Computer Methods in Civil Engineering - lecture handouts

Sławomir Milewski, CUT-FCE-L10, slawek@L5.pk.edu.pl

Contents

1 Introduction **1**

2 1D linear elastic problems **2**
2.1 Static analysis of bar structures . . . 2
2.2 Hierarchical FE interpolation . . . 7
2.3 Dynamic analysis of bar structures - free vibrations . . . 8
2.4 Dynamic analysis of bar structures - forced vibrations . . . 10
2.5 Computational problems . . . 11
2.5.1 1D static with linear shape functions . . . 11
2.5.2 1D static with hierarchical quadratic shape functions . . . 12
2.5.3 1D second order boundary value problem . . . 13
2.5.4 1D natural vibrations . . . 14
2.5.5 1D forced vibrations . . . 16

3 Formulation of 3D linear elastic problem **17**

4 Selected 2D linear elastic problems **18**
4.1 Plain strain state . . . 18
4.2 Plane stress state . . . 19
4.3 FE approximation . . . 20
4.4 Elasto-plastic material model . . . 22

5 1D non-linear problems **24**

6 Exemplary problems **27**
6.1 Problems for computer implementation . . . 31

1 Introduction

Real physical phenomena, occurring in engineering structures, can be effectively analysed by building their respective models. The basic model from which this process begins is *mechanical model.* It is a set of hypotheses and simplifying assumptions referring to the material, geometry, support or load conditions. The simpler the model, the easier its later analysis, but also the larger error (called *unavoidable error*) that we are making at this initial stage. The next model is *mathematical model.* This is a description of the mechanical model in the formalism of mathematics, namely the relationships between various physical fields resulting from all the above assumptions of the mechanical model. The mathematical model can be either a variation principle, a problem of functional minimization or a system of partial differential equations, all with appropriate boundary and initial conditions (the so-called initial-boundary problem), as well as the problem of non-linear optimization in a constrained domain. Very rarely this problem can be solved using analytical methods and transformations (such as direct equation integration). This is possible only with an extremely simple form of the equation itself and the geometrical shape of the problem domain. Most often, we have to solve this problem numerically, in an approximate manner. The final result is therefore *numerical model*, which is the discrete (finite) equivalent of the mathematical model and can lead to, for example, a system of algebraic equations (linear or non-linear) or a generalized eigen problem. The error we provide at this stage may be associated with both discretization of the task area and approximation of the unknown function, hidden under the sign of the appropriate differential operator. It can be controlled much more easily than the inevitable error, improving both of these aspects of numerical modelling (i.e. improving discretization or/and increasing the degree of approximation). Next, we sometimes talk about the *IT model*, which is a computer program (own or commercial), needed to run calculations based on a numerical model. Let us remember that each numerical result obtained must be subjected to critical analysis to minimize the risk of incorrectly accepted model parameters and ordinary human error.

The oldest method, dating back to the end of the 19th century and used to build numerical models of boundary and initial-boundary problems is the Finite Difference Method (FDM). It is based on the discretization of the problem domain using meshes of nodes (in the classic version of the method: regular; later generalized to the meshless version - for arbitrarily irregular clouds) and the replacement of differential operators with differential ones. Despite its simplicity, the classic version of FDM could only be used to solve simple linear problems. That is why since the 1950s it has been gradually supplanted by the then emerging finite element method (FEM), which is much more general and easier to automate. Needless to say that the enormous effort that has been put into developing the FEM, has resulted in the fact that this method is the basis of the vast majority of computational engineering systems of all kinds, including of course civil engineering domain. The academic definition of FEM reads that in this method we divide the domain into simple geometric figures, called finite elements. This is basically true, although in FEM the following stages are also important, such as generating an approximation of an unknown function in an element, manners of this function differentiating and integrating, and building a system of algebraic equations. All this distinguishes FEM from other computational methods. It is worth mentioning that FEM has strong mathematical background, including convergence criteria of approximate solution, including the error analysis that may be carried out without knowing the exact solution.

2 1D linear elastic problems

One-dimensionality of the engineering problem means that all functions: unknown (displacements, stresses, deformations) and known (loads, material functions) will be functions of one independent variable x, and differential equations will be ordinary equations. In turn, the elastic problem means that the relationship between stress and strain will be fully reversible - namely, no permanent (plastic) deformations will occut in the structure, and after removing the load from it - it will return to its pre-deformation state. Consequently, it will not remember subsequent load attempts.

2.1 Static analysis of bar structures

Consider the problem of a bar structure subjected to axial loading (tension, compression), shown in Fig. 1a. The bar is clamped at its left end ($x = 0$) and has a free right end ($x = L$). Length of bar (L) is given. It is assumed that the cross-section area A as well as Young modulus E are the C^1 functions of x or they are scalar numbers (independent from x). The bar may be loaded by uniform load with intensity $q = q(x)$ being a C^0 function as well as concentrated force P, subjected at the free right end (at $x = L$). Horizontal displacements $u = u(x)$ as well as the axial (longitudinal) force distribution $N = N(x)$ are unknown. The mechanical model of a bar, presented above, is based upon numerous assumptions concerning its geometry, material, load and support. However, for effective analytical and/or numerical analysis, the relevant mathematical model has to be formulated. In most cases (mechanical, thermal, thermo-mechanical problems), we deal with three types of equations, relating known (load q, P, ascribed displacements) and unknown (displacements - u, force or stress - N or σ as well as strain - ε) fields, namely

1. *balance* equation, relating the force/stress with external load, and describing the balance of a infinitesimal bar segment with length dx (see Fig. 1b)

$$-N(x) + N(x) + dN + q(x)\,dx = 0 \tag{1}$$

which leads to the first order differential problem

$$-\frac{dN}{dx} = q(x), \quad N(L) = P \tag{2}$$

It holds under the assumption that displacements are small ($u_{\max} << L$, where $u_{\max}$ is the maximum horizontal displacement) namely the initial and deformed states may not be distinguished from one another; otherwise, in case of large displacement, the boundary condition from (2) has to be formulated in deformed state (i.e., L is the deformed bar length),

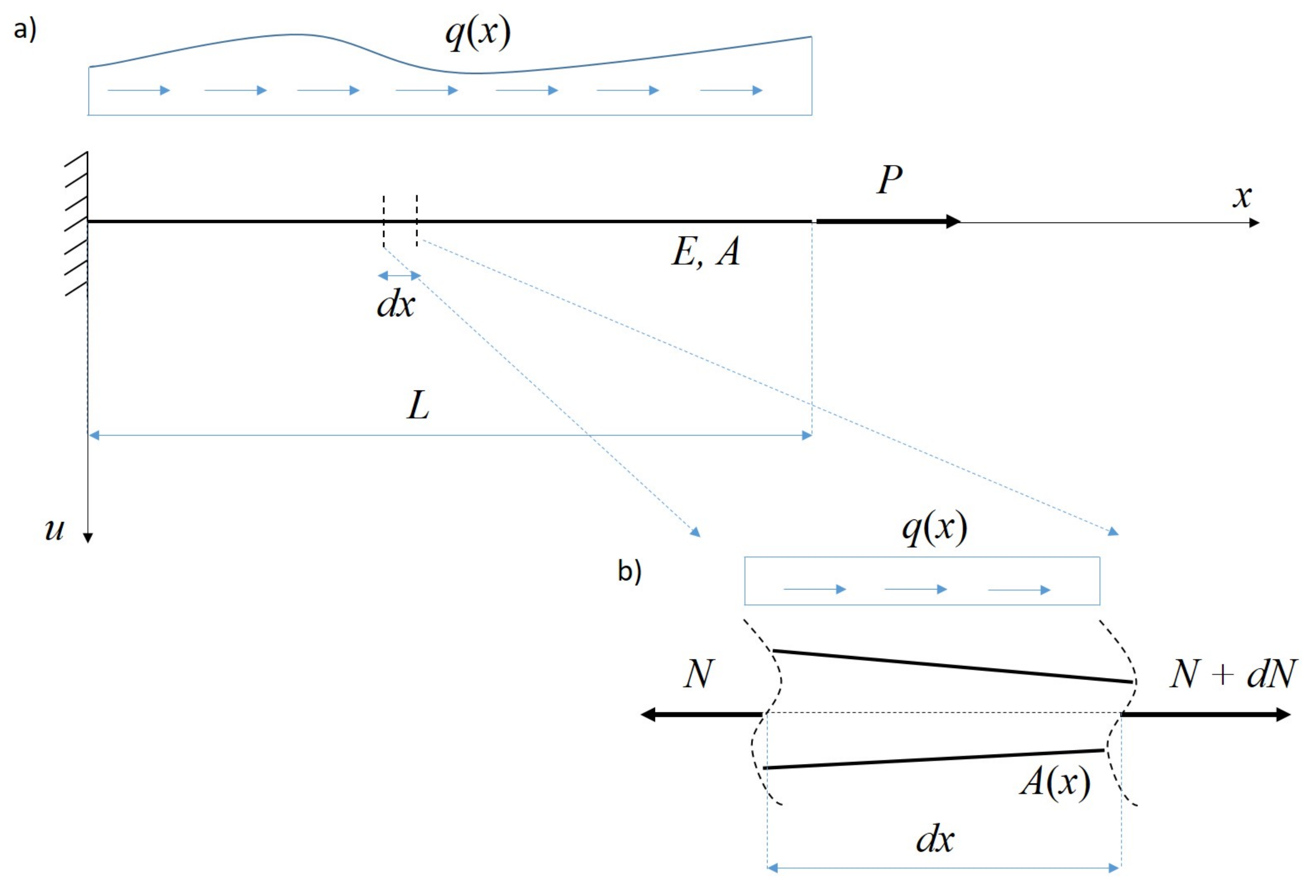

Figure 1: Mechanical model of a bar under tension, a) general view of the entire structure, b) force balance on the interval of small dimensions

2. *geometric* equation, relating strain with displacement and describing the deformed state under ascribed motion function

$$\varepsilon(x) = \frac{du}{dx} + \frac{1}{2}\left(\frac{du}{dx}\right)^2 \approx \frac{du}{dx}, \quad u(0) = 0 \tag{3}$$

The second component in (3) may be omitted providing the strains are small (i.e., $\frac{du}{dx} << 1$),

3. *physical* (constitutive) equation, relating stress and strain and describing the material behaviour (σ) subjected to external load (ε)

$$\sigma(x) = E(x)\,\varepsilon(x) \tag{4}$$

Here, linear elastic material has been assumed. Moreover, from the definition of σ, one has

$$\sigma(x) = \frac{N(x)}{A(x)} \tag{5}$$

By substituting σ from (5) to (4) and using relation for ε from (3) to (2), we obtain

$$-\frac{d}{dx}\left(EA\frac{d}{dx}u\right) = q(x) \tag{6}$$

Assuming that tensile stiffness is constant ($EA = const.$, where E is a Young modulus, and A - area of a cross-section of a prismatic bar), we have

$$-EA\frac{d^2u}{dx^2} = q(x), \quad x \in (0, L), \quad u(0) = 0, \quad EA\frac{du}{dx}(L) = P \tag{7}$$

or using simplified notation for derivatives of u with respect to space (x)

$$-EAu'' = q(x), \quad x \in (0, L), \quad u(0) = 0, \quad EAu'(L) = P \tag{8}$$

Mathematical formulation (8) is named **local** (since differential equations are satisfied at every point of the domain and its boundary separately) and **strong** (since there are strong continuity requirements ascribed to $u \in C^2$). Analytical approach as well as numerical approach based upon finite difference method (MFD) may use (8), however for the finite element method (FEM), its further modifications have to be performed. First of all, we multiply both sides of (8) by arbitrary *test function* (v, and u is named *trial function*)

$$v\left(-EAu''\right) = vq, \quad \forall v \tag{9}$$

and integrate over the entire domain

$$-EA\int_0^L vu''dx = \int_0^L vqdx, \quad \forall v \tag{10}$$

Then we integrate the left-hand side component by parts, incorporating the natural boundary condition from (8) and assuming that $v(0) = 0$ (which makes v a displacement variation)

$$\begin{aligned} L = -EA\int_0^L vu''dx = -EA\left| \begin{matrix} f = v & f' = v' \\ g' = u'' & g = u' \end{matrix} \right| = -EA\left(\left[vu'\right]|_0^L - \int_0^L v'u'dx\right) = \\ = -EA\left(v(L)\,u'(L) - \underbrace{v(0)}_{=0}u'(0) - \int_0^L v'u'dx\right) = -\underbrace{EAu'(L)}_{=P}v(L) + EA\int_0^L v'u'dx = \\ = -Pv(L) + EA\int_0^L v'u'dx \end{aligned} \tag{11}$$

Eventually, we are looking for $u \in H_0^1(u)$ which satisfies

$$EA\int_0^L v'u'dx = \int_0^L vqdx + Pv(L), \quad \forall v \in H_0^1(v) \tag{12}$$

Here, $H_0^1(u)$ is a Hilbert space of functions satisfying the homogeneous essential boundary condition and with the first derivative integrable in the considered domain. Mathematical formulation (12) is named **global** (since it is satisfied globally, "in one piece"), **weak** (since continuity requirements for u are weaker than in case of local formulation) and **variational** (since variation of displacement - test function v - appears). It may be directly applied for the FE analysis.

Let us assume that the problem domain (interval $x \in [0, L]$) is discretized using N finite elements (Fig. 2a). Therefore, $n = N + 1$ nodes $\mathbf{X} = \{x_i\}, \quad i = 1, ..., n$ and n nodal degrees of freedom (as nodal horizontal displacements $\mathbf{U} = \{U_i\}, \quad i = 1, ..., n$) are introduced. Nodes and elements are consequently numbered from 1 to n and from 1 to N, respectively. Let us pay attention on the single $e = i$-th element (Fig. 2b), located between $x_1 = x_{i-1}$ and $x_2 = x_i$ nodes, and having two degrees of freedom $U_1 = U_{i-1}$ and $U_2 = U_i$. Besides global coordinate system (x, u), the local element coordinate system $(x^{(e)}, u)$, with the origin at x_1, may be applied. In that case, element may be defined as the subinterval $x^{(e)} = [0, h]$, where $h = x_2 - x_1$ is the length of the element, and $x^{(e)} = x - d$. The simplest interpolation possible is based upon two linear shape functions of Lagrange type, assigned to each element degree of freedom, namely

$$N_1(x) = \frac{x - x_2}{x_1 - x_2} = 1 - \frac{x}{h}, \quad N_2(x) = \frac{x - x_1}{x_2 - x_1} = \frac{x}{h} \tag{13}$$

Note that $N_j(x_i) = \left\{ \begin{matrix} 0, & i \neq j, \\ 1, & i = j \end{matrix} \right., \quad i, j = 1, 2$. Therefore, those basis functions may be applied for both geometry

$$x = N_1(x)\,x_1 + N_2(x)\,x_2 = \mathbf{N}(x)\,\mathbf{X}^{(e)} \tag{14}$$

and unknown function interpolation (isoparametric C^0 element)

$$u(x) = N_1(x)\,U_1 + N_2(x)\,U_2 = \mathbf{N}(x)\,\mathbf{U}^{(e)} \tag{15}$$

where $\mathbf{N}(x) = [N_1(x) \;\; N_2(x)]$, $\mathbf{X}^{(e)} = [x_1 \; x_2]^{\mathrm{T}}$ and $\mathbf{U}^{(e)} = [U_1 \; U_2]^{\mathrm{T}}$. First derivative of u may be expressed as

$$u'(x) = \frac{d}{dx}N_1(x)\,U_1 + \frac{d}{dx}N_2(x)\,U_2 = \frac{d}{dx}\mathbf{N}(x)\,\mathbf{U} = \mathbf{B}(x)\,\mathbf{U} \tag{16}$$

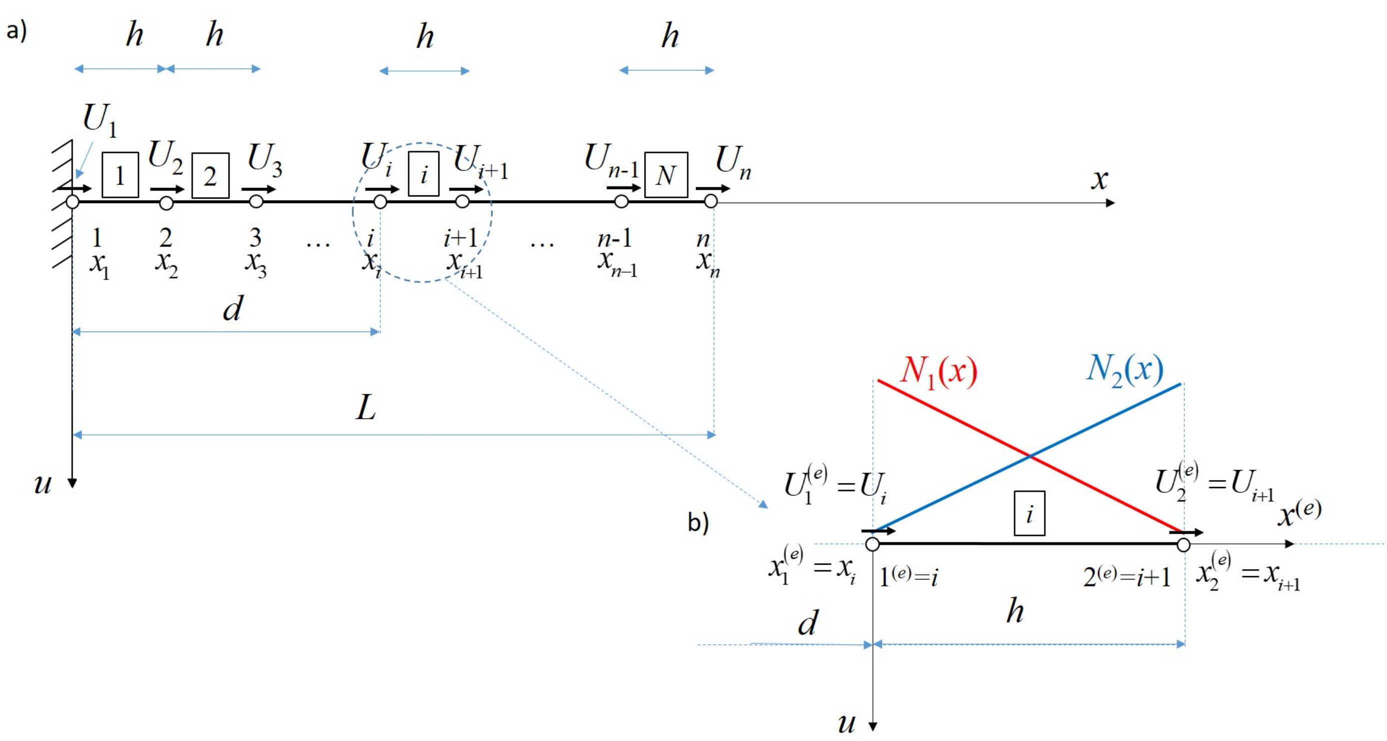

Figure 2: FE numerical model of a bar under tension: a) the entire FE mesh, b) selected finite element and its shape functions

where $\mathbf{B}(x) = \left[-\frac{1}{h}\ \frac{1}{h}\right]$ is the matrix of shape functions derivatives. Similarly, according to Bubnov-Galerkin concept, interpolation schemes of test function v and its first derivative are obtained

$$v(x) = \mathbf{N}(x)\mathbf{V} = \mathbf{V}^{\mathrm{T}}\mathbf{N}^{\mathrm{T}}(x), \quad v'(x) = \mathbf{B}(x)\mathbf{V} = \mathbf{V}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}(x) \tag{17}$$

