Master Advanced Materials

—

Computational Methods in Materials Science

Winter Term 2013/2014

Introduction to the Finite Element Analysis (FEA)

Dr.-Ing. Ulrich Simon

Dipl.-Inf. Frank Niemeyer

Dipl.-Ing. Malte Steiner

www.UZWR.de

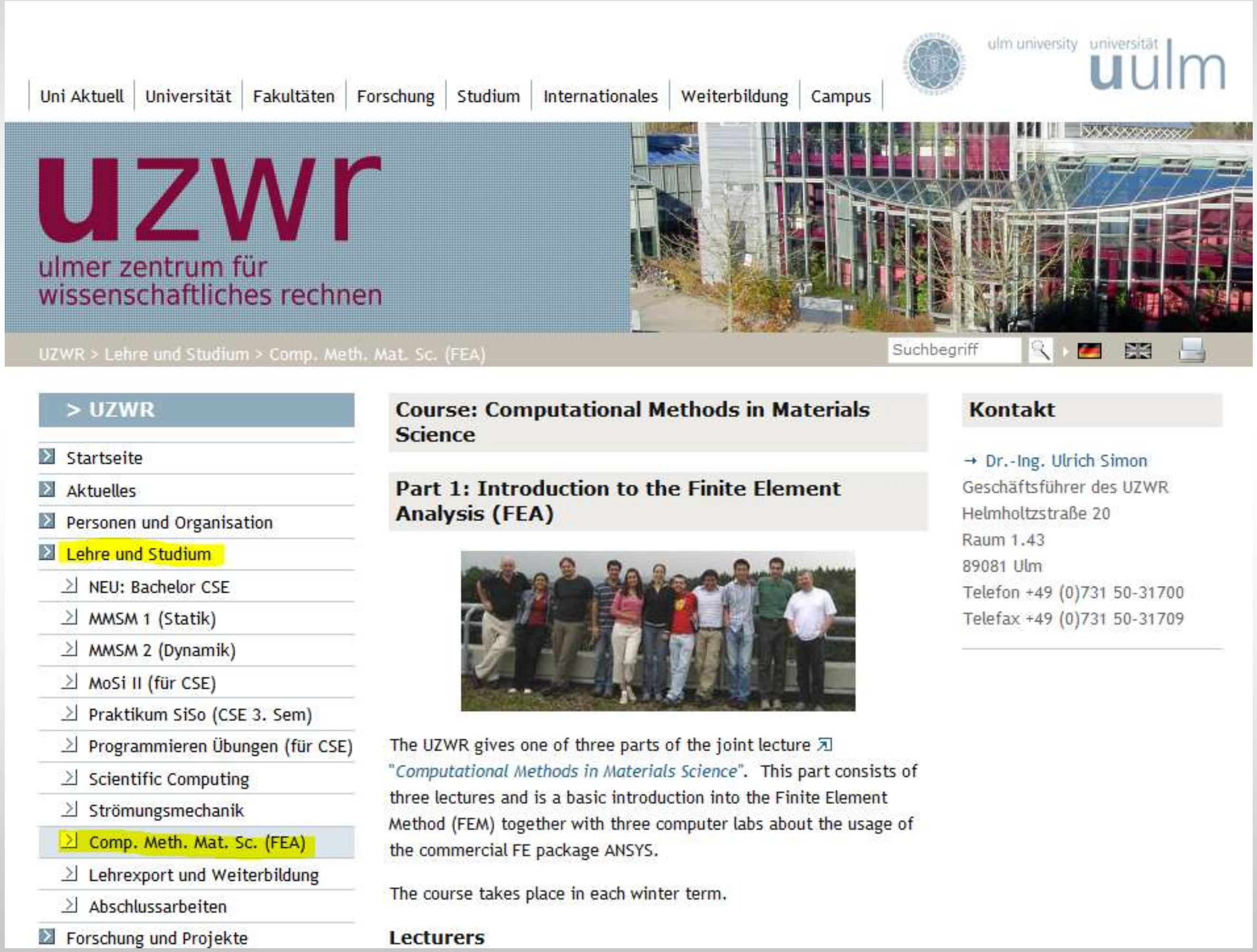
Schedule

Schedule for WS 2013/2014

date	lecturer	lab
17.10.2013	Krill	–
24.10.2013	Simon	–
31.10.2013	Simon	Simon
07.11.2013	Simon	Simon
14.11.2013	Krill	Simon
21.11.2013	Krill	–
28.11.2013	Krill	Krill
05.12.2013	Krill	Krill
12.12.2013	Krill	Krill
19.12.2013	no lecture!	–
09.01.2014	Herr	–
16.01.2014	Herr	Herr
23.01.2014	Herr	Herr
30.01.2014	(Herr)	Herr
06.02.2014	no lecture	no lab
13.02.2014	no lecture	no lab

} Niemeyer

www.uzwr.de



Uni Aktuell | Universität | Fakultäten | Forschung | Studium | Internationales | Weiterbildung | Campus

ulm university universität uulm

uzwr
ulmer zentrum für wissenschaftliches rechnen

UZWR > Lehre und Studium > Comp. Meth. Mat. Sc. (FEA)

Suchbegriff

> UZWR

- Startseite
- Aktuelles
- Personen und Organisation
- Lehre und Studium
  - NEU: Bachelor CSE
  - MMSM 1 (Statik)
  - MMSM 2 (Dynamik)
  - MoSi II (für CSE)
  - Praktikum SiSo (CSE 3. Sem)
  - Programmieren Übungen (für CSE)
  - Scientific Computing
  - Strömungsmechanik
  - Comp. Meth. Mat. Sc. (FEA)
  - Lehrexport und Weiterbildung
  - Abschlussarbeiten
- Forschung und Projekte

Course: Computational Methods in Materials Science

Part 1: Introduction to the Finite Element Analysis (FEA)

The UZWR gives one of three parts of the joint lecture "*Computational Methods in Materials Science*". This part consists of three lectures and 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 takes place in each winter term.

Lecturers

Kontakt

→ Dr.-Ing. Ulrich Simon
Geschäftsführer des UZWR
Helmholtzstraße 20
Raum 1.43
89081 Ulm
Telefon +49 (0)731 50-31700
Telefax +49 (0)731 50-31709

Contents

- General Introduction to FEA
- Mechanical Basics
- FE Theory, Ultra Simple Introduction

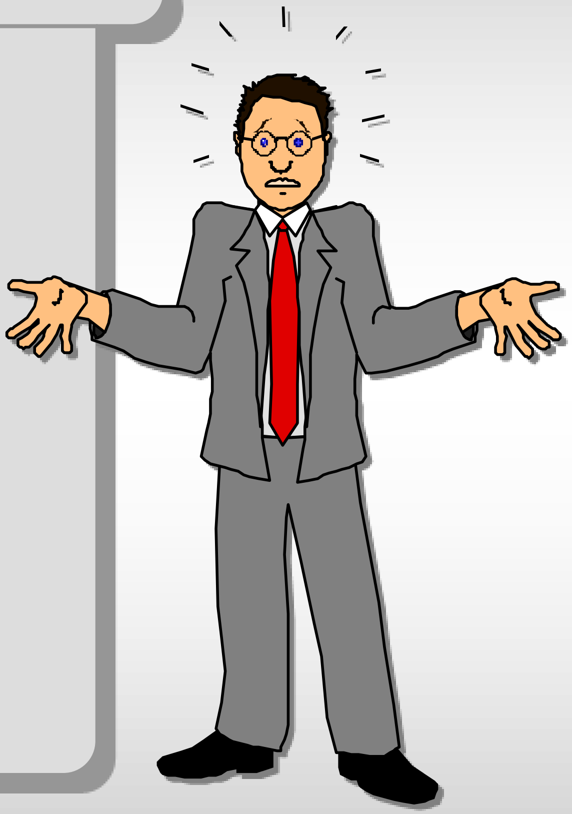
General Introduction to FEA

FE Explanation in one sentence

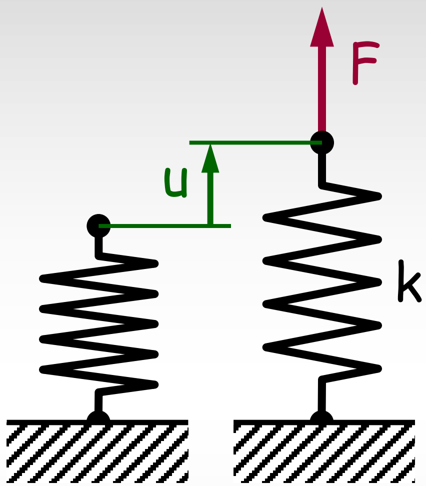
Finite Element Methode

=

Numerical Method
to solve partial differential
equations (PDEs) approximately

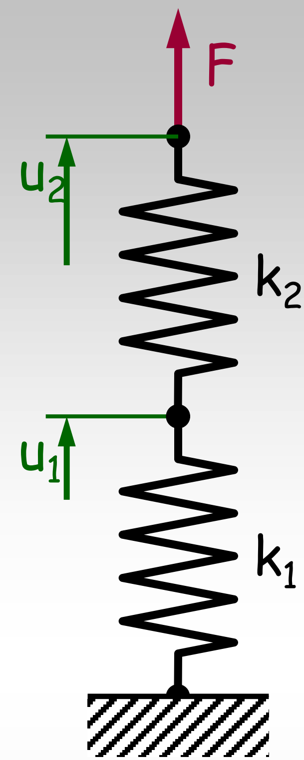


FE Explanation on one slide



$$k \cdot u = F$$

$$u = k^{-1} F$$



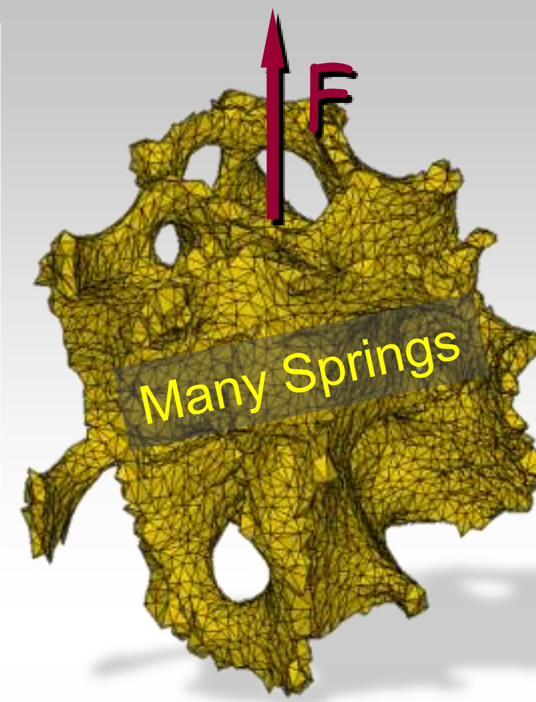
$$k_1 u_1 = k_2 (u_2 - u_1)$$

$$k_2 (u_2 - u_1) = F$$

$$\underbrace{\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}}_{\underline{\underline{K}}} \cdot \underbrace{\begin{bmatrix} u_1 \\ u_2 \end{bmatrix}}_{\underline{u}} = \underbrace{\begin{bmatrix} 0 \\ F \end{bmatrix}}_{\underline{F}}$$

$$\underline{u} = \underline{\underline{K}}^{-1} \underline{F}$$

$\Leftrightarrow$



FE-Software

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

FE-Software

$$\underline{u} = \underline{\underline{K}}^{-1} \underline{F}$$

Fields

Dynamics
- Implicit: Modal analysis
- Explicit: transient time dependent (crash)

Statics, Elasticity
- Stresses, Strains

Nonlinearities:
- Contact, Friction
- Plasticity, Hardening
- Fatigue, Fracture mechanics
- Shape optimization

