

Master in Advanced Materials
 —
 Computational Methods in Materials Science
 Winter term 2011/2012

Introduction to the Finite Element Analysis (FEA)

Dr.-Ing. Ulrich Simon
M.Sc. Ouz Yurdalan
B.Sc. Benedikt Holl
 UZWR

Schedule for WS 2010/11

<i>Date</i> <i>Thursday</i>	<i>Lecture</i> 10:15 – 11:45 45.2.102	<i>Lab</i> 13:15 – 15:30 45.2.104
20.10.2011	Krill	no lab
27.10.2011	Simon	Simon
03.11.2011	Simon	Simon
10.11.2011	Simon	no lab
17.11.2011	no lecture	no lab
24.11.2011	Krill	Simon
01.12.2011	Krill	no lab

) ?

www.UZWR.de



 ulm university | universität | fakultäten | uulm

[Uni Aktuell](#) | [universität](#) | [Fakultäten](#) | [Forschung](#) | [Studium](#) | [Internationales](#) | [Weiterbildung](#) | [Campus](#)

uzwr

ulmer zentrum für wissenschaftliches rechnen



UZWR > Lehre > Introduction to FEA

> UZWR

- Aktuelles
- Personen und Organisation
- Lehre**
- NEU: Studiengang CSE
- AMISM 1 (Statik)
- AMISM 2 (Dynamik)
- Scientific Computing
- Strömungsmechanik
- Introduction to FEA**
- Abschlussarbeiten
- Forschung und Projekte
- Industrie-Kooperationen
- Hardware und Software
- Kontakt
- Links

 Search provided by [vista](#)
 [\[Suchoptionen\]](#)

Introduction to the Finite Element Analysis (FEA)



(Part of the lecture "Computational Methods in Materials Science")
 The UZWR gives three lessons of the lecture "Computational Methods in Materials Science". This part is a basic introduction into the Finite Element Method (FEM) together with three computer labs about the usage of the commercial FE package ANSYS.

The course will take place in the coming winter term.

Lecturers:
 Dr. Ulrich Simon, Arif Ouz Yurdalan

Participants:
 For Students of [International Master of Advanced Materials Science](#).

Lectures:

1. introduction into basic mechanical concepts and terms.
2. introduction into the FE theory, very simple example.
3. Material laws and properties.

Kontakt

→ Dr.-Ing. Ulrich Simon
 Geschäftsführer des UZWR
 Helmoltzstraße 20
 Raum 114
 89081 Ulm
 Telefon +49 (0)731 500-31700
 Telefax +49 (0)731 500-31709

Contents

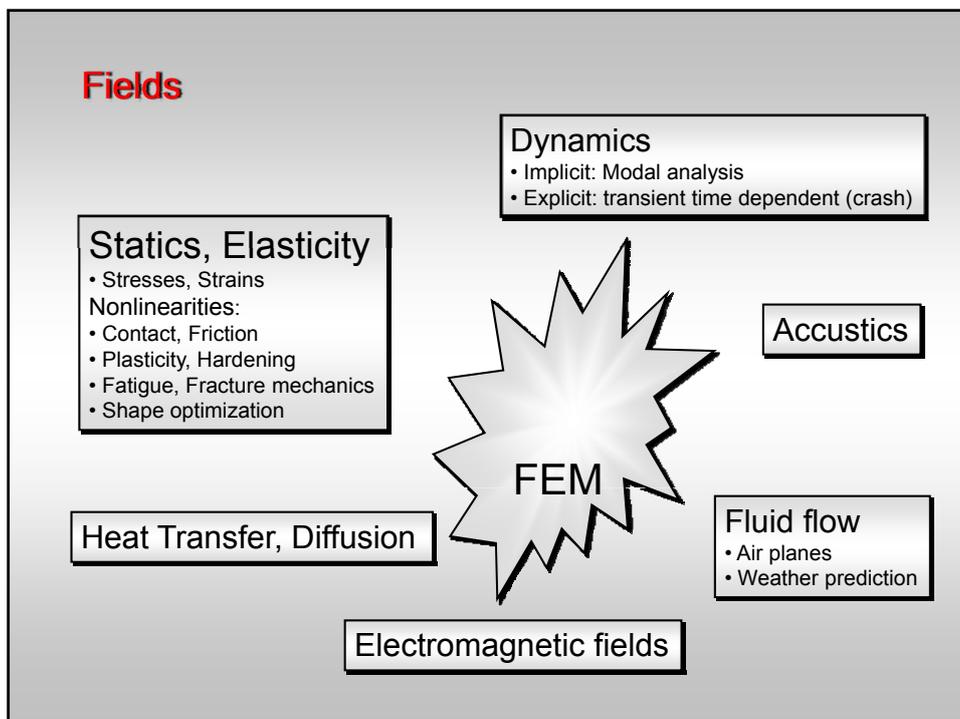
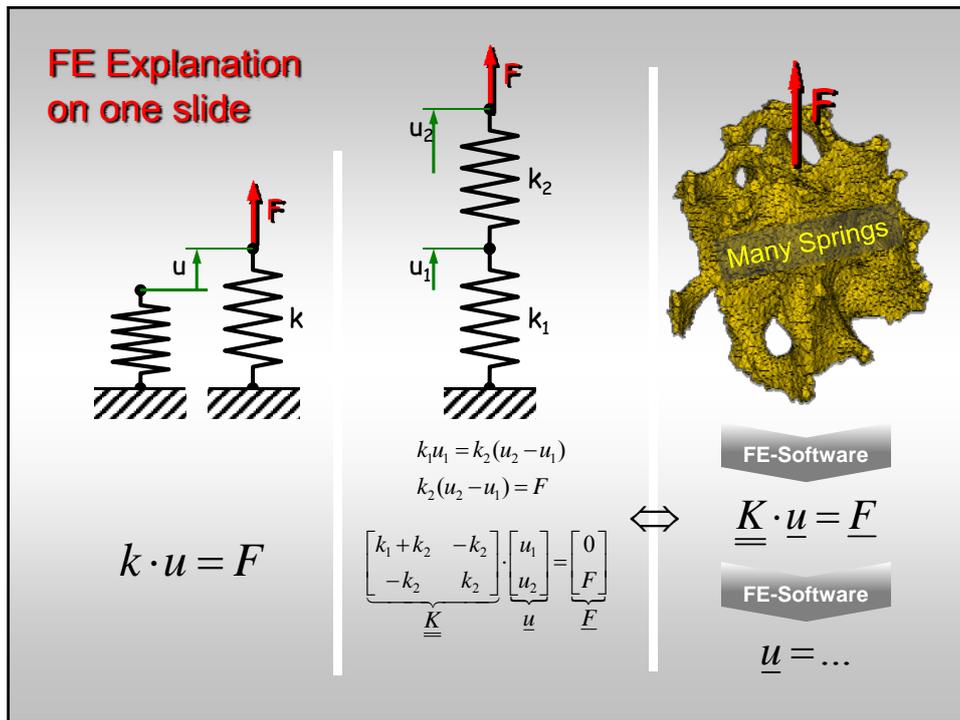
- Basic mechanical concepts and terms (force, moment, stress, strain)
- Introduction into the FE method
- FE theory with an simple example
- Complex material properties