After substitution of (15), (16) and (17) into (12), one gets

$$EA\int_0^h \mathbf{V}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}\mathbf{B}\mathbf{U}^{(e)}dx = \int_0^h \mathbf{V}^{\mathrm{T}}\mathbf{N}^{\mathrm{T}}q\,dx + P\mathbf{V}^{\mathrm{T}}\mathbf{N}^{\mathrm{T}}(L), \quad \forall\mathbf{V}^{\mathrm{T}} \tag{18}$$

in which the last component is non-zero for the last N-th element only and is usually included in the final system of equations, after an assembling process. Taking advantage from arbitrary selection of the test function, the final element equation is obtained

$$\mathbf{K}^{(e)}\mathbf{U}^{(e)} = \mathbf{F}^{(e)} \tag{19}$$

where $\mathbf{K}^{(e)}$ and $\mathbf{F}^{(e)}$ are local (element) stiffness matrix and load vector, respectively

$$\mathbf{K}^{(e)} = EA\int_0^h \mathbf{B}^{\mathrm{T}}\mathbf{B}dx = EA\int_0^h \begin{bmatrix} -\frac{1}{h} \\ \frac{1}{h} \end{bmatrix}\left[-\frac{1}{h}\ \frac{1}{h}\right]dx = \frac{EA}{h}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{20}$$

$$\mathbf{F}^{(e)} = \int_0^h \mathbf{N}^{\mathrm{T}}q(x+d)\,dx \tag{21}$$

More detailed form of the load vector depends on the load intensity function q. For the simplest case (constant load intensity q_0 over the entire element, Fig. 3a), we have

$$\mathbf{F}^{(e)} = q_0\int_0^h \begin{bmatrix} 1-\frac{x}{h} \\ \frac{x}{h} \end{bmatrix}dx = \frac{q_0 h}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \tag{22}$$

For the linear load distribution (e.g., according to $q(x) = ax + b$ formula, Fig. 3b), we have two options, namely

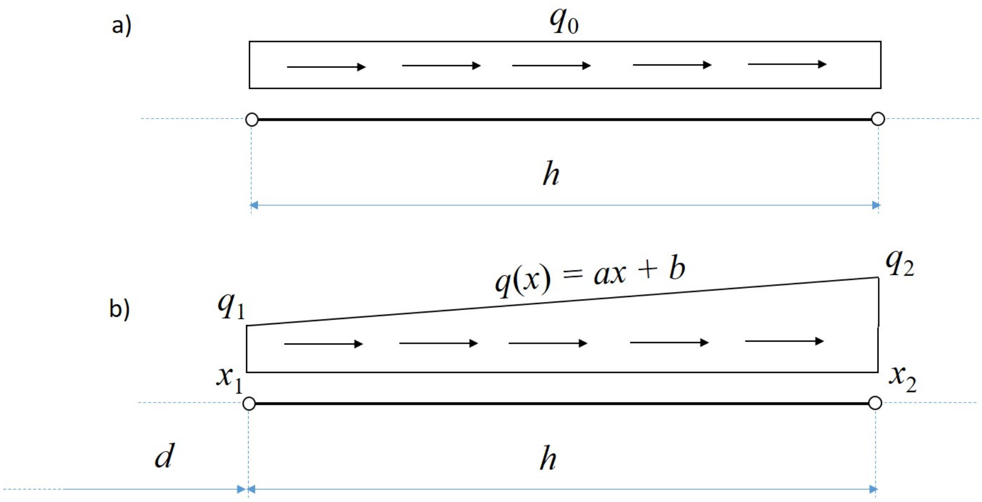

Figure 3: Simple cases of the continuous load over the finite element: a) constant intensity, b) linearly dependent intensity

- evaluate load vector according to (21)

$$\mathbf{F}^{(e)} = \int_0^h \begin{bmatrix} 1 - \frac{x}{h} \\ \frac{x}{h} \end{bmatrix} (a(x+d)+b)\, dx \tag{23}$$

- use isoparametric interpolation for load intensity $q(x) = N_1(x)\, q_1 + N_2(x)\, q_2$, based upon its nodal values $q_1 = ax_1 + b$ and $q_2 = ax_2 + b$

$$\mathbf{F}^{(e)} = \int_0^h \begin{bmatrix} 1 - \frac{x}{h} \\ \frac{x}{h} \end{bmatrix} \left(\left(1 - \frac{x}{h}\right) q_1 + \frac{x}{h} q_2\right) dx = \frac{h}{6} \begin{bmatrix} 2q_1 + q_2 \\ q_1 + 2q_2 \end{bmatrix} \tag{24}$$

Determination of element stiffness matrices and load vectors is followed by the assembling process (i.e., summation of element quantities into the global system, according to mesh topology), yielding the global system of equations

$$\mathbf{KU} = \mathbf{F} \tag{25}$$

Assembling process, for the considered problem, is done according to the schedule shown in Fig. 4.

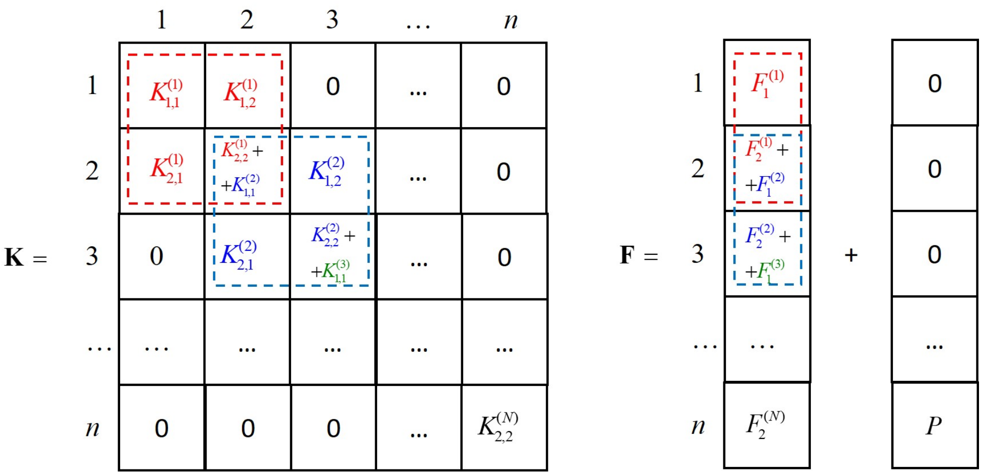

Figure 4: Assembling scheme of the global stiffness matrix **K** as well as the global load vector **F**

The global stiffness matrix **K** remains singular unless essential boundary conditions are fulfilled ($U_1 = 0$ and $V_1 = 0$). Technically:

1. in case of manual calculations: one may directly cross-out those rows and columns of (25) which correspond to homogeneous boundary conditions, or

2. in case of computer implementation: one may fill with zeros those rows, columns and elements which correspond to zero conditions and place 1 at the main diagonal elements.

In case of our problem, modifications should be applied to: first row and first columns of $\mathbf{K}$ as well as first element of $\mathbf{F}$ have to be 1) cross-outed or 2) filled with zeros; in the second variant, diagonal element $K_{1,1} = 1$. In more general case, in which the boundary conditions are non-homogeneous (non zero ones, e.g., geometrical loading $U_1 \neq 0$ though still $V_1 = 0$), first we need to transform the system using the formula $\mathbf{F} = \mathbf{F} - \mathbf{K}\bar{\mathbf{U}}$, where $\bar{\mathbf{U}}$ is the vector of known degrees of freedom (values of known horizontal displacements and zeros otherwise), and we may apply any of those two above-listed variants.

Afterwards, the system may be solved, yielding one unique solution $\mathbf{U} = \mathbf{K}^{-1}\mathbf{F}$, aka nodal horizontal displacements (primary unknowns) as well as reaction forces ($\mathbf{R} = \mathbf{KU} - \mathbf{F}$, secondary unknowns). The post-processing stage may include determination of displacements and normal (axial, longitudinal) forces for each element, namely

$$u^{(e)}(x) = \mathbf{N}(x-d)\,\mathbf{U}^{(e)}, \quad N^{(e)}(x) = EA\frac{du^{(e)}}{dx} = EA\,\mathbf{B}(x-d)\,\mathbf{U}^{(e)} \tag{26}$$

2.2 Hierarchical FE interpolation

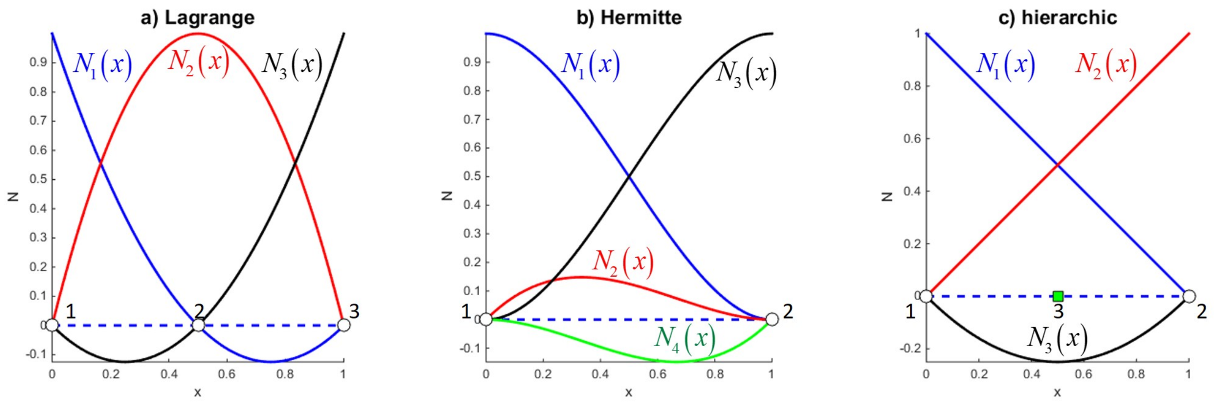

Figure 5: Higher order interpolation techniques for the finite element: a) Lagrange, b) Hermite, c) hierarchic

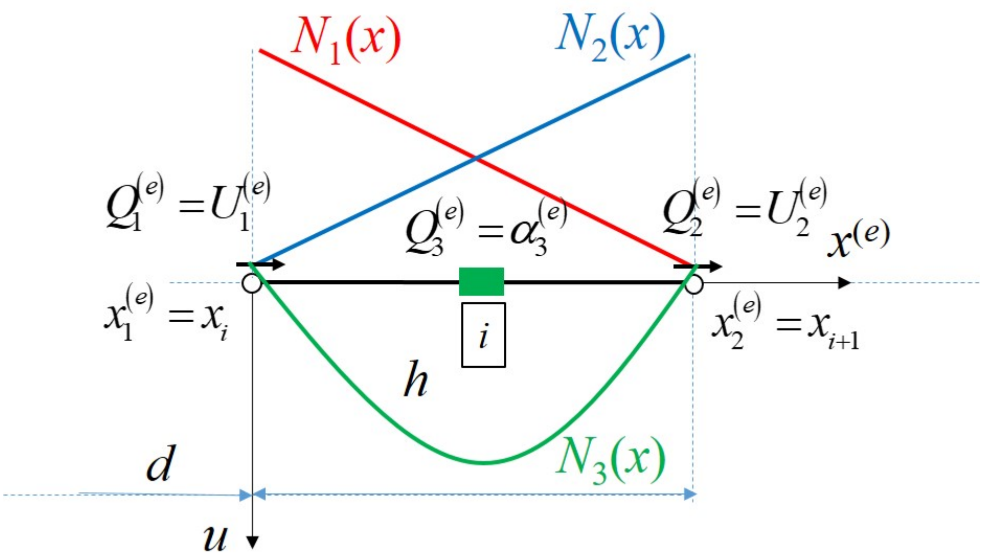

Figure 6: Hierarchic interpolation in the finite element

Though very simple, linear FE interpolation leads to low accuracy results, especially when solution derivative is taken into account. Therefore, higher order, quadratic interpolation is considered. At the level of finite element, it may be implemented using three various techniques, (i) namely using additional internal node and three Lagrange basis function of the 2^{nd} order (Fig. 5a), (ii) using additional degrees of freedom (displacement derivatives) at existing nodes and Hermitte basis functions of the 3^{rd} order (Fig. 5b) and (iii) hierarchic shape functions (Fig. 5c and Fig. 6), which include the basic linear ones and one additional bubble function

$$N_3(x) = (x - x_1)(x - x_2) = x(x - h) \tag{27}$$

which vanishes at element's nodes, namely $N_3(x_1) = N_3(x_2) = 0$, however it is not a shape function is a strict sense. Therefore, enhanced FE interpolation schemes of the unknown displacement function and its derivative are as follows

$$u(x) = N_1(x)U_1 + N_2(x)U_2 + N_3(x)\alpha_3 = \begin{bmatrix} 1 - \frac{x}{h} & \frac{x}{h} & x^2 - xh \end{bmatrix} \mathbf{Q} = \mathbf{N}(x)\mathbf{Q}^{(e)} \tag{28}$$

and

$$u'(x) = N_1'(x)U_1 + N_2'(x)U_2 + N_3'(x)\alpha_3 = \begin{bmatrix} -\frac{1}{h} & \frac{1}{h} & 2x - h \end{bmatrix} \mathbf{Q} = \mathbf{B}(x)\mathbf{Q}^{(e)} \tag{29}$$

using additional mathematical degree of freedom α_3, without any specified location in the finite element. Here, $\mathbf{Q}^{(e)} = [U_1\ U_2\ \alpha_3]^{\mathrm{T}}$. The modified element stiffness matrix and load vector (for $q = q_0 = const.$) are

$$\mathbf{K}^{(e)} = \frac{EA}{h}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & \frac{h^4}{3} \end{bmatrix}, \quad \mathbf{F}^{(e)} = \frac{q_0 h}{2}\begin{bmatrix} 1 \\ 1 \\ -\frac{h^2}{3} \end{bmatrix} \tag{30}$$

Note that in the stiffness matrix, the only new non-zero element is the diagonal element $K_{33}^{(e)}$, while the other, off-diagonal elements are 0. This is due to the fact that the derivative of the bubble function is orthogonal to the derivatives of the linear shape functions, namely $\int_0^h N_i'(x)\, N_3'(x)\, dx = 0, \quad i = 1, 2.$

2.3 Dynamic analysis of bar structures - free vibrations

The problem of determining the natural (normal, eigen) vibrations of a structure is related to the determination of their natural frequencies, which can be treated as the velocity of these vibrations. In this analysis, we are not interested in the cause that caused the vibrations, therefore we are not able to unambiguously determine the form of such vibrations, for any time moment. During vibrations, only two forces act on the structure, the interaction of which knock the structure out of a stable state of equilibrium, namely the elastic force associated with the properties of the material, and the inertia force associated with the Earth's gravitational field. On the other hand, we will show that the natural vibration parameters (i.e., frequency f measured in Hertz ($1Hz = 1/s$) and circular frequency $\omega = 2\pi f$ (also called angular frequency or pulsation), measured in rad/s with the period $T = \frac{1}{f}$ measured in seconds), depend on the geometrical and material parameters of the structure. They are determined in the design phase so that the designed structure, exposed to dynamic impacts (e.g. bridge, footbridge, cable, mast, all exposed to wind), is not subject to phenomena unfavourable both for itself and for living beings within it. These adverse effects include resonance and beat. *Resonance* consists in a rapid increase in the amplitude of the forced vibration and occurs when the frequency of the excitation load (e.g., wind) is the same or very close to the frequency of the natural vibrations, which causes overlapping of vibrations. In turn, *beat* is an overlap of vibrations with the same or very close frequencies (with weak damping), occurring when the number of forcing factors is greater than one.

Let us examine the same bar structure as in the previous subsection, with length L, Young modulus E, cross-section area A and mass density ρ. The bar is clamped at the left end, and has a free, unloaded right end. By enforcing vibrations (though nature of such enforcement is not considered), we are imposing the acceleration to the structure and therefore, inertia force appears as for every non-inertia system in physics. Local formulation may be expressed as: find such displacement function $u = u(x,t) \in C^2$, that

$$-EAu'' + A\rho\ddot{u} = 0, \quad u(0,t) = 0, \quad u'(L,t) = 0 \tag{31}$$

Here, $u'' = \frac{\partial^2 u}{\partial x^2}$ and $\ddot{u} = \frac{\partial^2 u}{\partial t^2}$. By multiplying both sides of (31) by v, integrating over entire domain, integrating by parts the first integral and incorporating the homogeneous boundary conditions, we obtain the variational formulation

$$EA\int_0^L v'u'dx + A\rho\int_0^L v\ddot{u}dx = 0, \quad u \in H_0^1, \quad \forall v \in H_0^1 \tag{32}$$

At the element level, we decompose u into space (x) and time (t) one-dimensional functions and assume harmonic response of a vibrating bar

$$u(x,t) = \mathbf{N}(x)\,\mathbf{U}^{(e)}(t) = \mathbf{N}(x)\,\mathbf{U}^{(e)}\sin(\omega t + \phi) \tag{33}$$

with angular frequency ω and optional phase translation ϕ. In similar manner, derivatives of u with respect to space and time may be expressed

$$\begin{aligned} u'(x,t) &= \mathbf{B}(x)\,\mathbf{U}^{(e)}(t) = \mathbf{B}(x)\,\mathbf{U}^{(e)}\sin(\omega t + \phi) \\ \ddot{u}(x,t) &= -\omega^2\mathbf{N}(x)\,\mathbf{U}^{(e)}\sin(\omega t + \phi) \end{aligned} \tag{34}$$

Interpolation schemes of test function v and its derivative remain unmodified, according to (17). After substitution of FE schemes into (32), one obtains the variational element equation

$$EA\int_0^h \mathbf{V}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}\mathbf{B}\mathbf{U}^{(e)}\sin(\omega t + \phi)\,dx - \omega^2 A\rho\int_0^h \mathbf{V}^{\mathrm{T}}\mathbf{N}^{\mathrm{T}}\mathbf{N}\mathbf{U}^{(e)}\sin(\omega t + \phi)\,dx = \mathbf{0} \tag{35}$$

Knowing that $\mathbf{V}^{\mathrm{T}}\sin(\omega t + \phi)$ is the arbitrary non-zero function of t, we get

$$\left(\mathbf{K}^{(e)} - \omega^2\mathbf{M}^{(e)}\right)\mathbf{U}^{(e)} = \mathbf{0} \tag{36}$$

New element matrix is introduced, namely the consistent (full) mass matrix $\mathbf{M}^{(e)}$ and

$$\mathbf{M}^{(e)} = A\rho\int_0^h \mathbf{N}^{\mathrm{T}}\mathbf{N}dx \tag{37}$$

The consistent mass matrix corresponds to the continuous distribution of mass along the element. For linear interpolation, we have

$$\mathbf{M}^{(e)} = A\rho\int_0^h \begin{bmatrix} 1 - \dfrac{x}{h} \\ \dfrac{x}{h} \end{bmatrix} \begin{bmatrix} 1 - \dfrac{x}{h} & \dfrac{x}{h} \end{bmatrix} dx = \frac{A\rho h}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \tag{38}$$

Besides the consistent mass matrix, more simplified approach, typical for structure mechanics, may be applied in which the diagonal (lumped) mass matrix, resulting from the discrete distribution of mass at element's nodes, is introduced

$$\mathbf{M}_d^{(e)} = \frac{A\rho h}{2}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \tag{39}$$

Determination of mass and stiffness matrices in (36), for all elements, is followed by their assembling and enforcement of the homogeneous essential boundary conditions. The problem on the global level

$$\left(\mathbf{K} - \omega^2\mathbf{M}\right)\mathbf{U} = \mathbf{0} \tag{40}$$

constitutes the generalized eigen problem of the global stiffness matrix $\mathbf{K}$. In order to solve it, we should transform it into the standard eigen problem

$$\left(\mathbf{M}^{-1}\mathbf{K} - \omega^2\mathbf{I}\right)\mathbf{U} = \mathbf{0} \tag{41}$$

where $\mathbf{I}$ is the identity matrix. Neglecting trivial zero solution ($\mathbf{U} = \mathbf{0}$), the above problem has non-zero solution, providing that

$$\det\left(\mathbf{M}^{-1}\mathbf{K} - \omega^2\mathbf{I}\right) = 0 \tag{42}$$

In that case, non-zero $\mathbf{U}$ are called eigen vectors, while non-zero and positive ω^2 are called eigen values. For each frequency ω, the form $\mathbf{U}$ of free vibrations may be determined, however its amplitude and sense remain ambiguous.

