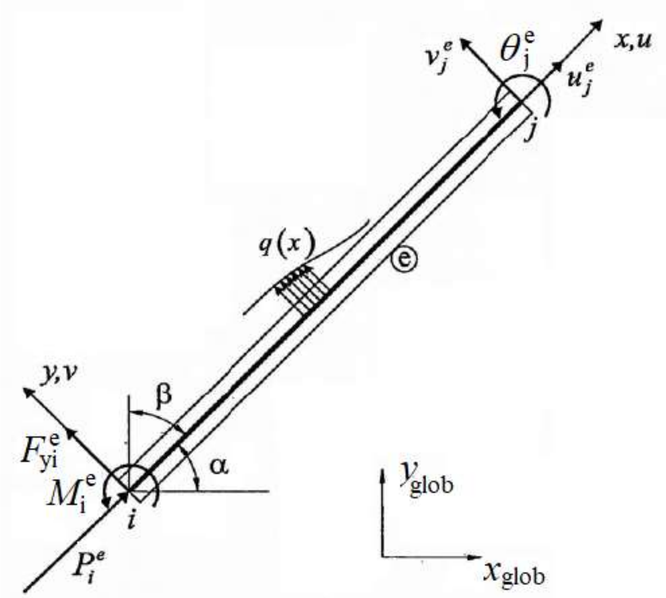

Fig. 3 2D - Beam element

The strain $\varepsilon_x(x,y)$ is defined from the equation (2)

$$\varepsilon_x(x,y)=\frac{du_s(x)}{dx}-y.\frac{d\theta_z(x)}{dx}. \tag{38}$$

The stress-strain relations are defined in eq. (8), where the matrix [*D*] is

$$[D]=\begin{bmatrix} E & 0 \\ 0 & G \end{bmatrix}. \tag{39}$$

The initial stress due to the temperature *T* (x, y) and the initial temperature T_o is defined as follows

$$\{\varepsilon_o\}=\{\alpha_T(T-T_o), \quad 0\}^T. \tag{40}$$

The internal forces on the beam *N*(*x*), *M*(*x*) and *V*(*x*) are defined as follows:

$$N(x)=\int_S \sigma_x(x).dS, \qquad M_z(x)=\int_S y.\sigma_x(x).dS,$$
$$V_y(x)=\int_S \tau_{xy}(x).dS \tag{41}$$

Considering eq. (8), (7) and (9), we have

$$N(x)=ES.\frac{du_s}{dx}-N_t, \qquad M_z(x)=EI_z.\frac{d\theta_z}{dx}-M_t,$$
$$V(x)=\kappa GS\left(\frac{dv_s}{dx}-\theta_z\right), \tag{42}$$

where *S* is the cross-section of the beam, I_z is the moment of inertia of the cross-section, and κ is the shear coefficient of the cross-section.

The internal forces of the beam $N_t(x)$, $M_t(x)$ caused by the influence of temperature are defined as follows:

$$N_t=ES\alpha_t.\left(\frac{T_d+T_h}{2}-T_o\right), \; M_t=EI_z\alpha_t.\frac{T_d-T_h}{h}. \tag{43}$$

The local vector of DOF 's $\{r\}$ and forces $\{F\}$ on the beam element in Fig. 3 is as follows:

$$\{r\}=\{u_i, \quad v_i, \quad \theta_i, \quad u_j, \quad v_j, \quad \theta_j\}^T,$$
$$\{F\}=\{F_{xi}, \quad F_{yi}, \quad M_{zi}, \quad F_{xj}, \quad F_{yj}, \quad M_{zj}\}^T. \tag{44}$$

The displacements *u(x)*, *v(x)*, and rotation $\theta_z(x)$ on the beam after the substitution of $x=(1+\xi)l/2$ are as follows:

$$u(\xi)=N_1u_i+N_2u_j, \; \theta(\xi)=N_1\theta_i+N_2\theta_j,$$
$$v(\xi)=N_3v_i+N_4\theta_i+N_5v_j+N_6\theta_j, \tag{45}$$

where the shape functions are as follows

$$N_1=(1-\xi)/2, \quad N_2=(1+\xi)/2, \quad N_3=1/2-\xi(3-\xi^2)/4,$$
$$N_4=L(1-\xi^2).(1-\xi)/8, \quad N_5=1/2+\xi(3-\xi^2)/4,$$
$$N_6=-L(1-\xi^2).(1+\xi)/8.$$

Then, the beam element shape matrix is as follows:

$$[N]=\begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 \\ 0 & N_3 & N_4 & 0 & N_5 & N_6 \\ 0 & 0 & N_1 & 0 & 0 & N_2 \end{bmatrix}. \tag{46}$$

The virtual work of the internal forces on deformations is

$$\delta A=\int_V \{\delta\varepsilon\}^T\{\sigma\}\,dV=\int_0^l\left(\int_S \{\delta\varepsilon\}^T\{\sigma\}\,dS\right)dx$$
$$=\int_0^l \{\delta q\}^T\{Q\}\,dx=\int_{-1}^{1}\{\delta q\}^T\{Q\}(l/2)\,d\xi, \tag{47}$$

where $\{q\}=\{u_{,x},\theta_{,x},v_{,x}-\theta\}^T$ is the deformation vector and $\{Q\}=\{N_x,M_z,V_y\}^T$ is the vector of the internal forces.