Introduction to FEA
1st Part

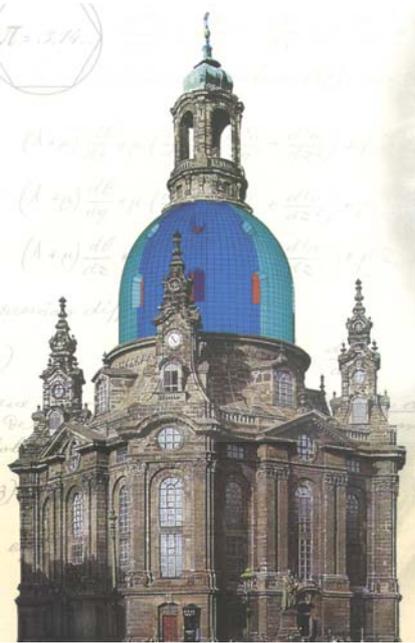
FE Explanation in one sentence

Finite Element Methode
=
Numerical Method
to solve partial differential
equations (PDEs) approximately





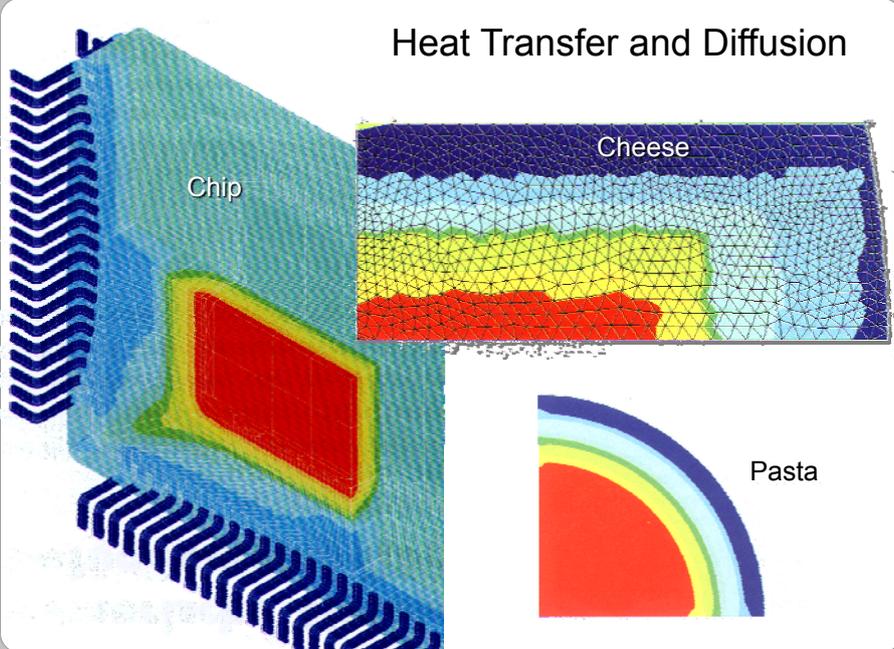
Statics, Elasticity



Frauenkirche, Dresden

The image shows the Frauenkirche in Dresden, a large Baroque church with a prominent blue dome. The dome is overlaid with a blue mesh, indicating a finite element analysis. The background features faint mathematical formulas, including $\pi = 3.14$, $(A + u) \frac{dA}{du}$, and $(A + u) \frac{dA}{dA}$.

Heat Transfer and Diffusion

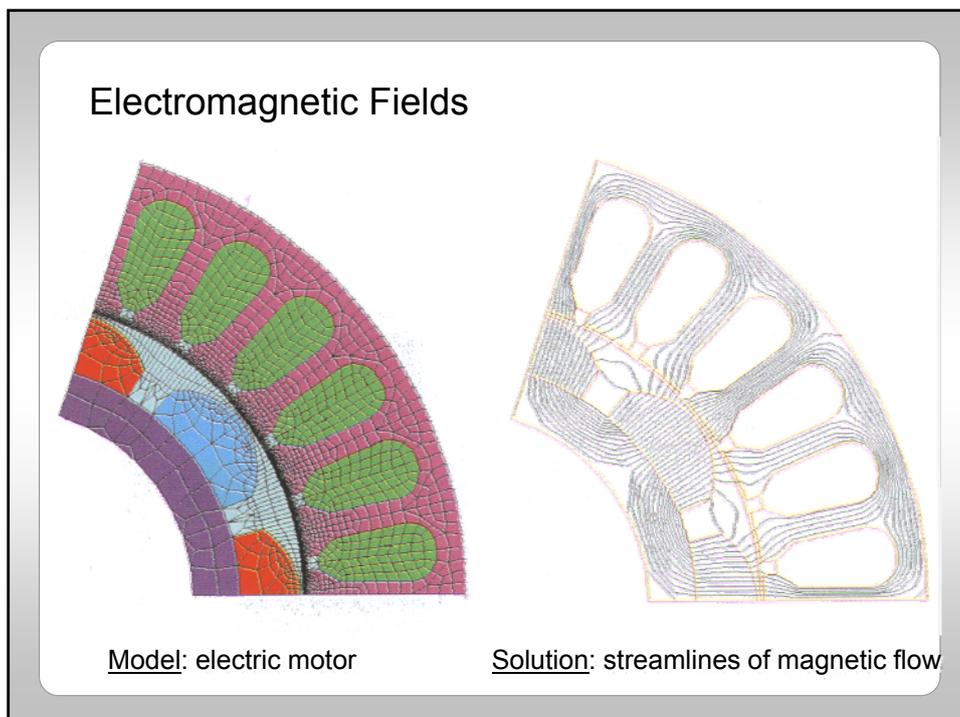
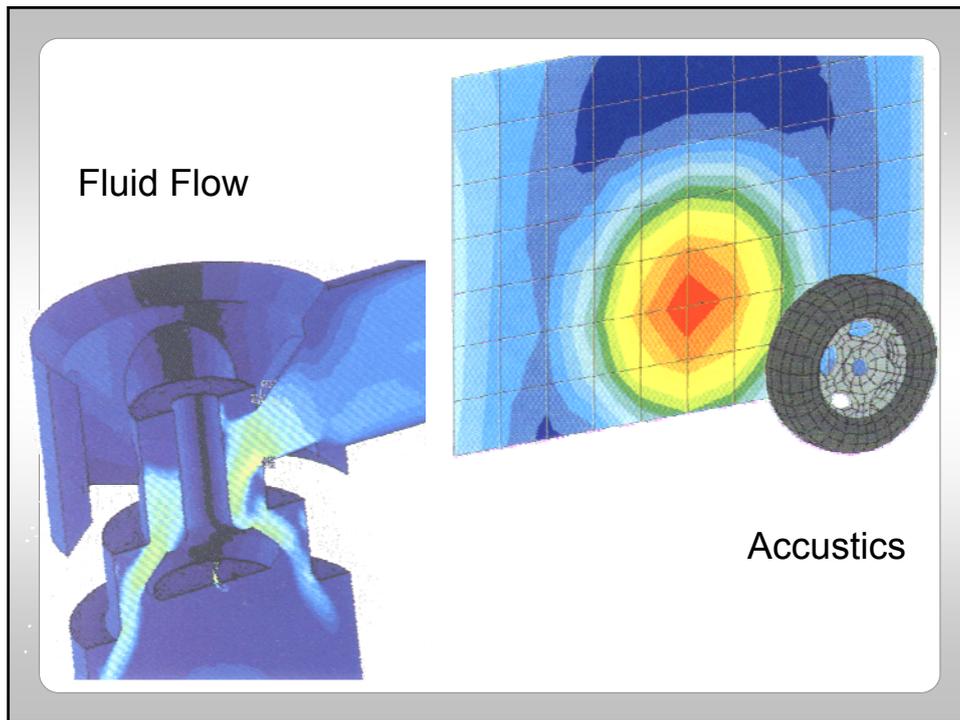