The specific forms of free vibrations (eigen modes) may be found as solutions of (41), for each eigen value ω_i^2 separately

$$\left(\mathbf{M}^{-1}\mathbf{K} - \omega_i^2\mathbf{I}\right)\mathbf{U}_i = \mathbf{0}, \quad i = 1, 2, ..., N \tag{43}$$

The number of determinable modes is equal to the number of free degrees of freedom, namely in this case $N = n - 1$. Since the above system of equations is a dependent system (with infinite number of solutions), at this stage additional assumptions are made for the $\mathbf{U}_i$ vibration vector, for example the unit length of this vector is forced, i.e. $\tilde{\mathbf{U}}_\mathbf{i} = \dfrac{\mathbf{U}_\mathbf{i}}{|\mathbf{U}_i\,|}$. It is worth stressing that eigen modes are orthogonal, namely

$$\mathbf{U}_i^{\mathrm{T}}\mathbf{K}\mathbf{U}_j = 0, \quad \mathbf{U}_i^{\mathrm{T}}\mathbf{M}\mathbf{U}_j = 0, \quad i \neq j, \quad i,j = 1, 2, ..., N \tag{44}$$

2.4 Dynamic analysis of bar structures - forced vibrations

In case no damping appears, the following hyperbolic problem is considered: find $u = u(x,t)$, with $u \in C^2$ that

$$-EAu'' + A\rho\ddot{u} = q(x,t) \tag{45}$$

with boundary

$$u(0,t) = 0, \quad EAu'(L,t) = P(t) \tag{46}$$

and initial conditions

$$u(x, t_0 = 0) = u_0, \quad \dot{u}(x, t_0 = 0) = v_0 \tag{47}$$

The appropriate variation formulation is derived similarly to the statics and dynamics of natural vibrations, remembering the non-zero right side and the heterogeneous boundary condition of the natural type

$$EA\int_0^L v'u'dx + A\rho\int_0^L v\ddot{u}dx = \int_0^L vqdx + P(t)\,v(L), \quad u \in H_0^1, \quad \forall v \in H_0^1 \tag{48}$$

Performing similar steps as already shown in previous sections (however, no specific displacement function is assumed), we obtain the global system of equations in the following form

$$\mathbf{K}^{(e)}\mathbf{U}^{(e)}(t) + \mathbf{M}^{(e)}\ddot{\mathbf{U}}^{(e)}(t) = \mathbf{F}^{(e)}(t) \tag{49}$$

with initial conditions

$$\mathbf{U}(t_0 = 0) = \mathbf{U}_0, \quad \dot{\mathbf{U}}(t_0 = 0) = \mathbf{V}_0 \tag{50}$$

and right-hand side dynamic load vector $\mathbf{F}(t)$ which includes assembled vector of element load vectors $\mathbf{F}^{(e)}(t)$, equal

$$\mathbf{F}^{(e)}(t) = \int_0^h \mathbf{N}^{\mathrm{T}} q(x,t)\,dx \tag{51}$$

as well as concentrated force $P(t)$, subjected to the last node. In general,

$$\mathbf{F}(t) = f(\mathbf{X},t) + \begin{bmatrix} 0 & 0 & \dots & 0 & P(t) \end{bmatrix}^{\mathrm{T}} \tag{52}$$

where $f(x,t)$ is the given global dynamic load. Stiffness and mass matrices are the same as in previous cases. In other words, the problem (45) is discretized in space direction, however it remains continuous in time direction. Therefore, appropriate time integration has to be provided, using principles of numerical analysis of initial problems (Euler or Runge-Kutta methods, explicit and implicit one-step or multi-step schemes). Time direction is discretized with Δt time step. Here, the Newmark method is applied, in which the current displacement and velocity (at $(k+1)$-th time level) are obtained using backward (implicit) Euler schemes, namely

$$\begin{aligned} \mathbf{V}_{k+1} &= \mathbf{V}_k + \Delta t\ddot{\mathbf{U}}_{k+1} \\ \mathbf{U}_{k+1} &= \mathbf{U}_k + \Delta t\mathbf{V}_k + \frac{1}{2}(\Delta t)^2\ddot{\mathbf{U}}_{k+1} \end{aligned} \tag{53}$$

By fulfilling the equation of motion (49) at unknown at $(k+1)$-th time level and substituting displacement from (53), we obtain

$$\mathbf{K}\left(\mathbf{U}_k + \Delta t\mathbf{V}_k + \frac{1}{2}(\Delta t)^2\ddot{\mathbf{U}}_{k+1}\right) + \mathbf{M}\ddot{\mathbf{U}}_{k+1} = \mathbf{F}_{k+1} \tag{54}$$

and finally, after simple transformations

$$\ddot{\mathbf{U}}_{k+1} = \left(\mathbf{M} + \frac{1}{2}(\Delta t)^2\mathbf{K}\right)^{-1}(\mathbf{F}_{k+1} - \mathbf{K}(\mathbf{U}_k + \Delta t\mathbf{V}_k)) \tag{55}$$

The most troublesome part of the Newmark algorithm concerns the computation of the matrix inversion. However, it is done only once, for the entire procedure.

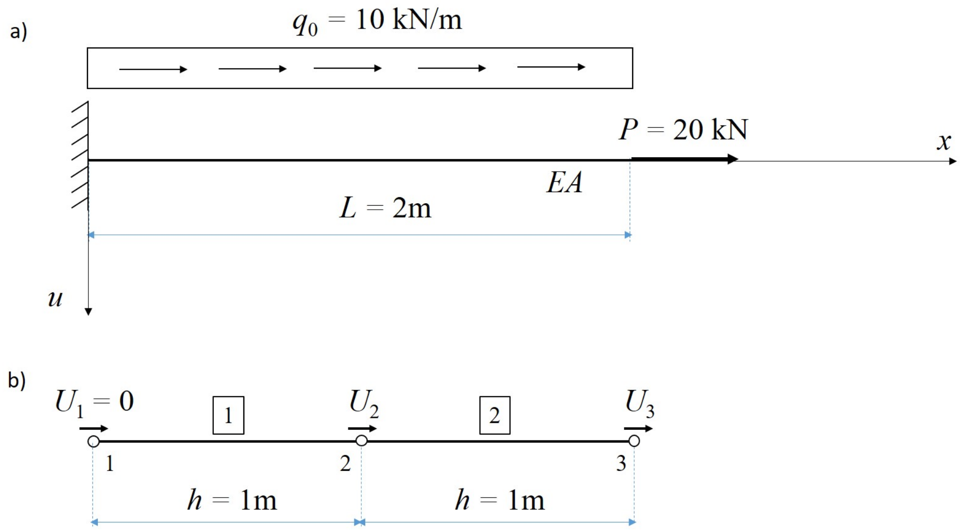

Figure 7: Illustration of the computational problem: a) mechanical model, b) FE numerical model

2.5 Computational problems

2.5.1 1D static with linear shape functions

For the considered bar, assume $L = 2m$, $q = q_0 = 10kN/m$, $P = 20kN$ and arbitrary stiffness EA (Fig. 7a). Find displacement and normal force using two finite elements with equal length and linear interpolation (Fig. 7b).

- element stiffness matrices and load vectors

$$\mathbf{K}^{(1)} = \mathbf{K}^{(2)} = EA \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}, \quad \mathbf{F}^{(1)} = \mathbf{F}^{(2)} = 5 \begin{bmatrix} 1 \\ 1 \end{bmatrix} kN \tag{56}$$

  The stiffness matrices of both elements are identical because elements have the same length (h) and the same stiffness (EA). In turn, the load vectors are identical, because the continuous load has a constant intensity of q_0.

- global system of equations with global stiffness matrix and load vector

$$\mathbf{KU} = \mathbf{F},$$

$$\mathbf{K} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}, \quad \mathbf{F} = 5 \begin{bmatrix} 1 \\ 2 \\ 1 \end{bmatrix} kN + \begin{bmatrix} 0 \\ 0 \\ 20 \end{bmatrix} kN = \begin{bmatrix} 5 \\ 10 \\ 25 \end{bmatrix} kN, \quad \mathbf{U} = \begin{bmatrix} U_1 \\ U_2 \\ U_3 \end{bmatrix} \tag{57}$$

- fulfilment of boundary conditions ($U_1 = V_1 = 0$) and solution of system of equations

$$EA \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} U_2 \\ U_3 \end{bmatrix} = \begin{bmatrix} 10 \\ 25 \end{bmatrix} kN \quad \rightarrow \quad \begin{bmatrix} U_1 \\ U_2 \\ U_3 \end{bmatrix} = \frac{1}{EA} \begin{bmatrix} 0 \\ 35 \\ 60 \end{bmatrix} \tag{58}$$

- determination of the reaction force

$$\mathbf{R} = \mathbf{KU} - \mathbf{F} = \mathbf{K} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \frac{1}{EA} \begin{bmatrix} 0 \\ 35 \\ 60 \end{bmatrix} - \begin{bmatrix} 5 \\ 10 \\ 25 \end{bmatrix} kN = \begin{bmatrix} -40 \\ 0 \\ 0 \end{bmatrix} kN \tag{59}$$

- determination of continuous interpolation of u for both elements

$$u(x) = \begin{cases} N_1(x)\,U_1 + N_2(x)\,U_2 = \dfrac{35}{EA}x, \quad e = 1 \\ N_1(x-d)\,U_2 + N_2(x-d)\,U_3 = \\ (1-(x-1))\dfrac{35}{EA} + (x-1)\dfrac{60}{EA} = \dfrac{25}{EA}x + \dfrac{10}{EA}, \quad e = 2 \end{cases} \tag{60}$$

- determination of discontinuous interpolation of N for both elements

$$N(x) = EAu'(x) = EA\mathbf{B}(x)\mathbf{U} = \begin{cases} 35kN, & e=1 \\ 25kN, & e=2 \end{cases} \tag{61}$$

Fig. 8 shows a comparison of the analytical solution and the FEM solution for $N = 2$ (example solved) and additionally for $N = 10$, assuming the non-physical value of $EA = 1$. It can be seen that the analytical displacement and FEM displacement values agree at nodes, and there is a non-zero difference between them outside the nodes. This is not a coincidence - this is the property of FEM for 1D linear tasks. In turn, FEM approximation of longitudinal force is a piecewise constant function, which means the discontinuity of such a solution at nodes.

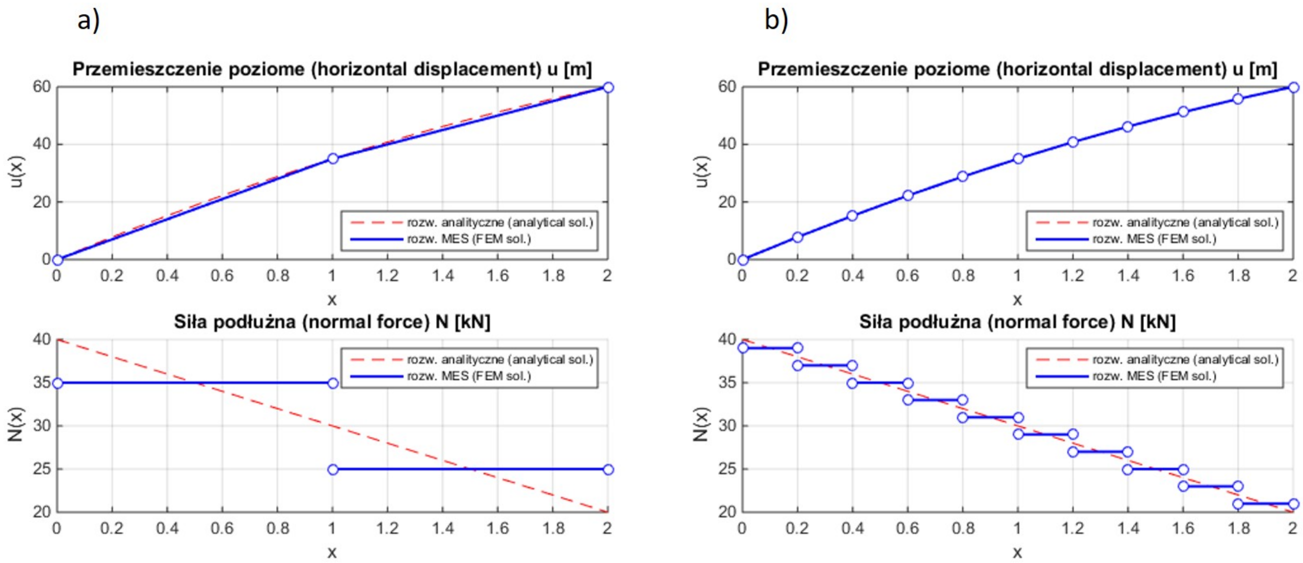

Figure 8: Graph of the horizontal displacement and normal force for a) FE mesh with 2 elements, b) FE mesh with 10 elements; with linear interpolation in each case

2.5.2 1D static with hierarchical quadratic shape functions

Next, we'll use 2nd order hierarchical interpolation to improve the quality of the FEM solution from the previous calculation example. We will take the same mechanical data for L, q_0 and P and unspecified EA. The division into 2 finite elements will be maintained, but each of them will be an element with three degrees of freedom. The numbering of nodes and degrees of freedom is shown in Fig. 9. Using the previously derived

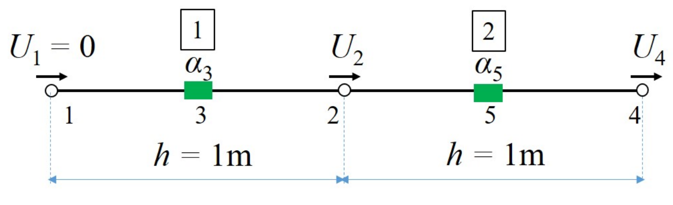

Figure 9: FE mesh for the computational problem with hierarchical interpolation of the second order

formulas for the local stiffness matrix and the load vector for a constant load, we get

$$\mathbf{K}^{(1)} = \mathbf{K}^{(2)} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & \frac{1}{3} \end{bmatrix}, \quad \mathbf{F}^{(1)} = \mathbf{F}^{(2)} = 5 \begin{bmatrix} 1 \\ 1 \\ -\frac{1}{3} \end{bmatrix} kN \tag{62}$$

After aggregating these quantities, we get a global system of equations

$$\mathbf{K}\mathbf{Q} = \mathbf{F},$$

$$\mathbf{K} = EA\begin{bmatrix} 1 & -1 & 0 & 0 & 0 \\ -1 & 2 & 0 & -1 & 0 \\ 0 & 0 & \frac{1}{3} & 0 & 0 \\ 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{3} \end{bmatrix}, \quad \mathbf{F} = 5\begin{bmatrix} 1 \\ 2 \\ -\frac{1}{3} \\ 1 \\ -\frac{1}{3} \end{bmatrix} kN + \begin{bmatrix} 0 \\ 0 \\ 0 \\ 20 \\ 0 \end{bmatrix} kN = \begin{bmatrix} 5 \\ 10 \\ -\frac{5}{3} \\ 25 \\ -\frac{5}{3} \end{bmatrix} kN, \quad \mathbf{Q} = \begin{bmatrix} U_1 \\ U_2 \\ \alpha_3 \\ U_4 \\ \alpha_5 \end{bmatrix} \tag{63}$$

whose solution, after considering the boundary conditions ($U_1 = V_1 = 0$), is a vector

$$\mathbf{Q} = \frac{1}{EA}\begin{bmatrix} 0 & 35 & -5 & 60 & -5 \end{bmatrix}^{\mathrm{T}} \tag{64}$$

On this basis, we find displacement functions

$$u(x) = \begin{cases} 0 + \frac{35}{EA}x - \frac{5}{EA}x(x-1) = \frac{1}{EA}\left(-5x^2 + 40x\right), & e = 1 \\ \frac{35}{EA}(1 - (x-1)) + \frac{60}{EA}(x-1) - \frac{5}{EA}(x-1)(x-2) = \frac{1}{EA}\left(-5x^2 + 40x\right), & e = 2 \end{cases} \tag{65}$$

and strength in the elements

$$N(x) = EAu'(x) = \begin{cases} -10x + 40, & e = 1 \\ -10x + 40, & e = 2 \end{cases} \tag{66}$$

which coincide with each other and with the analytical solution (since it is described by the same polynomial as FEM interpolation). Obviously, in this case the analytical solution and FEM will be identical, regardless of the number of elements. Therefore, to obtain the same result, one finite element $h = 2m$ with a second order hierarchical interpolation would suffice, which is worth checking yourself.