Then we have

$$\{Q\}=[\tilde{D}]\{q\}, \text{ where } [\tilde{D}]=\begin{bmatrix} ES & 0 & 0 \\ 0 & EI_z & 0 \\ 0 & 0 & \kappa GS \end{bmatrix} \tag{48}$$

The deformation vector $\{q\}$ is defined from the eq. (36) and (38) as follows:

$$\{q\}=[\tilde{\partial}]^T\{u\}=[\tilde{\partial}]^T[N]\{r\}=[\tilde{B}]\{r\}, \tag{49}$$

where the matrix $[\tilde{B}]$ is as follows

$$[\tilde{B}]=\begin{bmatrix} \frac{-1}{l} & 0 & 0 & \frac{1}{l} & 0 & 0 \\ 0 & 0 & \frac{-1}{l} & 0 & 0 & \frac{1}{l} \\ 0 & -\frac{3}{2l}(1-\xi^2) & \frac{3\xi^2-2\xi-1}{8} & 0 & \frac{3}{2l}(1-\xi^2) & \frac{-(3\xi^2-2\xi-1)}{8} \end{bmatrix}.$$

After substituting the eq. (49) to (47), we have

$$\delta A = \int_{-1}^{1}\{\delta q\}^T\{Q\}(l/2)d\xi = \{\delta r\}^T\left(\int_{-1}^{1}\left[\tilde{B}\right]^T\left[\tilde{D}\right]\left[\tilde{B}\right](l/2)d\xi\right)\{r\}$$
$$= \{\delta r\}^T[K]\{r\} \tag{50}$$

where $[K]$ is the stiffness matrix of the beam element,

$$[K] = k_o\begin{bmatrix} \frac{S(1+2\chi)}{2I_z} & 0 & 0 & -\frac{S(1+2\chi)}{2I_z} & 0 & 0 \\ & \frac{6}{l^2} & -\frac{3}{l} & 0 & -\frac{6}{l^2} & -\frac{3}{l} \\ & & (2+\chi) & 0 & \frac{3}{l} & (1-\chi) \\ & & & \frac{S(1+2\chi)}{2I_z} & 0 & 0 \\ & sym. & & & \frac{6}{l^2} & \frac{3}{l} \\ & & & & & (2+\chi) \end{bmatrix},$$

$$k_o = \frac{2EI_z}{l(1+2\chi)}, \quad \chi = 6EI_z/\kappa GSl^2 . \tag{51}$$

The vector of generalized forces from the continuous pressure on the element $p_y(x)$ is expressed from the relations (12) in the form of

$$\{F_p\} = \int_0^L [N]^T\{p\}dx = \int_0^L [N]^T\begin{Bmatrix} p_x \\ p_y \\ m_z \end{Bmatrix}dx = \int_0^L [N]^T\begin{Bmatrix} 0 \\ p_y \\ 0 \end{Bmatrix}dx , \tag{52}$$

$$\{F_p\} = \int_0^l \begin{bmatrix} 0 \\ 1-3x^2/L^2+2x^3/L^3 \\ x-2x^2/L+x^3/L^2 \\ 0 \\ 3x^2/L^2-2x^3/L^3 \\ -x^2/L+x^3/L^2 \end{bmatrix} p_y dx .$$

In the case of constant pressure $p_y(x) = p_y$ = const. we have

$$\{F_p\} = p_y\left[0 \quad L/2 \quad L^2/12 \quad 0 \quad L/2 \quad -L^2/12\right]^T . \tag{53}$$

The reaction vector is defined in the form:

$$\{R\} = \{R_{xi} \quad R_{yi} \quad M_{zi} \quad R_{xj} \quad R_{yj} \quad M_{zj}\}^T ,$$

with support, we get from the condition equations

$$\{R\} = [K]\{r\} - \{F\} . \tag{54}$$

The vector of internal forces on the beam $i - j$ is defined as follows:

$$\{p\} = \{N_i \quad V_i \quad M_i \quad N_j \quad V_j \quad M_j\}^T ,$$

$$\{p\} = [S]\{R\}, \text{ and } [S] = \begin{bmatrix} -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}, \tag{55}$$

where $[S]$ is the static matrix on the beam element in the local coordinate system.

The vector of DOF parameters and generalized forces in the global coordinate system in the case of a frame structure in the *XY* plane is of the form

$$\{r\} = [T].\{r\}_{glob}, \quad \{F\} = [T].\{F\}_{glob} \quad \text{or}$$
$$\{r\}_{glob} = [T]^T.\{r\}, \ \{F\}_{glob} = [T]^T.\{F\}, \tag{56}$$

where $[T] = \begin{bmatrix} [T_i] & [0] \\ [0] & [T_j] \end{bmatrix}$, $[T_i] = \begin{bmatrix} \cos(\alpha_i) & \sin(\alpha_i) & 0 \\ -\sin(\alpha_i) & \cos(\alpha_i) & 0 \\ 0 & 0 & 1 \end{bmatrix}$,

the angle α_i (or α_j) is the angle between the support at node *i* and the *x*-axis of the beam.

In the case of 3D straining of the beam, we have tension-compression, bending and shear,

$$\{u\} = \{u, v, w, \theta_x, \theta_y, \theta_z\}^T , \tag{57}$$
$$\{\varepsilon\} = \{\varepsilon_x, \gamma_{xy}\ \gamma_{yz}\ \gamma_{zx}\}^T, \quad \{\sigma\} = \{\sigma_x, \tau_{xy}, \tau_{yz}, \tau_{zx}\}^T .$$

Based on the hypothesis of preservation of cross-section planarity and rotation of the pseudo-normal, we have

$$u(x,y,z) = u_s(x) - y.\theta_z(x) + z.\theta_y(x), \tag{58}$$

where $u_s(x)$ are the displacements of the node on the axis of the beam and $\theta_y(x)$, $\theta_z(x)$, are the rotations of the beam cross-section around the Y and Z axes, while the rotation of the cross-section can be considered according to Euler-Bernoulli (the normal will retain its orientation after deformation) or according to Timoshenko's hypothesis (the rotation of the pseudo-normal consists of the rotation of the normal and also the skew angle due to shear deformation) similarly to the previous section.

The vector of DOF‘s parameters and the vector of generalized forces on an element in a local coordinate system are defined in the form

$$\{r\} = \{u_i \quad v_i \quad w_i \quad \theta_{xi} \quad \theta_{yi} \quad \theta_{zi} \quad u_j \quad v_j \quad w_j \quad \theta_{xj} \quad \theta_{yj} \quad \theta_{zj}\}^T,$$
$$\{F\} = \{F_i \quad F_j\}^T , \tag{59}$$
$$\{F_i\} = \{F_{xi} \quad F_{yi} \quad F_{zi} \quad M_{xi} \quad M_{yi} \quad M_{zi}\}^T ,$$
$$\{F_i\} = \{F_{xi} \quad F_{yi} \quad F_{zi} \quad M_{xi} \quad M_{yi} \quad M_{zi}\}^T .$$

V. Solving Truss Structure by Using FEM

Example V.1 - Calculate the value of displacements and internal forces under the action of singular forces using the FE method in the MathCAD software on the truss structure (Fig. 4).