Accustics

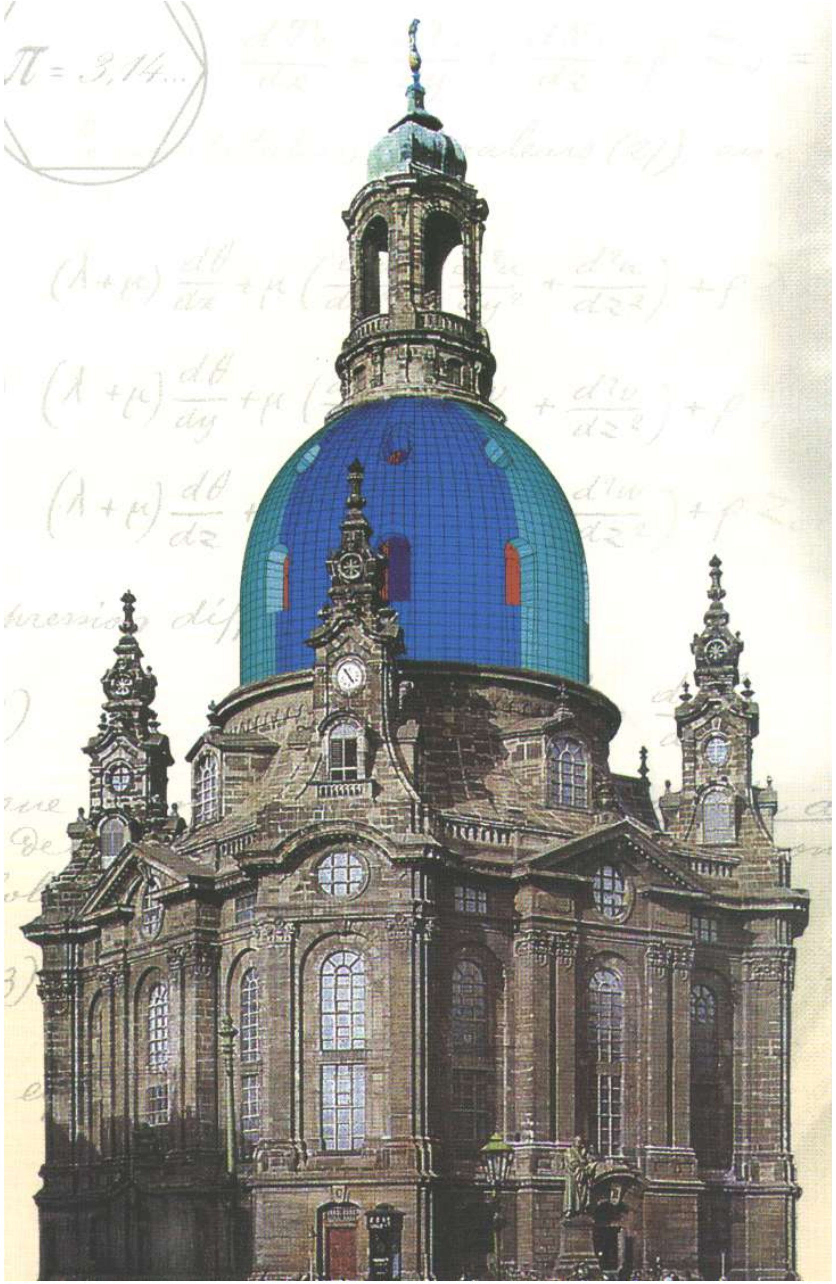
FEM

Heat Transfer, Diffusion

Fluid flow
- Air planes
- Weather prediction

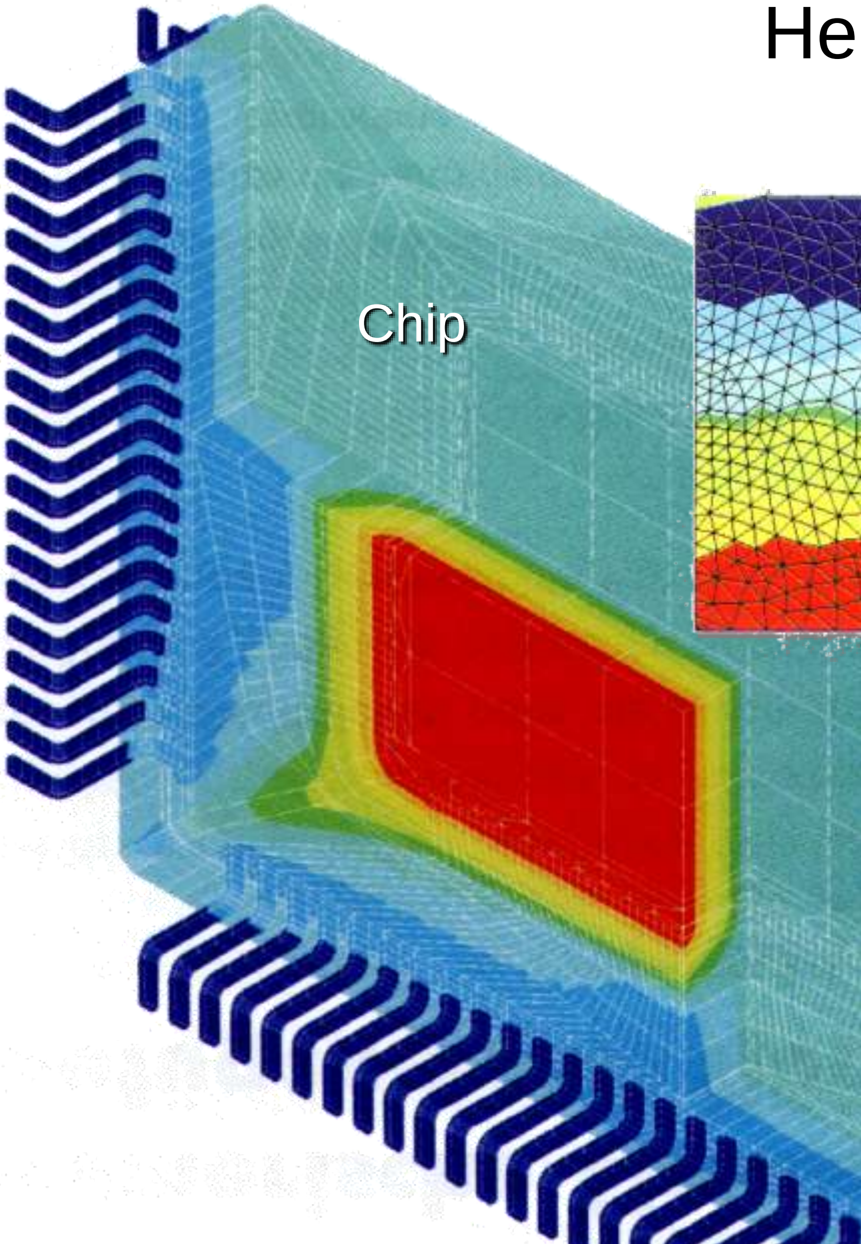
Electromagnetic fields

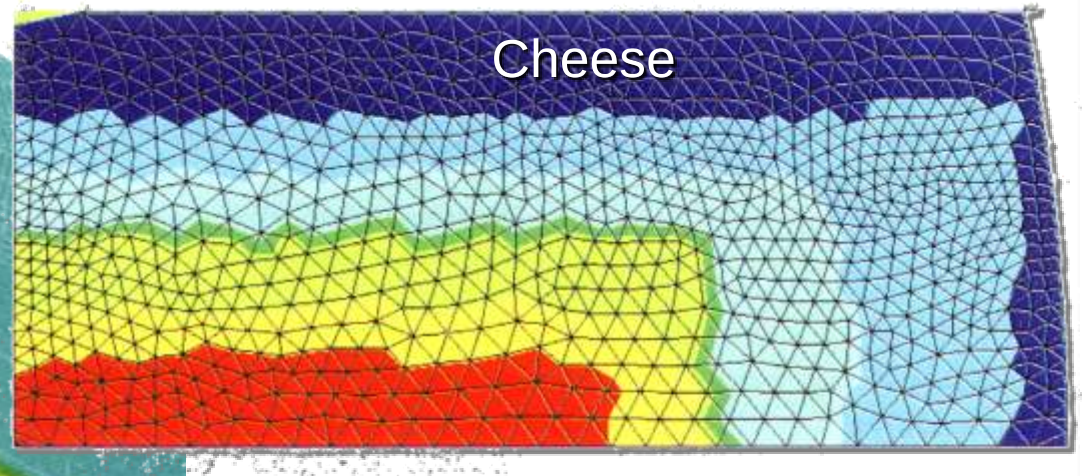
Statics, Elasticity

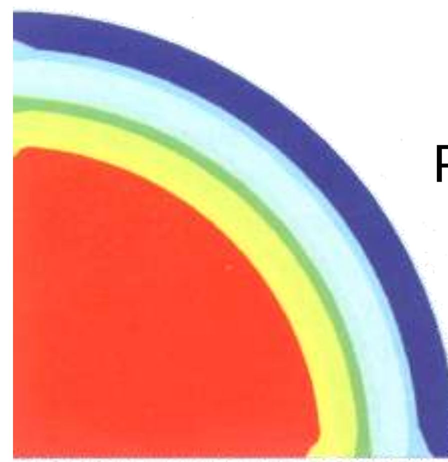


Frauenkirche, Dresden

Heat Transfer and Diffusion

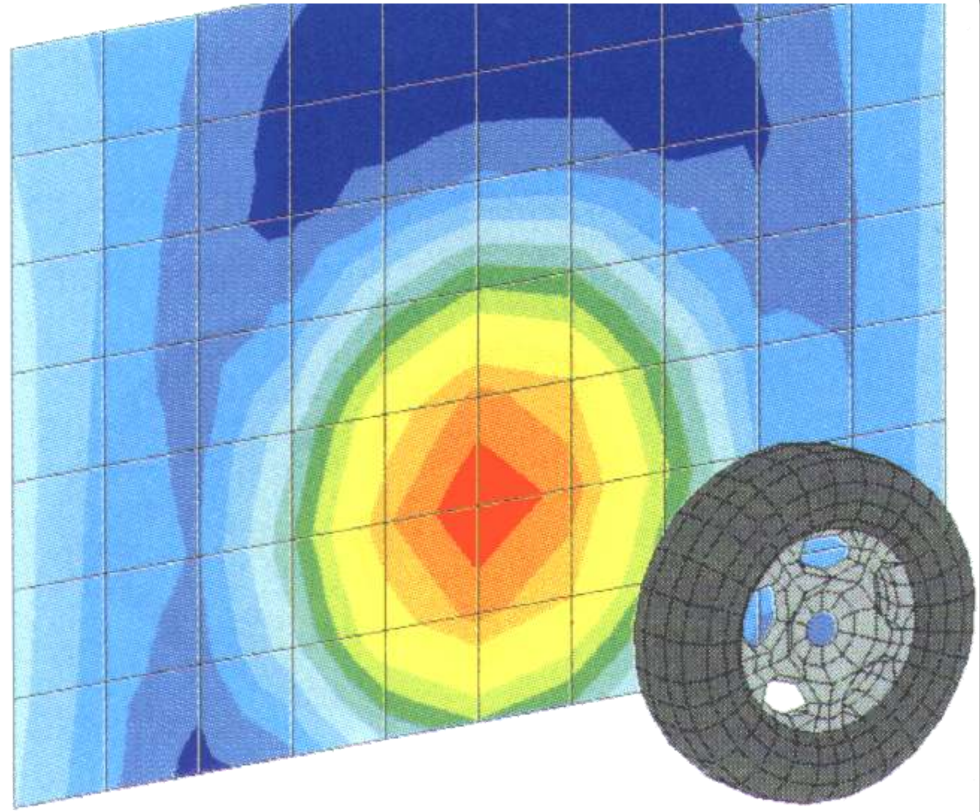






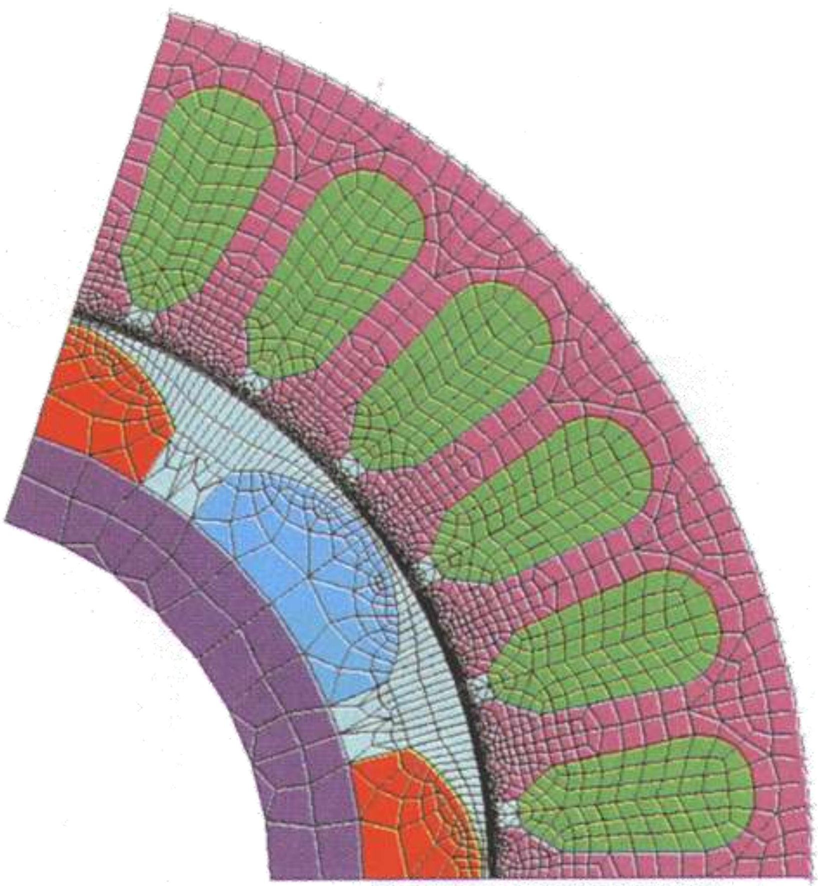
Pasta

Fluid Flow



Accustics

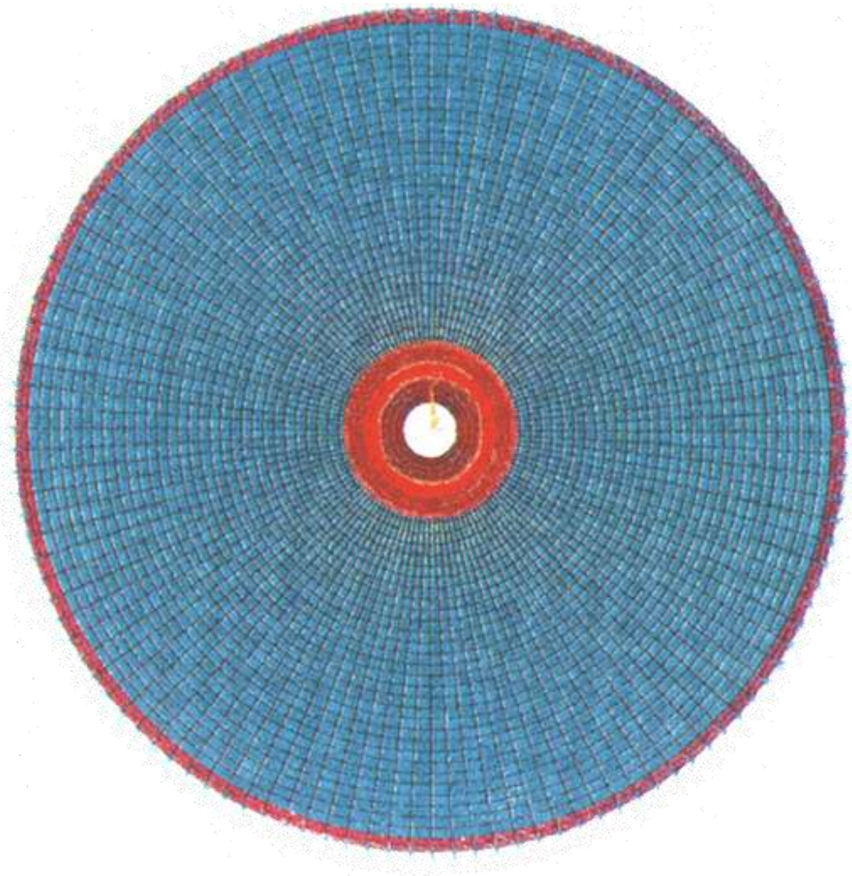
Electromagnetic Fields



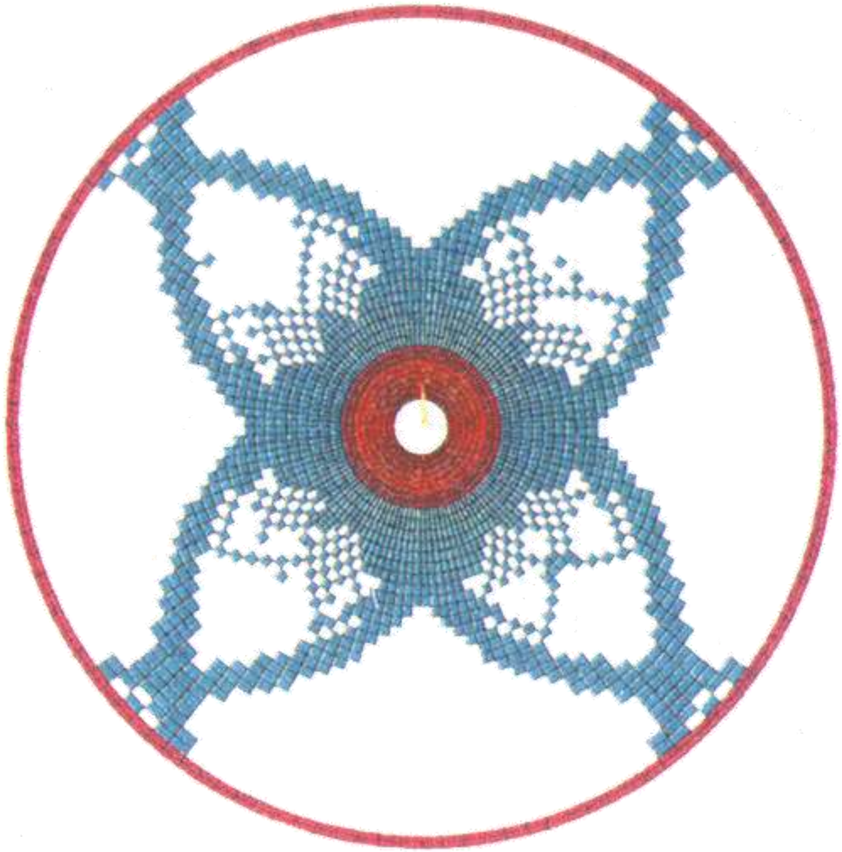
Model: electric motor

Solution: streamlines of magnetic flow

Shape optimization