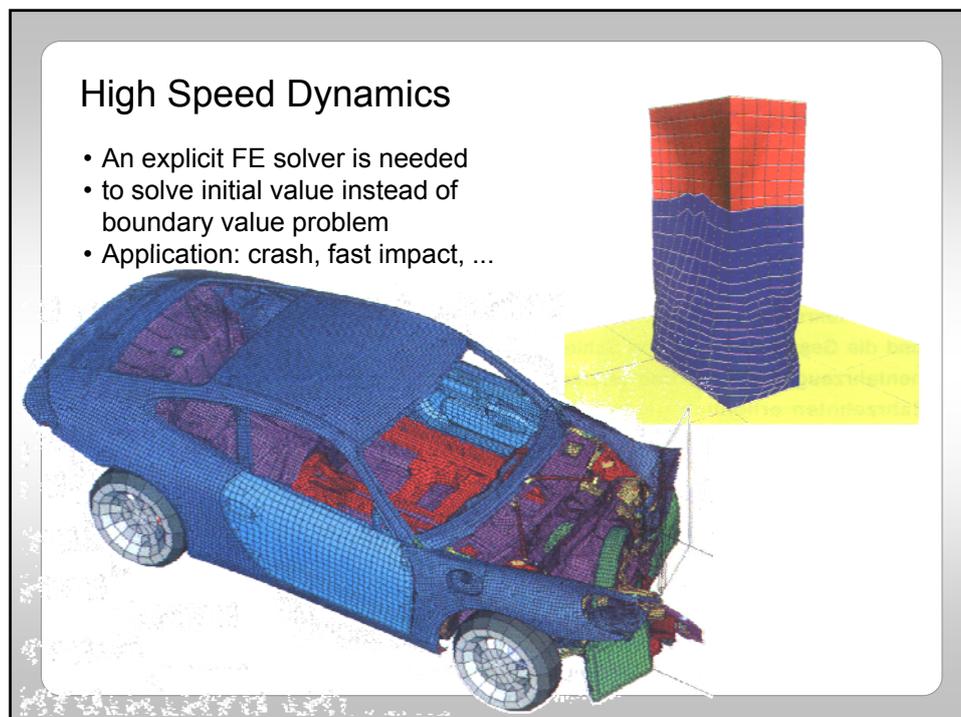
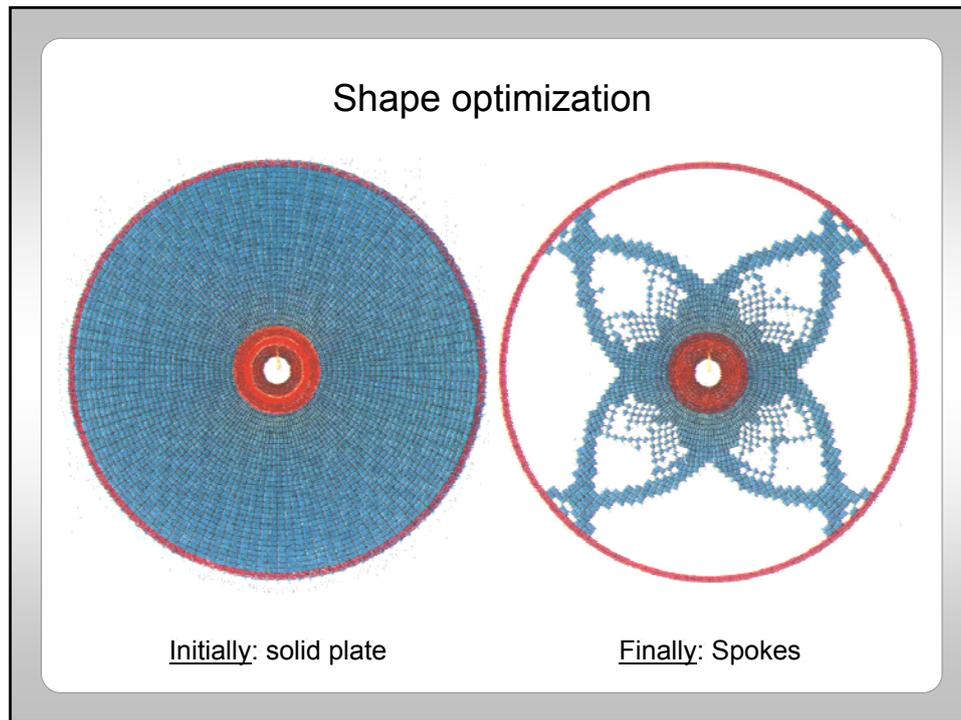
Chip

Cheese

Pasta

The image displays three finite element models illustrating heat transfer and diffusion. On the left is a 3D model of a chip with a red rectangular heat source on its surface. In the center is a cross-section of a sandwich with layers of cheese and pasta, showing a temperature gradient from red (hot) to blue (cold). On the right is a quarter-section of a pasta slice, also showing a temperature gradient.







Working Steps of a FEA

1. Preprocessor

- Geometry
- Mesh, Discretisation
- Material properties
- Load / boundary conditions

2. Solution

- Computer is working

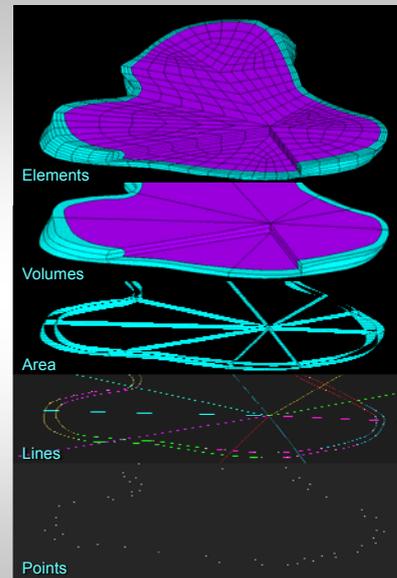
3. Postprocessor

- Verification, Validation
- Presenting results

Step 1: Preprocessor

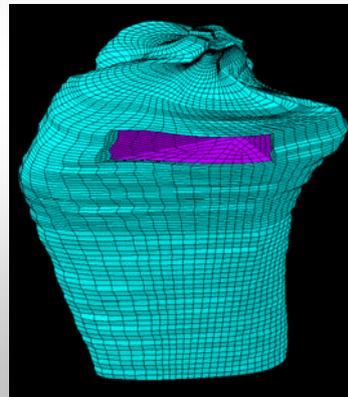
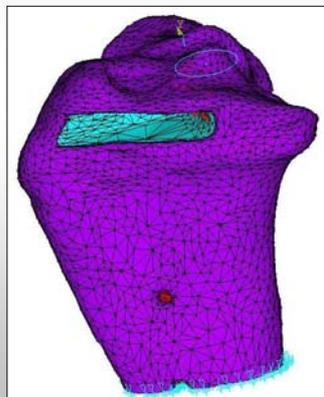
1.1. Generate/Import Geometry

- Botom-up Method
- CAD like Method: using Boolean operations: addition, subtraction of geometric primitives
- Direct generation of Elements: e.g. "Voxel Model"
- Import Geometry from CAD files



1.2. Meshing

- Tetraheadrons: better for complex geometry
- Hexaheadrons: better mechanical properties
- Convergence: better results with increasing number of elements, check it out!