2.5.3 1D second order boundary value problem

We will solve the boundary problem that has no mechanical interpretation, namely the second order differential equation, also containing the first order component.

$$y'' - xy' = 2, \quad x \in (-1, 2), \quad y(-1) = 1, \quad y(2) = -1 \tag{67}$$

We assume two finite elements, the first element $e = 1, x \in [-1, 0]$ has linear interpolation, the second element $e = 2, x \in [0, 2]$ has hierarchic quadratic interpolation. The relevant variational formulation is

$$\int_{-1}^{2}\left(v'y' + xvy'\right)dx = -2\int_{-1}^{2} vdx, \quad y \in H^1 + \bar{y}, \quad \forall v \in H_0^1 \tag{68}$$

whereas at the element level, we have

$$\mathbf{K}^{(e)}\mathbf{Y} = \mathbf{F}^{(e)}, \quad \mathbf{K}^{(e)} = \int_0^h \left(\mathbf{B}^{\mathrm{T}}\mathbf{B} + (x+d)\,\mathbf{N}^{\mathrm{T}}\mathbf{B}\right)dx, \quad \mathbf{F}^{(e)} = -2\int_0^h \mathbf{N}^{\mathrm{T}}dx \tag{69}$$

For the first element, we have $\mathbf{N} = \begin{bmatrix} 1-x & x \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} -1 & 1 \end{bmatrix}$, and

$$\mathbf{K}^{(1)} = \int_0^1 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + (x-1)\begin{bmatrix} x-1 & 1-x \\ -x & x \end{bmatrix} dx = \begin{bmatrix} \frac{4}{3} & -\frac{4}{3} \\ -\frac{5}{6} & \frac{5}{6} \end{bmatrix}, \quad \mathbf{F}^{(1)} = -\begin{bmatrix} 1 \\ 1 \end{bmatrix} \tag{70}$$

For the second element, we have $\mathbf{N} = \begin{bmatrix} 1-\frac{x}{2} & \frac{x}{2} & x(x-2) \end{bmatrix}$, $\mathbf{B} = \begin{bmatrix} -\frac{1}{2} & \frac{1}{2} & 2x-2 \end{bmatrix}$, and

$$\mathbf{K}^{(2)} = \int_0^2 \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & (2x-2)^2 \end{bmatrix} + x \begin{bmatrix} 1-\frac{x}{2} \\ \frac{x}{2} \\ x(x-2) \end{bmatrix} \begin{bmatrix} -\frac{1}{2} & \frac{1}{2} & 2x-2 \end{bmatrix} dx$$

$$= \begin{bmatrix} \frac{1}{2} & -\frac{1}{2} & 0 \\ -\frac{1}{2} & \frac{1}{2} & 0 \\ 0 & 0 & \frac{8}{3} \end{bmatrix} + \begin{bmatrix} -\frac{1}{3} & -\frac{1}{3} & 0 \\ -\frac{2}{3} & \frac{2}{3} & \frac{4}{3} \\ \frac{2}{3} & -\frac{2}{3} & -\frac{8}{15} \end{bmatrix} = \begin{bmatrix} \frac{1}{6} & -\frac{1}{6} & 0 \\ -\frac{7}{6} & \frac{7}{6} & \frac{4}{3} \\ \frac{2}{3} & -\frac{2}{3} & \frac{32}{15} \end{bmatrix}, \quad \mathbf{F}^{(2)} = -\begin{bmatrix} 2 \\ 2 \\ -\frac{8}{3} \end{bmatrix} \tag{71}$$

After assembling

$$\mathbf{K}\begin{bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ \alpha_4 \end{bmatrix} = \mathbf{F}, \quad \mathbf{K} = \begin{bmatrix} \frac{4}{3} & -\frac{4}{3} & 0 & 0 \\ -\frac{5}{6} & 1 & -\frac{1}{6} & 0 \\ 0 & -\frac{7}{6} & \frac{7}{6} & \frac{4}{3} \\ 0 & \frac{2}{3} & -\frac{2}{3} & \frac{32}{15} \end{bmatrix}, \quad \mathbf{F} = -\begin{bmatrix} \frac{1}{3} \\ 2 \\ -\frac{8}{3} \end{bmatrix} \tag{72}$$

After enforcing boundary conditions

$$\mathbf{F}_{bc} = \mathbf{F} - \mathbf{K}\begin{bmatrix} 1 \\ 0 \\ -1 \\ 0 \end{bmatrix} = \begin{bmatrix} -\frac{7}{3} \\ -\frac{7}{3} \\ -\frac{5}{6} \\ 2 \end{bmatrix}, \quad \begin{bmatrix} 1 & 0 \\ \frac{2}{3} & \frac{32}{15} \end{bmatrix}\begin{bmatrix} Y_2 \\ \alpha_4 \end{bmatrix} = \begin{bmatrix} -\frac{7}{3} \\ 2 \end{bmatrix} \quad \rightarrow \quad \begin{bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ \alpha_4 \end{bmatrix} = \begin{bmatrix} 1 \\ -\frac{7}{3} \\ -1 \\ \frac{5}{3} \end{bmatrix} \tag{73}$$

Solution interpolation at elements

$$y(x) = \begin{cases} (1-(x+1))\,Y_1 + (x+1)\,Y_2 = -\frac{10}{3}x - \frac{7}{3}, & e = 1, \\ \left(1-\frac{x}{2}\right)Y_2 + \frac{x}{2}Y_3 + x(x-2)\,\alpha_4 = \frac{5}{3}x^2 - \frac{8}{3}x - \frac{7}{3}, & e = 2 \end{cases} \tag{74}$$

Solution derivative interpolation at elements

$$y'(x) = \begin{cases} -\frac{10}{3}, & e = 1, \\ \frac{10}{3}x - \frac{8}{3}, & e = 2 \end{cases} \tag{75}$$

2.5.4 1D natural vibrations

We will determine the dynamic characteristics for the bar from the previous tasks, namely fixed at the left end, free at the right end, length $L = 2m$ and with unspecified E, A and ρ. We will use a FEM mesh consisting of two elements of equal lengths, linear interpolation and consistent mass matrices. The consecutive steps of the FEM analysis of natural vibrations of this bar are as follows

- generalized eigen problem at the global level

$$\left(EA\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} - \omega^2\frac{A\rho}{6}\begin{bmatrix} 2 & 1 & 0 \\ 1 & 4 & 1 \\ 0 & 1 & 2 \end{bmatrix} \right)\begin{bmatrix} U_1 \\ U_2 \\ U_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix} \tag{76}$$

- fulfilment of the boundary conditions ($U_1 = V_1 = 0$)

$$\left(EA\begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2\frac{A\rho}{6}\begin{bmatrix} 4 & 1 \\ 1 & 2 \end{bmatrix}\right)\begin{bmatrix} U_2 \\ U_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \tag{77}$$

- transformation into the standard eigen problem

$$\begin{bmatrix} 4 & 1 \\ 1 & 2 \end{bmatrix}^{-1} = \frac{1}{7}\begin{bmatrix} 2 & -1 \\ -1 & 4 \end{bmatrix} \tag{78}$$

$$\left(EA\frac{1}{7}\begin{bmatrix} 2 & -1 \\ -1 & 4 \end{bmatrix}\begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} - \omega^2\frac{A\rho}{6}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\right)\begin{bmatrix} U_2 \\ U_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \tag{79}$$

$$\left(\begin{bmatrix} 5 & -3 \\ -6 & 5 \end{bmatrix} - \lambda\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\right)\begin{bmatrix} U_2 \\ U_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \tag{80}$$

with $\lambda = \dfrac{7\omega^2\rho}{6E}$,

- characteristic equation and its solution

$$\begin{aligned} &\det\begin{bmatrix} 5-\lambda & -3 \\ -6 & 5-\lambda \end{bmatrix} = 0 \quad\rightarrow\quad (5-\lambda)^2 - 18 = 0 \quad\rightarrow \\ &\rightarrow\quad \lambda^2 - 10\lambda + 7 = 0 \quad\rightarrow\quad \lambda_{1,2} = 5 \mp 3\sqrt{2} \quad\rightarrow\quad \omega_{1,2} = \sqrt{\frac{6\lambda_{1,2}E}{7\rho}} = \sqrt{\frac{E}{\rho}}\begin{bmatrix} 0.8057 \\ 2.8147 \end{bmatrix} \end{aligned} \tag{81}$$

For the first, smallest and dominant frequency $\omega = \omega_1 = 0.8057\sqrt{\dfrac{E}{\rho}}$, we will determine the form of free vibrations. Its square $\omega^2 = \dfrac{6\left(5-3\sqrt{2}\right)E}{7\rho}$ is provided into the eigen problem (77). Since those two equations are dependent (this was the condition of zeroing the determinant of the matrix of coefficients), we take one of them, for instance the second equation, and complete it with the condition of unit length

$$\begin{cases} EA\left(-U_2 + U_3\right) - \dfrac{6\left(5-3\sqrt{2}\right)E}{7\rho}\dfrac{A\rho}{6}\left(U_2 + 2U_3\right) = 0 \\ U_2^2 + U_3^2 = 1 \end{cases} \tag{82}$$

After ordering the components, we get a simple non-linear system of equations and its solution

$$\begin{cases} U_2\left(\sqrt{2}-4\right) + U_3\left(2\sqrt{2}-1\right) = 0 \\ U_2^2 + U_3^2 = 1 \end{cases} \quad\rightarrow\quad \begin{cases} U_2 = \dfrac{1}{\sqrt{3}} = 0.5774 \\ U_3 = U_2\sqrt{2} = \sqrt{\dfrac{2}{3}} = 0.8165 \end{cases} \tag{83}$$

Therefore $\mathbf{U}_1 = \begin{bmatrix} 0 & 0.5774 & 0.8165 \end{bmatrix}^{\mathrm{T}}$. Similarly, the $\mathbf{U}_2$ can be determined for the second frequency ω_2.

Comparison of solutions (the first four - the lowest - natural frequencies and the corresponding forms of vibrations), for $N = 10$ elements and two different mass matrices are shown in Fig. 10. In addition to the previous length of $L = 2m$, the fixed values $E = 10^5 N/m$, $A = 10^{-3} m^2$ and $\rho = 10^3 kg/m$ were assumed for calculations. Eigen vectors have been normalized (they have unit lengths). The higher the vibration frequency is, the greater the difference between the full and lumped matrices results - and thus the lowest frequencies are most accurately calculated. It is noteworthy that for the first and second frequencies, the graphs almost coincide, but for the third and fourth frequencies, the vibration forms have opposite senses. However strange it looks, this is correct from the point of view of algebra. Eigen vectors - as already mentioned - are unambiguous to the direction only.

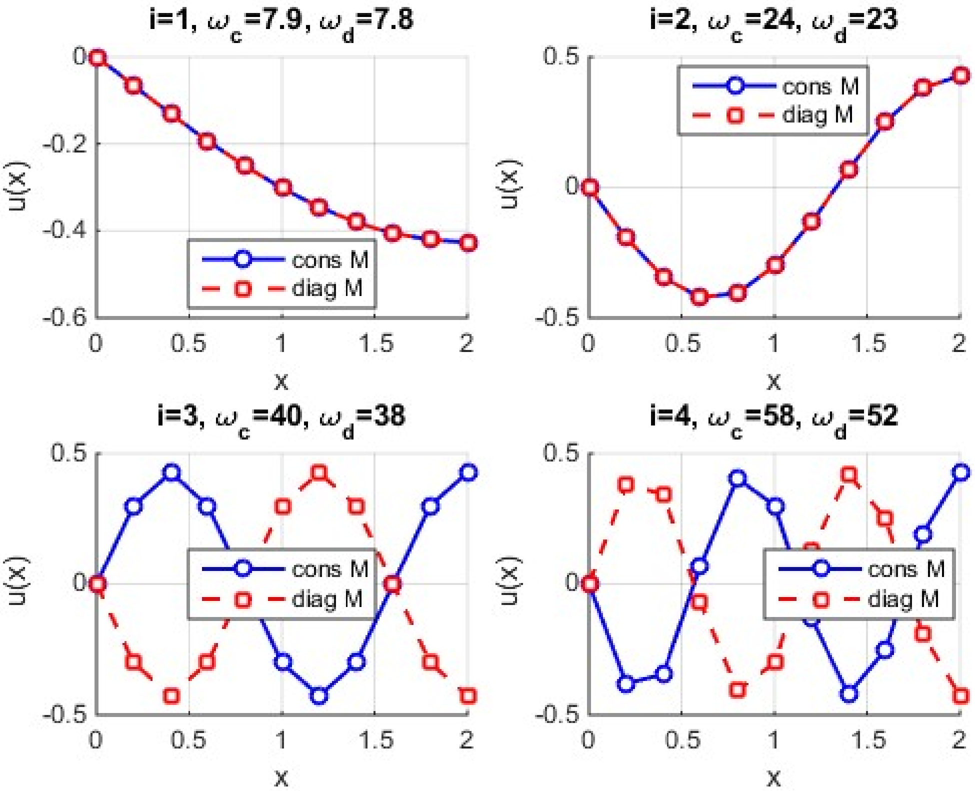

Figure 10: Four first forms and frequencies of the natural vibrations for the considered bar, for $N = 10$ finite elements and two mass matrices: consistent and lumped

2.5.5 1D forced vibrations

Next, the bar's forced vibration are analysed: we will determine displacements and velocities for two consecutive unknown time steps. We will use the same data for calculations as above. We assume the time step $\Delta t = 0.1s$, zero initial displacement and velocity, as well as the harmonic dynamic load function $f(x,t) = x\sin(10t)$, $P = 0$. Let us check that the assumed frequency of the forcing load, i.e. $\omega_f = 10$, is greater than the first, dominant natural vibration frequency, which, for the adopted data is $\omega_1 = 8.05$. Therefore, there is no direct danger of resonance (though such situations should be avoided in practice), as long as calculations for a denser FE mesh confirm that this result is correct and accurate enough. We are still using two finite elements of equal length, linear interpolation and consistent mass matrices. For such a coarse FE mesh, you guarantee the accuracy of the results. The coarse FE mesh is accepted only for practical reasons - to carry out manual calculations. Let us follow the subsequent calculation stages

- Newmark algorithm for acceleration for FE system with boundary condition included

$$\ddot{\mathbf{U}}_{k+1} = \left(\frac{A\rho}{6} \begin{bmatrix} 4 & 1 \\ 1 & 2 \end{bmatrix} + \frac{1}{2}(0.1)^2 EA \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \right)^{-1} \cdot \cdot \left(\begin{bmatrix} 1 \\ 2 \end{bmatrix} \sin(10 \cdot 0.1 \cdot (k+1)) - EA \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} (\mathbf{U}_k + 0.1\mathbf{V}_k) \right) \tag{84}$$

- Newmark algorithm for acceleration after substitution of data value

$$\ddot{\mathbf{U}}_{k+1} = \frac{1}{6} \begin{bmatrix} 10 & -2 \\ -2 & 5 \end{bmatrix}^{-1} \left(\begin{bmatrix} 1 \\ 2 \end{bmatrix} \sin(10 \cdot 0.1 \cdot (k+1)) - 10^2 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} (\mathbf{U}_k + 0.1\mathbf{V}_k) \right) = = \frac{1}{276} \begin{bmatrix} 5 & 2 \\ 2 & 10 \end{bmatrix} \left(\begin{bmatrix} 1 \\ 2 \end{bmatrix} \sin(10 \cdot 0.1 \cdot (k+1)) - 10^2 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} (\mathbf{U}_k + 0.1\mathbf{V}_k) \right) \tag{85}$$

- computations for the first unknown time level ($k = 0$, $t = 0.1s$)

$$\ddot{\mathbf{U}}_1 = \frac{1}{276} \begin{bmatrix} 5 & 2 \\ 2 & 10 \end{bmatrix} \left(\begin{bmatrix} 1 \\ 2 \end{bmatrix} \sin(1) - 10^2 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \left(\begin{bmatrix} 0 \\ 0 \end{bmatrix} + 0.1 \begin{bmatrix} 0 \\ 0 \end{bmatrix} \right) \right) = \begin{bmatrix} 0.0274 \\ 0.0671 \end{bmatrix} \frac{m}{s^2} \tag{86}$$

$$\mathbf{V}_1 = \begin{bmatrix} 0 \\ 0 \end{bmatrix} + 0.1 \begin{bmatrix} 0.0274 \\ 0.0671 \end{bmatrix} = \begin{bmatrix} 0.0027 \\ 0.0067 \end{bmatrix} \frac{m}{s} \tag{87}$$

$$\mathbf{U}_1 = \begin{bmatrix} 0 \\ 0 \end{bmatrix} + 0.1 \begin{bmatrix} 0 \\ 0 \end{bmatrix} + \frac{1}{2}(0.1)^2 \begin{bmatrix} 0.0274 \\ 0.0671 \end{bmatrix} = \begin{bmatrix} 0.0001 \\ 0.0003 \end{bmatrix} m \tag{88}$$

- computations for the second unknown time level ($k = 1$, $t = 0.2s$)

$$\begin{aligned} \ddot{\mathbf{U}}_2 = \frac{1}{276} \begin{bmatrix} 5 & 2 \\ 2 & 10 \end{bmatrix} \left(\begin{bmatrix} 1 \\ 2 \end{bmatrix} \sin(2) - 10^2 \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \left(\begin{bmatrix} 0.0001 \\ 0.0003 \end{bmatrix} + 0.1 \begin{bmatrix} 0.0027 \\ 0.0067 \end{bmatrix} \right) \right) \\ = \begin{bmatrix} 0.0296 \\ 0.0705 \end{bmatrix} \frac{m}{s^2} \end{aligned} \tag{89}$$

$$\mathbf{V}_2 = \begin{bmatrix} 0.0027 \\ 0.0067 \end{bmatrix} + 0.1 \begin{bmatrix} 0.0296 \\ 0.0705 \end{bmatrix} = \begin{bmatrix} 0.0057 \\ 0.0138 \end{bmatrix} \frac{m}{s} \tag{90}$$

$$\mathbf{U}_2 = \begin{bmatrix} 0.0001 \\ 0.0003 \end{bmatrix} + 0.1 \begin{bmatrix} 0.0027 \\ 0.0067 \end{bmatrix} + \frac{1}{2}(0.1)^2 \begin{bmatrix} 0.0296 \\ 0.0705 \end{bmatrix} = \begin{bmatrix} 0.0005 \\ 0.0013 \end{bmatrix} m \tag{91}$$

3 Formulation of 3D linear elastic problem

Let us start with the complete set of differential equations for the stationary 3D linear elastic problem. Let us assume that 3D domain Ω is subjected to mass forces $\mathbf{b}$ while on this boundary $\partial\Omega$, either known displacements $\bar{\mathbf{u}}$ (on $\partial\Omega_u$) or known traction forces $\mathbf{t}$ (on $\partial\Omega_\sigma$, in normal outward direction) are given. Note that $\partial\Omega_\sigma \cup \partial\Omega_u = \partial\Omega$. On other other hand, the following unknown mechanical fields appear, namely displacements (vector $\mathbf{u}$), stresses (tensor $\boldsymbol{\sigma}$) and strains (tensor $\boldsymbol{\varepsilon}$). Similarly as in 1D case, we deal with three types of equations, namely balance, geometric and physical (constitutive) equations, relating known ($\mathbf{b}$, $\bar{\mathbf{u}}$,$\mathbf{t}$) and unknown fields ($\mathbf{u}$, $\boldsymbol{\sigma}$, $\boldsymbol{\varepsilon}$). Balance (first Cauchy or Navier) equations, resulting from the conversation of momentum law, may be expressed using matrix

$$\mathrm{div}\boldsymbol{\sigma} + \mathbf{b} = \mathbf{0} \quad \forall x \in \Omega, \quad \boldsymbol{\sigma}\mathbf{n} = \mathbf{t} \quad \forall x \in \partial\Omega_\sigma \tag{92}$$

or index notation

$$\sigma_{ij,j} + b_i = 0 \quad \forall x \in \Omega, \quad \sigma_{ij} n_i = t_i \quad \forall x \in \partial\Omega_\sigma \tag{93}$$

under the assumptions of small displacements. Here, $\mathbf{n}$ is the versor, outward normal to the boundary $\partial\Omega_\sigma$. Moreover, from the conservation of the angular momentum law, the symmetry of the stress tensor may be derived (second Cauchy equations)