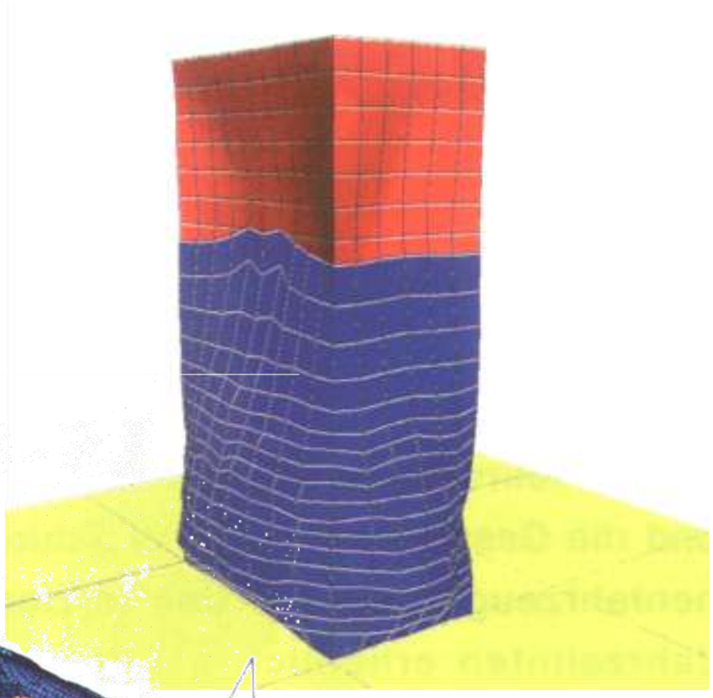
<u>Initially</u>: solid plate

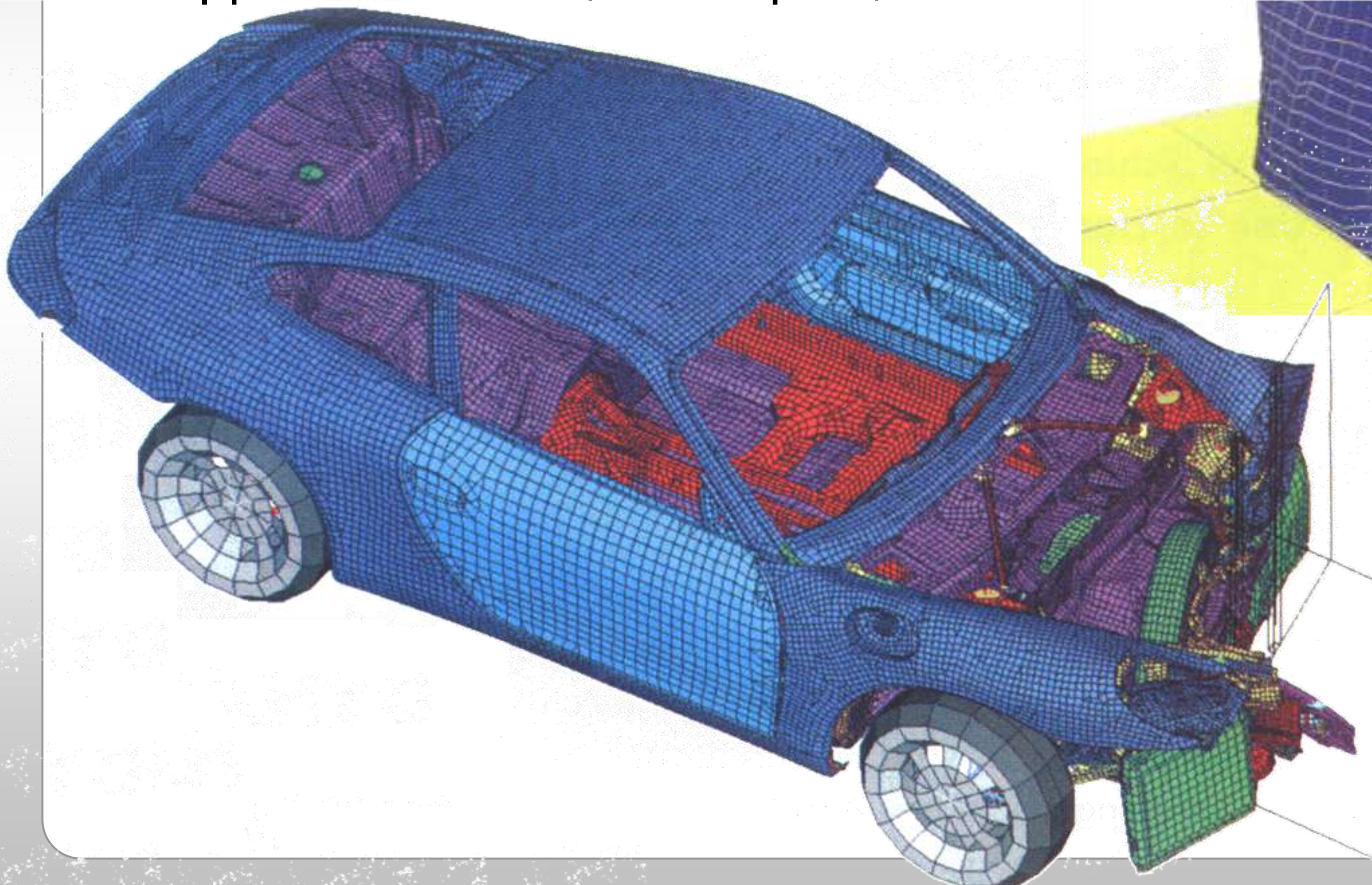


<u>Finally</u>: Spokes

High Speed Dynamics

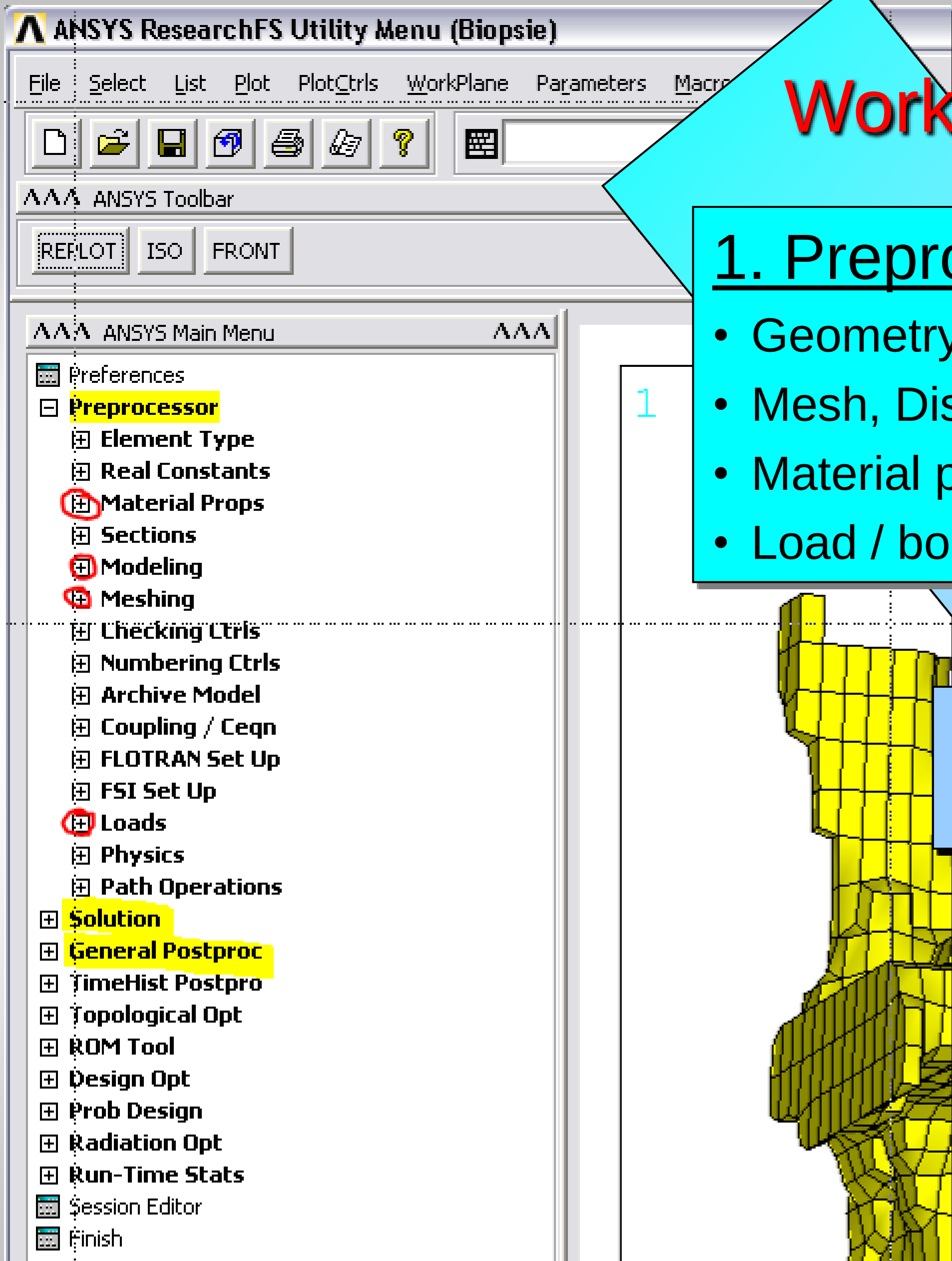
- An explicit FE solver is needed
- to solve initial value instead of boundary value problem
- Application: crash, fast impact, ...





Steps of a FEA

Working Steps of a FEA



1. Preprocessor

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

2. Solution

- Computer is working

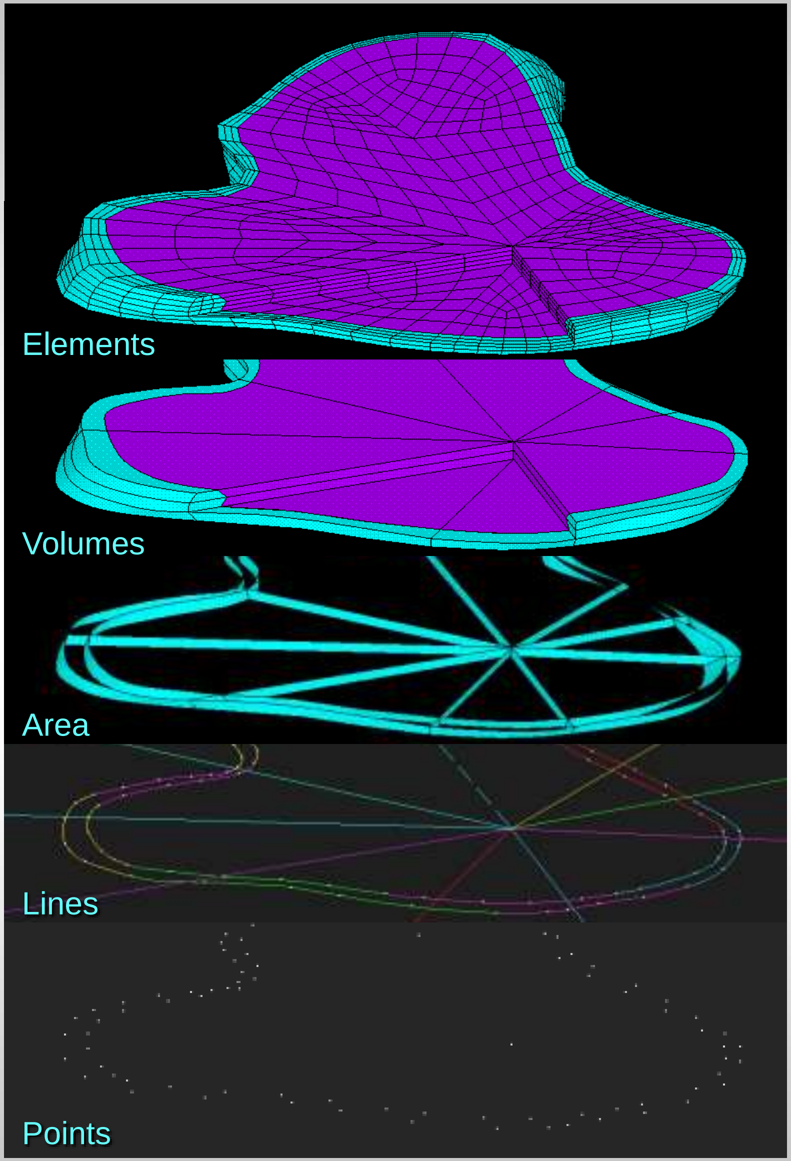
3. Postprocessor

- Verification,Validation
- Presenting results

Step 1: Preprocessor

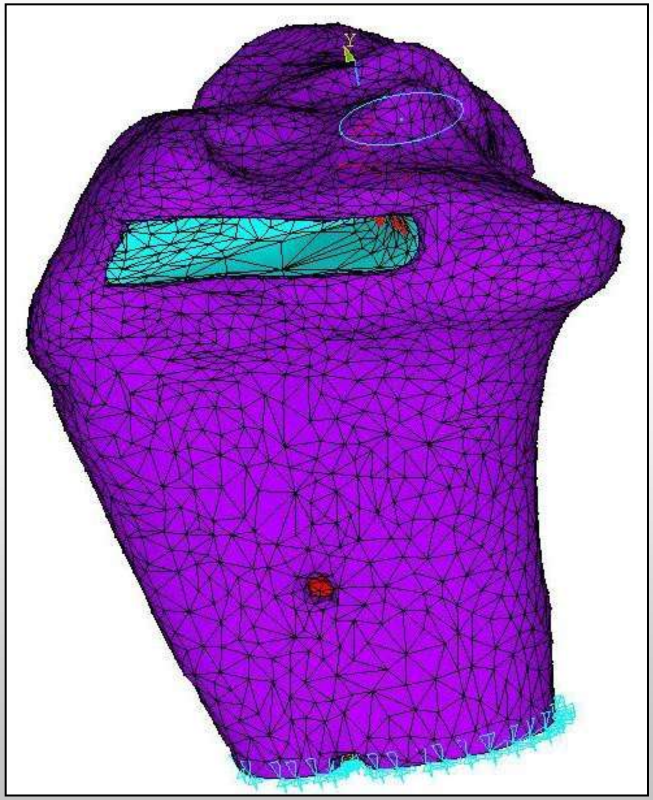
<u>1.1. Generate/Import Geometry</u>

