Laboratory exercise no 6: Simulation of stress and strain distribution using finite element method

Aim: The aim of the laboratory is application of finite element method to calculate spatial distribution of stress and strain in cantilevered beam.

Introduction: Historical the finite element method was first applied to study of strength of materials aimed at the calculation of stress and strain distribution in structural members, such as beams, columns, and shafts.

Stress is defined as the force ***F*** across a "small" boundary per unit area ***S*** of that boundary, for all orientations of the boundary.

$$T = \frac{F}{S}$$

In general, the stress T that a particle P applies on another particle Q across a surface S can have any direction relative to S. The vector T may be regarded as the sum of two components: the normal stress σ (compression or tension) perpendicular to the surface, and the shear stress τ that is parallel to the surface.

A strain is in general a tensor quantity. Physical insight into strains can be gained by observing that a given strain can be decomposed into normal and shear components. The amount of stretch or compression along material line elements or fibers is the normal strain, and the amount of distortion associated with the sliding of plane layers over each other is the shear strain, within a deforming body. The Cauchy strain or engineering strain ε is expressed as the ratio of total deformation to the initial dimension of the material body in which the forces are being applied.

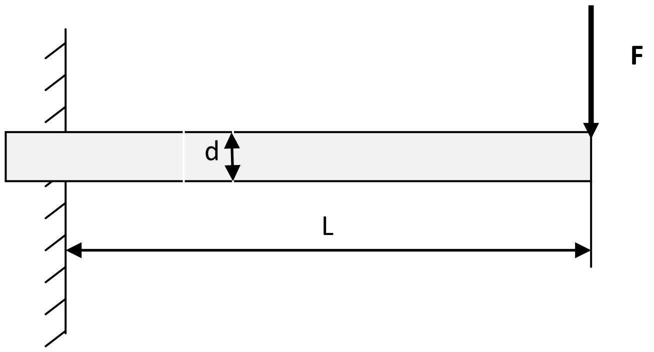
$$\varepsilon = \frac{\Delta L}{L_0}$$

In such method simplified to 2D case, two vector components of stress and strain have to be calculated, so each computational node has 4 degrees of freedom. We assume in the calculations that strain has elastic deformation character described by Hooke's law.

$$\sigma = E \cdot \varepsilon$$

where E is Young's modulus.

The simulation object is cantilevered beam of length L and thickness d supported from one side, loaded at the other end with force F, presented on fig.1



As a result of the load, the beam will deform at the end by a value h that can be calculated from the formula:

$$h = \frac{F \cdot L^3}{3 \cdot E \cdot J}$$

where:

- F – loading force
- L – beam length
- E – material Young's modulus
- J – moment of inertia of plane area

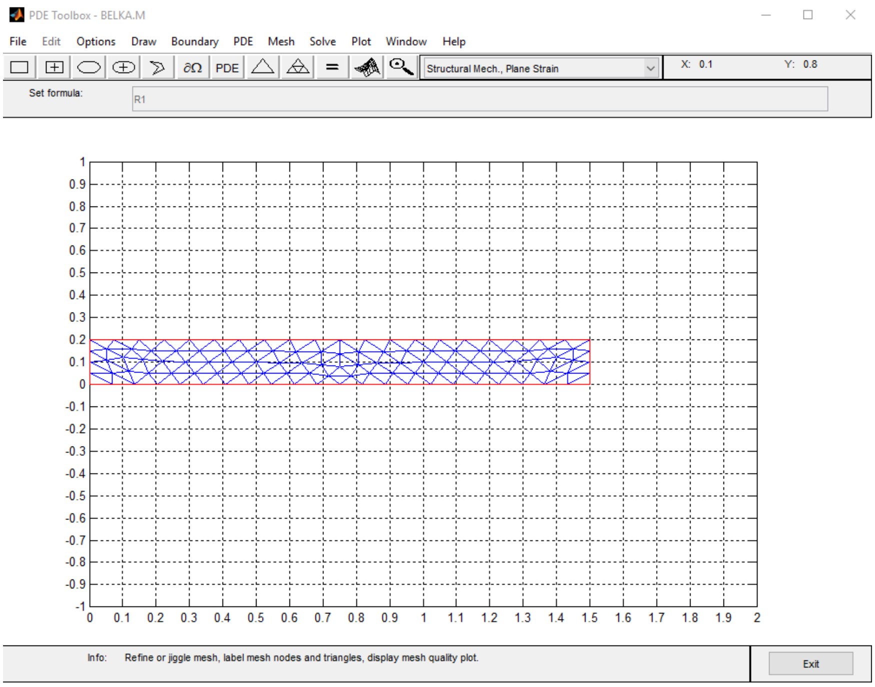
For the rectangular cross section elastic modulus can be calculated from the equation:

$$J = \frac{g \cdot d^3}{12}$$

where:

- g – beam thickness (for 2D problem we assume g equal to 1)
- d – beam width

The calculations will be performed using pdetool extension in MATLAB dedicated for solving two dimensional partial differentia equations using finite element method.



Laboratory programme:

1. Learning the methodology
2. Learning the pdetool
3. Designing the simulated element
4. Calculation of the distribution of stresses and strains
5. Comparison of the calculated beam deformation with the theory
6. Simulation of more complicated geometry

Laboratory outline:

1. Start pdetool
2. Adaptation of the environment to the simulation (determination of the boundary and resolution of the simulated area).
3. Designing the geometry of the simulated element.
4. Choosing the pre-defined computing problem.
5. Definition of boundary conditions (beam support, force load).
6. Determination of coefficients in equations.
7. Generating mesh.
8. Calculations.
9. Visualization of the obtained results.
10. Determination of beam deformation and comparison with theory.
11. Preparation and simulation of more complicated geometry.
12. The report must include the conclusion.