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}^{\mathrm{T}}, \quad \sigma_{ij} = \sigma_{ji} \tag{94}$$

Geometric equations result from deformation and kinematic theories, relating strains and displacements. Assuming small strains, they may be expressed using either matrix

$$\boldsymbol{\varepsilon} = \boldsymbol{\nabla}_s \mathbf{u} = \frac{1}{2}\left(\boldsymbol{\nabla}\mathbf{u} + \boldsymbol{\nabla}\mathbf{u}^{\mathrm{T}}\right), \quad \forall x \in \Omega \quad \mathbf{u} = \bar{\mathbf{u}}, \quad \forall x \in \partial\Omega_u \tag{95}$$

or index notation

$$\varepsilon_{ij} = \frac{1}{2}\left(u_{i,j} + u_{j,i}\right), \quad \forall x \in \Omega, \quad u_i = \bar{u}_i, \quad \forall x \in \partial\Omega_u \tag{96}$$

Eventually, physical equations may be expressed in general manner

$$\boldsymbol{\sigma} = \mathbf{D} : \boldsymbol{\varepsilon}, \quad \sigma_{ij} = D_{ijkl}\varepsilon_{kl} \tag{97}$$

where $\mathbf{D}$ is the 4^{th} rank tensor of 81 components. Assuming all known geometric symmetries, we have 21 independent components, constituting the set of material parameters for anisotropic material. Furthermore, if we assume the isotropic perfectly linear elastic behaviour, we may reduce the number of material constants, to two, namely Young modulus E and Poisson coefficient ν. In such a case, physical equations are usually expressed in the following manner

$$\sigma_{ij} = \lambda\varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij} \tag{98}$$

in which two Lame constants appear, namely

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}, \quad \mu = \frac{E}{2(1+\nu)} \tag{99}$$

The FE analysis requires relevant variational formulation. In order to derive it, we multiply (in a scalar manner) both sides of (92) by an arbitrary vector test function $\mathbf{v}$ and integrate over the domain Ω

$$\int_\Omega \mathbf{v} \cdot \mathrm{div}\boldsymbol{\sigma}\, d\Omega = -\int_\Omega \mathbf{v} \cdot \mathbf{b}\, d\Omega \tag{100}$$

By integration by parts and introduction of the natural boundary condition from (92), we obtain

$$\int_\Omega \boldsymbol{\nabla}\mathbf{v} : \boldsymbol{\sigma}\, d\Omega = \int_\Omega \mathbf{v} \cdot \mathbf{b}\, d\Omega + \int_{\partial\Omega_\sigma} \mathbf{v} \cdot \mathbf{t}\, d\partial\Omega, \quad \boldsymbol{\sigma} \in L^2, \quad \forall \mathbf{v} \in H_0^1 \tag{101}$$

Afterwards, we substitute $\boldsymbol{\sigma}$ from (97), which leads to

$$\int_\Omega \boldsymbol{\nabla}\mathbf{v} : \mathbf{D} : \boldsymbol{\varepsilon}\, d\Omega = \int_\Omega \mathbf{v} \cdot \mathbf{b}\, d\Omega + \int_{\partial\Omega_\sigma} \mathbf{v} \cdot \mathbf{t}\, d\partial\Omega, \quad \boldsymbol{\varepsilon} \in L^2, \quad \forall \mathbf{v} \in H_0^1 \tag{102}$$

Note that $\boldsymbol{\nabla}\mathbf{v}$ may be decomposed into symmetric and antisymmetric parts, namely

$$\boldsymbol{\nabla}\mathbf{v} = \boldsymbol{\nabla}_s\mathbf{v} + \boldsymbol{\nabla}_a\mathbf{v} = \frac{1}{2}\left(\boldsymbol{\nabla}\mathbf{v} + \boldsymbol{\nabla}\mathbf{v}^{\mathrm{T}}\right) + \frac{1}{2}\left(\boldsymbol{\nabla}\mathbf{v} - \boldsymbol{\nabla}\mathbf{v}^{\mathrm{T}}\right) \tag{103}$$

Since $\boldsymbol{\varepsilon}$ is the symmetric tensor, the scalar product of $\boldsymbol{\varepsilon}$ and $\boldsymbol{\nabla}_a\mathbf{v}$ is zero. Therefore, $\boldsymbol{\nabla}_s\mathbf{v}$ are strains induced on displacement variation $\mathbf{v}$. Taking this fact into account and substituting $\boldsymbol{\varepsilon}$ from (95), we obtain the final variational formulation of the 3D stationary linear elastic problem

$$\int_\Omega \boldsymbol{\nabla}_s\mathbf{v} : \mathbf{D} : \boldsymbol{\nabla}_s\mathbf{u}\, d\Omega = \int_\Omega \mathbf{v} \cdot \mathbf{b}\, d\Omega + \int_{\partial\Omega_\sigma} \mathbf{v} \cdot \mathbf{t}\, d\partial\Omega, \quad \mathbf{u} \in H_0^1 + \bar{\mathbf{u}}, \quad \forall \mathbf{v} \in H_0^1 \tag{104}$$

where $\boldsymbol{\nabla}_s\mathbf{u}$ is computed according to the right-hand side of (95).

4 Selected 2D linear elastic problems

In this section, two particular linear elastic states are considered, namely the plane strain and stress state, followed by general principles of FE interpolation of displacement function on triangular and quadrangular elements. Instead of axis numbering with integers $(1, 2, 3)$, axes names x, y, z will be introduced. Therefore, $x_1 = x, x_2 = y, x_3 = z$, and $\mathbf{x} = \begin{bmatrix} x & y & z \end{bmatrix}$. Moreover, $\frac{\partial}{\partial x} =_{,x}$ etc.

4.1 Plain strain state

In this case, we distinguish the plane (x, y) and the third axis z, perpendicular to this plane, along which all derivatives vanish ($\frac{\partial}{\partial z}(\bullet) = 0$). In civil engineering, it may be applied in case of long, prismatic structures, like dam, dyke or thrust shaft, which length L is much larger than the characteristic dimension of this cross-section area. Therefore, we may cut off the representative plane with thickness of $1m$. Moreover, we may assume that the displacement field has the form

$$\mathbf{u}(\mathbf{x}) = \begin{bmatrix} u_x(x,y) \\ u_y(x,y) \\ 0 \end{bmatrix} \tag{105}$$

On the basis of geometric equations (95), we obtain the vector of non-zero independent strain components (according to Voight notation), namely

$$\boldsymbol{\varepsilon}(\mathbf{x}) = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} = \gamma_{xy} \end{bmatrix} = \begin{bmatrix} u_{x,x} \\ u_{y,y} \\ u_{x,y} + u_{y,x} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix} = \mathbf{L}\,\mathbf{u}(\mathbf{x}) \tag{106}$$

while other strain components are zero ($\varepsilon_{xz} = \varepsilon_{yz} = \varepsilon_{zz} = 0$). Similarly, from (97), we obtain non-zero independent stress components

$$\boldsymbol{\sigma}(\mathbf{x}) = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} \lambda(\varepsilon_{xx}+\varepsilon_{yy}) + 2\mu\varepsilon_{xx} \\ \lambda(\varepsilon_{xx}+\varepsilon_{yy}) + 2\mu\varepsilon_{yy} \\ 2\mu\varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} \lambda & \lambda+2\mu & 0 \\ \lambda+2\mu & \lambda & 0 \\ 0 & 0 & \mu \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} =$$
$$= \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \dfrac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} = \mathbf{D}\,\boldsymbol{\varepsilon}(\mathbf{x}) \tag{107}$$

while other are $\sigma_{xz} = \sigma_{yz} = 0$, though

$$\sigma_{zz} = \lambda(\varepsilon_{xx}+\varepsilon_{yy}) = \frac{\lambda}{2(\lambda+\mu)}(\sigma_{xx}+\sigma_{yy}) = \nu(\sigma_{xx}+\sigma_{yy}) \tag{108}$$

Additional, non-zero σ_{zz} component may be considered as the cross-section stress (− reaction), caused by the rejection of left and right parts of a structure, which prevent the considered plane model from deformation along the $x_3 = z$ axis.

Eventually, the balance equations (92) have to be satisfied, namely

$$\begin{cases} \sigma_{xx,x} + \sigma_{xy,y} + b_x = 0 \\ \sigma_{yx,x} + \sigma_{yy,y} + b_y = 0 \\ \sigma_{zz,z} + b_z = 0 \end{cases} \tag{109}$$

Note that $\sigma_{yx} = \sigma_{xy}$. From the last equation, since $\sigma_{zz,z} = 0$, we have the requirement for the mass force intensity $b_z = 0$, in plane strain state.

4.2 Plane stress state

In this case, we consider a structure which thickness h is much smaller that the characteristic dimension d of the plane cross-section. Therefore, we may assume that the load and stress tensor have non-zero plane components only, namely

$$\mathbf{b} = \begin{bmatrix} b_x \\ b_y \\ 0 \end{bmatrix}, \quad \mathbf{t} = \begin{bmatrix} t_x \\ t_y \\ 0 \end{bmatrix}, \quad \boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} \tag{110}$$

Automatically, we obtain two balance equations

$$\begin{cases} \sigma_{xx,x} + \sigma_{xy,y} + b_x = 0 \\ \sigma_{yx,x} + \sigma_{yy,y} + b_y = 0 \end{cases} \tag{111}$$

with $\sigma_{yx} = \sigma_{xy}$. On the other hand, when we express stresses using strains from (97), we have

$$\begin{cases} \sigma_{xx} = \lambda(\varepsilon_{xx}+\varepsilon_{yy}+\varepsilon_{zz}) + 2\mu\varepsilon_{xx} \\ \sigma_{yy} = \lambda(\varepsilon_{xx}+\varepsilon_{yy}+\varepsilon_{zz}) + 2\mu\varepsilon_{yy} \\ \sigma_{xy} = 2\mu\varepsilon_{xy} \end{cases} \quad \begin{cases} 0 = \lambda(\varepsilon_{xx}+\varepsilon_{yy}+\varepsilon_{zz}) + 2\mu\varepsilon_{zz} \\ 0 = 2\mu\varepsilon_{xz} \\ 0 = 2\mu\varepsilon_{yz} \end{cases} \tag{112}$$

As we may have suspected, $\varepsilon_{xz} = \varepsilon_{yz} = 0$, however ε_{zz} is not necessarily a zero component. In fact, we have

$$\varepsilon_{zz} = -\frac{\lambda(\varepsilon_{xx}+\varepsilon_{yy})}{\lambda+2\mu} = -\frac{\nu}{1-\nu}(\varepsilon_{xx}+\varepsilon_{yy}) \tag{113}$$

which may be substituted to formulas for σ_{xx} and σ_{yy}, yielding

$$\boldsymbol{\sigma}(\mathbf{x}) = \frac{2\mu}{\lambda+2\mu} \begin{bmatrix} 2\mu+1+\lambda & \lambda & 0 \\ \lambda & 2\mu+1+\lambda & 0 \\ 0 & 0 & \dfrac{1}{2}(\lambda+2\mu) \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} =$$
$$= \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} = \mathbf{D}\,\boldsymbol{\varepsilon}(\mathbf{x}) \tag{114}$$

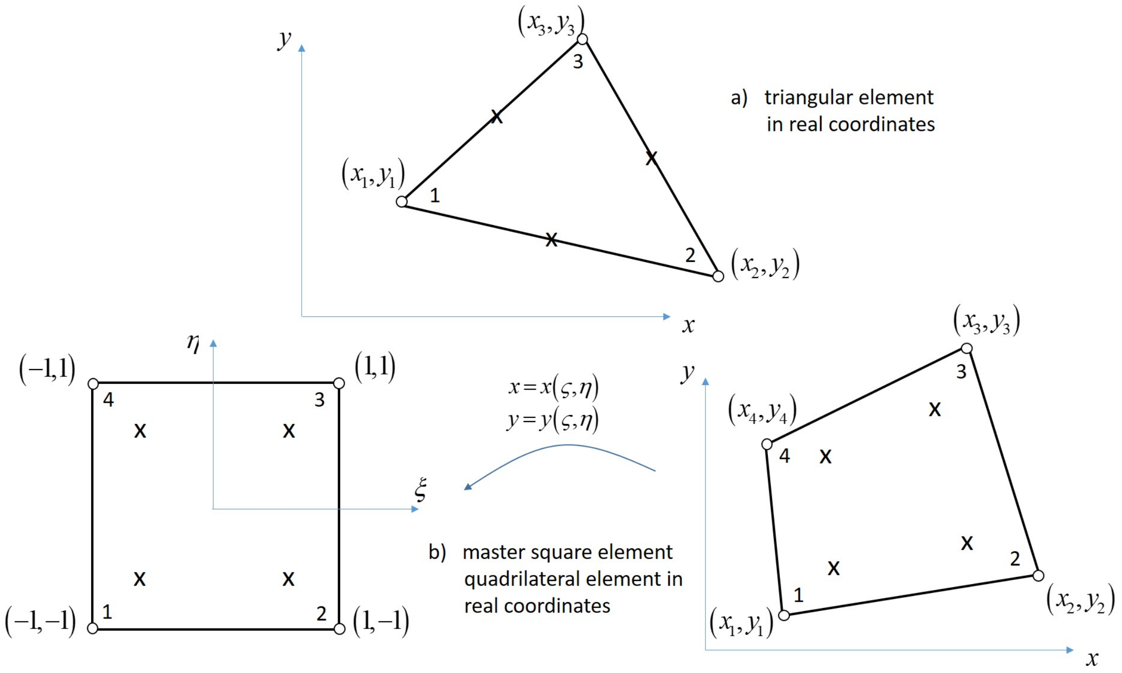

Figure 11: Finite elements for 2D analysis: a) triangular real elements, b) master square element and quadrilateral real element

Additional, non-zero ε_{zz} corresponds to the free displacement of a structure in the unblocked z direction, caused by in-plane loading. This non-zero displacement u_z may be observed in geometric equations, namely

$$\boldsymbol{\varepsilon}(\mathbf{x}) = \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix} = \mathbf{L}\,\mathbf{u}(\mathbf{x}) \tag{115}$$

as well as

$$\varepsilon_{zz} = u_{z,z} \quad \rightarrow \quad u_z = \varepsilon_{zz} z + u_z(0) \tag{116}$$

Regardless of the plane state type, the variational equation (104) has the following form

$$h \int_\Omega (\mathbf{Lv})^{\mathrm{T}} \mathbf{D}\,\mathbf{Lu}\, d\Omega = \int_\Omega \mathbf{v} \cdot \mathbf{b}\, d\Omega + \int_{\partial\Omega_\sigma} \mathbf{v} \cdot \mathbf{t}\, d\partial\Omega, \quad \mathbf{u} \in H_0^1 + \bar{\mathbf{u}}, \quad \forall \mathbf{v} \in H_0^1 \tag{117}$$

For the plane strain state, we put $h = 1m$.

4.3 FE approximation

The most commonly applied finite elements in 2D analysis are triangles and quadrangles (including squares, rectangles, rhombuses, parallelograms and trapezes). In the simplest case, FE interpolation uses nodes (degrees of freedom) located at their vertices. Therefore, linear or bi-linear interpolation schemes may be produced. On the other hand, additional nodes/degrees of freedom are required for higher order interpolation. They are usually located on elements sides and inside elements. Let us examine two isoparametric elements, namely 3-noded triangle and 4-noded quadrangle (Fig. 11). For the triangular elements, we will use the real coordinates (x, y) for generation of interpolation base $\mathbf{N}(x, y) = \begin{bmatrix} N_1(x,y) & N_2(x,y) & N_3(x,y) \end{bmatrix}$, namely three linear shape functions, assigned to each of element's vertices

$$N_j(x, y) = a_j x + b_j y + c_j, \quad j = 1, 2, 3 \tag{118}$$

Interpolation coefficients may be found from conditions

$$N_j(x_i, y_i) = \begin{cases} 0, & i \neq j, \\ 1, & i = j \end{cases} \tag{119}$$

which lead to the system of equations with three right-hand sides

$$\begin{bmatrix} x_1 & y_1 & 1 \\ x_2 & y_2 & 1 \\ x_3 & y_3 & 1 \end{bmatrix} \begin{bmatrix} a_1 & a_2 & a_3 \\ b_1 & b_2 & b_3 \\ c_1 & c_2 & c_3 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{120}$$

Similarly, gradient of the vector $\mathbf{N}$, of shape functions, may be determined using direct analytical differentiation

$$\boldsymbol{\nabla}\mathbf{N} = \begin{bmatrix} \dfrac{\partial \mathbf{N}}{\partial x} \\ \dfrac{\partial \mathbf{N}}{\partial y} \end{bmatrix} = \begin{bmatrix} a_1 & a_2 & a_3 \\ b_1 & b_2 & b_3 \end{bmatrix} \tag{121}$$

Integration over the finite element e

$$I = \int_e f(x, y)\, dx dy \tag{122}$$

of an arbitrary function $f = f(x, y)$ may be done twofold, either using analytical or numerical approach. In case of numerical approach, one may use Gauss formula, in which

$$I \approx J \sum_{k=1}^{N_G} f(x_k, y_k)\, \omega_k \tag{123}$$

in which J is the area of a triangle, (x_k, y_k) are coordinates of Gauss points and ω_k are integration weights. One of the simplest configurations, shown in Fig. 11a, consists of three integration points, located at triangle sides' centres and three weights equal $\frac{1}{3}$.