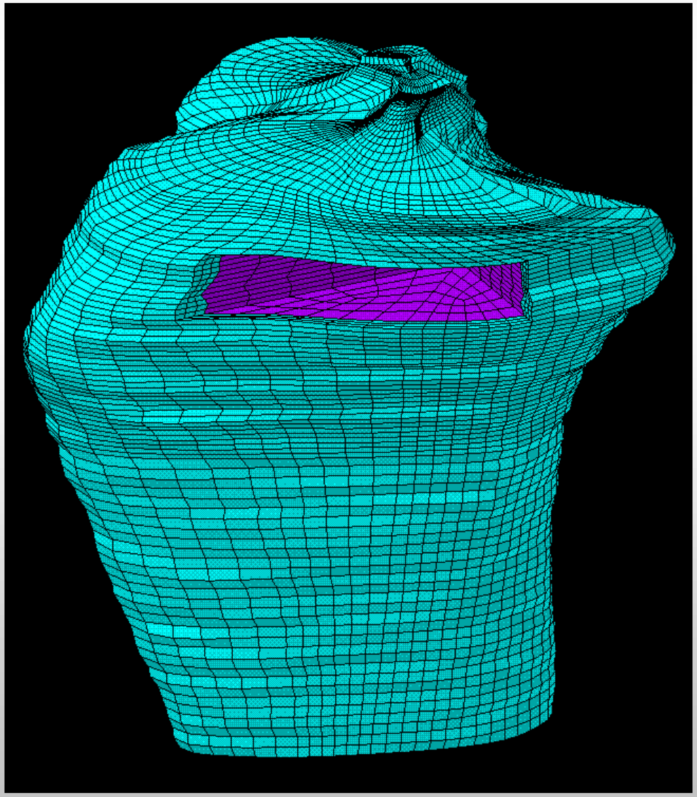
- Botom-up Method
- CAD like Method: using Boolean operations: addition, subtraction of geometric primitives
- Direct generation of Elements: e.g. “Voxel Model”
- Import Geometry from CAD files



1.2 Meshing

- Tetraeheadrons: 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

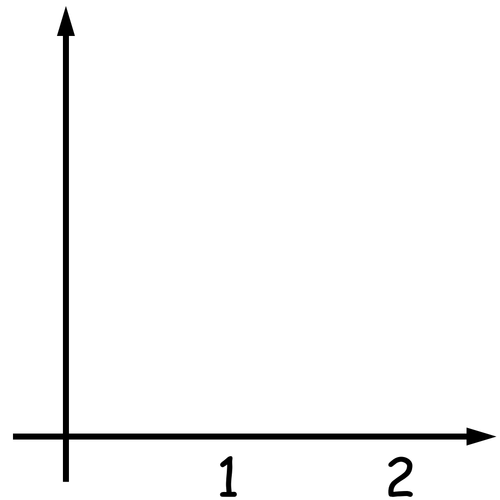
Standard: ISO 31, DIN 1313

Variable $=$ Number $\cdot$ Unit

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

{Variable} $=$ Number

[Variable] $=$ Unit



~~Length L [m]~~

Length L / m

Length L in m

Three mechanical SI-Units:

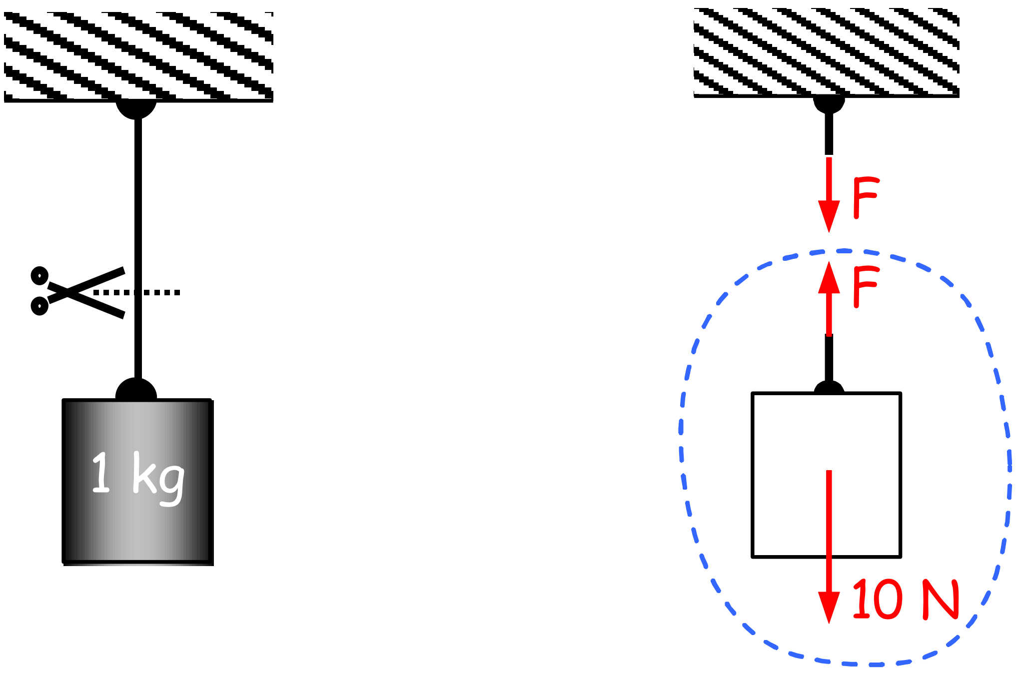
m (Meter)