1.3 Material laws and properties

- Simplest: Linear elastic, isotropic: E modulus E and Poisson's ratio ν
- More complex: Non-linear elastic, plastic, hardening, fatigue, cracks
- Anisotropic: Transverse Isotropic (wood), Orthotropic, ...
- Biphasic: Porous media

1.4 Load and Boundary Conditions (BC)

- Apply forces and/or displacements (or pressures, temperatures, ...)
- Forces can be applied to nodes
- Some programs allow application of line- area- ore volume-forces. The program will then distribute these forces to the underlying nodes automatically.
- Displacement BC are: fixations, supports, symmetries, constraints

Step 2: Solution

- The computer is doing the work
- Solver for linear systems: direct solver or iterative solver
- Solver for non-linear systems: iterativ, Newton-Raphson

Step 3: Post-Processor

- Presenting the results (important message)
- Displacements
- Strains, stresses
- Interpretation
- Verification (check code, convergence, plausibility, ...)
- Validation (compare with experiments)

Mechanical Basics

Variables, Dimensions and Units

Standard: ISO 31, DIN 1313

Variable = Number · Unit

Length $L = 2 \cdot \text{m} = 2 \text{ m}$

{Variable} = Number

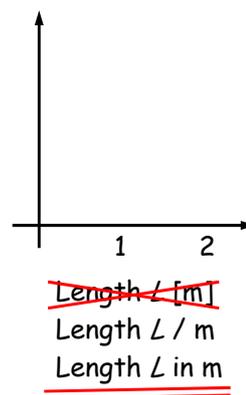
[Variable] = Unit

Three mechanical SI-Units:

m (Meter)

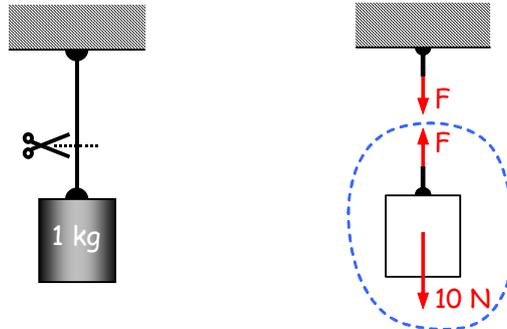
kg (Kilogram)

s (Seconds)



THE FORCE

Method of Sections [Schnittprinzip]



Note to Remember:

First, cut the system, then include forces and moments.

Free-body diagram = completely isolated part.

Units of Force

Newton

$$N = \text{kg} \cdot \text{m}/\text{s}^2$$

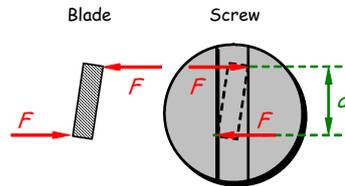
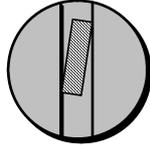


Note to Remember:

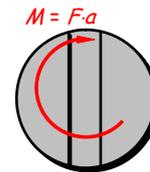
1 Newton \approx Weight of a bar of chocolate (100 g)

THE MOMENT [Das Moment]

Slotted screw with
screwdriver blade



Force Couples (F, a)



Moment M

Note to remember:

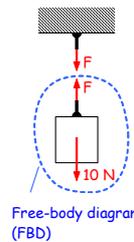
The moment $M = F \cdot a$ is equivalent to a force couple (F, a).

A moment is the cause for angular acceleration or angular deformation (Torsion, Bending) of a body.

Static Equilibrium

Important:

First free-body diagram (FBD), then equilibrium!



For 2D Problems max. 3 equations for each FBD:

The sum of all forces in x -direction equals zero: $F_{1,x} + F_{2,x} + \dots = 0$

The sum of all forces in y -direction equals zero: $F_{1,y} + F_{2,y} + \dots = 0$

The sum of Moments with respect to P equals zero: $M_{1,z}^P + M_{2,z}^P + \dots = 0$

(For 3D Problems max. 6 equations for each FBD)

Recipe for Solving Problems in Statics

Step 1: Model building. Generate a simplified replacement model (diagram with geometry, forces, constraints).

Step 2: Cutting, Free-body diagram. Cut system and develop free-body diagrams. Include forces and moments at cut, as well as weight.

Step 3: Equilibrium equations. Write the force- and moment equilibrium equations (only for free-body diagrams).

Step 4: Solve the equations. One can only solve for as many unknowns as equations, at most.

Step 5: Display results, explain, confirm with experimental comparisons. Are the results reasonable?

STRESSES

... to account for the loading of the material !

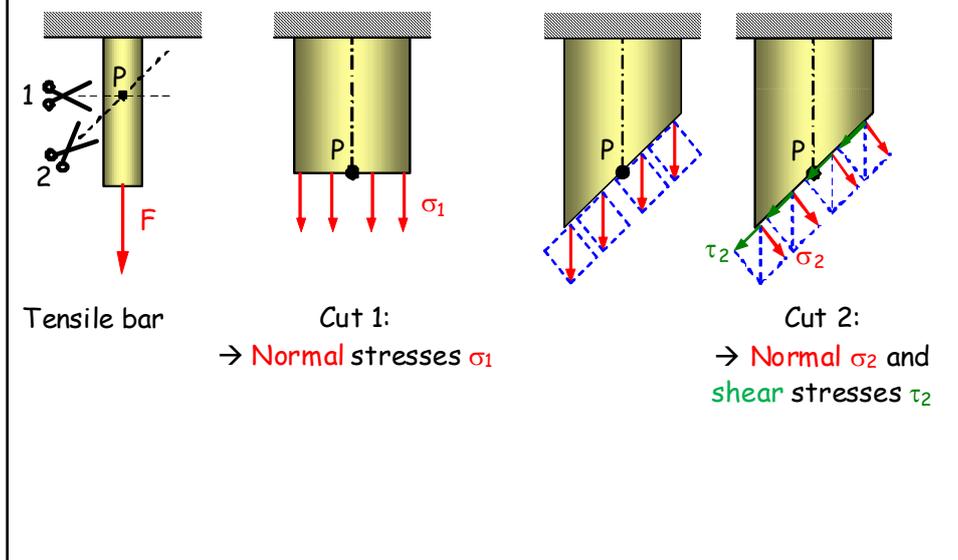


Note to Remember:

Stress = „smeared“ force

Stress = Force per Area or $\sigma = F/A$

Normal and Shear Stresses



Stress

Note to Remember:

First, you must choose a point and a cut through the point, then you can specify (type of) stresses at this point in the body.

