Introduction to Finite Element Method

Direct Approach to FEM

Tomáš Krejčí

Department of Mechanics
Faculty of Civil Engineering
Czech Technical University in Prague
Czech Republic

132NAST - Numerical analysis of structures
Prague – 21 September 2022

Outline

1. Course information
2. Introduction to FEM
3. Direct Approach for discrete systems

Course information

- Lecturer: Tomáš Krejčí
- Room: D2029
- Office hours: Thursday from 12:00 to 13:30 in D2029
- Email: krejci@fsv.cvut.cz
- Web page: http://mech.fsv.cvut.cz/~krejci/TEACHING/NAST/index.html
- Credit requirements:
  - Minimal 10 points
  - Mid-term test 20 points
  - Homeworks (optional), each 2 points
  - Semester projects (optional), each 5 points
- Exam requirements:
  - Credit has to be given
  - Grades:
    - A 90 - 100 points
    - B 80 - 89 points
    - C 70 - 79 points
    - D 60 - 69 points
    - E 50 - 59 points

http://mech.fsv.cvut.cz/∼krejci/TEACHING/NAST/index.html

132NAST - Numerical analysis of structures

Lecturer: Tomáš Krejčí

Subject anotation

- Subject code: 132NAST
- No. of credits: 2 + 2
- Credit and Exam

Short description: Overview of direct stiffness method of structural mechanics. Weak solution of one-dimensional elasticity equations. Galerkin method, Gauss integration, principle of the Finite Element method. Steady state heat conduction in one dimension. Two-dimensional heat conduction problem, triangular finite elements. Two-dimensional elasticity problems. Convergence of FEM, error estimates.

References:

- J. Fish and T. Belytschko:A First Course in Finite Elements,John Wiley & Sons, 2007 (free access @CTU)
- O. C. Zienkewicz and R. L. Taylor: The Finite Element Method, Volume 1, The Basis, Fifth Edition, Butterworth-Heinemann, 2000
- C. Felippa:Introduction to Finite Element Methods,Department of Aerospace Engineering Sciences,University of Colorado at Boulder
- Z. Bittnar and J. Šejnoha:Numerical Methods in Structural Mechanics, ASCE Press, 1996

MATLAB links:

- G. Strang:Short MATLAB tutorial
- C. Moler:Numerical Computing with MATLAB

- Grading policy
- Semester projects (optional)
- Homework assignments
- Lectures
- Tutorials
- Mid-term review test
- Exams (obsolete)

- **Course Schedule:**

Date	Lecture	Tutorial
September 21	Introduction to FEM, Overview of Direct Stiffness Method - 1D Elasticity	Introduction to Programming - Matlab, Octave, Excel; Localization
September 28	**Public Holiday**	**Public Holiday**
October 5	Strong and Weak Forms, Weighted Residual Method, Lagrange Principle	1D Elastic Element: Localization, Benchmarks
October 12	Aproximation Functions and Numerical Integration - Gauss Quadrature, Finite Elements	2D Trusses
October 19	Finite Element Formulation for One-Dimensional Problems - Linear Elasticity	Linear and Quadratic Bar Elements, Gauss Quadrature, Natural Coordinates
October 26	Finite Element Formulation for One-Dimensional Heat Conduction Problems	1D Heat Conduction
November 2	Finite Element Formulation for One-Dimensional Nonstationary Heat Conduction Problems	1D Nonstationary Heat Transfer
November 9	Two-Dimensional Heat Conduction Problems	2D Heat Conduction
November 16	**Mid-term test**; Consolidation	2D Heat Conduction
November 23	Finite Element Formulation for Two-Dimensional Problems - Linear Elasticity	2D Problems - Plane Stress and Plane Strain Problems
December 1	Finite Element Formulation for Beams	Beams
December 7	Numerical Aspects of FEM Part I	Beams
December 14	Numerical Aspects of FEM Part II	A posteriori Error Estimation

Downloads:

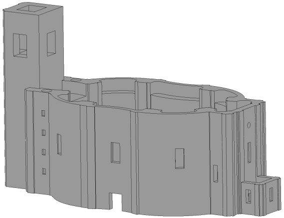
Links:

Introduction to Finite Element Method

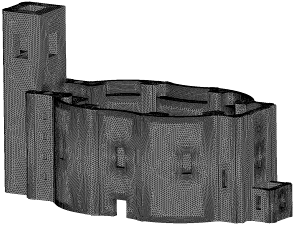
- Many physical phenomena in engineering and science can be described in terms of partial differential equations
  - Stress analysis, Heat transfer, Fluid flow and Electromagnetics
- Solving these equations by classical analytical methods for arbitrary shapes is almost impossible
- The finite element method (FEM) is a numerical approach by which these partial differential equations can be solved approximately
- Millions of engineers and scientists worldwide use the FEM
- Google - “FEM” about 533,000,000 results
- 69,000 FEM books
- Google - “FEM software” 23,900,000 results
- https://en.wikipedia.org/wiki/List_of_finite_element_software_packages

Real structure →

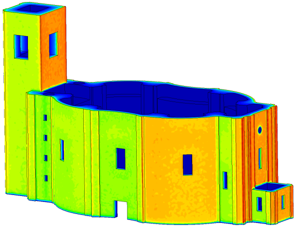
Model - Problem - partial differential equations →



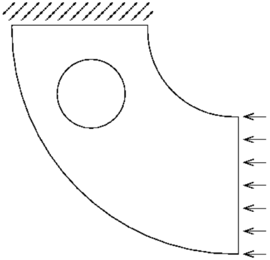
Discretization by finite elements →



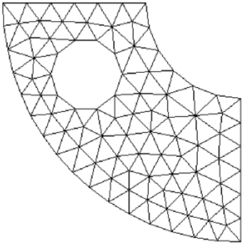
Weak solution - approximation



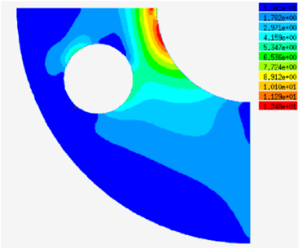
Problem, differential equations →



Discretization by finite elements →



Weak solution



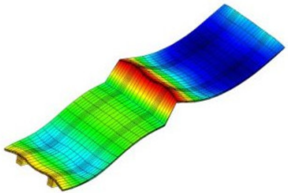
History of FEM

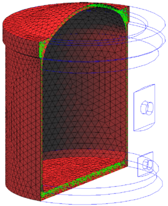
- 1943: Courant published a paper in which he used tringular elements with variational principles to solve vibration problems
- 1950: The FEM was developed in the aerospace industry - Boeing and Bell Aerospace (USA), Rolls Royce (UK)
- 1956: M.J.Turner, R.W.Clough (Berkeley, Boeing), and M.C.Martin, L. J. Topp published one of the first papers that laid out the major ideas
- Berkeley: E. Wilson, R.L.Taylor, professors at Berkeley, and their students PhD students: T.J.R. Hughes, C. Felipa, K.J. Bathe
- Swansea: O.C. Zienkiewicz, B. Irons, R.Owen
- 1960: E. Wilson developed one of the first finite element programs that was widely used
- 1965: NASA funded a project to develop a general-purpose finite element program Nastran
- 1969: ANSYS program - focused on linear and nonlinear applications, has a capitalization of $1.8 billion (in 2006)
- 1978: ABAQUS software - initially focused on nonlinear applications
- Nowadays: FEM applications in many areas
  - Fast computer development
  - Software: NASTRAN, ANSYS, ABAQUS, LS-DYNA, COMSOL, NEMETSCHEK, SCIA Engineer, RFEM, ATENA