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

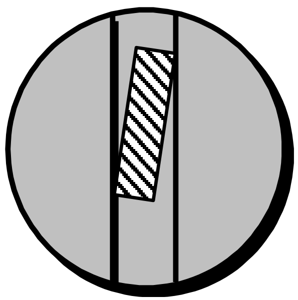
N = kg·m/s^2

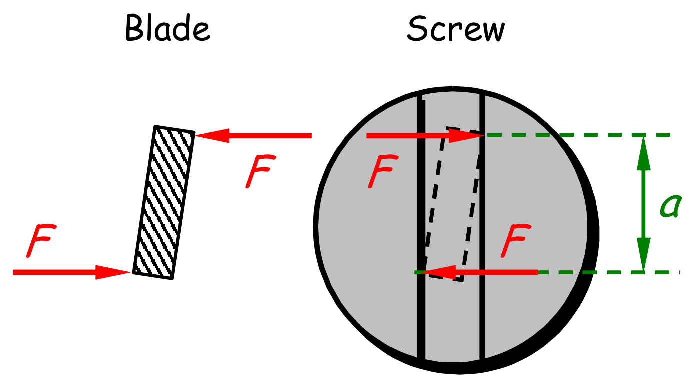
Note to Remember:

1 Newton ≈ Weight of a bar of chocolate (100 g)

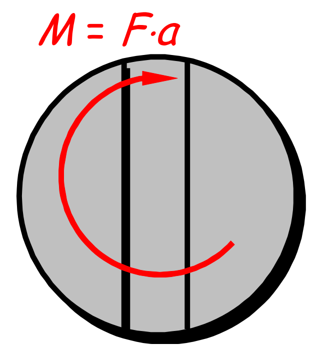
THE MOMENT [Das Moment]

Slotted screw with screwdriver blade





Force Couples (F, a)



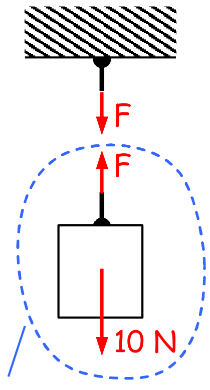
Moment M

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



Free-body diagram (FBD)

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:	$F_{1,x} + F_{2,x} + ... \overset{!}{=} 0$
The sum of all forces in *y*-direction equals zero:	$F_{1,y} + F_{2,y} + ... \overset{!}{=} 0$
The sum of Moments with respect to P equals zero:	$M_{1,z}^{P} + M_{2,z}^{P} + ... \overset{!}{=} 0$

(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

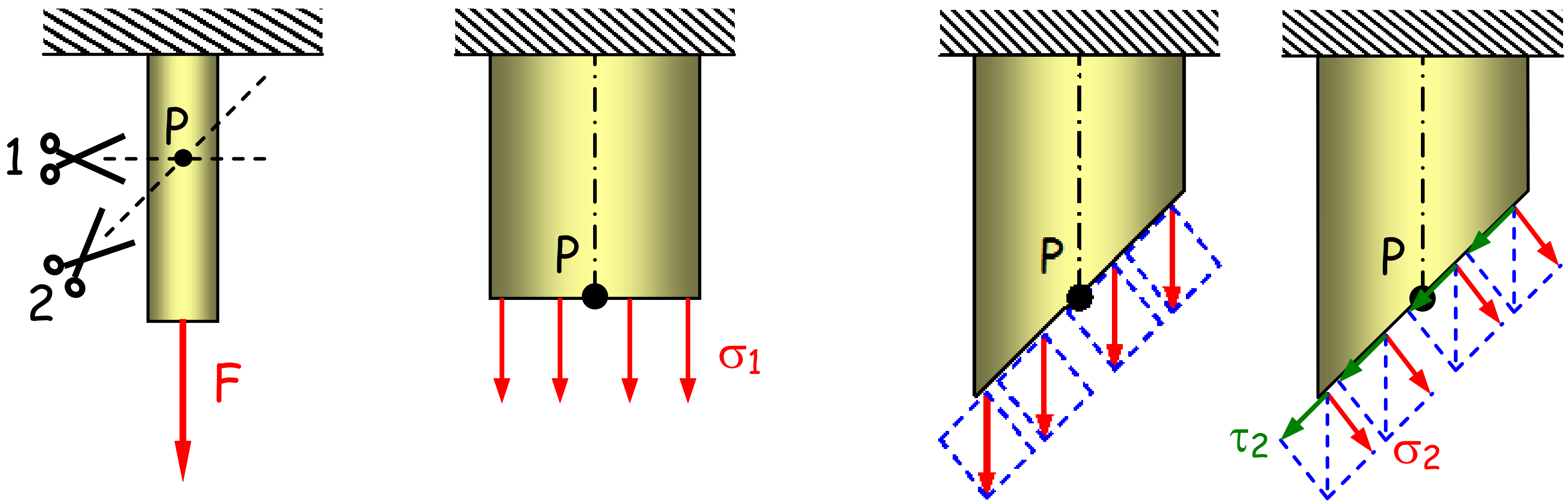
… to account for the loading of the material !

Note to Remember:

Stress = „smeared" force

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

Normal and Shear Stresses



Tensile bar

Cut 1:
→ Normal stresses σ_1

Cut 2:
→ Normal σ_2 and shear stresses τ_2

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:

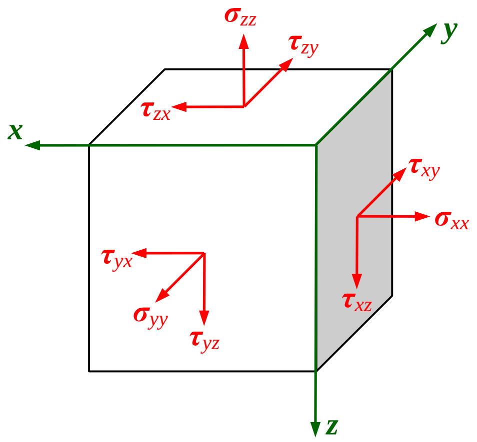
- 3 stress components in one cut (normal stress, 2x shear stress)

    times
- 3 cuts

    result in
- 9 stress components, but only
- 6 of these components will be independent (eq. of shear stresses)

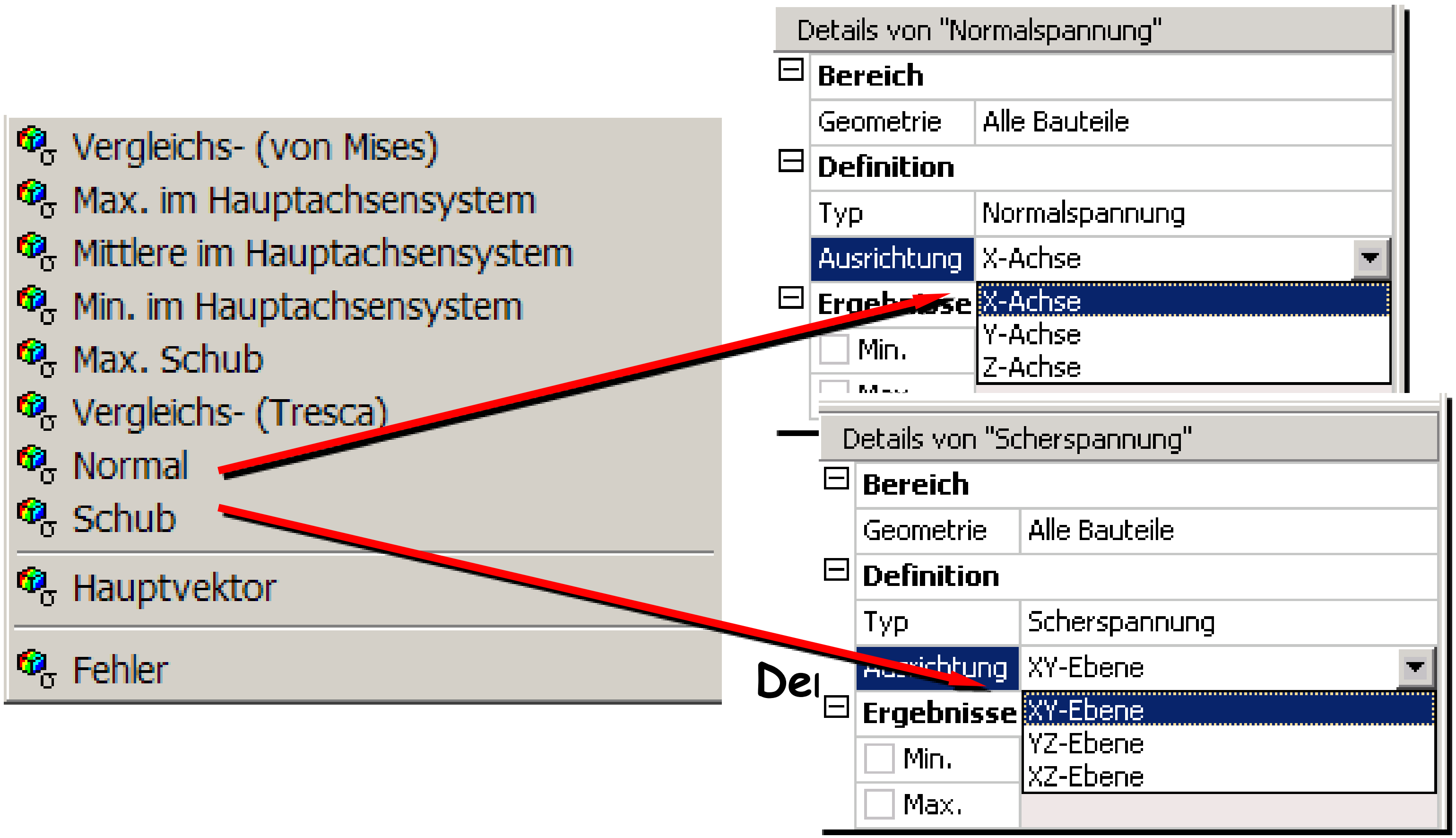
The „stress tensor“

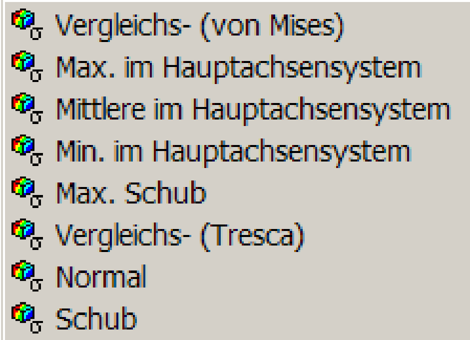
$$\underline{\underline{\sigma}} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix} = \begin{bmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_{yy} & \tau_{yz} \\ \tau_{xz} & \tau_{yz}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