In case of quadrangles, it is more convenient to derive interpolation schemes on a square master element and to transform coordinates between master and real elements using transformation formulas

$$\left\{ \begin{array}{l} x = x(\xi, \eta) \\ y = y(\xi, \eta) \end{array} \right. \tag{124}$$

In order to determine them, let us define four element shape functions on the master element, using Lagrange interpolation formula, namely

$$\begin{gathered} \mathbf{N}(\xi, \eta) = \begin{bmatrix} \frac{1}{4}(\xi - 1)(\eta - 1) & -\frac{1}{4}(\xi + 1)(\eta - 1) & \frac{1}{4}(\xi + 1)(\eta + 1) & -\frac{1}{4}(\xi - 1)(\eta + 1) \end{bmatrix} = \\ = \frac{1}{4} \begin{bmatrix} -\xi - \eta + \xi\eta + 1 & \xi - \eta - \xi\eta + 1 & \xi + \eta + \xi\eta + 1 & -\xi + \eta - \xi\eta + 1 \end{bmatrix} \end{gathered} \tag{125}$$

According to isoparametric features, real geometry may be interpolated as follows

$$\left\{ \begin{array}{l} x(\xi, \eta) = \mathbf{N}(\xi, \eta)\, \mathbf{X} \\ y(\xi, \eta) = \mathbf{N}(\xi, \eta)\, \mathbf{Y} \end{array} \right. \tag{126}$$

where $\mathbf{X}$ and $\mathbf{Y}$ are vectors containing real x and y coordinates, respectively. Let us examine small increments of both coordinates

$$\begin{bmatrix} dx \\ dy \end{bmatrix} = \begin{bmatrix} \dfrac{\partial x}{\partial \xi} d\xi + \dfrac{\partial x}{\partial \eta} d\eta \\ \dfrac{\partial y}{\partial \xi} d\xi + \dfrac{\partial y}{\partial \eta} d\eta \end{bmatrix} = \begin{bmatrix} x'_\xi & x'_\eta \\ y'_\xi & y'_\eta \end{bmatrix} \begin{bmatrix} d\xi \\ d\eta \end{bmatrix} = \mathbf{J}(\xi, \eta) \begin{bmatrix} d\xi \\ d\eta \end{bmatrix} \tag{127}$$

where $\mathbf{J}$ is the Jacobi matrix. Differentiation of an arbitrary function $f = f(x, y)$, defined on the real element, may be performed as

$$\begin{bmatrix} \dfrac{\partial f}{\partial \xi} \\ \dfrac{\partial f}{\partial \eta} \end{bmatrix} = \begin{bmatrix} \dfrac{\partial f}{\partial x}\dfrac{\partial x}{\partial \xi} + \dfrac{\partial f}{\partial y}\dfrac{\partial y}{\partial \xi} \\ \dfrac{\partial f}{\partial x}\dfrac{\partial x}{\partial \eta} + \dfrac{\partial f}{\partial y}\dfrac{\partial y}{\partial \eta} \end{bmatrix} = \begin{bmatrix} x'_\xi & y'_\xi \\ x'_\eta & y'_\eta \end{bmatrix} \begin{bmatrix} \dfrac{\partial f}{\partial x} \\ \dfrac{\partial f}{\partial y} \end{bmatrix} = \mathbf{J}^{\mathrm{T}} \begin{bmatrix} \dfrac{\partial f}{\partial x} \\ \dfrac{\partial f}{\partial y} \end{bmatrix} \tag{128}$$

Therefore

$$\nabla f(x,y) = \nabla f(x(\xi,\eta), y(\xi,\eta)) = \mathbf{J}^{-\mathrm{T}} \begin{bmatrix} \dfrac{\partial f}{\partial \xi} \\ \dfrac{\partial f}{\partial \eta} \end{bmatrix} \tag{129}$$

Similarly, the gradient of the vector of shape functions may be computed as

$$\nabla \mathbf{N}(x,y) = \mathbf{J}^{-\mathrm{T}} \begin{bmatrix} \mathbf{N}'_{\xi} \\ \mathbf{N}'_{\eta} \end{bmatrix} = \frac{\mathbf{J}^{-\mathrm{T}}}{4} \begin{bmatrix} \eta-1 & 1-\eta & \eta+1 & -\eta-1 \\ \xi-1 & -1-\xi & \xi+1 & 1-\xi \end{bmatrix} \tag{130}$$

Finally, definite integral of function f may be computed as

$$I = \int_e f(x,y)\, dxdy = \int_{-1}^{1}\int_{-1}^{1} f(x(\xi,\eta), y(\xi,\eta)) \det \mathbf{J}\, d\xi d\eta \approx \det \mathbf{J} \sum_{k=1}^{N_G} f(x(\xi_k,\eta_k), y(\xi_k,\eta_k))\, \omega_k \tag{131}$$

The most commonly applied configuration consists of 4 Gauss points, shown in Fig. 11b, namely

$$\left\{ \left(-\frac{1}{\sqrt{3}}, -\frac{1}{\sqrt{3}}\right) \; \left(\frac{1}{\sqrt{3}}, -\frac{1}{\sqrt{3}}\right) \; \left(\frac{1}{\sqrt{3}}, \frac{1}{\sqrt{3}}\right) \; \left(-\frac{1}{\sqrt{3}}, \frac{1}{\sqrt{3}}\right) \right\} \tag{132}$$

with all weights equal 1.

All components of the displacement field are interpolated by means of the same shape functions, namely

$$\mathbf{u}(x,y) = \begin{bmatrix} u_x(x,y) \\ u_y(x,y) \end{bmatrix} = \begin{bmatrix} \mathbf{N}(x,y)\,\mathbf{U}_x \\ \mathbf{N}(x,y)\,\mathbf{U}_y \end{bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \dots & N_m & 0 \\ 0 & N_1 & 0 & N_2 & 0 & \dots & N_m \end{bmatrix} \mathbf{Q} = \mathbf{N}^{(e)}(x,y)\,\mathbf{Q} \tag{133}$$

where $\mathbf{U}_x$ and $\mathbf{U}_y$ are column vectors of element degrees of freedom (i.e., horizontal and vertical displacements, respectively) whereas $\mathbf{Q} = \begin{bmatrix} U_{x(1)} & U_{y(1)} & U_{x(2)} & U_{y(2)} & \dots & U_{x(m)} & U_{y(m)} \end{bmatrix}^{\mathrm{T}}$ and m is the number of element nodes ($m = 3$ for triangle, $m = 4$ for quadrangle). The evaluation of strains requires the application of the matrix of shape functions' derivatives which may be expressed as

$$\boldsymbol{\varepsilon}(x,y) = \mathbf{L}\,\mathbf{u}(x,y) = \begin{bmatrix} N_{1,x} & 0 & N_{2,x} & 0 & \dots & N_{m,x} & 0 \\ 0 & N_{1,y} & 0 & N_{2,y} & 0 & \dots & N_{m,y} \\ N_{1,y} & N_{1,x} & N_{2,y} & N_{2,x} & \dots & N_{m,y} & N_{m,x} \end{bmatrix} \mathbf{Q} = \mathbf{B}(x,y)\;\mathbf{Q} \tag{134}$$

By assuming Bubnov-Galerkin approach (test function $\mathbf{v}$ is interpolated using the same shape functions), substituting FE interpolation schemes for strains, induced on $\mathbf{v}$ and $\mathbf{u}$, to (117), and using arbitrariness of test function selection, we have the element equation

$$\mathbf{K}^{(e)}\mathbf{Q} = \mathbf{F}^{(e)}, \quad \mathbf{K}^{(e)} = h\int_e \mathbf{B}^{\mathrm{T}}\mathbf{D}\mathbf{B}\, dxdy, \quad \mathbf{F}^{(e)} = \int_e \left(\mathbf{N}^{(e)}\right)^{\mathrm{T}} \mathbf{b}\, dxdy + \int_{\partial e} \left(\mathbf{N}^{(e)}\right)^{\mathrm{T}} \mathbf{t}\, ds \tag{135}$$

4.4 Elasto-plastic material model

The material non-linearity appears when the stress-strain relation is non-linear. It usually appears for all engineering materials (e.g., concrete, steel) at high load levels. The most commonly applied non-linear material models assume:

- non-linear elasticity (the relation stress-strain is non-linear, though no irreversible strain appears after full unloading),
- elasto-plasticity (the relation stress-strain is non-linear and irreversible plastic strain appears, with a certain hardening assumed),
- linear elastic-perfect plasticity (the stress-strain is linear up to the specified stress point, above which the plastic flow appears with constant stress),
- linear elastic-plastic with linear hardening (the stress-strain is linear up to the specified stress point, above which the plastic hardening appears with constant rate) and

- rigid-perfectly plastic (elastic range is neglected).

The most important issues of the computational plasticity are:

- definition of **the yield criterion** which defines the limit at which the material becomes plastic,
- **the flow rule** which describes the relation stress-strain in a plastic range,
- **the consistency condition** which prevents stresses from exceeding the yield limit.

The yield condition is usually expressed by means of the scalar function $f(\boldsymbol{\sigma}, \boldsymbol{\sigma}_Y)$, where $\boldsymbol{\sigma}$ contains six independent stress components whereas $\boldsymbol{\sigma}_Y$ defines specific material parameters. The yield function f defines the surface in stress space. Three situations may be distinguished, namely

- $f < 0$ - elastic range,
- $f > 0$ - inadmissible state,
- $f = 0$ - plastic range (maximal permissible stresses).

The yield function f is usually defined in terms of principal stresses $(\sigma_1, \sigma_2, \sigma_3)$ or certain stress invariants (I_1, I_2, I_3) in order to avoid the influence of a rotation of the coordinate system. Moreover, since the hydrostatic state does not provide plastic strains, invariants (J_1, J_2, J_3) of deviatoric stress tensor are applied, namely

$$\sigma_{ij} = s_{ij} + \sigma_0 \delta_{ij}, \quad \sigma_0 = \frac{1}{3}(\sigma_{11} + \sigma_{22} + \sigma_{33}) = \frac{1}{3} I_1 \tag{136}$$

Here, s_{ij} are components of a stress deviator tensor (or pure shear stress tensor) which corresponds to the shape change of the stressed body, whereas $\sigma_0 \delta_{ij}$ are components of a hydrostatic stress tensor (or volumetric stress tensor, or mean normal stress tensor), which corresponds to its volume change. For instance, the second invariant J_2 is given by

$$\begin{aligned} J_2 &= \frac{1}{6}\left((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right) = \\ &= \frac{1}{6}\left((\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2\right) + \sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2 \end{aligned} \tag{137}$$

One of the oldest yielding criterion for metal is Tresca criterion according to which the yielding appears as the results of the maximal shear stress reaching a critical value (yield limit). Thought it predicts the yielding of metals quite satisfactory, it defines non-smooth yielding function as well as it does not take into account two lesser shear stresses. Therefore, more accurate criterion has been independently proposed by von Mises and Huber. In this case, the yield function is given by

$$f(\boldsymbol{\sigma}, \sigma_Y) = \sigma_e - \sigma_Y \tag{138}$$

where σ_e is the equivalent (effective, von Mises, Huber-Mises-Hencky) stress given by

$$\begin{aligned} \sigma_e &= \sqrt{3J_2} = \sqrt{\frac{1}{2}\left((\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right)} = \\ &= \sqrt{\frac{1}{2}\left((\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2\right) + 3\sigma_{12}^2 + 3\sigma_{23}^2 + 3\sigma_{13}^2} \end{aligned} \tag{139}$$

While yielding independence on hydrostatic pressure holds for metals, it is usually not true for other materials as concrete and soils. Therefore the modified von Mises criterion may be provided (the so called Drucker-Prager criterion), namely

$$f(\boldsymbol{\sigma}, \alpha, \sigma_Y) = \sigma_e + \alpha \sigma_m - \sigma_Y \tag{140}$$

in which the yield function depends on the mean stress $\sigma_m = \sigma_0$. Two material constants, α and σ_Y, are usually expressed in terms of the cohesion and the angle of internal friction.

5 1D non-linear problems

Non-linearity in mechanical models may have two different sources, namely

1. geometric non-linearity in which we assume large displacements and/or large strains, and
2. physical non-linearity in which we assume the non-linear physical law (i.e., non-linear strain-stress relation).

Obviously, both types of non-linearity may appear separately or altogether. In each case, the appropriate numerical treatment has to be performed, namely derivation of the linearized model and its solution by means of the Newton-Raphson method.

Let us pay attention on the large strains (with large displacements and linear elasticity) for the bar problem considered in the first chapter. The complete set of equations is as follows

$$\begin{cases} -\dfrac{dN}{dx} = q, \quad N(L) = P, \\ \varepsilon = \dfrac{du}{dx} + \dfrac{1}{2}\left(\dfrac{du}{dx}\right)^2, \quad u(0) = 0, \\ \sigma = E\varepsilon \quad \rightarrow \quad N = EA\varepsilon \end{cases} \tag{141}$$

Here, L denotes the length of the deformed bar, as the balance equation has to be formulated in the current, deformed state, not the initial one, in which the initial bar length L_0 is known.

By multiplying the balance equation by arbitrary test function v and integrating over the entire domain $[0, L]$, we have

$$-\int_0^L v\frac{dN}{dx}dx = \int_0^L vq\,dx \tag{142}$$

After integration by parts of the first integral, we have

$$\int_0^L \frac{dv}{dx}N\,dx - v(L)\,N(L) + v(0)\,N(0) = \int_0^L vq\,dx \tag{143}$$

Substituting $N(L) = P$ and assuming $v(0) = 0$, we have

$$\int_0^L \frac{dv}{dx}N\,dx = \int_0^L vq\,dx + Pv(L) \tag{144}$$

The final variational formulation is obtained after substitution the formula for N on the basis of physical and geometric equations, namely

$$EA\int_0^L \frac{dv}{dx}\left(\frac{du}{dx} + \frac{1}{2}\left(\frac{du}{dx}\right)^2\right)dx = \int_0^L vq\,dx + Pv(L), \quad u \in H_0^1, \quad \forall u \in H_0^1 \tag{145}$$

It may be observed that, comparing with the full linear model, we deal with a completely new non-linear term, containing the displacement's first derivative squared. Let us follow the standard FE interpolation schemes, namely

$$u(x) = \mathbf{N}(x)\,\mathbf{U}^{(e)}, \quad u'(x) = \mathbf{B}(x)\,\mathbf{U}^{(e)}, \quad v(x) = \left(\mathbf{V}^{(e)}\right)^{\mathrm{T}}\mathbf{N}^{\mathrm{T}}(x), \quad v'(x) = \left(\mathbf{V}^{(e)}\right)^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}(x) \tag{146}$$

After their substitution into the final variational form, and taking advantage from the free selection of degrees of freedom $\mathbf{V}$, we have

$$EA\int_0^h \mathbf{B}^{\mathrm{T}}\left(\mathbf{B}\mathbf{U}^{(e)} + \frac{1}{2}\left(\mathbf{B}\mathbf{U}^{(e)}\right)^2\right)dx = \int_0^h \mathbf{N}^{\mathrm{T}}q\,dx + P\mathbf{N}^{\mathrm{T}}(L) \tag{147}$$

Since the left-hand side of the above equation is non-linear, we may express it in the residual form, namely

$$\mathbf{R}^{(e)}\left(\mathbf{U}^{(e)}\right) = EA\int_0^h \mathbf{B}^{\mathrm{T}}\left(\mathbf{B}\mathbf{U}^{(e)} + \frac{1}{2}\left(\mathbf{B}\mathbf{U}^{(e)}\right)^2\right)dx - \int_0^h \mathbf{N}^{\mathrm{T}}q\,dx - P\mathbf{N}^{\mathrm{T}}(L) \tag{148}$$

Following the well-known linearization according to the Newton-Raphson method, we obtain the iterative scheme

$$\mathbf{K}_{\mathrm{T}}^{(k)}\Delta\mathbf{U}^{(k+1)} = -\mathbf{R}^{(k)} \tag{149}$$

in which $\mathbf{K}_{\mathrm{T}}$ is the tangent stiffness matrix, given by

$$\mathbf{K}_{\mathrm{T}}^{(e)}\left(\mathbf{U}^{(e)}\right) = \frac{\partial \mathbf{R}^{(e)}}{\partial \mathbf{U}^{(e)}} = EA\int_0^h \mathbf{B}^{\mathrm{T}}\mathbf{B}\left(1+\mathbf{B}\mathbf{U}^{(e)}\right)dx \tag{150}$$

and $\Delta\mathbf{U}^{(k+1)} = \mathbf{U}^{(k+1)} - \mathbf{U}^{(k)} \quad \rightarrow \quad \mathbf{U}^{(k+1)} = \mathbf{U}^{(k)} + \Delta\mathbf{U}^{(k+1)}$. In case of large displacements, the bar length has to be updated as well, namely $L = L_0 + U_n^{(k+1)}$, where $U_n^{(k+1)}$ is the displacement of the last right-end node. Obviously, new FE mesh has to be generated, yielding new element length h.

Moreover, the initial solution $\mathbf{U}^{(0)}$ (e.g., corresponding to the linear model as well as satisfying original, essential and natural, boundary conditions), admissible error as well as maximum number of iterations have to be assumed. Iterations are performed as long as $e < e_{adm}$, where $e = \dfrac{\|\Delta\mathbf{U}^{(k+1)}\|}{\|\mathbf{U}^{(k+1)}\|}$.

In the similar manner, the numerical scheme for the bar under tension, with non-linear elastic law, may be derived. The appropriate problem formulation, with physical power law, may be as follows

$$\begin{cases} -\dfrac{dN}{dx} = q, \quad N(L) = P, \\ \varepsilon = \dfrac{du}{dx}, \quad u(0) = 0, \\ \sigma = E\varepsilon^p \quad \rightarrow \quad N = EA\varepsilon^p \end{cases} \tag{151}$$

in which the exponent $p \neq 0$. It may be negative however under the assumption that the strain $\varepsilon > 0$ at every point of the construction (pure tension). Mostly, for real engineering materials, we use $p = 2$ or $p = \dfrac{1}{2}$. In the first case, the construction would be less stiff, and in the second case - more stiff, than it corresponds to the linear elastic law (for $p = 1$). The model of non-linear elasticity is a much simpler case than the model with large displacements and/or strains, since the stiffening principle may hold (L back means the initial length of the bar). Hence, the FEM mesh does not change during the iterative Newton-Raphson method.

Let us examine the above-formulated algorithm for the well-known bar construction from the previous sections, though this time we assume large strains-displacements. Again, FE mesh consists of two elements with equal length h and linear shape functions. We set initial length $L_0 = 2m$, $P = 20\ kN$, $q_0 = 10\ kN/m$ and $EA = 100kN$ (which fully justifies the application of the geometrically non-linear model). Therefore, the initial solution for $L = L_0 = 2m$ and $h = 1m$ is

$$\mathbf{U}^{(0)} = \frac{1}{EA}\begin{bmatrix} 0 \\ 35 \\ 60 \end{bmatrix} = 10^{-2}\begin{bmatrix} 0 \\ 35 \\ 60 \end{bmatrix} m \tag{152}$$

Hence, the new bar length is $L = L_0 + 0.6m = 2.6m$ whereas finite elements have the length of $h = 1.3m$. The tangent stiffness matrices and residual vectors for the element with linear shape functions are

$$\begin{aligned} \mathbf{K}_{\mathrm{T}}^{(e)} &= EA\int_0^h \begin{bmatrix} -\dfrac{1}{h} \\ \dfrac{1}{h} \end{bmatrix}\begin{bmatrix} -\dfrac{1}{h} & \dfrac{1}{h} \end{bmatrix}\left(1 + \begin{bmatrix} -\dfrac{1}{h} & \dfrac{1}{h} \end{bmatrix}\begin{bmatrix} U_1^{(e)} \\ U_2^{(e)} \end{bmatrix}\right)dx = \\ &= \frac{EA}{h^2}\left(h - U_1^{(e)} + U_2^{(e)}\right)\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{aligned} \tag{153}$$

and

$$\begin{aligned} \mathbf{R}^{(e)} &= EA\int_0^h \begin{bmatrix} -\dfrac{1}{h} \\ \dfrac{1}{h} \end{bmatrix}\left(\begin{bmatrix} -\dfrac{1}{h} & \dfrac{1}{h} \end{bmatrix}\begin{bmatrix} U_1^{(e)} \\ U_2^{(e)} \end{bmatrix} + \frac{1}{2}\left(\begin{bmatrix} -\dfrac{1}{h} & \dfrac{1}{h} \end{bmatrix}\begin{bmatrix} U_1^{(e)} \\ U_2^{(e)} \end{bmatrix}\right)^2\right)dx - \frac{q_0 h}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} = \\ &= \frac{EA}{h^2}\left(hU_2^{(e)} - hU_1^{(e)} + \frac{1}{2}\left(U_1^{(e)}\right)^2 - U_1^{(e)}U_2^{(e)} + \frac{1}{2}\left(U_2^{(e)}\right)^2\right)\begin{bmatrix} -1 \\ 1 \end{bmatrix} - \frac{q_0 h}{2}\begin{bmatrix} 1 \\ 1 \end{bmatrix} \end{aligned} \tag{154}$$

In the residual vector, the last component, corresponding to the concentrated force, has been omitted. However, it is going to be included on the global level.