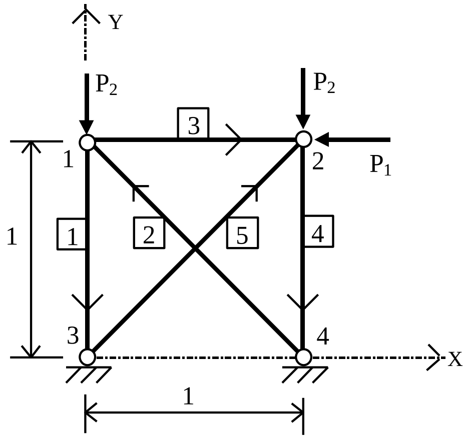

Fig. 4 Scheme of the truss structure

A) Input data:

The structural geometry, loads, boundary conditions, and identification of nodes and elements are shown in Fig. 4. Geometrical and physical input data are defined in further procedures:

- Geometric characteristics of cross sections:

$$A^T := (500\ 250\ 500\ 500\ 250)\cdot mm^2. \tag{60}$$

- Material characteristics:

$$E := 210.10^6.kPa. \tag{61}$$

- Load defined in nodes on the global x and y axis:

$$L_{Fu} := \begin{pmatrix} 1 \\ 2 \end{pmatrix} \quad F_u := \begin{pmatrix} 0 & -88 \\ -44 & -88 \end{pmatrix}.kN. \tag{62}$$

- Node coordinate system:

$$\alpha_u := (30.\deg), \qquad n_{rot} := rows(\alpha_u). \tag{63}$$

- Boundary conditions:

(0–the shift is bound, 1-the shift is free)

Bounds are defined in the form: Node – X bound– Y bound

$$Bounds := \begin{pmatrix} 3 & 0 & 0 \\ 4 & 0 & 0 \end{pmatrix} \tag{64}$$

- Geometry description:

Node coordinates are defined in the following matrix:

$$Nodes^T := \begin{pmatrix} 0 & 1 & 0 & 1 \\ 1 & 1 & 0 & 0 \end{pmatrix}\cdot m \tag{65}$$

The numbering of nodes by elements is defined in the localization matrix:

$$Local^T := \begin{pmatrix} 1 & 4 & 1 & 2 & 3 \\ 3 & 1 & 2 & 4 & 2 \end{pmatrix} \tag{66}$$

- The number of structural nodes can be determined as follows:

$$n_{nodes} := rows(Nodes) \tag{67}$$

- The number of structural elements can be determined as follows:

$$n_{elem} := rows(Local) \tag{68}$$

- The number of nodes on an element is defined as follows:

$$nn_{pe} := 2 \tag{69}$$

- The number of degrees of freedom (DOF) in nodes is defined as follows:

$$ndof_{pn} := 2 \tag{70}$$

- The number of DOF on an element is defined as follows:

$$ndof_{pe} := nn_{pe}.ndof_{pn} \tag{71}$$

- The element length is calculated as follows:

$$i := 1...n_{elem} \tag{72}$$

$$L_i := \left(\left(Nodes_{Local_{i,2},1} - Nodes_{Local_{i,1},1} \right)^2 + \right.$$

$$\left. + \left(Nodes_{Local_{i,2},2} - Nodes_{Local_{i,1},2} \right)^2 \right)^{\frac{1}{2}}$$

- The orientation of the elements is defined as follows:

$$c_i := \frac{Nodes_{Local_{i,2},1} - Nodes_{Local_{i,1},1}}{L_i} \tag{73}$$

$$s_i := \frac{Nodes_{Local_{i,2},2} - Nodes_{Local_{i,1},2}}{L_i}. \tag{74}$$

B) Definition of procedures for calculating the numerical code of nodes and elements

The code number of nodes and elements is calculated using MathCAD program procedures.

- Procedures for calculating the numerical code of nodes:

$$Codenod(Bounds, Nodes) :=$$

```
n ← 0
for i ∈ 1..rows(Nodes)
  for j ∈ 1..cols(Nodes)
    n ← n+1
    A_{i,j} ← n
    for k ∈ 1..rows(Bounds)
      n ← n−1 if (Bounds_{k,1} = i)·(Bounds_{k,j+1} = 0)
      A_{i,j} ← 0 if (Bounds_{k,1} = i)·(Bounds_{k,j+1} = 0)
A
```

(75)

- Procedures for calculating element code numbers:

$$Codeele(Local, Code_{Nod}) :=$$

```
for i ∈ 1..rows(Local)
 for j ∈ 1..cols(Local)
  for k ∈ 1..cols(Code_{Nod})
    l ← (j−1)·cols(Cede_{Nod})+k
    A_{i,l} ← Code_{Nod_{(Local_{i,j}),k}}
A
```

(76)

The global force vector is calculated using the code number vector.

$$
\begin{aligned}
&Fglob(F_u, L_u, Code) :=\\
&\quad nel \leftarrow cols(Code)\\
&\quad for\ \ i \in 1..\max(Code)\\
&\qquad A_i \leftarrow 0\\
&\quad for\ \ i \in 1..rows(Code)\\
&\qquad for\ \ j \in 1..cols(Code)\\
&\qquad\quad for\ \ k \in 1..rows(L_u)\\
&\qquad\qquad A_{Code_{i,j}} \leftarrow A_{Code_{i,j}} + F_{u_{k,j}}\ \ if\ \ (Code_{i,j}) \neq 0 \wedge L_{u_k} = i\\
&\quad A
\end{aligned} \tag{77}
$$