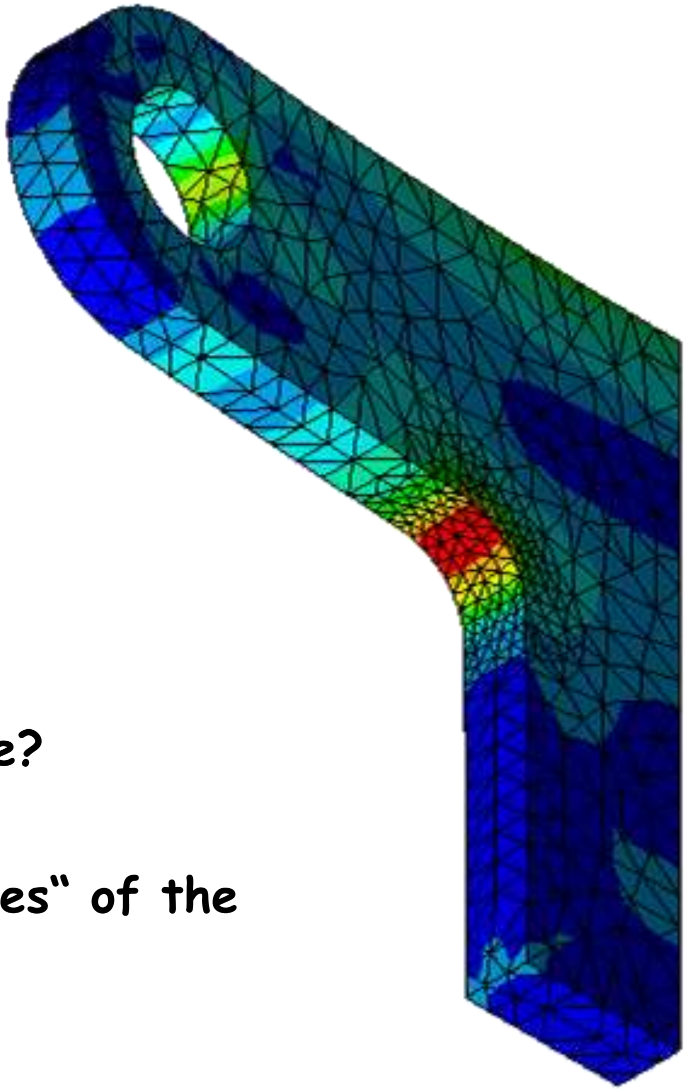


General 3D Stress State

6 Components







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

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

- Local, (internal) strains

Units of Strain

without a unit

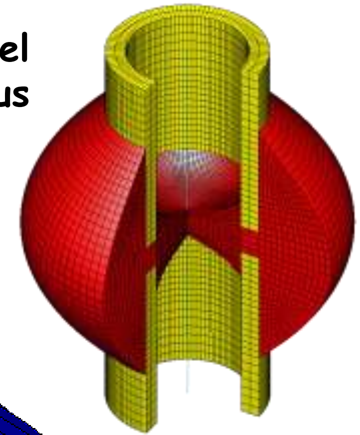
1

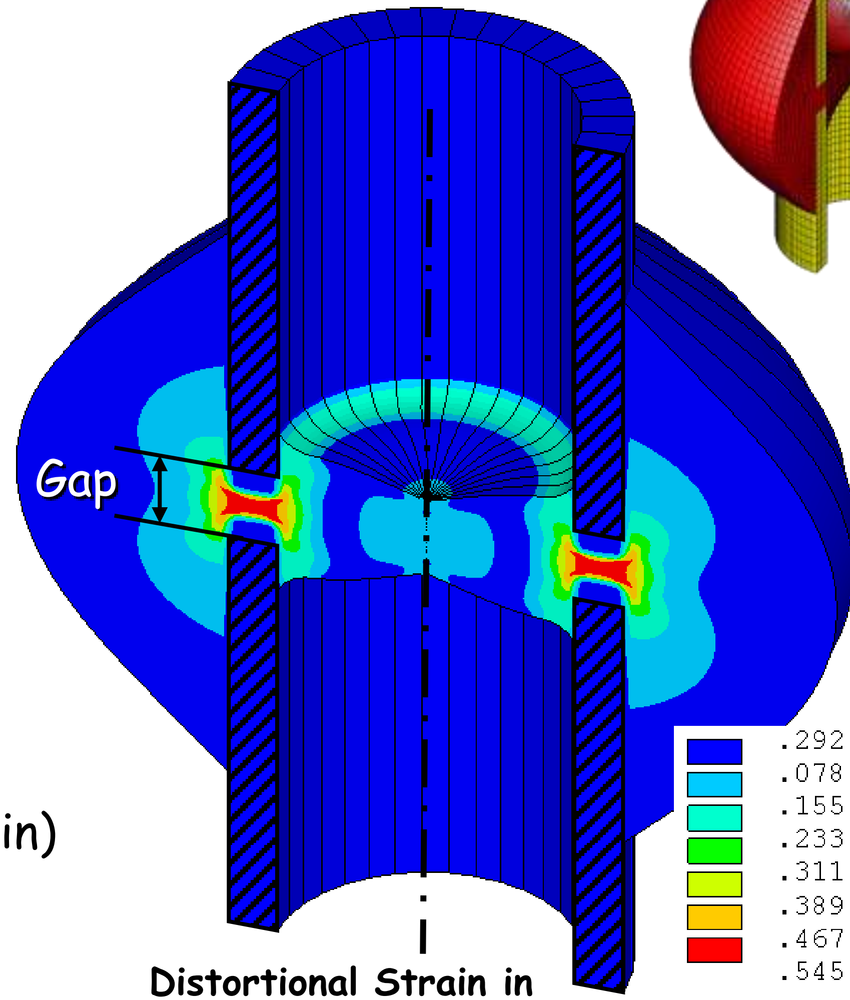
1/100 = %

1/1.000.000 = με (micro strain)

= 0,0001 %

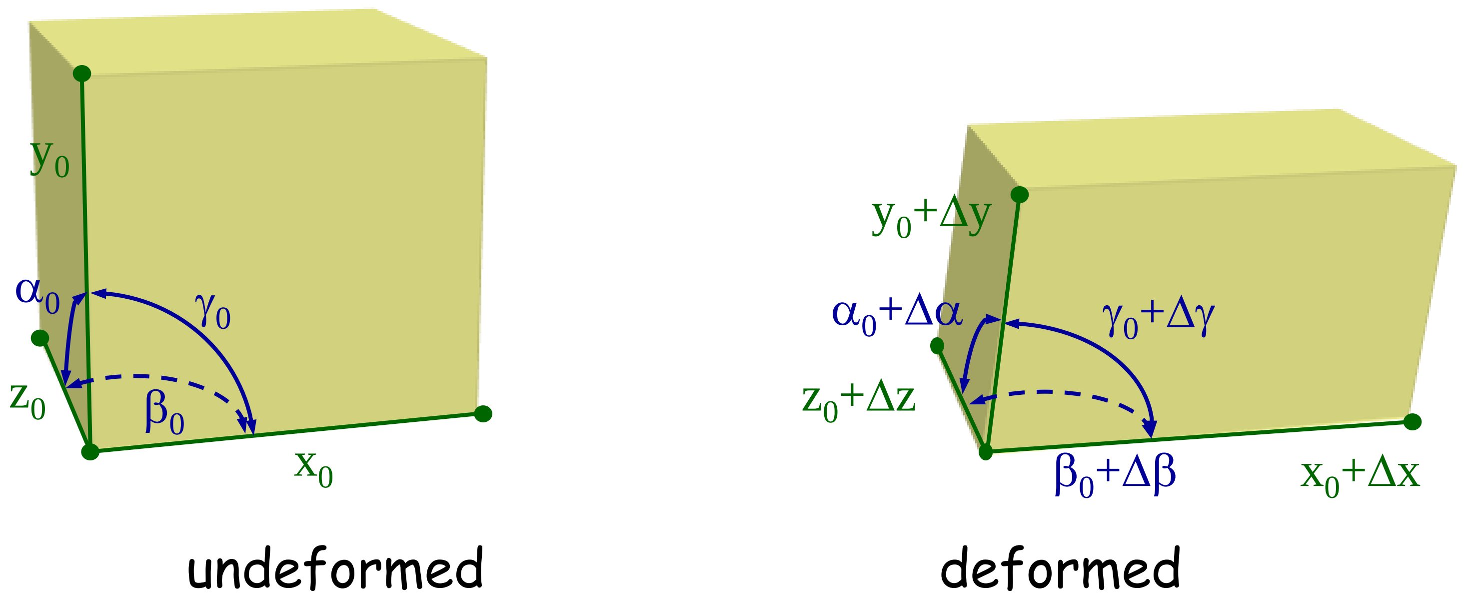
Finite element model of the fracture callus





Distortional Strain in a fracture callus

Definition of the Local Strain State



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

$$\gamma_{xy} = \frac{1}{2} \cdot \Delta\varphi, \quad \gamma_{xz} = \ldots$$

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

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:

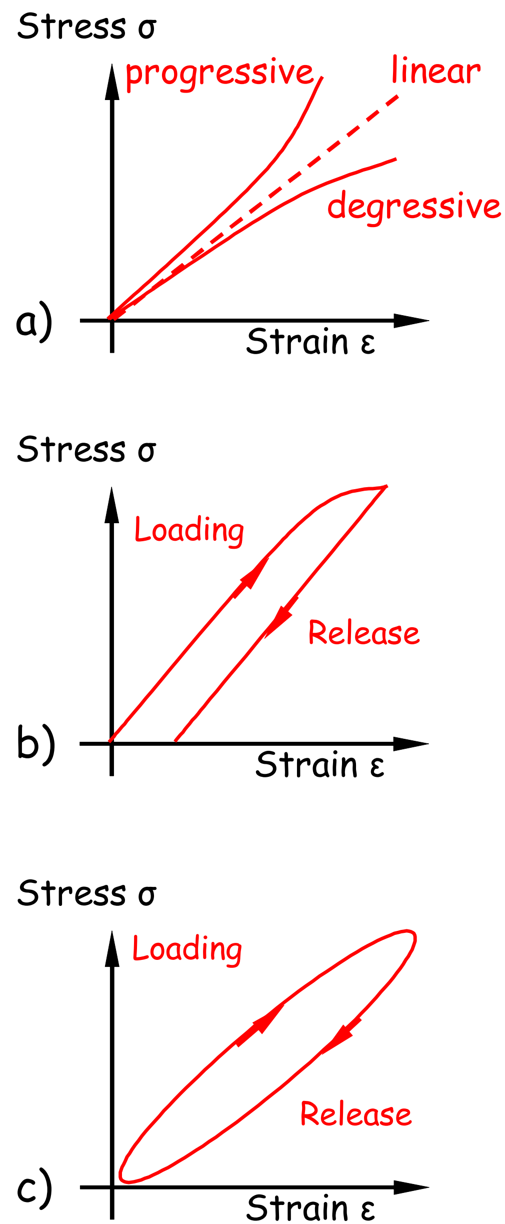
Young's Modulus E (Elastic Modulus) [Elastizitäts-Modul]

Shear Modulus G [Schubmodul]

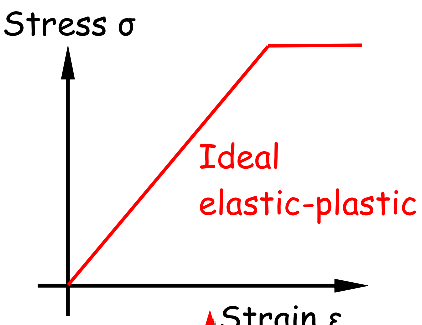
Poisson's ratio ν [Querkontraktionszahl]

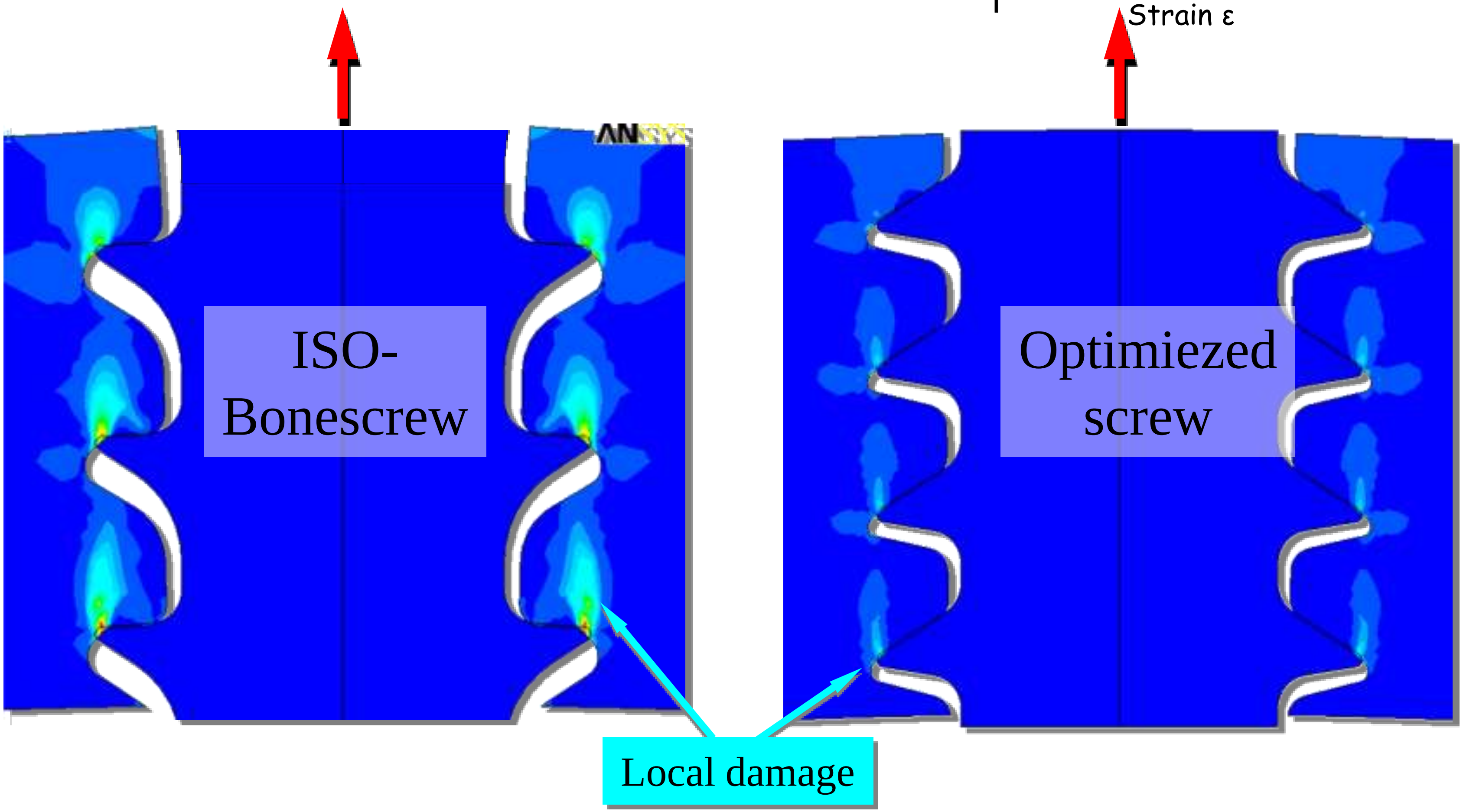
Complex Material Laws:

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



Example: Plastic Strain





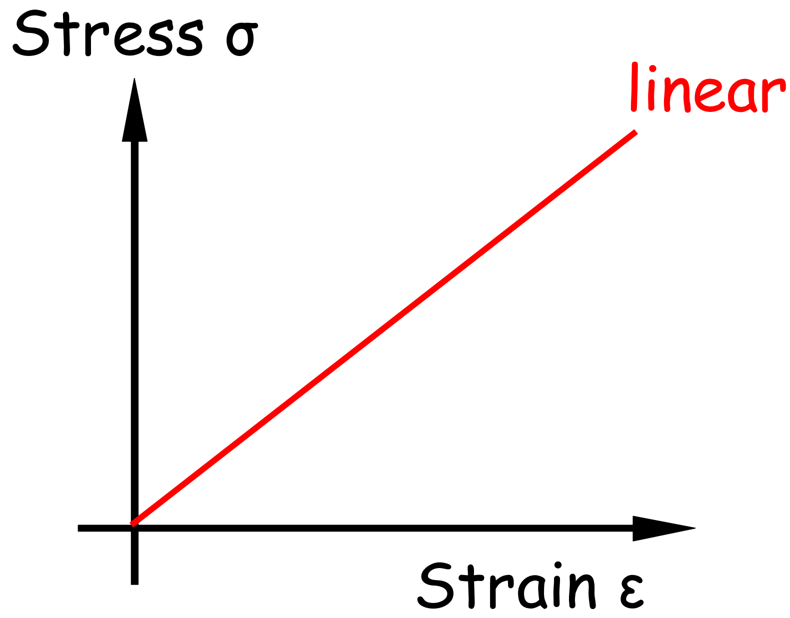
Isotropic vs. Anisotropic (linear elastic)

Linear stress-strain relation

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

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

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



- Full 3^4 material properties tensor of 4^{th} 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)

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

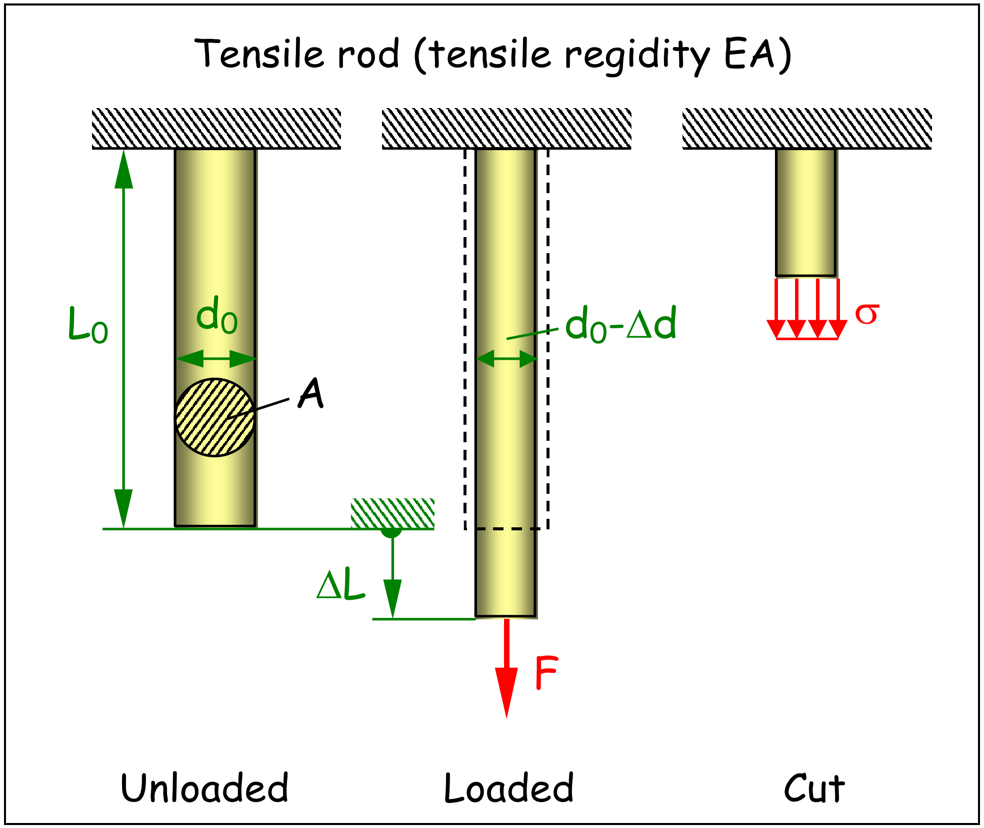
$$\begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \frac{E}{(1+v)\cdot(1-2v)} \cdot \begin{bmatrix} (1-v) & v & v & 0 & 0 & 0 \\ & (1-v) & v & 0 & 0 & 0 \\ & & (1-v) & 0 & 0 & 0 \\ & & & \frac{(1-2v)}{2} & 0 & 0 \\ & & & & \frac{(1-2v)}{2}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

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

← 2 of these

Simple Load Cases for 1D objects

1 Tension and Compression



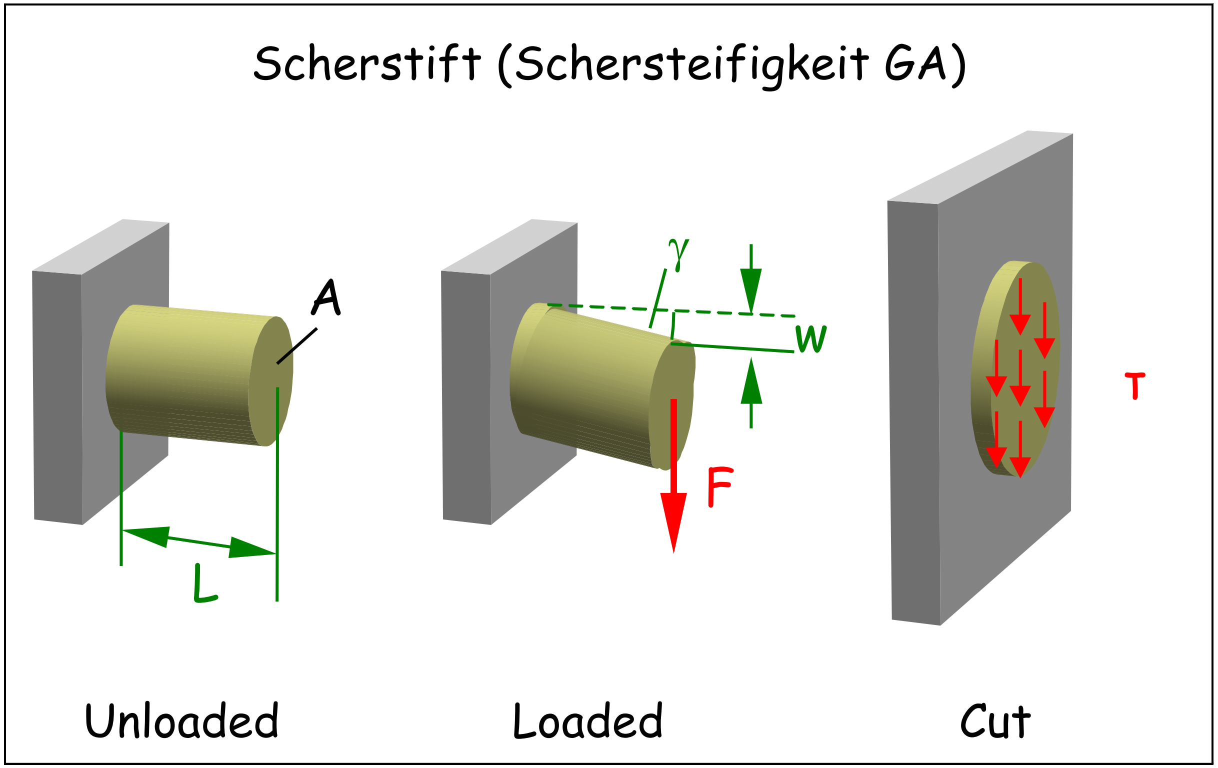
Global behavior, stiffness

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

Stresses in transverse cut

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

Shear



Global behavior, stiffness

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

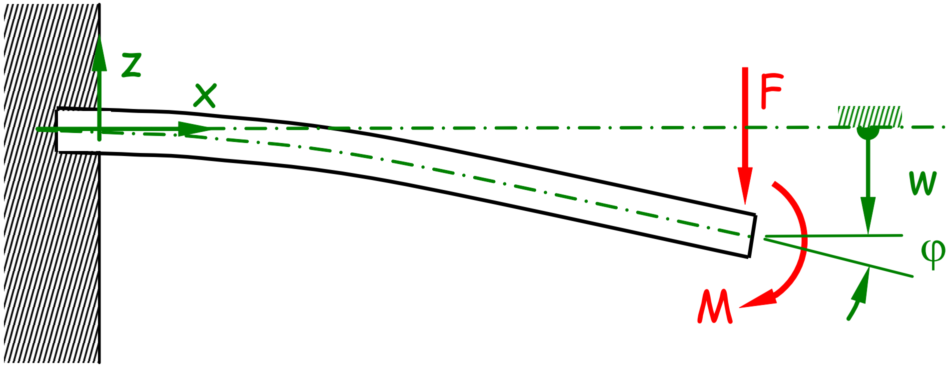
Stresses in transverse cut

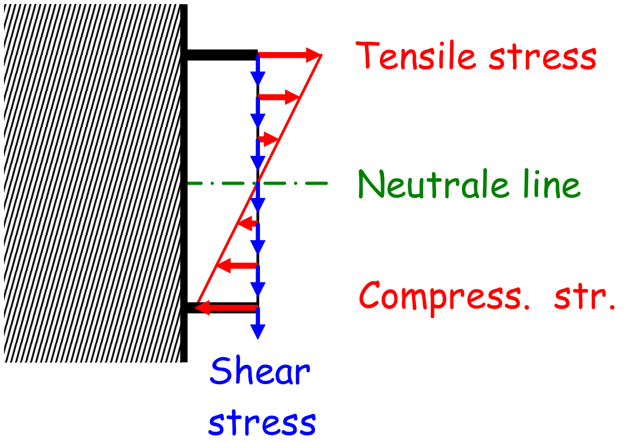
$$\tau = \frac{F}{A}$$

Bending (Cantilever beam)

Cantilever beam (Bending regidity EI_a, Length L)

Cut





Global behavior, compliance

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

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

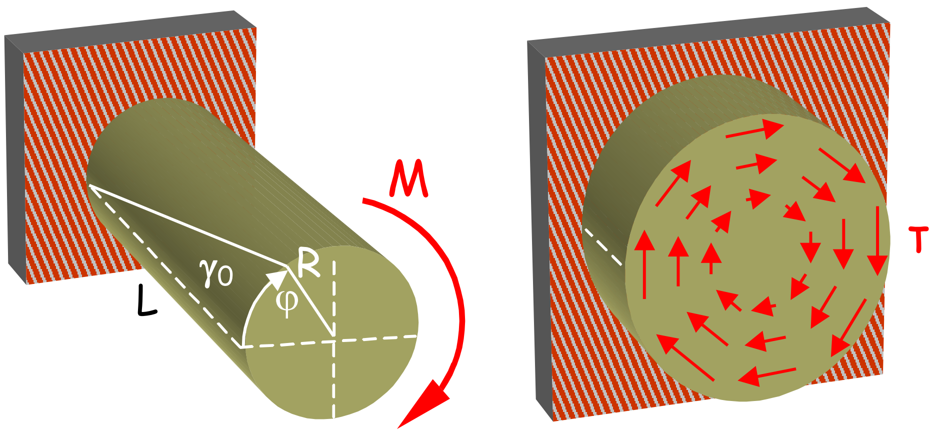
Local stress in transverse cut:

(normal stress)

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

Torsion

Torsional rod (torsional regidity GI_T)