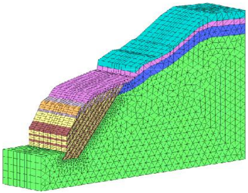
Applications of FEM

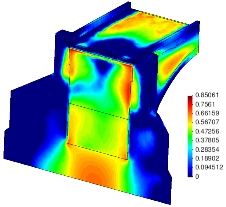
- Stress and thermal analyses of industrial parts such as electronic chips, electric devices, valves, pipes, pressure vessels, automotive engines and aircraft
- Seismic analysis of dams, power plants, cities and high-rise buildings
- Crash analysis of cars, trains and aircraft
- Fluid flow analysis of coolant ponds, pollutants and contaminants, and air in ventilation systems
- Electromagnetic analysis of antennas, transistors and aircraft signatures
- Analysis of surgical procedures such as plastic surgery, jaw reconstruction, correction of scoliosis and many others

- Civil Engineering:
  Statics



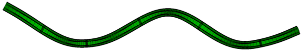


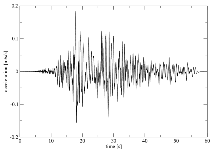


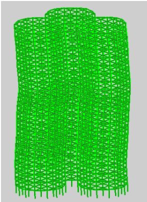


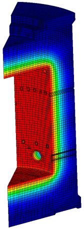
Dynamics

Coupled analysis



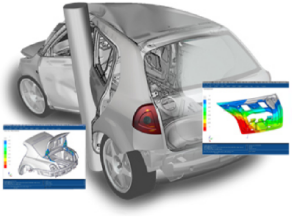


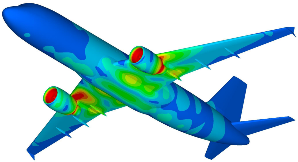




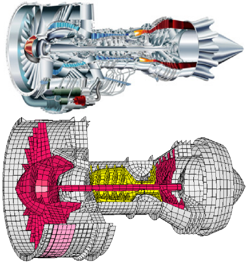
Applications of FEM

- Mechanical Engineering:



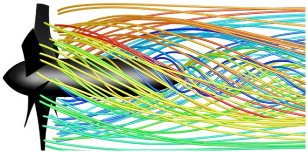


source:stressebook.com

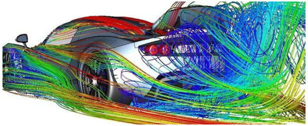


source:semanticscholar.org

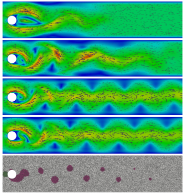
- Fluid Flow:



source:simtec-europe.com

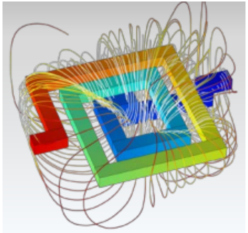


source:simutechgroup.com

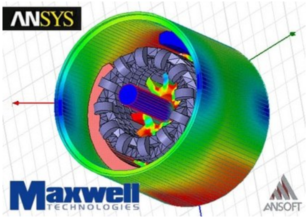


source:https://doi.org/10.3390/fluids4010005

- Electrical engineering:
  Electromagnetics simulations

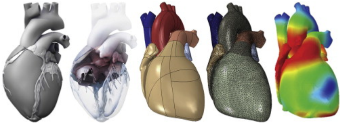


source:comsol.com

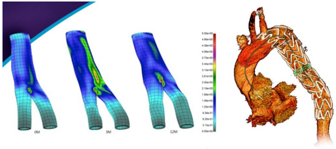


source:https://solenoidsystems.com

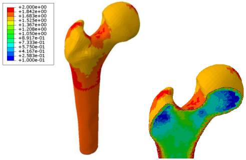
- Medical research:



source:https://doi.org/10.1016/j.euromechsol.2014.04.001



source:http://www.cardiomedtech.com/



source:semanticscholar.org

The finite element method (FEM) consists of the following five steps:

- 1. Preprocessing: subdividing the problem domain into finite elements
- 2. Element formulation: development of equations for elements
- 3. Assembly: obtaining the equations of the entire system from the equations of individual elements
- 4. Solving the equations
- 5. Postprocessing: determining quantities of interest, such as stresses and strains, and obtaining visualizations of the response

Customary matrix notation is used:

Matrices are denoted by uppercase boldface italic letters $\boldsymbol{A}, \boldsymbol{B}$,

Lowercase boldface italic letters stand for vectors $\boldsymbol{\sigma}, \boldsymbol{r}$,

Scalars A, b,

Variables A, b,

- Truss structures

Highway truss bridge over the Mississippi River, photo by Highsmith, Carol M.