- Procedures for calculating the total stiffness matrix of the structure using the numerical code of the elements:

$$
\begin{aligned}
&Kglob(KT, Code) :=\\
&\quad nel \leftarrow cols(Code)\\
&\quad for\ i \in 1..\max(Code)\\
&\qquad for\ \ j \in 1..\max(Code)\\
&\qquad\quad A_{i,j} \leftarrow 0\\
&\quad for\ i \in 1..rows(Code)\\
&\qquad for\ \ j \in 1..cols(Code)\\
&\qquad\quad for\ \ k \in 1..cols(Code)\\
&\qquad\qquad A_{Code_{i,j},Code_{i,k}} \leftarrow A_{Code_{i,j},Code_{i,k}} + KT_{j,(i-1)\cdot ndof_{pe}+k}\ \ /\\
&\qquad\qquad\quad /\ if\ \ (Code_{i,j} \cdot Code_{i,k}) \neq 0
\end{aligned} \tag{78}
$$

- Procedure of decomposition of the vector of deformation parameters for elements:

$$
\begin{aligned}
&r\_elem(r_{glob}, Code) :=\\
&\quad for\ i \in 1..rows(Code)\\
&\qquad for\ \ j \in 1..cols(Code)\\
&\qquad\quad A_{j,i} \leftarrow 0\\
&\quad for\ i \in 1..rows(Code)\\
&\qquad for\ \ j \in 1..cols(Code)\\
&\qquad\quad A_{j,i} \leftarrow r_{glob_{Code_{i,j}}}\ \ if\ \ (Code_{i,j}) \neq 0\\
&\quad A
\end{aligned} \tag{79}
$$

C) Calculation of the local and global stiffness of the matrix

- The element stiffness matrix in the local coordinate system is equal to

$$
ko_i := E \cdot \frac{A_i}{L_i} \qquad Kel(i) := ko_i \cdot \begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix}. \tag{80}
$$

- The element transformation matrix is defined as follows:

$$
T(i) := \begin{pmatrix} c_i & s_i & 0 & 0 \\ 0 & 0 & c_i & s_i \end{pmatrix}. \tag{81}
$$

In case of a rotated support, we define the node coordinate system in the given node α_u. There are procedures for calculating transformation relations.

$$
\begin{aligned}
&Tu\cos(c, Local, \alpha) :=\\
&\quad n_{rot} \leftarrow rows(\alpha)\\
&\quad for\ i \in 1..rows(Local)\\
&\qquad for\ \ j \in 1..cols(Local)\\
&\qquad\quad A_{i,j} \leftarrow c_i\\
&\qquad\quad for\ k \in 1..n_{rot}\\
&\qquad\qquad A_{i,j} \leftarrow \cos(a\cos(c_i) - \alpha_{k,2})\ if\ \ /\\
&\qquad\qquad\quad /\ \ \alpha_{k,1} = Local_{i,j} \wedge \alpha_{k,2} \neq 0\\
&\quad A
\end{aligned} \tag{82}
$$

$$
\begin{aligned}
&Tu\sin(s, Local, \alpha) :=\\
&\quad n_{rot} \leftarrow rows(\alpha)\\
&\quad for\ i \in 1..rows(Local)\\
&\qquad for\ \ j \in 1..cols(Local)\\
&\qquad\quad A_{i,j} \leftarrow s_i\\
&\qquad\quad for\ k \in 1..n_{rot}\\
&\qquad\qquad A_{i,j} \leftarrow \sin(a\sin(s_i) - \alpha_{k,2})\ if\ \ /\\
&\qquad\qquad\quad /\ \ \alpha_{k,1} = Local_{i,j} \wedge \alpha_{k,2} \neq 0\\
&\quad A
\end{aligned} \tag{83}
$$

- The oriented cosine of an angle is defined as follows:

$$
Cu := Tu\cos(c, Local, \alpha_u). \tag{84}
$$

- The oriented sinus of an angle is defined as follows:

$$
Su := Tu\sin(s, Local, \alpha_u). \tag{85}
$$

$$
T(i) := \begin{pmatrix} Cu_{i,1} & Su_{i,1} & 0 & 0 \\ 0 & 0 & Cu_{i,2} & Su_{i,2} \end{pmatrix}. \tag{86}
$$

- The element stiffness matrix in the global coordinate system is calculated from the local stiffness matrix and transformation matrix:

$$
K_{elg}(i) := T(i)^T . K_{el}(i) . T(i). \tag{87}
$$

- The element load vector in the global coordinate system is calculated from the vector in the local coordinate system:

$$
L_{Fu} := \begin{pmatrix} 1 \\ 2 \end{pmatrix} \quad F_u := \begin{pmatrix} 0 & -88 \\ -44 & -88 \end{pmatrix} . kN, \tag{88}
$$

$$
F_{glob} := Fglob(F_u, L_{F_u}, Code_{Nod}). \tag{89}
$$

- The boundary conditions are defined in the following matrix:

$$
Bounds = \begin{pmatrix} 3 & 0 & 0 \\ 4 & 0 & 0 \end{pmatrix}. \tag{90}
$$

- Structure node code numbers are calculated using the following procedure:

$$
Code_{nod} := Codenod(Bounds, Nodes), \tag{91}
$$

$$
Code_{nod}^{\ T} = \begin{pmatrix} 1 & 3 & 0 & 0 \\ 2 & 4 & 0 & 0 \end{pmatrix}. \tag{92}
$$

- Structure element code numbers are calculated using the following procedure:

$$Code_{ele} := Codele(Local, Code_{Nod}), \quad (93)$$

$$Code_{ele}^{\ T} := \begin{pmatrix} 1 & 0 & 1 & 3 & 0 \\ 2 & 0 & 2 & 4 & 0 \\ 0 & 1 & 3 & 0 & 3 \\ 0 & 2 & 4 & 0 & 4 \end{pmatrix}. \quad (94)$$

- The maximum number of unknown parameters is calculated from the matrix of code numbers:

$$n_{tot} := \max(Code_{ele}), \quad n_{tot} = 4. \quad (95)$$

- The global element stiffness matrices are inserted into one KT-matrix:

$$q := 1..ndof, \quad (96)$$

$$KT^{\langle q+ndof_{pe}.(i1)\rangle} := K_{elg}(i)^{\langle q\rangle} \quad (97)$$