Length L, radius r

Cut

Global behavior, stiffness

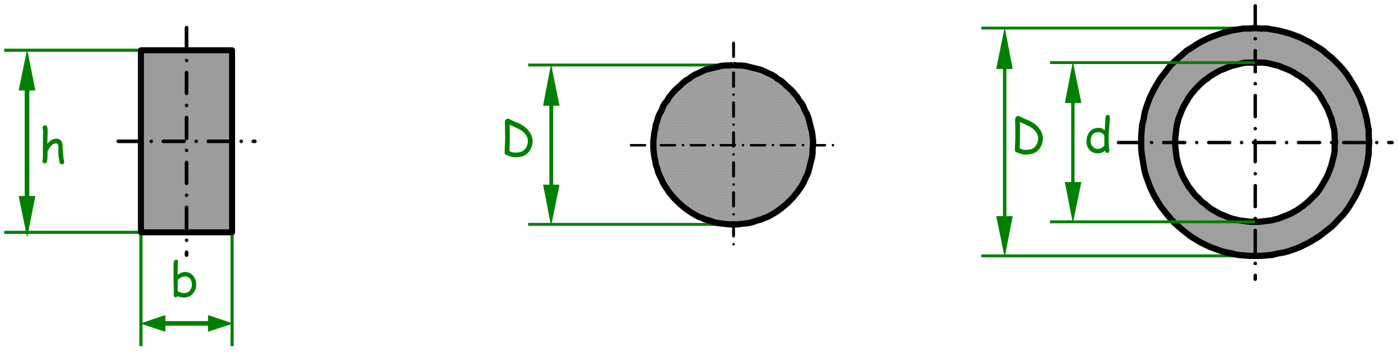
$$M = \frac{GI_T}{L}\varphi, \quad c = \frac{GI_T}{L}$$

Stresses in transverse cut

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

$\rho =$ Distance from Center

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



Axial moment of area (bending)

$$I_a = \frac{b \cdot h^3}{12} \qquad I_a = \frac{\pi}{64} D^4 \qquad I_a = \frac{\pi}{64}(D^4 - d^4)$$

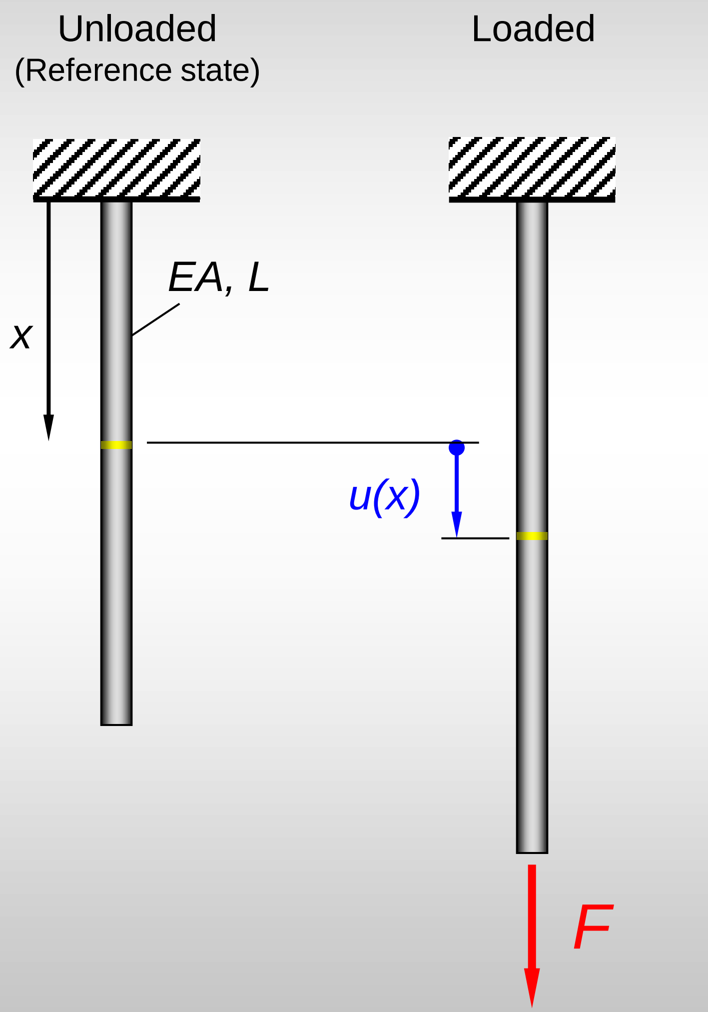
Polar moment of area (torsion)

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

Theory of the
Finite Element Method
using a ‘super simple’ example

Hier geht
es weiter!

Example: Tensile Rod



Given:

Rod with …

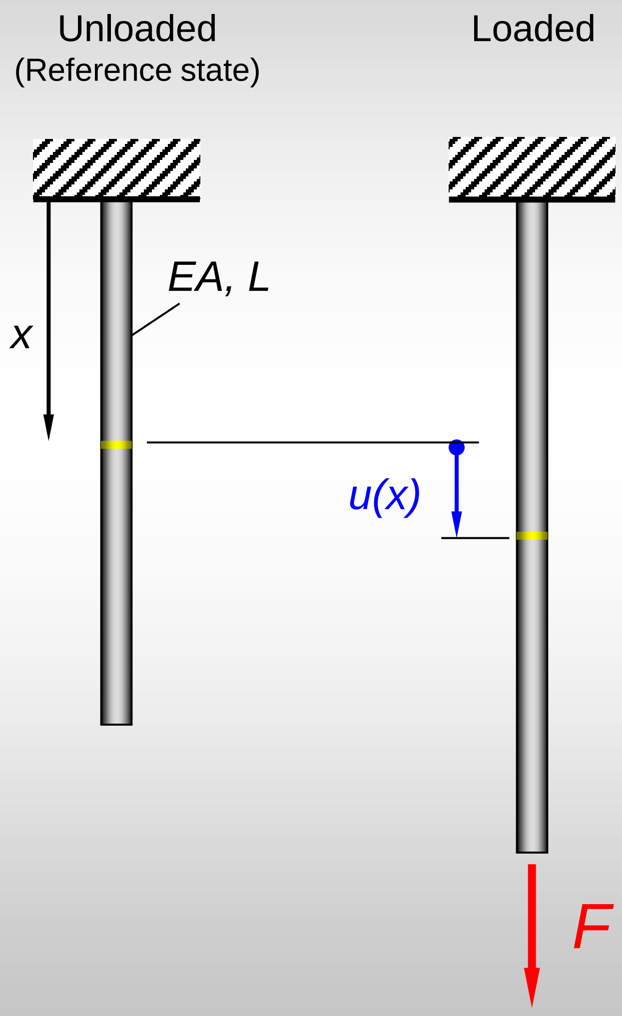
- Length L
- Cross-section A (constant)
- E-modulus E (constant)
- Force F (axial)
- Upper end fixed

To determine:

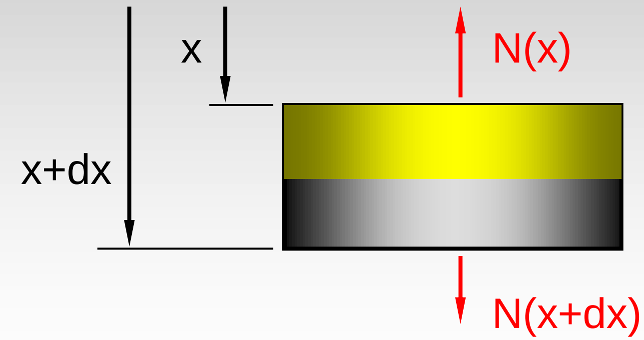
Deformation of the loaded rod:

Displacement function $u(x)$

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



Differential Element
(infinitesimale Higth dx)



Generate the Differential Equation

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

$\Rightarrow$ DGl: $(EA\,u')' = 0$

If EA = const then

$u'' = 0$

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

Solve the Differential Equation

$u''(x) = 0$

Integrate 2 times: $u'(x) = C_1$

$u(x) = C_1 * x + C_2$ (General Solution)

Adjust to Boundary Conditions

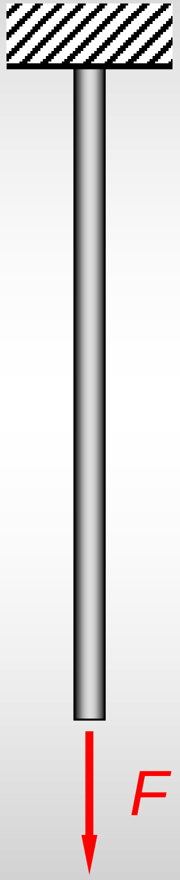
Top (Fixation): $u(0) = 0 \Rightarrow C_2 = 0$

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

$\Rightarrow C_1 = F/EA$

Adjusted Solution

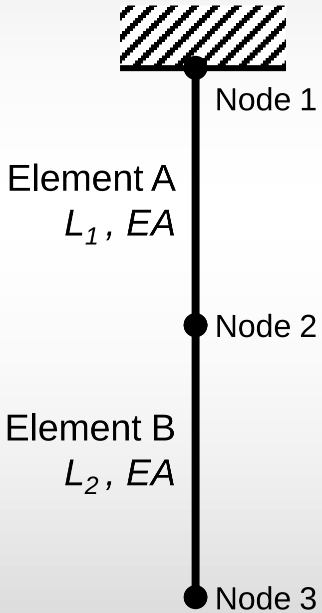
$$\mathbf{u(x) = (F/EA)*x}$$



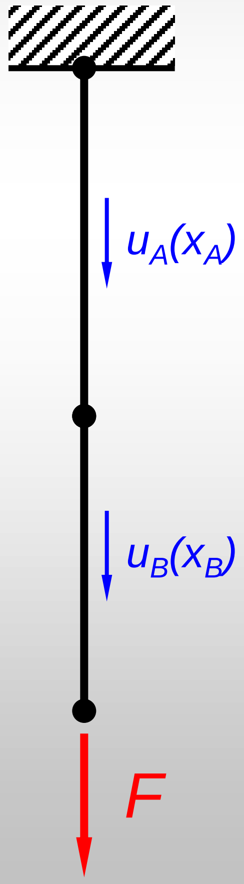
B) Solution with FEM

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

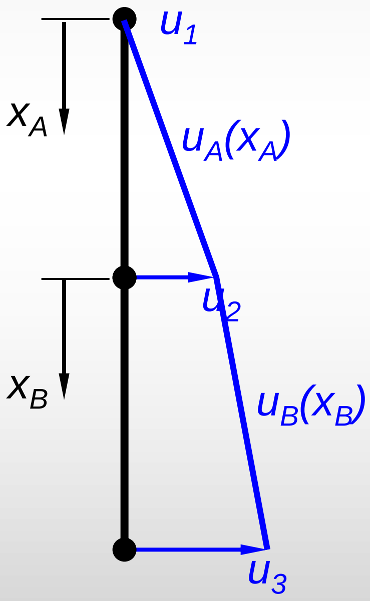
Unloaded:
(Reference condition)



Loaded:



Ansatz functions (linear) for the unknown displacements *u*



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.

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

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

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.

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

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:

$$\delta W = N_A(\delta\hat{u}_2 - \delta\hat{u}_1) + N_B(\delta\hat{u}_3 - \delta\hat{u}_2) - F\delta\hat{u}_3$$
$$= \frac{EA}{L_A}(\hat{u}_2 - \hat{u}_1)(\delta\hat{u}_2 - \delta\hat{u}_1) + \frac{EA}{L_B}(\hat{u}_3 - \hat{u}_2)(\delta\hat{u}_3 - \delta\hat{u}_2) - F\delta\hat{u}_3$$

$$\delta W = \delta\hat{u}_1\left(+\frac{EA}{L_A}\hat{u}_1 - \frac{EA}{L_A}\hat{u}_2\right)$$
$$+\delta\hat{u}_2\left(-\frac{EA}{L_A}\hat{u}_1 + \frac{EA}{L_A}\hat{u}_2 + \frac{EA}{L_B}\hat{u}_2 - \frac{EA}{L_B}\hat{u}_3\right)$$
$$+\delta\hat{u}_3\left(-\frac{EA}{L_B}\hat{u}_2 + \frac{EA}{L_B}\hat{u}_3 - F\right) = 0$$

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:

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

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 chose 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!

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

… 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
$\underline{F}$ - 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 …

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

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

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.

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

Strains — Stresses

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

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 solution run the program first creates the(big) linear system of equations based on that known template structure and than solves the system in terms of nodal displacements.

General Hints and Warnings

- FEA is a tool, not an solution
- Take care about nice pictures („GiGo“)
- Parameter
- Verification

} needs experiments

- FE models are case (question) specific

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/