Truss structure in the atelier at Faculty of Civil Engineering

- Direct derivation of FEM for bar elements - deformation method
- Expression of forces according to nodal displacements

- Equilibrium of the element internal forces and nodal forces

$$F_1^e = -\sigma^e A^e, \qquad F_2^e = \sigma^e A^e, \qquad F_1^e + F_2^e = 0$$

- The elastic stress–strain law, known as Hooke's law

$$\sigma^e = E^e \varepsilon^e$$

- The deformation of the structure must be compatible
- Strain definition as the ratio of the elongation to the original element length

$$\varepsilon^e = \frac{\Delta l^e}{l^e}$$

Behavior of a single bar element

- Equation for stress

$$\sigma^e = E^e \varepsilon^e = E^e \frac{\Delta l^e}{l^e} = E^e \frac{u_2^e - u_1^e}{l^e}$$

- Equations for nodal forces

$$F_1^e = -\sigma^e A^e = \frac{E^e A^e}{l^e}(u_1^e - u_2^e), \qquad F_2^e = \sigma^e A^e = \frac{E^e A^e}{l^e}(u_2^e - u_1^e)$$

- Matrix form of nodal forces

$$\left\{ \begin{array}{c} F_1^e \\ F_2^e \end{array} \right\} = \left[\begin{array}{cc} k^e & -k^e \\ -k^e & k^e \end{array} \right] \left\{ \begin{array}{c} u_1^e \\ u_2^e \end{array} \right\},$$

where

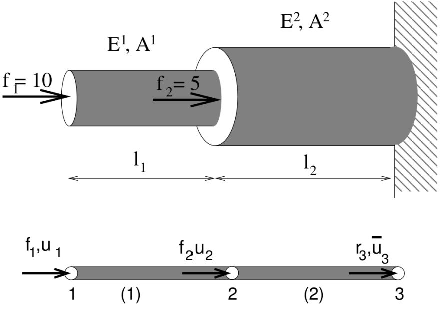
$$k^e = \frac{EA}{l^e}$$

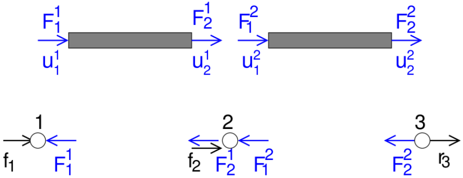
- Matrix form of nodal forces

$$\boldsymbol{F}^e = \boldsymbol{K}^e \boldsymbol{d}^e$$

$$\left\{ \begin{array}{c} F_1^e \\ F_2^e \end{array} \right\} = \left[\begin{array}{cc} k^e & -k^e \\ -k^e & k^e \end{array} \right] \left\{ \begin{array}{c} u_1^e \\ u_2^e \end{array} \right\}$$

- Stifness matrix $\boldsymbol{K}^e$ is symmetric and singular
- Linearity of constitutive, geometrical, and balance equations
- Rigid body motion: If $u_1^e = u_2^e$, then $F_1^e = F_2^e = 0$





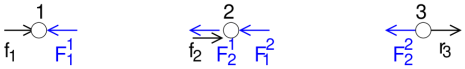
- Force - displacement relations for both elements

$$\boldsymbol{F}^1 = \boldsymbol{K}^1 \boldsymbol{d}^1, \qquad \boldsymbol{F}^2 = \boldsymbol{K}^2 \boldsymbol{d}^2$$