- The global structure stiffness matrix is calculated from the following relations using the matrix of code numbers of the Structure:

$$K_{glob} := Kglob(KT, Code_{ele}), \quad (98)$$

$$K_{glob} = \begin{pmatrix} 123.6 & -18.6 & -105 & 0 \\ -18.6 & 123.6 & 0 & 0 \\ -105 & 0 & 123.6 & 18.6 \\ 0 & 0 & 18.6 & 123.6 \end{pmatrix}.MN.m^{-1}. \quad (99)$$

- The unknown DOF can be calculated as follows:

$$r_{glob} := K_{glob}^{\ -1}.F_{glob}, \quad (100)$$

$$r_{glob} = (-1.36 \quad -0.92 \quad -1.43 \quad -0.5).mm. \quad (101)$$

- Calculation of displacement vectors in elements in LCS:
 Displacement vectors in elements are defined as follows:

$$r_{tot} := r\_elem(r_{glob}, Code_{ele}), \quad (102)$$

$$r_{tot} = \begin{pmatrix} -1.356 & 0 & -1.356 & -1.434 & 0 \\ -0.916 & 0 & -0.916 & -0.497 & 0 \\ 0 & -1.356 & -1.434 & 0 & -1.434 \\ 0 & -0.916 & -0.497 & 0 & -0.497 \end{pmatrix}.mm. \quad (103)$$

The vectors of degrees of freedom of the system (DOF) by elements are defined as follows:)

$$r_{ele}^{\langle i\rangle} := T(i).r_{tot}, \quad (104)$$

$$r_{ele} = \begin{pmatrix} 0.916 & 0 & -1.356 & 0.497 & 0 \\ 0 & 0.311 & -1.434 & 0 & -1.365 \end{pmatrix} \cdot mm. \quad (105)$$

The element vectors of forces in the LCS are calculated as follows:

$$F_{ele}^{\langle i\rangle} := K_{el}(i) \cdot r_{ele}^{\langle i\rangle}, \quad (106)$$

$$F_{ele} = \begin{pmatrix} 96.2 & -11.6 & 8.2 & 52.2 & 50.7 \\ -96.2 & 11.6 & -8.2 & -52.2 & -50.7 \end{pmatrix} \cdot kN. \quad (107)$$

- Graphic control of the truss deformations

To display the deformed shape of the structure, it is necessary to calculate the relative limits of the structure.

We determine the maximum displacement value as follows:

$$m_r := max(max(r_{tot}), |min(r_{tot})|),$$

$$m_r = 1.434x10^{-3} m. \quad (108)$$

The interpretation scale is:

$$p := \left(\frac{\max(L)}{m_r}.m_k\right).m^{-1}, \quad p = 98.644m^{-1}. \quad (109)$$

The geometry of the deformed structure is presented in Fig. 5.

The geometry of the structure deformation shape is defined as follows:

$$x_{plot_{i,1}} := x_{n_{Lokal_{i,1}}} + p \cdot r_{tot_{1,i}}$$

$$y_{plot_{i,1}} := y_{n_{Lokal_{i,1}}} + p \cdot r_{tot_{2,i}}$$

$$x_{plot_{i,2}} := x_{n_{Lokal_{i,2}}} + p \cdot r_{tot_{3,i}}$$

$$y_{plot_{i,2}} := y_{n_{Lokal_{i,2}}} + p \cdot r_{tot_{4,i}}$$

$$x_{plot_{i,3}} := x_{n_{Lokal_{i,1}}} + p \cdot r_{tot_{1,i}}$$

$$y_{plot_{i,3}} := y_{n_{Lokal_{i,1}}} + p \cdot r_{tot_{2,i}}$$

$$i := 1..n_{elem} \qquad j := 1..cols(x_{plot}) \qquad k := 1..n_{node} \quad (110)$$

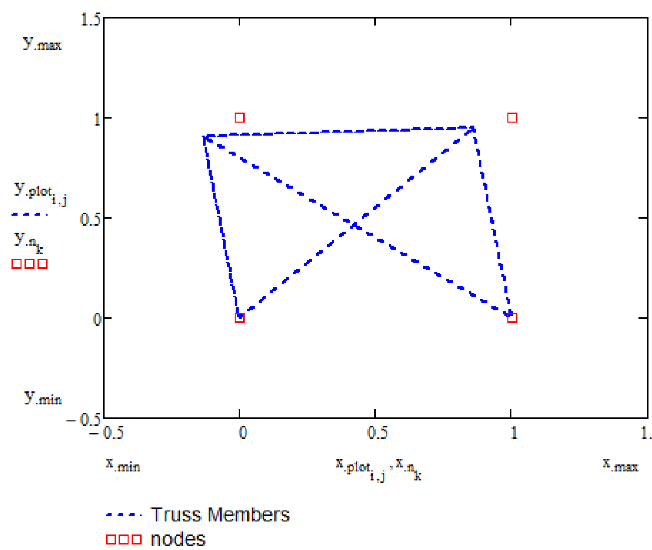

Fig. 5 Original and deformed structure of the truss

VI. Frame Solution Using FEM

Example VI.1 - On the frame structure (Fig. 6), calculate the value of displacements and internal forces under the action of singular forces using the FE method in the MathCAD software. The numbers of nodes and elements, as well as the orientation of the local x-axis, are marked in the Fig. 6.

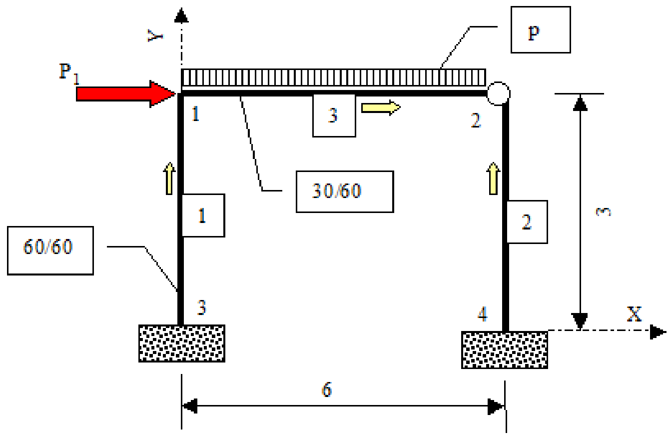

Fig. 6 Static scheme of the frame with loads

A) Input data:

The structural geometry, loads, boundary conditions, and identification of nodes and elements are shown in Fig. 6. Geometrical and physical input data are defined in further procedures:

- The geometrical characteristics of the cross-sections are defined for a rectangular cross-section using *b* and *h*:

$$b^T := (0.6 \quad 0.6 \quad 0.3).m\,, \tag{111}$$

$$h^T := (0.6 \quad 0.6 \quad 0.6).m\,. \tag{112}$$

The number of elements is calculated as follows:

$$n_{elem} := rows(b), \qquad i := 1..n_{elem}\,. \tag{113}$$

The cross-sectional area A and moment of inertia of the cross-section I are calculated as follows:

$$A_i := b_i \cdot h_i\,, \tag{114}$$

$$I_i := \frac{1}{12} \cdot b_i \cdot (h_i)^3\,. \tag{115}$$

- Material properties are defined as follows:

$$E := 30 \cdot GPa \quad G := 0.43 \cdot E \quad \kappa := \frac{5}{6}\,. \tag{116}$$

- Element load is defined as follows:

$$p_e := 10 \cdot kN \cdot m^{-1} \quad L_p := (3)\,, \tag{117}$$

$$n_p := rows(L_p)\,. \tag{118}$$

- Node load is defined as follows:

$$P_1 := 100 \cdot kN \quad L_{F_u} := (1) \quad F_u := \begin{pmatrix} \frac{-P_1}{kN} & 0 & 0 \end{pmatrix}, \tag{119}$$

$$n_F := rows(L_{F_u})\,. \tag{120}$$

The definition of the code coordinate system is the same as in (61). (0–the DOF is bound, 1-the DOF is free, 2-DOF is independent)

Kinematic boundaries are defined in the following form:
Bounds: Node – U – V – Φ,

$$Bounds := \begin{pmatrix} 2 & 1 & 1 & 2 \\ 3 & 0 & 0 & 0 \\ 4 & 0 & 0 & 0 \end{pmatrix}. \tag{121}$$

Static boundaries are defined as follows:
(0-the bound is free, 1-the bound is bound)

Release: element – u1, v1, φ1, u2, v2, φ2

$$Release : \begin{pmatrix} 2 & 0 & 0 & 0 & 0 & 1 \\ 3 & 0 & 0 & 0 & 0 & 2 \end{pmatrix}. \tag{122}$$

- Description of frame geometry:
The node coordinates are defined in the following matrix:

$$Nodes^T := \begin{pmatrix} 0 & 6 & 0 & 6 \\ 3 & 3 & 0 & 0 \end{pmatrix} \cdot m\,. \tag{123}$$

The location of the elements and nodes is defined by the following matrix:

$$Local^T := \begin{pmatrix} 3 & 1 & 4 \\ 1 & 2 & 2 \end{pmatrix}. \tag{124}$$

The number of structure nodes is the same as in (65). The number of elements is the same as in (66). The number of element nodes is the same as in (65).

The number of degrees of freedom in nodes is:

$$ndof_{pn} := 3 \tag{125}$$

The calculation of the beam length is the same as in (70), and the orientation of the beams is the same as in (71, 72).

B) Definition of procedures for calculating the numerical code of nodes and elements:

Code numbers by nodes are defined by the following procedure:

$$
\begin{aligned}
&Codenod(Bounds, Nodes) := \\
&\quad n \leftarrow 0 \\
&\quad \text{for } i \in 1..rows(Nodes) \\
&\qquad \text{for } j \in 1..cols(Nodes)-1 \\
&\qquad\quad n \leftarrow n+1 \\
&\qquad\quad A_{i,j} \leftarrow n \\
&\qquad\quad \text{for } k \in 1..rows(Bounds) \\
&\qquad\qquad n \leftarrow n-1 \text{ if } (Bounds_{k,1} = i)\cdot(Bounds_{k,j+1} = 0) \\
&\qquad\qquad A_{i,j} \leftarrow n \text{ if } (Bounds_{k,1} = i)\cdot(Bounds_{k,j+1} \geq 2) \\
&\qquad\qquad n \leftarrow n + Bounds_{k,j+1} - 1 \text{ if } (Bounds_{k,1} = i)\cdot \quad / \\
&\qquad\qquad\qquad / \quad \cdot(Bounds_{k,j+1} \geq 2) \\
&\qquad\qquad A_{i,j} \leftarrow 0 \text{ if } (Bounds_{k,1} = i)\cdot(Bounds_{k,j+1} = 0) \\
&\quad A
\end{aligned}
\tag{126}
$$

The definition of the procedure for calculating beam code numbers is the same as in (75). The definition of the procedure for calculating the numerical release codes is as follows:

$$kod\,\mathrm{mod}(\mathrm{Re}lease, Code) :=$$

$$\begin{array}{|l} \text{for } i \in 1..rows(Code) \\ \text{for } j \in 1..cols(Code) \\ A_{i,j} \leftarrow Code_{i,j} \\ \text{for } k \in 1..rows(\mathrm{Re}lease) \\ A_{i,j} \leftarrow Code_{i,j} + \mathrm{Re}lease_{k,j+1} - 1 \text{ if } (\mathrm{Re}lease_{k,1} = i)\cdot \\ \quad \cdot(\mathrm{Re}lease_{k,j+1} \succ 0) \end{array} \tag{127}$$

The procedure for calculating the total stiffness matrix is the same as in (77).

The global load vector at the nodes is calculated by the following procedure:

$$Fglob(A, F_u, L_u, Code) :=$$

$$\begin{array}{|l} \text{for } i \in 1..rows(Code) \\ \text{for } j \in 1..cols(Code) \\ \text{for } k \in 1..rows(L_u) \\ A_{Code_{i,j}} \leftarrow A_{Code_{i,j}} + F_{u_{k,j}} \text{ if } (Code_{i,j}) \neq 0 \wedge L_{u_k} = i \\ A \end{array} \tag{128}$$

The global load vector at the elements is calculated by the following procedure:

$$Pglob(A, F_e, Code) :=$$

$$\begin{array}{|l} \text{for } i \in 1..rows(Code) \\ \text{for } j \in 1..cols(Code) \\ \text{for } k \in 1..rows(F_e) \\ A_{Code_{i,j}} \leftarrow A_{Code_{i,j}} + F_{e_{j,i}} \text{ if } (Code_{i,j}) \neq 0 \\ A \end{array} \tag{129}$$

The decomposition of the vector of deformation parameters for the elements is the same as in (78).