Normal stresses (tensile and compressive stress) are oriented perpendicular to the cut-surface.

Shear stresses lie tangential to the cut-surface.

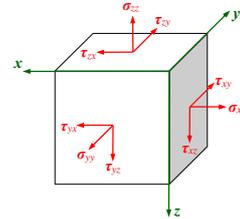
General 3D Stress State

... in a point of the body:

- 3 stress components in one cut (normal stress, 2x shear stress)
times
- 3 cuts
result in
- 9 stress components, but only
- 6 of these components will be independent (eq. of shear stresses)

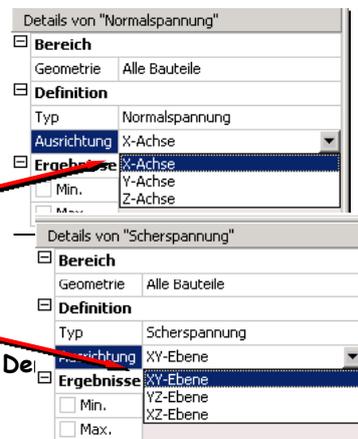
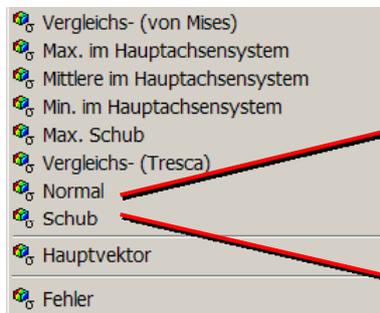
The „stress tensor“

$$\sigma = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} = \begin{bmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_{yy} & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_{zz} \end{bmatrix}$$

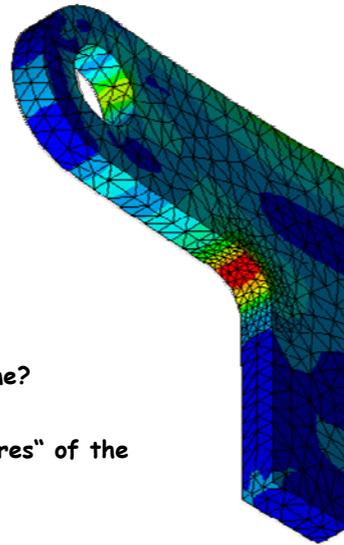


General 3D Stress State

6 Components



- ☞ Vergleichs- (von Mises)
- ☞ Max. im Hauptachsensystem
- ☞ Mittlere im Hauptachsensystem
- ☞ Min. im Hauptachsensystem
- ☞ Max. Schub
- ☞ Vergleichs- (Tresca)
- ☞ Normal
- ☞ Schub



Problem:

- How to produce nice Pictures?
- Which component should I use?
- Do I need 6 pictures at the same time?

So called „Invariants“ are „smart mixtures“ of the components

$$\sigma_{Mises} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{xx}\sigma_{zz} - \sigma_{yy}\sigma_{zz} + 3\tau_{xy}^2 + 3\tau_{xz}^2 + 3\tau_{yz}^2}$$

Strains

- Global, (external) strains

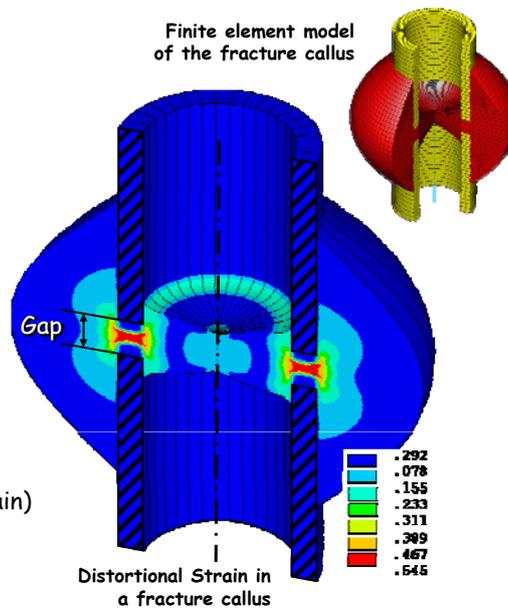
$$\epsilon := \frac{\text{Change in length}}{\text{Original length}} = \frac{\Delta L}{L_0}$$

- Local, (internal) strains

Units of Strain

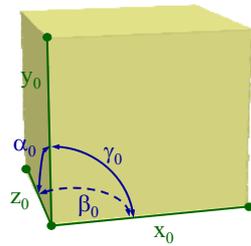
- without a unit
- 1
- 1/100 = %
- 1/1.000.000 = $\mu\epsilon$ (micro strain)
- = 0,1 %

Finite element model of the fracture callus

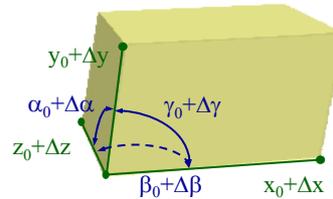


Distortional Strain in a fracture callus

Definition of the Local Strain State



undeformed



deformed

$$\varepsilon_{xx} = \frac{\Delta x}{x_0}, \quad \varepsilon_{yy} = \frac{\Delta y}{y_0}, \quad \dots$$

$$\gamma_{xy} = \frac{1}{2} \cdot \Delta\phi, \quad \gamma_{xz} = \dots$$

$$\underline{\underline{\varepsilon}} = \begin{bmatrix} \varepsilon_{xx} & \gamma_{xy} & \gamma_{xz} \\ \gamma_{xy} & \varepsilon_{yy} & \gamma_{yz} \\ \gamma_{xz} & \gamma_{yz} & \varepsilon_{zz} \end{bmatrix}$$

The strain tensor

Strain

Note to Remember:

Strain is relative change in length (and shape)

Strain = Change in length / Original length

Material Laws

... relation between stresses and strains

Linear-Elastic, Isotropic Material Law:

Two of the following three parameters are necessary:

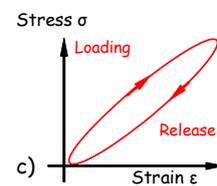
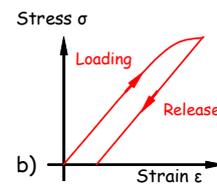
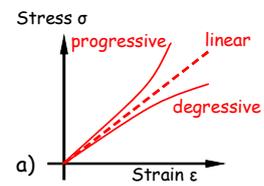
Young's Modulus E (Elastic Modulus) [Elastizitäts-Modul]

Shear Modulus G [Schubmodul]