- Compatibility of displacements

$$u_1^1 = u_1, \quad u_2^1 = u_1^2 = u_2, \quad u_2^2 = u_3 \tag{1}$$

Stiffnes matrix assembly - localization



- Equilibrium equations for three nodes

$$\begin{aligned}
(1) \quad &\rightarrow \quad F_1^1 - f_1 = 0 \\
(2) \quad &\rightarrow \quad F_2^1 + F_1^2 - f_2 = 0 \\
(3) \quad &\rightarrow \quad F_2^2 - r_3 = 0
\end{aligned}$$

- Matrix form

$$\left\{ \begin{array}{c} F_1^1 \\ F_2^1 \\ 0 \end{array} \right\} + \left\{ \begin{array}{c} 0 \\ F_1^2 \\ F_2^2 \end{array} \right\} = \left\{ \begin{array}{c} f_1 \\ f_2 \\ r_3 \end{array} \right\}$$

- Element internal nodal forces are expressed by global displacements

$$\left\{ \begin{array}{c} F_1^1 \\ F_2^1 \end{array} \right\} = \boldsymbol{K}^1 \left\{ \begin{array}{c} u_1^1 \\ u_2^1 \end{array} \right\} = \boldsymbol{K}^1 \left\{ \begin{array}{c} u_1 \\ u_2 \end{array} \right\}$$

$$\left\{ \begin{array}{c} F_1^2 \\ F_2^2 \end{array} \right\} = \boldsymbol{K}^2 \left\{ \begin{array}{c} u_1^2 \\ u_2^2 \end{array} \right\} = \boldsymbol{K}^2 \left\{ \begin{array}{c} u_2 \\ u_3 \end{array} \right\}$$

- Local vectors of nodal forces and stifness matrices are augmented by adding zeros

$$\underbrace{\begin{Bmatrix} F_1^1 \\ F_2^1 \\ 0 \end{Bmatrix}}_{\tilde{\boldsymbol{F}}^1} = \underbrace{\begin{Bmatrix} k^1 & -k^1 & 0 \\ -k^1 & k^1 & 0 \\ 0 & 0 & 0 \end{Bmatrix}}_{\tilde{\boldsymbol{K}}^1} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

$$\underbrace{\begin{Bmatrix} 0 \\ F_1^2 \\ F_2^2 \end{Bmatrix}}_{\tilde{\boldsymbol{F}}^2} = \underbrace{\begin{Bmatrix} 0 & 0 & 0 \\ 0 & k^2 & -k^2 \\ 0 & -k^2 & k^2 \end{Bmatrix}}_{\tilde{\boldsymbol{K}}^2} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$

- Augmented vectors are introduced into the equilibrium equations for the nodes

$$\underbrace{\begin{Bmatrix} k^1 & -k^1 & 0 \\ -k^1 & k^1 & 0 \\ 0 & 0 & 0 \end{Bmatrix}}_{\tilde{\boldsymbol{K}}^1} \underbrace{\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}}_{\boldsymbol{d}} + \underbrace{\begin{Bmatrix} 0 & 0 & 0 \\ 0 & k^2 & -k^2 \\ 0 & -k^2 & k^2 \end{Bmatrix}}_{\tilde{\boldsymbol{K}}^2} \underbrace{\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}}_{\boldsymbol{d}} = \underbrace{\begin{Bmatrix} f_1 \\ f_2 \\ r_3 \end{Bmatrix}}_{\boldsymbol{f}}$$

$$\left(\tilde{\boldsymbol{K}}^1 + \tilde{\boldsymbol{K}}^2\right)\boldsymbol{d} = \boldsymbol{f} \qquad \rightarrow \qquad \boldsymbol{K}\boldsymbol{d} = \boldsymbol{f}$$