The node code number matrix is defined as follows:

$$Code_{nod} := Codenod(Bounds, Nodes)$$

$$Code_{nod}{}^{T} = \begin{pmatrix} 1 & 4 & 0 & 0 \\ 2 & 5 & 0 & 0 \\ 3 & 6 & 0 & 0 \end{pmatrix} \tag{130}$$

The element code number matrix is defined as follows: (126)

$$Code_{ele} := Codeele(Local, Code_{nod}) \quad (127)$$

$$Code_{ele} = \begin{pmatrix} 0 & 0 & 0 & 1 & 2 & 3 \\ 1 & 2 & 3 & 4 & 5 & 6 \\ 0 & 0 & 0 & 4 & 5 & 6 \end{pmatrix} \quad (130)\ (131)$$

The maximum number of unknown parameters is as follows:

$$n_{tot} := \max(Code_{ele}) \quad n_{tot} = 7 \tag{132}$$

- Definition of the beam stiffness matrix in the local coordinate system (LCS):

The relative stiffness in bending and shear is defined by the following parameter:

$$\chi_i := \frac{6 \cdot E \cdot I_i}{\kappa \cdot G \cdot A_i \cdot L_i} \cdot \frac{1}{m} \tag{133}$$

$$A := A \cdot \frac{1}{m^2} \quad L := L \cdot \frac{1}{m} \quad I := I \cdot \frac{1}{m^4} \quad E := E \cdot \frac{1}{kPa} \tag{134}$$

(133)

134)

$$K_{ei}(i) := \frac{2 \cdot E \cdot I_i}{L_i \cdot (1 + 2 \cdot \chi_i)} \cdot \begin{bmatrix} \frac{A_i \cdot (1 + 2 \cdot \chi_i)}{2 \cdot I_i} & 0 & 0 & \frac{-A_i \cdot (1 + 2 \cdot \chi_i)}{2 \cdot I_i} & 0 & 0 \\ 0 & \frac{6}{(L_i)^2} & \frac{3}{L_i} & 0 & \frac{-6}{(L_i)^2} & \frac{3}{L_i} \\ 0 & \frac{3}{L_i} & (2 + \chi_i) & 0 & \frac{-3}{L_i} & (1 - \chi_i) \\ \frac{-A_i}{L_i} & 0 & 0 & \frac{A_i \cdot (1 + 2 \cdot \chi_i)}{2 \cdot I_i} & 0 & 0 \\ 0 & \frac{-6}{(L_i)^2} & \frac{3}{L_i} & 0 & \frac{6}{(L_i)^2} & \frac{-3}{L_i} \\ 0 & \frac{-3}{L_i} & (1 - \chi_i) & 0 & \frac{-3}{L_i} & (2 + \chi_i) \end{bmatrix} \tag{135}$$

(VII…29)

- The beam transformation matrix is defined in the following matrix:

$$T(i) := \begin{pmatrix} c_i & s_i & 0 & 0 & 0 & 0 \\ s_i & c_i & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & c_i & s_i & 0 \\ 0 & 0 & 0 & s_i & c_i & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{pmatrix}. \tag{136}$$

(136)

In the case of rotated support, we define the node coordinate system at the given node α_u: same as in (81, 82, 83, 84)

$$T(i) := \begin{pmatrix} Cu_{i,1} & Su_{i,1} & 0 & 0 & 0 & 0 \\ Su_{i,1} & Cu_{i,1} & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & Cu_{i,2} & Su_{i,2} & 0 \\ 0 & 0 & 0 & Su_{i,2} & Cu_{i,2} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{pmatrix}. \quad (137)$$

The calculation of the beams stiffness matrix in the global coordinate system is the same as in (87). (137)

The beam load vector in the global coordinate system is defined as follows:

$$p_e := p_e \cdot \frac{m}{kN}, \quad (138)$$

$$j := 1..n_p \quad k := L_{p_{n_p}}, \quad (139)$$

$$F_{pe_{2,L_{p_j}}} := \frac{-p_e \cdot L_{\left(L_{p_j}\right)}}{2}, \quad (140)$$

$$F_{pe_{3,L_{p_j}}} := \frac{-p_e \cdot \left(L_{\left(L_{p_j}\right)}\right)^2}{12},$$

$$F_{pe_{5,L_{p_j}}} := F_{pe_{2,L_{p_j}}} \quad F_{pe_{6,L_{p_j}}} := -F_{pe_{3,L_{p_j}}}, \quad (141)$$

$$F_{el}(i) := F_{pe}^{\langle i \rangle}.$$

The transformation of the beam load vector to the global coordinate system is as follows:

$$j := 1..n_{tot} \quad F_{glob_j} := 0, \quad (142)$$

$$K_{elg}(i) := T(i)^T \cdot F_{el}(i), \quad (143)$$

$$F_e^{\langle i \rangle} := F_{elg}(i), \quad (144)$$

$$F_{glob} := P_{glob}\left(F_{glob}, F_e, Code_{ele}\right). \quad (145)$$

The global frame load vector from the singular forces at the nodes is defined as follows:

$$F_{glob} := Fglob\left(F_{glob}, F_u, L_{F_u}, Code_{nod}\right), \quad (146)$$

$$F_{glob}^T = \begin{pmatrix} -100 & 0 & 0 & -90 & 0 & 45 \end{pmatrix}. \quad (147)$$

The calculation of the global structure stiffness matrix is the same as in (96, 97).

The composition of the matrix of the total stiffness of the structure from the stiffness matrices of the elements is the same as in (98).