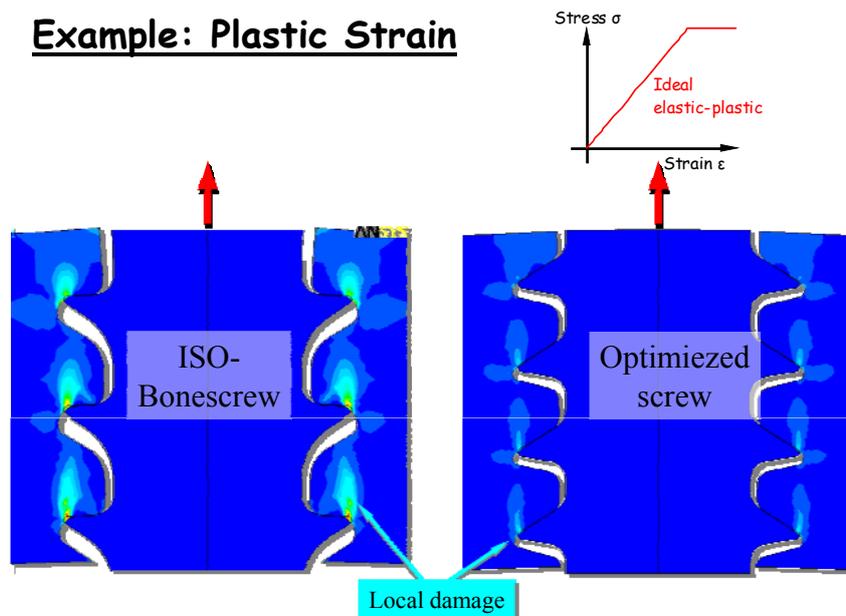
Poisson's ratio ν [Querkontraktionszahl]

Complex Material Laws:

- Non-linear (a)
- Non-elastic, plastic (b)
- Visco-elastic, Type: internal damping (c)
- Visco-elastic, Type: memory effect (c)
- Anisotropy



Example: Plastic Strain



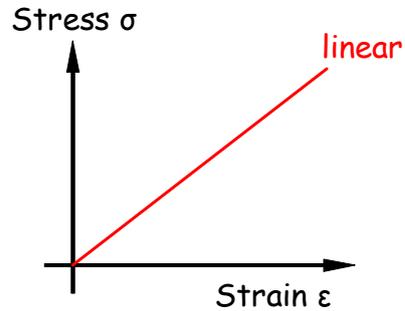
Isotropic vs. Anisotropic (linear elastic)

Linear stress-strain relation

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

$$\underline{\underline{\sigma}} = \underline{\underline{E}} \cdot \underline{\underline{\varepsilon}} \quad (81 \text{ Param.})$$

$$\underline{\sigma} = \underline{E} \cdot \underline{\varepsilon} \quad (36 \text{ Param.})$$



- Full 3⁴ material properties tensor of 4th order (81 Param.)
- Equality of shear stresses (Boltzmann Continua) and strains: (36 Param.)
- Reciprocity Theorem from Maxwell → fully anisotropic: (21 Param.)
- Orthotropic: (9 Param.)
- Transverse isotropic: (5 Param.)
- Full isotropic: (2 Param.)

Isotropic vs. Anisotropic (linear elastic)

$$\underline{\underline{\sigma}} = \underline{\underline{E}} \cdot \underline{\underline{\varepsilon}}$$

$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \frac{E}{(1+\nu) \cdot (1-2\nu)} \cdot \begin{bmatrix} (1-\nu) & \nu & \nu & 0 & 0 & 0 \\ & (1-\nu) & \nu & 0 & 0 & 0 \\ & & (1-\nu) & 0 & 0 & 0 \\ & & & \frac{(1-2\nu)}{2} & 0 & 0 \\ & & & & \frac{(1-2\nu)}{2} & 0 \\ & & & & & \frac{(1-2\nu)}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix}$$

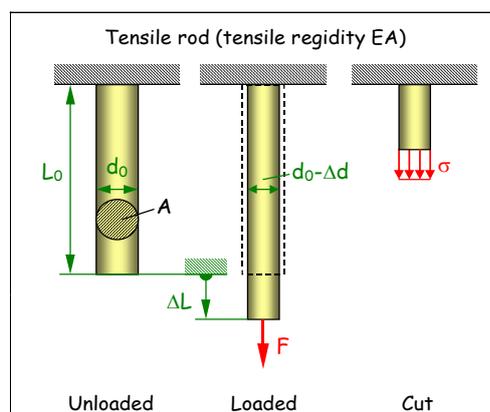
sym

- E - Young's modulus
- ν - Poisson's ratio (0 ... 0.5)
- G - Shear modulus
- K - Bulk modulus
- μ, λ - Lamé Constants

← 2 of these

Simple Load Cases for 1D objects

1 Tension and Compression



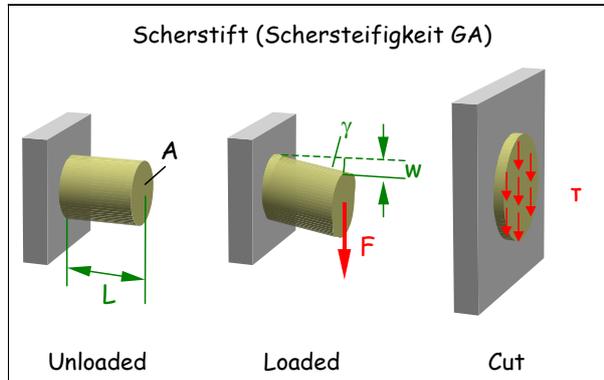
Global behavior, stiffness

$$F = \frac{EA}{L_0} \Delta L, \quad k = \frac{EA}{L_0}$$

Stresses in transverse cut

$$\sigma = \frac{F}{A}$$

Shear



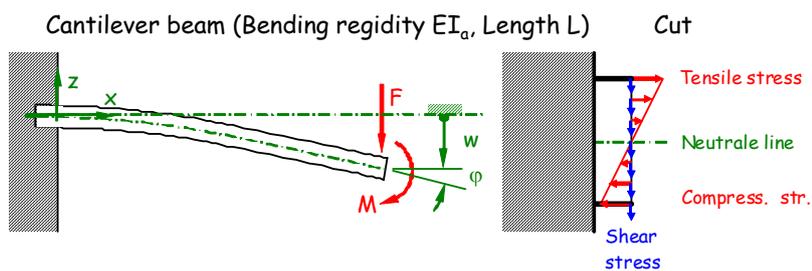
Global behavior, stiffness

$$F = \frac{GA}{L} w, \quad k = \frac{GA}{L}$$

Stresses in transverse cut

$$\tau = \frac{F}{A}$$

Bending (Cantilever beam)



Global behavior, compliance

$$w = \frac{L^3}{3EI_a} F + \frac{L^2}{2EI_a} M$$

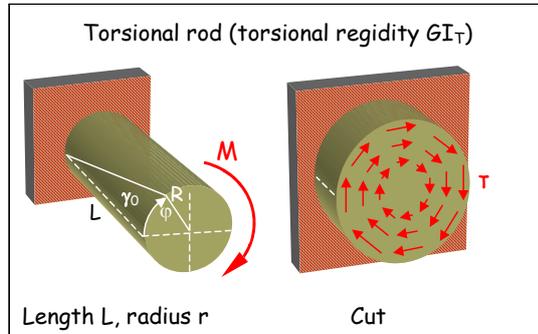
$$\varphi = \frac{L^2}{2EI_a} F + \frac{L}{EI_a} M$$