For the first finite element ($h = 1.3m$, $\mathbf{U}^{(1)} = 10^{-2} \left[\begin{array}{cc} 0 & 35 \end{array} \right]^{\mathrm{T}} m$), we have

$$\mathbf{K}_{\mathrm{T}}^{(1)} = 97.6331 \left[\begin{array}{cc} 1 & -1 \\ -1 & 1 \end{array} \right] kN/m, \quad \mathbf{R}^{(1)} = \left[\begin{array}{c} -37.0473 \\ 24.0473 \end{array} \right] kN \tag{155}$$

whereas for the second one ($h = 1.3m$, $\mathbf{U}^{(2)} = 10^{-2} \left[\begin{array}{cc} 35 & 60 \end{array} \right]^{\mathrm{T}} m$), we have

$$\mathbf{K}_{\mathrm{T}}^{(2)} = 91.7160 \left[\begin{array}{cc} 1 & -1 \\ -1 & 1 \end{array} \right] kN/m, \quad \mathbf{R}^{(2)} = \left[\begin{array}{c} -27.5799 \\ 14.5799 \end{array} \right] kN \tag{156}$$

Afterwards, the assembly procedure is performed, yielding the following global system of FE equations

$$\left[\begin{array}{ccc} 97.6331 & -97.6331 & 0 \\ -97.6331 & 189.3491 & -91.7160 \\ 0 & -91.7160 & 91.7160 \end{array} \right] \left[\begin{array}{c} \Delta U_1 \\ \Delta U_2 \\ \Delta U_3 \end{array} \right] = - \left(\left[\begin{array}{c} -37.0473 \\ -3.5325 \\ 14.5799 \end{array} \right] - \left[\begin{array}{c} 0 \\ 0 \\ 20 \end{array} \right] \right) = \left[\begin{array}{c} 37.0473 \\ 3.5325 \\ 5.4201 \end{array} \right] \tag{157}$$

After fulfilment of the boundary conditions (increment $\Delta U_1 = 0$, independently from the known value of U_1; and $V_1 = 0$), we have

$$\left[\begin{array}{cc} 189.3491 & -91.7160 \\ -91.7160 & 91.7160 \end{array} \right] \left[\begin{array}{c} \Delta U_2 \\ \Delta U_3 \end{array} \right] = \left[\begin{array}{c} 3.5325 \\ 5.4201 \end{array} \right] \quad \rightarrow \quad \left[\begin{array}{c} \Delta U_2 \\ \Delta U_3 \end{array} \right] = \left[\begin{array}{c} 0.0917 \\ 0.1508 \end{array} \right] m \tag{158}$$

Hence, the current solution is

$$\mathbf{U}^{(1)} = \mathbf{U}^{(0)} + \mathbf{\Delta U} = 10^{-2} \left[\begin{array}{c} 0 \\ 35 \\ 60 \end{array} \right] m + \left[\begin{array}{c} 0 \\ 0.0917 \\ 0.1508 \end{array} \right] m = \left[\begin{array}{c} 0 \\ 0.4417 \\ 0.7508 \end{array} \right] m \tag{159}$$

with updated $L = 2.7508m$ and $h = 1.3754m$. Further steps depend on the Newton-Raphson method variant. In the **regular method**, we recalculate both tangent stiffness matrices and residual vectors for all elements, while in the **modified method** we use the once calculated tangent stiffness matrix from (157), and calculate only residual vectors for individual elements and assembly them into a global system . Assuming that the admissible calculation error is $e_{\mathrm{dop}} = 10^{-6}$, the final solution of $\mathbf{U} = \left[\begin{array}{ccc} 0 & 0.4998 & 0.8436 \end{array} \right]^{\mathrm{T}} m$ is obtained for both the regular and the modified methods after $k = 15$ iterations. Fig. 12 presents graphic results of calculations in the form of a displacement (upper graph) and deformation (lower graph) graph, for $N = 2$ and for $N = 10$, for a geometrically linear and geometrically non-linear models.

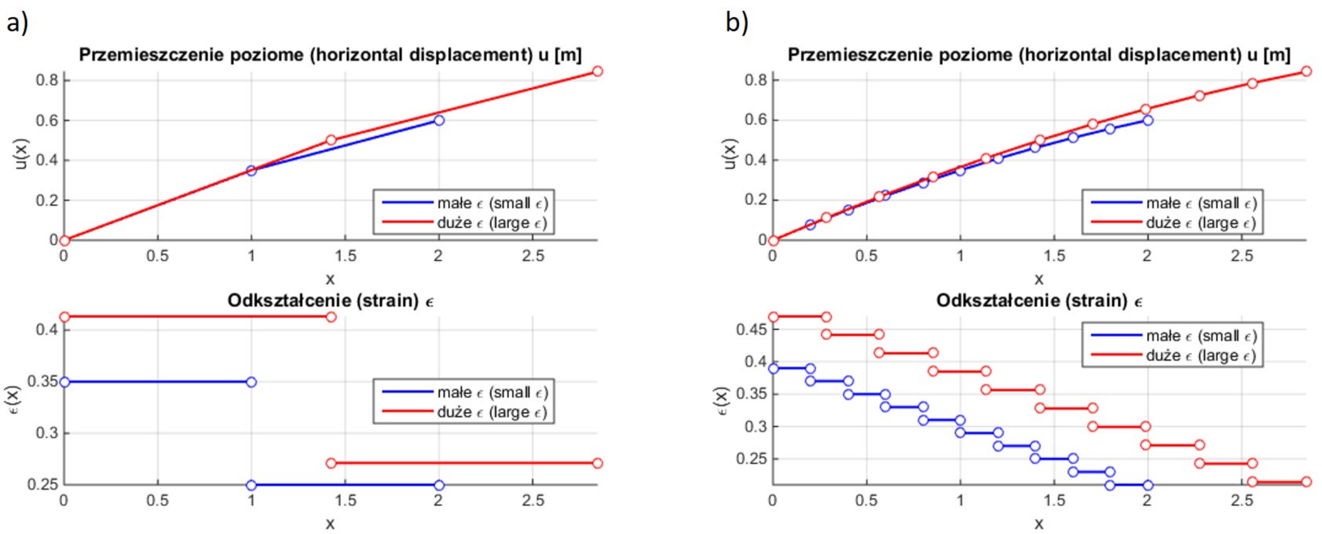

Figure 12: Calculation results for a geometrically non-linear tension bar problem: comparison of displacements and strains for a linear and non-linear models, with a) $N = 2$ and b) $N = 10$

6 Exemplary problems

1. For the considered tensile bar, assume $L = 3m$, $q = q_0 = 10kN/m$, $P = 20kN$ and arbitrary stiffness EA. Find displacement and normal force using three finite elements with equal length and linear interpolation. Compare the numerical and analytical solutions for u and N - using graphs and values in the middle of each finite elements.

2. Determine element load vectors for parabolic load intensities, shown in Fig. 13, assuming linear FE interpolation.

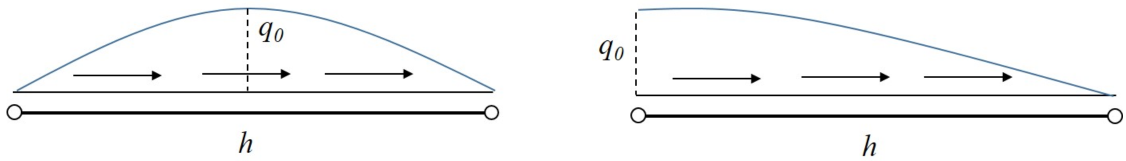

Figure 13: Parabolic loadings

3. For the considered bar structure, assume variable cross-section area $A = A(x) \in C^1$; formulate local and variational boundary value problem; determine the general type of element stiffness matrix.

4. Use derivations and formulas from the previous problem in order to determine displacement and normal force for bar structure with $L = 2m$, E, $A(x) = x + 1$, $q = 0$ and $P = -10kN$. Use two elements with linear FE interpolation.

5. Determine element load vectors for parabolic load intensities, shown in Fig. 13, assuming hierarchic FE interpolation, with linear and quadratic basis functions.

6. Determine displacement and normal force for various types of loadings and FE mesh, shown in Fig. 14. Assume $EA = 10^6 kN$, $L = 1m$, $P = 10kN$, $q = 10kN/m$.

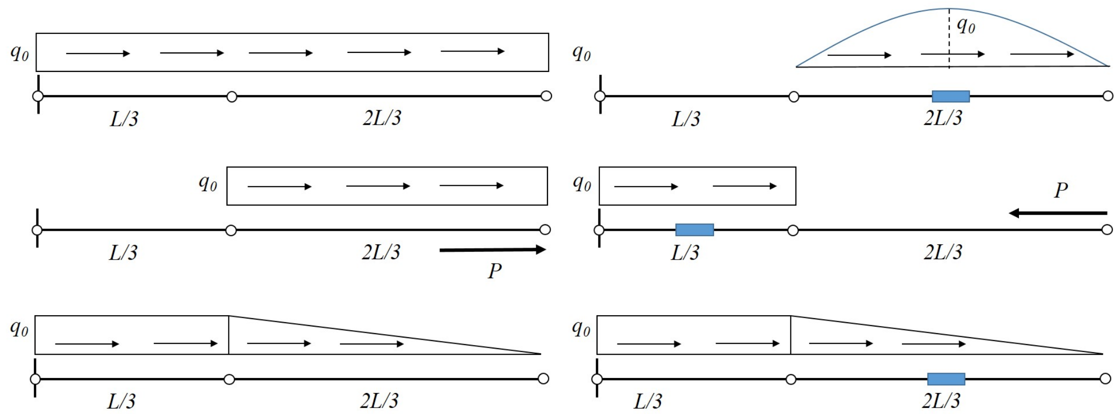

Figure 14: Various types of loading and FE mesh

7. Determine the mass matrix for the FE element with hierarchic interpolation.

8. Determine the natural frequencies of a bar structure with given $L = 2m$, E, A, ρ. Use
   - two finite elements of equal length, linear FE interpolation and consistent mass matrices,
   - two finite elements of equal length, linear FE interpolation and diagonal mass matrices,
   - one finite element with hierarchic interpolation and appropriate mass matrix.

9. The bar structure is analysed, with $L = 2m$, $E = 10^5 kN/m^2$, $A = 10^{-3} m^2$, $\rho = 10^3 kg/m^3$. Determine nodal displacements and velocities for two unknown time steps. Assume $\Delta t = 0.1s$, zero initial displacements and velocities, and harmonic load $f(x,t) = x \sin(10t)$, $P = 0$. Use one finite element with linear FE interpolation and consistent mass matrix.

10. Solve the previous problem, though with two finite elements of uneven lengths and with linear FE interpolation and consistent mass matrices

11. Solve the previous problem, though with diagonal mass matrices and one or two FE elements with linear interpolation.

12. Solve the previous problem, though with $f = 0$ and $P(t) = 10t$, using one or two finite elements with linear interpolation.

13. Solve previous problems (for two loading cases: $f(x,t) = x\sin(10t)$ and $P = 0$, as well as $f = 0$ and $P(t) = 10t$), using one finite element with hierarchic interpolation.

14. Solve previous problem, though with $f = 0$, $P = 0$, $u_0 = x^2$, $v_0 = 0$, using one or two finite elements.

15. Solve previous problem, though with $f = 0$, $P = 0$, $u_0 = 0$, $v_0 = \sin\dfrac{x\pi}{2L}$, using one or two finite elements.

16. Given are the following differential equations (no physical interpretation). Derive relevant weak variational forms (on the continuous and element levels) and solve them by FEM in order to obtain the solution and solution derivative approximation. Use from one (if possible) to three finite elements with homogeneous/mixed linear and/or hierarchic interpolation.

    - $y'' - xy' = 2, \quad x \in (-1,2)\,, \quad y(-1) = 1, \quad y(2) = -1.$
    - $-xy'' + y = x, \quad x \in (-2,0)\,, \quad y(-2) = 0, \quad y'(0) = -2,$
    - $y'' + y' + y = x, \quad x \in (-1,1)\,, \quad y'(-1) = 0, \quad y(1) = 0,$
    - $y'' = 2, \quad x \in (0,2)\,, \quad y(0) = 3, \quad y(2) = -2,$

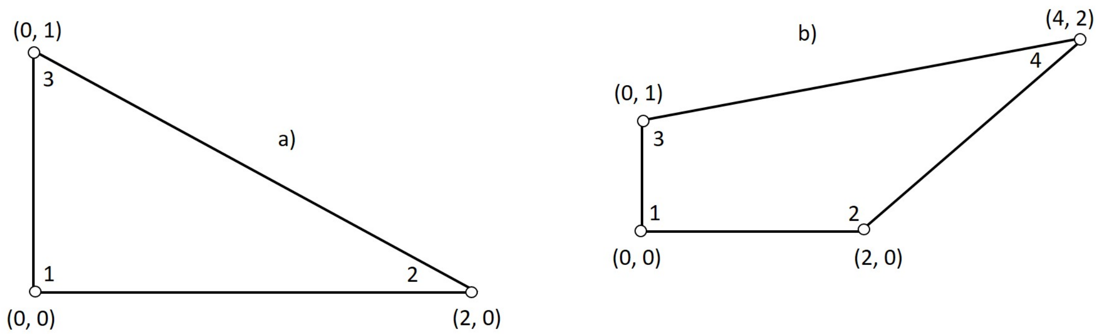

Figure 15: Examples of 2D finite elements : a) triangular element, b) quadrilateral element

17. Find shape functions, gradient of shape functions as well as compute, analytically and numerically, definite integral of $f(x,y) = x + y$ on the triangular element, shown in Fig. 15a.

    - element shape functions

$$
\begin{aligned}
&\left\{\begin{array}{l} N_1(0,0) = 1, \\ N_1(2,0) = 0, \\ N_1(0,1) = 0, \end{array}\right. \rightarrow \left\{\begin{array}{c} c_1 = 1, \\ 2a_1 + c_1 = 0, \\ 1b_1 + c_1 = 0 \end{array}\right. \rightarrow N_1(x,y) = -\frac{1}{2}x - y + 1 \\
&\left\{\begin{array}{l} N_2(0,0) = 0, \\ N_2(2,0) = 1, \\ N_2(0,1) = 0, \end{array}\right. \rightarrow \left\{\begin{array}{c} c_2 = 0, \\ 2a_2 + c_2 = 1, \\ 1b_2 + c_2 = 0 \end{array}\right. \rightarrow N_2(x,y) = \frac{1}{2}x \\
&\left\{\begin{array}{l} N_3(0,0) = 0, \\ N_3(2,0) = 0, \\ N_3(0,1) = 1, \end{array}\right. \rightarrow \left\{\begin{array}{c} c_3 = 0, \\ 2a_3 + c_3 = 0, \\ 1b_3 + c_3 = 1 \end{array}\right. \rightarrow N_3(x,y) = y \\
&\mathbf{N}(x,y) = \left[\begin{array}{ccc} -\frac{1}{2}x - y + 1 & \frac{1}{2}x & y \end{array}\right]
\end{aligned}
\tag{160}
$$

- gradient of the vector of shape functions

$$\nabla \mathbf{N} = \begin{bmatrix} \dfrac{\partial \mathbf{N}}{\partial x} \\ \dfrac{\partial \mathbf{N}}{\partial y} \end{bmatrix} = \begin{bmatrix} -\dfrac{1}{2} & \dfrac{1}{2} & 0 \\ -1 & 0 & 1 \end{bmatrix} \tag{161}$$

- definite integral of $f(x,y) = x + y$ using analytical integration

$$I = \int_e x + y\, dxdy = \int_0^2 \int_0^{1-\frac{1}{2}x} x + y\, dydx = \int_0^2 x\left(1 - \frac{1}{2}x\right) + \frac{\left(1 - \frac{1}{2}x\right)^2}{2} dx = 1 \tag{162}$$

- definite integral of $f(x,y) = x + y$ using numerical integration

$$I = \frac{1}{2} \cdot 2 \cdot 1 \cdot \frac{1}{3}\left(1 + \left(1 + \frac{1}{2}\right) + \frac{1}{2}\right) = 1 \tag{163}$$

18. For the triangle from the previous problem, find its stiffness matrix. Assume plane stress state with $h = 0.1m$, $E = 10^6 kN/m^2$ and $\nu = 0.1$ as well as metres as a unit of all distances.

$$\mathbf{D} = 1.010 \cdot 10^6 \begin{bmatrix} 1 & 0.1 & 0 \\ 0.1 & 1 & 0 \\ 0 & 0 & 0.450 \end{bmatrix} kN/m^2, \quad \mathbf{B} = \begin{bmatrix} -0.5 & 0 & 0.5 & 0 & 0 & 0 \\ 0 & -1 & 0 & 0 & 0 & 1 \\ -1 & -0.5 & 0 & 0.5 & 1 & 0 \end{bmatrix} 1/m \tag{164}$$

$$\begin{aligned} \mathbf{B}^{\mathrm{T}}\mathbf{D}\mathbf{B} &= 1.010 \cdot 10^6 \begin{bmatrix} -0.5 & 0 & -1 \\ 0 & -1 & -0.5 \\ 0.5 & 0 & 0 \\ 0 & 0 & 0.5 \\ 0 & 0 & 1 \\ 0 & 1 & 0 \end{bmatrix} \begin{bmatrix} -0.5 & -0.1 & 0.5 & 0 & 0 & 0.1 \\ -0.05 & -1 & 0.05 & 0 & 0 & 1 \\ -0.450 & -0.225 & 0 & 0.225 & 0.450 & 0 \end{bmatrix} = \\ &= 1.010 \cdot 10^6 \begin{bmatrix} 0.7000 & 0.2750 & -0.2500 & -0.2250 & -0.4500 & -0.0500 \\ & 1.1125 & -0.0500 & -0.1125 & -0.2250 & -1.0000 \\ & & 0.2500 & 0 & 0 & 0.0500 \\ & \mathit{symmetry} & & 0.1125 & 0.2250 & 0 \\ & & & & 0.4500 & 0 \\ & & & & & 1.0000 \end{bmatrix} kN/m^4 \end{aligned} \tag{165}$$

$$\mathbf{K}^{(e)} = 0.1m \cdot 0.5 \cdot 2m \cdot 1m \cdot \mathbf{B}^{\mathrm{T}}\mathbf{D}\mathbf{B} =$$

$$= 1.010 \cdot 10^5 \begin{bmatrix} 0.7000 & 0.2750 & -0.2500 & -0.2250 & -0.4500 & -0.0500 \\ & 1.1125 & -0.0500 & -0.1125 & -0.2250 & -1.0000 \\ & & 0.2500 & 0 & 0 & 0.0500 \\ & \mathit{symmetry} & & 0.1125 & 0.2250 & 0 \\ & & & & 0.4500 & 0 \\ & & & & & 1.0000 \end{bmatrix} kN/m \tag{166}$$

19. For the triangle from the previous problem, find components of the displacement, strain and stress at its center point, assuming its degrees of freedom vector $\mathbf{Q} = \begin{bmatrix} 0 & 0 & 0.1 & -0.2 & 0 & 0.2 \end{bmatrix}^{\mathrm{T}} m$.