- Calculation of displacement vectors in beams in LCS:

The displacement matrix in the elements is calculated as well in the relation (102)

$$r_{tot} := \begin{pmatrix} 0 & -9.093\cdot10^{-5} & 0 \\ 0 & 3.598\cdot10^{-7} & 0 \\ 0 & -2.706\cdot10^{-5} & 0 \\ -9.093\cdot10^{-5} & -3.839\cdot10^{-5} & -3.839\cdot10^{-5} \\ 3.598\cdot10^{-7} & 6.711\cdot10^{-7} & 6.711\cdot10^{-7} \\ -2.706\cdot10^{-5} & -8.398\cdot10^{-7} & 2.389\cdot10^{-4} \end{pmatrix}. \quad (148)$$

The beam force matrix at the elements in LCS is calculated as well in the relation (106)

$$F_{ele} = \begin{pmatrix} -1.295 & -94.569 & -1.208 \\ 5.431 & -1.295 & 21.399 \\ 15.447 & -5.224 & -0.13 \\ 1.295 & -68.601 & 1.208 \\ -5.431 & -1.208 & -21.399 \\ 5.224 & 0 & 45 \end{pmatrix}. \quad (149)$$

- Graphical control of the frame deformations is based on the calculation of the maximum value of the displacements as follows:

$$m_r := \max\left(\max\left(r_{tot}\right), \left|\min\left(r_{tot}\right)\right|\right)$$

$$m_r = 2.389x10^{-4}. \quad (150)$$

The interpretation scale is defined as follows:

$$p := \frac{\max(L)}{m_r}.m_k \qquad p = 2.511x10^3. \quad (151)$$

The geometry of the deformed structure in Fig. 7 is defined by the following relations:

$$x_{plot_{i,1}} := x_{nLokal_{i,1}} + p\cdot r_{tot_{1,i}}$$
$$y_{plot_{i,1}} := y_{nLokal_{i,1}} + p\cdot r_{tot_{2,i}}$$
$$x_{plot_{i,2}} := x_{nLokal_{i,2}} + p\cdot r_{tot_{3,i}}$$
$$y_{plot_{i,2}} := y_{nLokal_{i,2}} + p\cdot r_{tot_{4,i}}$$
$$x_{plot_{i,3}} := x_{nLokal_{i,1}} + p\cdot r_{tot_{1,i}}$$
$$y_{plot_{i,3}} := y_{nLokal_{i,1}} + p\cdot r_{tot_{2,i}}$$
$$i := 1..n_{elem} \qquad j := 1..cols\left(x_{plot}\right) \qquad k := 1..n_{node}. \quad (152)$$

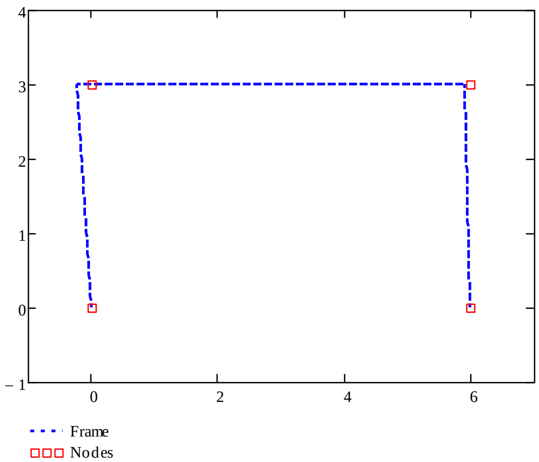

Fig. 7 Original and deformed frame structure

VII. Conclusion

This paper presents the experience of teaching FEM at the university. We currently use mathematical software such as MathCAD to solve truss and frame structures. Not knowing the essence of the problems leads to unprofessional and ineffective solutions to the static problem. The authors offer the readers of this article, those interested in the presented problems, the original library of applications in MathCAD software created by the authors and used at the university for teaching the finite element method for free.

Acknowledgment

The project was implemented with the financial support of the Grant Agency SR VEGA 1/0453/20.

References

[1] R. Courant, Variational methods for the solution of problems of equilibrium and vibrations, Bulletin of the American Mathematical Society. Vol 49, pp. 1–23. 1943.
[2] O.C. Zienkiewicz, R.L. Taylor, “The Finite Element Method. Vol. 1 The Basis, Vol .2 Solid Mechanics”, First ed. by McGraw-Hill 1967, Fifth Ed. by Butterworth-Heinemann 2000.
[3] E. Madenci, I. Guven, “The Finite Element Method and Applications in Engineering Using ANSYS”, Ed. Springer, pp. 653, 2015.
[4] Z. Bittnar, J. Šejnoha, “Numerical Methods in Structural Mechanics”, ASCE, pp. 470, 1996.
[5] J. Králik, “Reliability Analysis of Structures Using Stochastic Finite Element Method”, Monograph. Published by STU Bratislava, pp. 143, 2009.
[6] J. Králik, “Analysis of Structures – Finite Element Method”, Ed. STU Bratislava, pp. 253, 2018. (in Slovak)
[7] J. Králik, J. Králik, jr., Methodology of teaching methods of FEM analysis using software ANSYS, In International Journal of Education and Transformation Technologies, Vol. 16, Art. #2, pp. 12-28, 2022.
[8] W. K. Chen, Linear Networks and Systems (Book style). Belmont, CA: Wadsworth, pp. 123–135, 1993.
[9] S. M. Abdullaeva, U. N. Sultonova, “Use of MathCAD software in the preparation of students majoring in engineering”, International Journal of Scientific and Research Publications, Volume 10, Issue 12, 2020.
[10] B. Maxfield, “Engineering with Mathcad, Using Mathcad to create and organize your engineering calculations”, Ed. Elsevier, pp. 494, 2006.
[11] B. Maxfield, “Essential PTC Mathcad Prime 3.0”, Ed. Elsevier, pp. 563, 2014.
[12] D. J. Inman, “A Mathcad manual for engineering mechanics: Dynamics - Computational edition”, pp. 144, 2007.

Contribution of individual authors to the creation of a scientific article (ghostwriting policy)

We confirm that all Authors equally contributed in the present research, at all stages from the formulation of the problem to the final findings and solution.

Sources of funding for research presented in a scientific article or scientific article itself

The project was implemented with the financial support of the Grant Agency SR VEGA 1/0453/20.

Conflict of Interest

The authors have no conflict of interest to declare that is relevant to the content of this article.