18. Finite element method

Among the up-to-date methods of stress state analysis, the finite element method (abbreviated as FEM below) dominates clearly nowadays; it is used also in other fields of engineering analyses (heat transfer, convection of liquids, electricity and magnetism etc.). In mechanics, the FEM enables us to solve the following types of problems:

- stress-state analysis under static, cyclic or dynamic loading, incl. various non-linear problems;
- natural as well as forced vibrations, with or without damping;
- contact problem (contact pressure distribution);
- stability problems (buckling of structures);
- stationary or non-stationary heat transfer and evaluation of temperature stresses (incl. residual stresses).

The fundamentals of FEM are quite different from the analytical methods of stress-strain analysis. While the analytical methods of stress-strain analysis are based on the differential and integral calculus, the FEM is based on the variation calculus which is generally not so well known; it seaks for a minimum of some **functional**.

Note:

Function - *is a mapping between sets of numbers. It is a mathematical term for a rule which enables us to assign unambiguously some numerical value (from the image of mapping) to an inital numerical value (from the domain of mapping).*

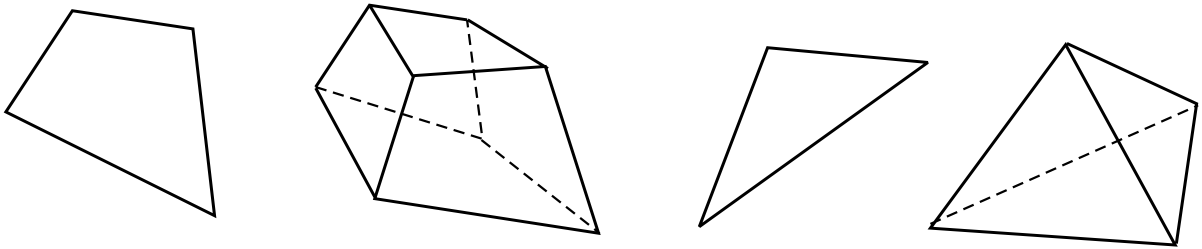
Functional - *is a mapping from a set of functions to a set of numbers. It is a rule which enables us to assign unambiguously some numerical value to a function (on the domain of the function or on its part). Definite integral is example of a functional.*

Strain energy represents the basic functional in stress-strain analysis of bodies. It is the work done during the deformation of the body; it is reversible in the case of elastic deformation, i.e. this work can be recuperated if the body (e.g. a spring) returns to its initial (undeformed) state. According to the above definition of a functional, it is a numerical value assigned to the functions describing the displacements of the individual points of the body; in the most common so called deformational variant of FEM, displacements are basic unknown functions. The strain energy can be calculated for any deformed shape of the body from strains and stresses in all the points of the body. The body cannot adopt a random shape (under certain loads and with certain supports) but its deformed shape is unambiguously determined (except some stability problems); it is the shape which requires the minimal work for deformation, i.e. the shape with the least strain energy. This fact is mathematically expressed by the **principle of minimum of the quadratic functional**. This principle is generally valid in the nature and it tells us that only that of the possible processes comes into being which requires the minimal consumption of energy (e.g. the rasor-edge cuts the material always through the way having the minimal resistance). Consequently, only that of the possible deformed shapes of the body (i.e. being in accordance with the given boundary conditions - loads and supports) will be realized, which has the minimal strain energy. The total energy potential Π of the body represents the corresponding functional, the minimum of which defines the real deformed shape of the body; this energy potential is defined as the total strain energy W of the body after subtraction of energy potential P of the external loads:

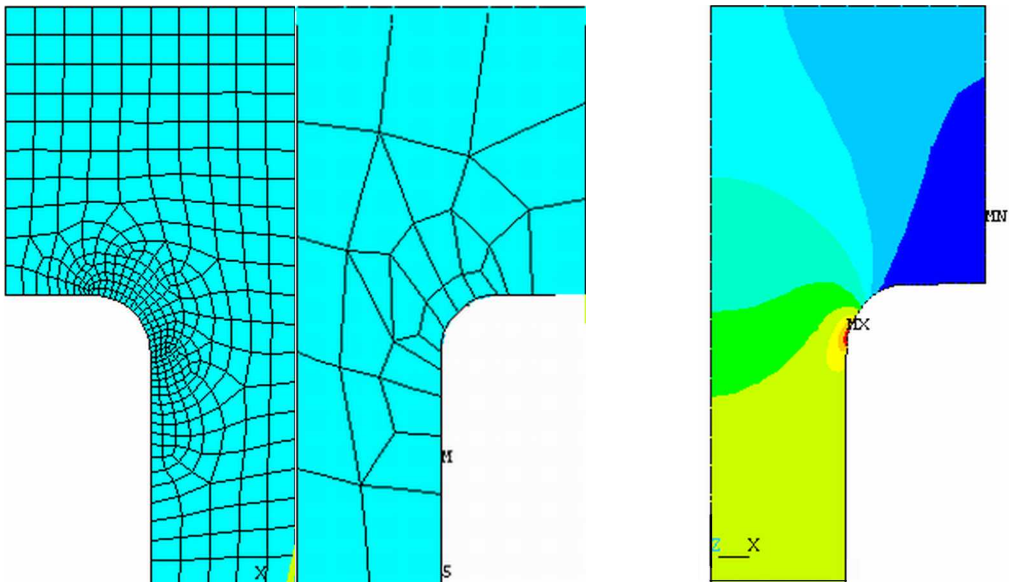
$$\Pi = W - P$$

Naturally, the total energy potential of the body is function of displacements of its individual points. Variation methods of mathematics enable us to find a minimum of the functional Π, i.e. to find such a shape, in which this functional will be minimal under the given boundary conditions (loads, supports); this shape will be the only one to come into being.

It is possible to evaluate the strain components from the displacements of the individual points and, consequently, the stress components using constitutive equations (with the known material characteristics). In practice the calculation is carried out in such a way that a geometrical model of the body or the structure is created using preprocessing (i.e. a computer programme for input data processing); this model must be continuously (without any residues) divided into finite elements. The basic in plane element is a quadrangle, in a 3D space it is a hexahedron (brick); sometimes simplified element shapes are used (triangle, tetrahedron).



The corners of these elements, if need be also some other points, represent nodes of the mesh in which the unknown displacements are calculated. The edges of the elements create the mesh, the density of which is decissive for the accuracy of the results. These edges are mostly straight but, at quadratic elements, also curvilinear edges can be realized. The quadratic elements have, in addition to the nodes in their corners, also some more nodes in the midpoints of the edges; in this way we obtain an eight-node-element in plane and a twenty-node-brick in the 3D space. These elements are able to describe the local stress concentration much better even if the mesh is rather rough (see the following example).



The distribution of the first principal stress is demonstrated in the right-hand colloured figure for a symmetric half of a notched shaft (shaft shoulder); the red colour corresponds to the highest stress level (in the root of the notch). The mean normal longitudinal stress in the cross section of the notch has the value of 1 MPa, the maximum stress value is

1,676 MPa. The folowing table shows that, in the case of a fine mesh (left half of the left-hand figure), the quadratic as well as linear elements give correct results, while the error is much higher in the case of linear elements than for quadratic elements, if the rough mesh (corresponding to the right half of the left-hand figure) is used.

Type of element	mesh density	calculated maximum stress [MPa]
linear - four nodes	rough	1,28
linear - four nodes	fine	1,67
quadratic - eight nodes	rough	1,59
quadratic - eight nodes	fine	1,67

The table presents the maximum stress value in the root of the notch (shaft shoulder) under a tensional load, calculated using various types of elements and various mesh densities corresponding to the above figure.

In practice, the number of calculated points cannot be infinite. Therefore the density of the created mesh is determined by the engineer and the quality of the mesh depends on his experience. If the mesh is too fine (dense), the solution is too time-consuming; in opposite, if the mesh is too rough, the calculated stresses can be substantially lower than the real values and the maximum stress can be underestimated. This can happen especially in the case of a pronounced local extreme of stresses (notch root in the above example). The up-to-date programme systems are able to create the mesh automatically, but the estimation of the sufficient mesh density must be made by the engineer in any case; moreover, a mesh created by an experienced engineer is almost always much better (shows lower computer time and memory consumption) than an automatically created mesh. Each node of the mesh represents three unknown parameters (displacements in the three perpendicular directions) in the case of a 3D model. Standard PCs are able to solve models counting tens or hundreds thousands equations (unknown parameters) in reasonable computational times.

It is necessary to define material parameters (modulus of elasticity and Poisson's ratio in the case of a linear elastic isotropic material) for all the elements of the model. Furher, the boundary conditions (loads and supports) must be defined; these boundary conditions should ensure (for a static problem) an immovable position of the body in the space (all the degrees of freedom restricted). An eventual deformation restriction (statically indeterminately supported body) does not make the problem more complex; only some more boundary conditions are prescribed. In the second step, the solver is activated; this is a programme which assembles the system of equations with the unknown displacements and calculates the strains and stresses of them. The solver cannot be activated without definition of all the input data (geometry of the model, material properties, supports and loads); therefore inverse problems (i.e. problems in which some parameters of geometry, material, loads or supports are not known) cannot be solved by finite element method.

Checking question

The last component of the program system is the postprocessing, i.e. a programme for presentation of results. It enables us to represent any of the output parameters (e.g. displacements, stresses etc.) in the solved body or in its part in various ways. Also some reduced stresses or other values necessary for evaluations of safety factors can be calculated.