Local stress in transverse cut:

(normal stress)

$$\sigma_{xx}(x, z) = \frac{M + F(x-l)}{I_a} z$$

Torsion



Global behavior, stiffness

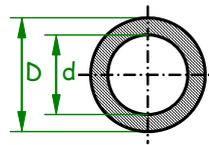
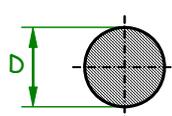
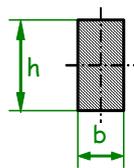
$$M = \frac{GI_T}{L} \phi, \quad c = \frac{GI_T}{L}$$

Stresses in transverse cut

$$\tau = \frac{M}{I_T} \rho$$

$\rho = \text{Distance from Center}$

Second Moment of Area I (SMA) [Flächenmoment zweiten Grades]



Axial moment of area (bending)

$$I_a = \frac{b \cdot h^3}{12}$$

$$I_a = \frac{\pi}{64} D^4$$

$$I_a = \frac{\pi}{64} (D^4 - d^4)$$

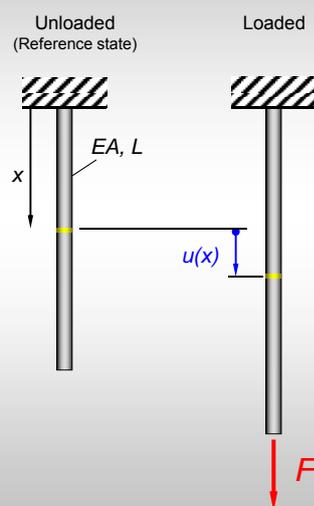
Polar moment of area (torsion)

$$I_T = I_p = \frac{\pi}{32} D^4$$

$$I_T = I_p = \frac{\pi}{32} (D^4 - d^4)$$

Theory of the Finite Element Method using a 'super simple' example

Example: Tensile Rod



Given:

Rod with ...

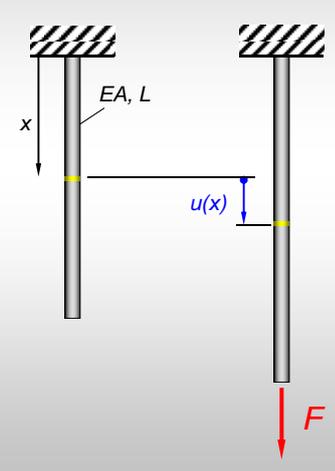
- Length L
- Cross-section A (constant)
- E-modulus E (constant)
- Force F (axial)
- Upper end fixed

To determine:

Deformation of the loaded rod:

Displacement function $u(x)$

A) Classical Solution (Method of „infinite“ Elements)

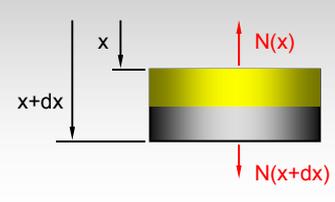


EA, L

$u(x)$

F

Differential Element
(infinitesimale High dx)



x

$x+dx$

$N(x)$

$N(x+dx)$

Generate the Differential Equation

1. Kinematics: $\epsilon = u'$
2. Material: $\sigma = E\epsilon \Rightarrow N = EAu'$
3. Equilibrium: $N' = 0$

\Rightarrow DGI: $(EA u')' = 0$

If $EA = \text{const}$ then

$u'' = 0$

A) Classical Solution (Method of „infinite“ Elements)

Solve the Differential Equation

$$u''(x) = 0$$

Integrate 2 times:

$$u'(x) = C_1$$

$$u(x) = C_1 * x + C_2 \quad (\text{General Solution})$$

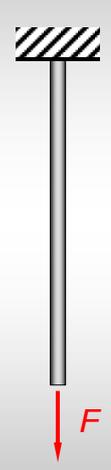
Adjust to Boundary Conditions

Top (Fixation): $u(0) = 0 \Rightarrow C_2 = 0$

Bottom (open, Force): $N(L) = F \Rightarrow u'(L) = F/(EA)$
 $\Rightarrow C_1 = F/EA$

Adjusted Solution

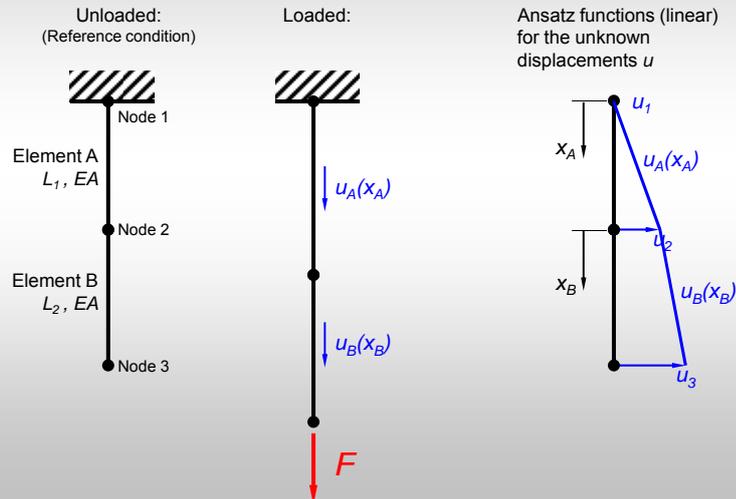
$$u(x) = (F/EA) * x$$



F

B) Solution with FEM

Discretization: We divide the rod into (only) two finite (= not infinitesimal small) **Elements**. The Elements are connected at their **nodes**.



The unknown displacement function of the entire rod is described with a series of simple (linear) **ansatz functions** (see figure). This is the **basic concept** of FEM.

$$u_A(x_A) = \hat{u}_1 + (\hat{u}_2 - \hat{u}_1) \frac{x_A}{L_A} = \hat{u}_1 \left(1 - \frac{x_A}{L_A}\right) + \hat{u}_2 \frac{x_A}{L_A}$$

$$u_B(x_B) = \hat{u}_2 + (\hat{u}_3 - \hat{u}_2) \frac{x_B}{L_B} = \hat{u}_2 \left(1 - \frac{x_B}{L_B}\right) + \hat{u}_3 \frac{x_B}{L_B}$$

The remaining unknowns are the three "nodal displacements" $\hat{u}_1, \hat{u}_2, \hat{u}_3$ and a no longer a whole function $u(x)$. Now we introduce the so-called "**virtual displacements (VD)**". These are additional, virtual, arbitrary displacements $\delta \hat{u}_1, \delta \hat{u}_2, \delta \hat{u}_3$. Basically: we "waggle" the nodes a bit.