- Formalising with help of a distribution matrix ("gather" matrix) $\boldsymbol{L}^e$ for each element, $\boldsymbol{d}^e = \boldsymbol{L}^e \boldsymbol{d}$
- For example, displacements for the first element read

$$\boldsymbol{d}^1 = \left\{ \begin{array}{c} u_1^1 \\ u_2^1 \end{array} \right\} = \underbrace{\left\{ \begin{array}{ccc} 1 & 0 & 0 \\ 0 & 1 & 0 \end{array} \right\}}_{\boldsymbol{L}^1} \left\{ \begin{array}{c} u_1 \\ u_2 \\ u_3 \end{array} \right\} = \boldsymbol{L}^1 \boldsymbol{d}$$

- The same for vector of nodal forces

$$\boldsymbol{F}^1 = \boldsymbol{L}^e \tilde{\boldsymbol{F}}^1$$

- Equilibrium equations for one element can be expressed via a global displacement vector

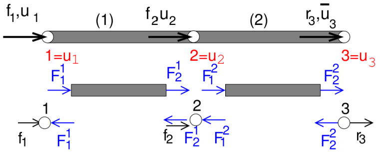
$$\boldsymbol{K}^e \boldsymbol{d}^e = \boldsymbol{F}^e \qquad \rightarrow \qquad \boldsymbol{K}^e \boldsymbol{L}^e \boldsymbol{d} = \boldsymbol{F}^e$$

- Multiplying by transpose matrix $\boldsymbol{L}^{e\mathrm{T}}$

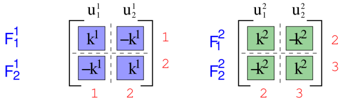
$$\underbrace{\boldsymbol{L}^{e\mathrm{T}} \boldsymbol{K}^e \boldsymbol{L}^e}_{\tilde{\boldsymbol{K}}^e} \boldsymbol{d} = \underbrace{\boldsymbol{L}^{e\mathrm{T}} \boldsymbol{F}^e}_{\boldsymbol{f}}$$

Stiffnes matrix assembly - algorithmization

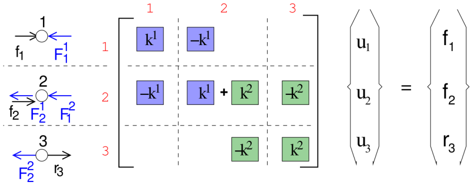
- Code numbers represent numbers of unknown displacements



- Assignment of code numbers to local displacements and forces on elements



- Local stiffness matrices are added to global stiffness matrix



- Code numbers represent mapping between local numbering on elements and global numbering for the whole structure
- Formally we can write:

$$\boldsymbol{K} = \sum_i \tilde{\boldsymbol{K}}^i$$

- The stiffness matrix is symmetric and singular
- Regularization of the matrix is obtained by the setting of boundary conditions
- The system of equations is rearanged for free degrees of freedom (unknown displacments) and prescribed displacements

$$\begin{bmatrix} \boldsymbol{K}_{uu} & \boldsymbol{K}_{up} \\ \boldsymbol{K}_{pu} & \boldsymbol{K}_{pp} \end{bmatrix} \begin{Bmatrix} \boldsymbol{u} \\ \bar{\boldsymbol{u}} \end{Bmatrix} = \begin{Bmatrix} \boldsymbol{f} \\ \boldsymbol{r} \end{Bmatrix}$$

- The vector of unknown displacements is calculated from the first row

$$\boldsymbol{u} = \boldsymbol{K}_{uu}^{-1} (\boldsymbol{f} - \boldsymbol{K}_{up}\bar{\boldsymbol{u}})$$

- Unknown reactions are obtained from the second row

$$\boldsymbol{r} = \boldsymbol{K}_{up}\boldsymbol{u} + \boldsymbol{K}_{up}\bar{\boldsymbol{u}}$$

References

- English course of "Numerical analysis of structures" by J. Zeman (jan.zeman@fsv.cvut.cz)
- Czech course of "Numerická analýza konstrukcí" (Numerical analysis of structures) by B. Patzák (borek.patzak@fsv.cvut.cz)
- J. Fish and T. Belytschko: A First Course in Finite Elements, John Wiley & Sons, 2007