20. Find transformation formulas for coordinates, gradient of the first shape function in real coordinates as well as compute, analytically and numerically, definite integral of $f(x,y) = x - 2y$ on the quadrilateral element, shown in Fig. 15b.

- transformation formulas for coordinates

$$\begin{cases} x(\xi,\eta) = 0 \cdot N_1(\xi,\eta) + 2 \cdot N_2(\xi,\eta) + 4 \cdot N_3(\xi,\eta) + 0 \cdot N_4(\xi,\eta) = \dfrac{1}{2}\xi\eta + \dfrac{3}{2}\xi + \dfrac{1}{2}\eta + \dfrac{3}{2}, \\ y(\xi,\eta) = 0 \cdot N_1(\xi,\eta) + 0 \cdot N_2(\xi,\eta) + 2 \cdot N_3(\xi,\eta) + 1 \cdot N_4(\xi,\eta) = \dfrac{1}{4}\xi\eta + \dfrac{1}{4}\xi + \dfrac{3}{4}\eta + \dfrac{3}{4} \end{cases} \tag{167}$$

- transformation matrix, its determinant and inverse transpose

$$\mathbf{J} = \begin{bmatrix} \frac{1}{2}\eta + \frac{3}{2} & \frac{1}{2}\xi + \frac{1}{2} \\ \frac{1}{4}\eta + \frac{1}{4} & \frac{1}{4}\xi + \frac{3}{4} \end{bmatrix}, \quad \det \mathbf{J} = \frac{1}{4}\xi + \frac{1}{4}\eta + 1, \quad \mathbf{J}^{-\mathrm{T}} = \frac{1}{\frac{1}{4}\xi + \frac{1}{4}\eta + 1} \begin{bmatrix} \frac{1}{4}\xi + \frac{3}{4} & -\frac{1}{4}\eta - \frac{1}{4} \\ -\frac{1}{2}\xi - \frac{1}{2} & \frac{1}{2}\eta + \frac{3}{2} \end{bmatrix} \tag{168}$$

- gradient of the first shape function in real coordinates

$$\nabla N_1(x,y) = \frac{1}{4\left(\frac{1}{4}\xi + \frac{1}{4}\eta + 1\right)} \begin{bmatrix} \frac{1}{4}\xi + \frac{3}{4} & -\frac{1}{4}\eta - \frac{1}{4} \\ -\frac{1}{2}\xi - \frac{1}{2} & \frac{1}{2}\eta + \frac{3}{2} \end{bmatrix} \begin{bmatrix} \eta - 1 \\ \xi - 1 \end{bmatrix} = \frac{1}{2\xi + 2\eta + 8} \begin{bmatrix} -\xi + 2\eta - 1 \\ 4\xi - 2\eta - 2 \end{bmatrix} \tag{169}$$

- definite integral of $f(x,y) = x - 2y$ using analytical integration

$$\begin{aligned} I &= \int_e x - 2y\, dxdy = \int_{-1}^{1}\int_{-1}^{1} (\xi - \eta)\left(\frac{1}{4}\xi + \frac{1}{4}\eta + 1\right) d\xi d\eta = \frac{1}{4}\int_{-1}^{1}\int_{-1}^{1} \left(\xi^2 - \eta^2 + 4\xi - 4\eta\right) d\xi d\eta = \\ &= \frac{1}{4}\left(\int_{-1}^{1}\left(\frac{\xi^3}{3} - \eta^2\xi + 2\xi^2 - 4\eta\xi\right)|_{-1}^{1} d\eta\right) = \frac{1}{4}\left(\int_{-1}^{1}\frac{2}{3} - 2\eta^2 - 8\eta\, d\eta\right) = 0 \end{aligned} \tag{170}$$

- definite integral of $f(x,y) = x - 2y$ using numerical integration

$$\begin{aligned} I = \frac{1}{4}\Bigg(&\left(\left(\frac{1}{\sqrt{3}}\right)^2 - \left(\frac{1}{\sqrt{3}}\right)^2 - \frac{4}{\sqrt{3}} + \frac{4}{\sqrt{3}}\right) + \left(\left(\frac{1}{\sqrt{3}}\right)^2 - \left(\frac{1}{\sqrt{3}}\right)^2 + \frac{4}{\sqrt{3}} + \frac{4}{\sqrt{3}}\right) + \\ &+ \left(\left(\frac{1}{\sqrt{3}}\right)^2 - \left(\frac{1}{\sqrt{3}}\right)^2 + \frac{4}{\sqrt{3}} - \frac{4}{\sqrt{3}}\right) + \left(\left(\frac{1}{\sqrt{3}}\right)^2 - \left(\frac{1}{\sqrt{3}}\right)^2 - \frac{4}{\sqrt{3}} - \frac{4}{\sqrt{3}}\right)\Bigg) = 0 \end{aligned} \tag{171}$$

21. For the quadrangle from the previous problem, find its matrix of shape functions derivatives **B** in master and real coordinates.

22. For the quadrangle from the previous problem, find components of its displacement strain and stress at the point (0.5,0.5), knowing that nodal degrees of freedom are $\mathbf{Q} = \begin{bmatrix} 0 & 0 & 0.1 & -0.2 & -0.05 & -0.1 & 0 & 0 \end{bmatrix}^{\mathrm{T}} m$. Assume plain stress, $E = 10^6 kN/m^2$, $\nu = 0.1$ and $h = 0.1m$.

    from transformation formulas

$$\begin{cases} 0.5 = \frac{1}{2}\xi\eta + \frac{3}{2}\xi + \frac{1}{2}\eta + \frac{3}{2} \\ 0.5 = \frac{1}{4}\xi\eta + \frac{1}{4}\xi + \frac{3}{4}\eta + \frac{3}{4} \end{cases} \rightarrow \quad \xi = \frac{-\frac{1}{2}\eta - 1}{\frac{1}{2}\eta + \frac{3}{2}} \quad \rightarrow \quad 2\eta^2 + 7\eta + 1 = 0 \quad \rightarrow \quad \rightarrow \begin{cases} \eta = \dfrac{-7 + \sqrt{41}}{4} = -0.1492m \\ \xi = -0.6492m \end{cases} \tag{172}$$

    displacement components at $(-0.6294, -0.1492)$

$$\begin{bmatrix} u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 0 \cdot N_1(\xi,\eta) + 0.1 \cdot N_2(\xi,\eta) - 0.05 \cdot N_3(\xi,\eta) + 0 \cdot N_4(\xi,\eta) \\ 0 \cdot N_1(\xi,\eta) - 0.2 \cdot N_2(\xi,\eta) - 0.1 \cdot N_3(\xi,\eta) + 0 \cdot N_4(\xi,\eta) \end{bmatrix} = \begin{bmatrix} 0.0063m \\ -0.0276m \end{bmatrix} \tag{173}$$

    average displacement at $(-0.6294, -0.1492)$

$$u_{av}(0.5, 0.5) = \sqrt{0.0063^2 + 0.00276^2} = 0.0283m \tag{174}$$

    Jacoby matrix, Jacobian, inverse transpose of **J** at $(-0.6294, -0.1492)$

$$\mathbf{J} = \begin{bmatrix} 1.4254 & 0.1754 \\ 0.2127 & 0.5877 \end{bmatrix}, \quad \det \mathbf{J} = 0.8004, \quad \mathbf{J}^{-\mathrm{T}} = \begin{bmatrix} 0.7343 & -0.2657 \\ -0.2191 & 1.7809 \end{bmatrix} \tag{175}$$

gradients of N_2 and N_3 at $(-0.6294, -0.1492)$

$$\boldsymbol{\nabla} N_2 = \begin{bmatrix} 0.2343 \\ -0.2191 \end{bmatrix}, \quad \boldsymbol{\nabla} N_3 = \begin{bmatrix} 0.1329 \\ 0.1096 \end{bmatrix} \tag{176}$$

strain components at $(-0.6294, -0.1492)$

$$\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} 0.1 \cdot 0.2343 - 0.05 \cdot 0.1329 = 0.0168 \\ -0.2 \cdot (-0.2191) - 0.1 \cdot 0.1096 = 0.0329 \\ 0.1 \cdot (-0.2191) - 0.2 \cdot 0.2343 - 0.05 \cdot 0.1096 - 0.1 \cdot 0.1329 = -0.0875 \end{bmatrix} \tag{177}$$

as well as $\varepsilon_{zz} = -\dfrac{\nu}{1-\nu}(\varepsilon_{xx} + \varepsilon_{yy}) = -0.0055$;
stress components

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \mathbf{D} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{bmatrix} = \frac{10^6}{1-0.1^2} \begin{bmatrix} 1 & 0.1 & 0 \\ 0.1 & 1 & 0 \\ 0 & 0 & \frac{0.9}{2} \end{bmatrix} \begin{bmatrix} 0.0168 \\ 0.0329 \\ -0.0875 \end{bmatrix} = 10^4 \begin{bmatrix} 2.0293 \\ 3.4929 \\ -3.9773 \end{bmatrix} kPa \tag{178}$$

23. Determine shape functions, their derivatives as well as definite integrals of functions $f(x, y) = x^2 + y^2$ and $f(x, y) = \sin(x + y)$, defined on the following finite elements

    - equilateral triangle with a given side length a,
    - right-angled triangle with both legs equal a,
    - triangle given by points with coordinates (-1,0), (1,-1) and (1,1),
    - square with a given side length a,
    - rectangle with given side lengths a and b,
    - rhombus with a given side length a,
    - parallelogram with given side lengths a and b,
    - right trapezoid with given height h and side lengths a and b,
    - a quadrangle given by points with coordinates (-1,0), (1,0), (1.5,1) and (-1.5,1).

    In case there is no location of a figure specified, assume the origin of the real coordinate system in such a manner that the resulting formulas are as simple as possible.

24. Use FEM for a numerical solution of the 1D bar problem assuming geometric (large strains) and/or physical non-linearity (non-linear strain-stress relation), with specified type of load (concentrated and/or uniform load) and number of elements (1-2-3, with linear and/or quadratic interpolation).

25. Solve the following non-linear 1D boundary value problems using FEM and specified number of elements (1-2-3, with linear and/or quadratic interpolation).

    - $y'' - x\,(y')^2 = 1, \quad y(-2) = 0, \quad y'(0) = 0$
    - $y'' + y^2 = x, \quad y(0) = 0, \quad y(1) = -1$
    - $y'' + yy' = 0, \quad y'(0) = -1 \quad y(2) = 0$

7 Problems for computer implementation

1. Write a code in Matlab/Octave for determination of the nodal displacements/reaction forces of the considered bar under static load. Assume appropriate data and linear shape functions. Compare the nodal displacements with analytical solution. Plot the analytical axial force as well. Use the code structure given below.

2. Extend the code from the previous example for "return to element" segment: determine the displacement function as well as axial force for each finite element and plot them on existing figures. Use the code structure given below.

3. Extend the previous code for the hierarchic interpolation of the second order.
4. Use a full code from a static analysis of a bar with linear shape functions and rework it towards analysis of natural vibrations. Use the code structure given below.
5. Use a full code from a dynamic analysis of a bar with linear shape functions and rework it towards analysis of forced vibrations. Assume zero initial displacement and velocity, $P = 0$, dynamic load $f(x,t) = x \sin(\omega_f t)$, length of a time step Δt as well as the number of time steps. Use the code structure given below.
6. Use a full code from a static analysis of a bar with linear shape functions and rework it towards analysis of non-linear geometrical model. Assume large strains.

```
%% CODE SCHEME FOR PROBLEM #1 - STATIC ANALYSIS OF BAR with LINEAR SHAPE
%% FUNCTIONS
%
%% DATA
L = ; % beam length [m]
P = ; % concentrated force [kN] at x = L
q0 = ; % maximum intensity of uniform load [kN/m]
E = ; % Young modulus [kN/m^2]
A = ; % cross-section area [m^2]
N = ; % number of finite elements

%% CALCULCATIONS
EA = ; % tensile stiffness (product of Young modulus and cross-section area)
h = ; % mesh modulus (beam length over number of elements)
n = ; % number of nodes
dof = ; % number of degrees of freedom

K = ; % fill global stiffness matrix (dof rows x dof columns) with zeros
F = ; % fill global load vector (dof rows x 1) with zeros
F() = ; % place the concentrated force
X = ; % generate nodes coordinates (sequence from 0 by h to L)

for i = % loop over all elements
    e = []; % numbers of element's degrees of freedom
    x1 = X(); % coordinate of the first (left) element node
    x2 = X(); % coordinate of the second (right) element node
    Ke = ; % explicit formula for the element stiffness matrix
    q1 = ; q2 = ; % load intensities at element's ends (applicable for
    % linear load distribution)
    Fe = ; % explicit formula for the element load vector
    K( , ) = K( , ) + Ke; % assembly the stiffness matrix
    F( ) = F( ) + Fe; % assembly the load vector
end

Kbc = K;
Fbc = F;

Kbc( , ) = ; % fill the first row of Kbc with zeros
Kbc( , ) = ; % fill the first column of Kbc with zeros
Kbc( , ) = ; % place 1 at the first main diagonal element
Fbc( ) = ; % place 0 at the first element

U = ; % solve the system of equations Kbc * U = Fbc (determine displacements)
R = ; % determine reaction forces

x = linspace(0,L,100);
ua = ; % formula for analytical displacement
na = ; % formula for analytical normal force

close all
subplot(2,1,1);
hold on
plot(X,U,'bo-','markerfacecolor','r'); % plot of nodal displacements

plot(x,ua,'r-'); % plot of analytical displacements

subplot(2,1,2);
hold on
plot(x,na,'r-'); % plot of axial force
```

```
%% CODE SCHEME FOR PROBLEM #2 - EVALUATION OF DISPLACEMENT AND
%% AXIAL FORCE FOR EACH FINITE ELEMENTS + GRAPHS

for i = % loop over all elements
    e = []; % numbers of element's degrees of freedom
    x1 = X(); % coordinate of the first (left) element node
    x2 = X(); % coordinate of the second (right) element node
    Ue = U(); % displacements of the e-th element
    x = linspace(x1,x2,10);
    d = ; % distance between the 1st node of element and origin of global
    % coordinate system
    N1 = ; % first shape function
    N2 = ; % second shape function
    ue = % formula for displacements of the element
    ne = % formula for axial force of the element
    subplot(2,1,1);
    plot(x,u,'b-','linewidth',1.5)
    plot(x([1 end]),ue([1 end]),'bo','markerfacecolor','w')
    subplot(2,1,2);
    plot(x,ne,'b-','linewidth',1.5)
    plot(x([1 end]),ne([1 end]),'bo','markerfacecolor','w')
end
```

```
%% CODE SCHEME FOR PROBLEM #4 - DYNAMIC ANALYSIS of BAR with LINEAR SHAPE
%% FUNCTIONS - EIGEN & FORCED VIBRATIONS
%
%% GENERAL DATA
L = ; % beam length [m]
E = ; % Young modulus [N/m^2]
A = ; % cross-section area [m^2]
N = ; % number of finite elements
rho = ; % mass density [kg/m^3]

%% CALCULCATIONS FOR EIGEN VIBRATIONS
EA = ; % tensile stiffness (product of Young modulus and cross-section area)
h = ; % mesh modulus (beam length over number of elements)
n = ; % number of nodes
dof = ; % number of degrees of freedom

K = ; % fill global stiffness matrix (dof rows x dof columns) with zeros
M = ; % fill global mass matrix (dof rows x dof columns) with zeros
X = ; % generate nodes coordinates (sequence from 0 by h to L)

for i = % loop over all elements
    e = []; % numbers of element's degrees of freedom
    Ke = ; % explicit formula for the element stiffness matrix
    Me = ; % explicit formula for the element consistent mass matrix
    K( , ) = K( , ) + Ke; % assembly the stiffness matrix
    M( , ) = M( , ) + Me; % assembly the mass matrix
end

K( , ) = ; % fill the first row of K with zeros
K( , ) = ; % fill the first column of K with zeros
K( , ) = ; % place 1 at the first main diagonal element

M( , ) = ; % fill the first row of M with zeros
M( , ) = ; % fill the first column of M with zeros
M( , ) = ; % place 1 at the first main diagonal element

[U,Lambda] = eig(K,M);
Omega = sqrt(diag(Lambda));

close all
figure(1);
for k = 1:min([N 4])
    subplot(2,2,k);
    hold on
    grid on
    title(['i=',num2str(k)',', \omega=',num2str(Omega(k+1),2)])
    plot(X,U(:,k+1),'bo-','markerfacecolor','w','linewidth',1.5);
    xlabel('x');
    ylabel('u(x)');
end

%% DATA AND CALCULATIONS FOR FORCED VIBRATIONS

omega_f = ; % angular frequency of the dynamic load
dt = ; % time step length
time_steps = ; % number of time steps

U = zeros(; % displacement matrix (dof x time_steps)
V = zeros(; % velocity matrix (dof x time_steps)
Udd = zeros(; % acceleration matrix (dof x time_steps)
```

```
MKinv = ; % inverse of coefficient matrix

for k =  % loop over time steps (from 1 to time_steps)
    t = ; % recent time
    F = ; % load for fixed time moment
    Udd(:,k+1) = ; % recent acceleration
    V(:,k+1) = ; % recent velocity
    U(:,k+1) = ; % recent displacement
end

figure(2);

for k = 1:time_steps
    subplot(2,1,1);
    plot(X,U(:,k),'ro-');
    hold on
    grid on
    title('FORCED VIBRATIONS - HORIZONTAL DISPLACEMENT');
    xlabel('x');
    ylabel('u');
    axis([0 L min(min(U)) max(max(U))]);

    subplot(2,1,2);
    plot(X,V(:,k),'ro-');
    title('FORCED VIBRATIONS - VELOCITY');
    hold on
    grid on
    xlabel('x');
    ylabel('v');
    axis([0 L min(min(V)) max(max(V))]);

    pause(0.1);
    subplot(2,1,1);
    hold off
    subplot(2,1,2);
    hold off
end

figure(3);
hold on
grid on
title('HISTORY OF DISPLACEMENT FOR THE LAST RIGHT-END NODE');
xlabel('t');
ylabel('u(x=L)');
plot(0:dt:(time_steps)*dt,U(dof,:),'ro-');
```