Now the **Principle of Virtual Displacements (PVD)** applies: A mechanical system is in equilibrium when the total work (i.e. elastic minus external work) due to the virtual displacements consequently disappears.

$$\delta W = 0 \Rightarrow \delta W_{el} - \delta W_a = 0$$

For our simple example we can apply:

virt. elastic work = normal force N times VD

virt. external work = external force F times VD

The normal force N can be replaced by the expression EA/L times the element elongation. Element elongation again can be expressed by a difference of the nodal displacements:

$$\begin{aligned}\delta W &= N_A(\delta \hat{u}_2 - \delta \hat{u}_1) + N_B(\delta \hat{u}_3 - \delta \hat{u}_2) - F \delta \hat{u}_3 \\ &= \frac{EA}{L_A}(\hat{u}_2 - \hat{u}_1)(\delta \hat{u}_2 - \delta \hat{u}_1) + \frac{EA}{L_B}(\hat{u}_3 - \hat{u}_2)(\delta \hat{u}_3 - \delta \hat{u}_2) - F \delta \hat{u}_3 \\ \delta W &= \delta \hat{u}_1 \left(\frac{EA}{L_A} \hat{u}_1 - \frac{EA}{L_A} \hat{u}_2 \right) \\ &\quad + \delta \hat{u}_2 \left(-\frac{EA}{L_A} \hat{u}_1 + \frac{EA}{L_A} \hat{u}_2 + \frac{EA}{L_B} \hat{u}_2 - \frac{EA}{L_B} \hat{u}_3 \right) \\ &\quad + \delta \hat{u}_3 \left(-\frac{EA}{L_B} \hat{u}_2 + \frac{EA}{L_B} \hat{u}_3 - F \right) = 0\end{aligned}$$

With this principle we unfortunately have only one equation for the three unknown displacements $\hat{u}_1, \hat{u}_2, \hat{u}_3$. **What a shame!** However, there is a trick...

Abbreviated we write:

$$\delta \hat{u}_1(\dots)_1 + \delta \hat{u}_2(\dots)_2 + \delta \hat{u}_3(\dots)_3 = 0$$

The virtual displacements can be chosen independently of one another. For instance all except one can be zero. Then the term within the bracket next to this not zero VD has to be zero, in order to fulfill the equation. However, as we can choose the VD we want and also another VD could be chosen as the only non-zero value, consequently all three brackets must individually be zero. We get three equations. **Juhu!**

$$(\dots)_1 = 0; \quad (\dots)_2 = 0; \quad (\dots)_3 = 0$$

... which we can also write down in matrix form:

$$\begin{bmatrix} \frac{EA}{L_1} & -\frac{EA}{L_1} & 0 \\ -\frac{EA}{L_1} & \frac{EA}{L_1} + \frac{EA}{L_2} & -\frac{EA}{L_2} \\ 0 & -\frac{EA}{L_2} & \frac{EA}{L_2} \end{bmatrix} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ F \end{bmatrix}$$

$$\begin{bmatrix} \frac{EA}{L_1} & -\frac{EA}{L_1} & 0 \\ -\frac{EA}{L_1} & \frac{EA}{L_1} + \frac{EA}{L_2} & -\frac{EA}{L_2} \\ 0 & -\frac{EA}{L_2} & \frac{EA}{L_2} \end{bmatrix} \begin{bmatrix} \hat{u}_1 \\ \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ F \end{bmatrix}$$

Or in short:

$$\underline{\underline{K}} \underline{\hat{u}} = \underline{\underline{F}}$$

$\underline{\underline{K}}$ - Stiffness matrix
 $\underline{\hat{u}}$ - Vector of the unknown nodal displacement
 $\underline{\underline{F}}$ - Vector of the nodal forces

This is the classical fundamental equation of a structural mechanics, linear FE-analysis. A **linear system of equations** for the unknown nodal displacements

We still have to account for the **boundary conditions**. The rod is fixed at the top end. As a consequence node 1 cannot be displaced:

$$\hat{u}_1 = 0$$

Because the virtual displacements also have to fulfill the boundary conditions we have $\delta \hat{u}_1 = 0$. Therefore we need to eliminate the first line in the system of equations, as this equation does no longer need to be fulfilled. The first column of the matrix can also be removed, as these elements are in any case multiplied by zero. So it becomes ...

$$\begin{bmatrix} \frac{EA}{L_1} + \frac{EA}{L_2} & -\frac{EA}{L_2} \\ -\frac{EA}{L_2} & \frac{EA}{L_2} \end{bmatrix} \begin{bmatrix} \hat{u}_2 \\ \hat{u}_3 \end{bmatrix} = \begin{bmatrix} 0 \\ F \end{bmatrix}$$

$$u(x) = (F/EA) \cdot x$$

We **solve** the system of equations and obtain the nodal displacements

$$\hat{u}_2 = \frac{L_A}{EA} F \quad \text{und} \quad \hat{u}_3 = \frac{L_A + L_B}{EA} F$$

Here the FE-solution corresponds exactly with the (existing) analytical solution. In a more complex example this would not be the case. Generally, it applies that the convergence of the numerical solution with the exact solution continually improves with an increasing number of finite elements. For extremely complicated problems there is no longer an analytical solution; for such cases one needs FEM!

From the nodal displacements one can also determine strains and stresses in a subsequent calculation. In our example strains and stresses stay constant within the elements.

$$\varepsilon_A(x_A) = \frac{\hat{u}_2 - \hat{u}_1}{L_A}$$

$$\varepsilon_B(x_B) = \frac{\hat{u}_3 - \hat{u}_2}{L_B}$$

Strain

Finished!

Summary

The **essential steps** and ideas of FEM are thus:

- Discretization: Division of the spatial domain into finite elements
- Choose simple ansatz functions (polynomials) for the unknown variables within the elements. This reduces the problem to a finite number of unknowns.
- Write up a mechanical principle (e.g. PVD, the mathematician says “weak formulation” of the PDE) and
- From this derive a system of equations for the unknown nodal variables
- Solve the system of equations

Many of these steps will no longer be apparent when using a commercial FE program. With the selection of an analysis and an element type the underlying PDE and the ansatz functions are implicitly already chosen. The mechanical principle was only being used during the development of the program code in order to determine the template structure of the stiffness matrix. During the program run no mechanical principle like PVD will be performed.

General Hints and Warnings

- FEA is a tool, not an solution
 - Take care about nice pictures („GiGo“)
 - Parameter
 - Verification
 - FE models are case (question) specific
- } needs experiments

Literature and Links reg. FEM

Books:

- Zienkiewicz, O.C.: „Methode der finiten Elemente“; Hanser 1975 (engl. 2000).
The bible of FEM (German and English)
- Bathe, K.-J.: „Finite-Elemente-Methoden“; erw. 2. Aufl.; Springer 2001
Textbook (theory)
- Dankert, H. and Dankert, J.: „Technische Mechanik“; Statik, Festigkeitslehre, Kinematik/Kinetik, mit Programmen; 2. Aufl.; Teubner, 1995.
German mechanics textbook incl. FEM, with nice homepage
<http://www.dankertdankert.de/>
- Müller, G. and Groth, C.: „FEM für Praktiker, Band 1: Grundlagen“, mit ANSYS/ED-Testversion (CD). (Band 2: Strukturdynamik; Band 3: Temperaturfelder)
ANSYS Intro with examples (German)
- Smith, I.M. and Griffiths, D.V.: „Programming the Finite Element Method“
From engineering introduction down to programming details (English)
- Young, W.C. and Budynas, G.B.: „Roark's Formulas for Stress and Strain“
Solutions for many simplified cases of structural mechanics (English)

Links:

- Z88 Free FE-Software: <http://z88.uni-bayreuth.de/>

