Vorlesung Wintersemester 2013/2014:

Die Finite-Elemente-Methode in der Biomechanik

Dr. Fabio Galbusera

Institut für Unfallchirurgische Forschung und Biomechanik
im Zentrum für Muskuloskeletale Forschung (zmfu)
Universitätsklinikum Ulm

Gliederung

- Anwendungsgebiete aus der Biomechanik
- Ablauf einer FE-Analyse
- Solution (Was macht der Solver?)
- Modellverifikation und -validierung

Warum brauchen wir (mathematische) Modelle?

In-vivo-Untersuchungen

• realistischste Bedingung, da physiologische Verhältnisse vorliegen

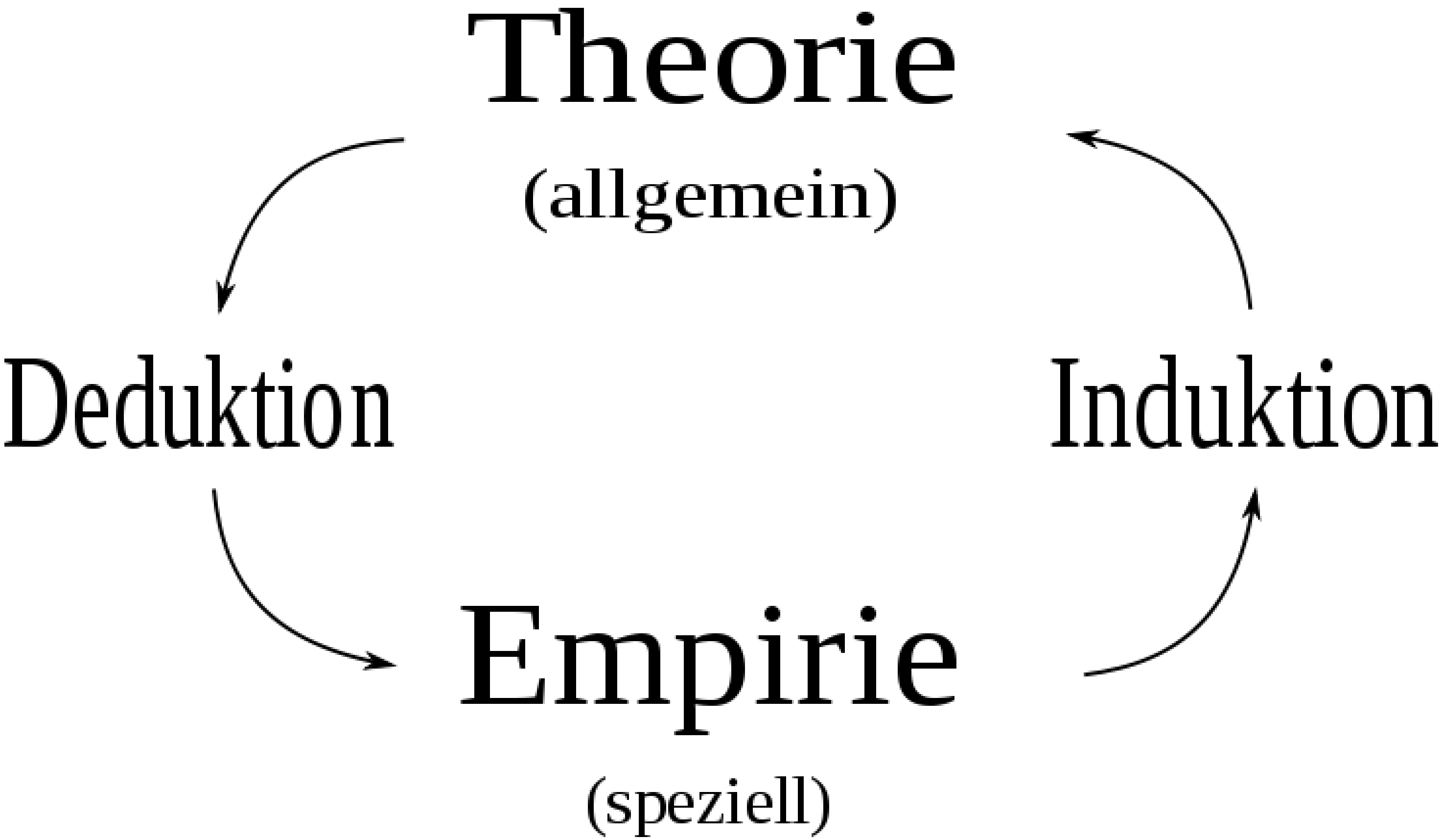
In-vitro-Untersuchungen

• Messaufnehmer direkt am Objekt
→ Quantifizierung einzelner Strukturen isoliert oder im Verbund

- Grenzen der Messtechnik
- Starke interindividuelle Variabilität
- Insbesondere bei In-vivo-Untersuchungen: ethischen Vertretbarkeit?
- Tierschutzrechtliche Grenzen bei In-vivo-Untersuchungen

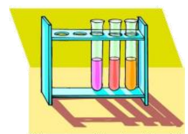
(Theoretische) Modelle:
- Erklärungen („Wieso passiert das?“)
- Vorhersagen („Was wäre wenn?“)

Warum brauchen wir (mathematische) Modelle?

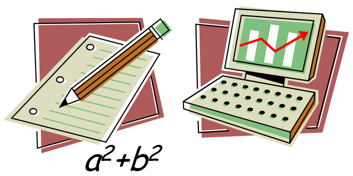


Unterteilung mathematischer Modelle

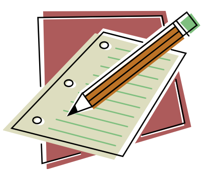
„Experimentell“
In vivo, In vitro



„Theoretisch“
Mathematisch

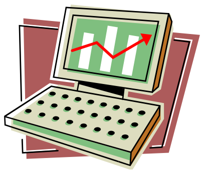


„Analytisch“
⇒ Rechnen mit Variablen



$F = ma$

„Numerisch“
⇒ Rechnen mit Zahlen



1.37491

- (Mathematisch) exakte Lösung
- Nur für wenige einfache Probleme möglich

- Oft Näherungsverfahren (Newton-Methode, Taylor-Formel)
- Näherungslösung für viele komplexe Probleme bekannt

Grundbegriffe

Wirklichkeit

↓

Abstraktion

↓

Analytisches Modell

z.B. gekoppelte, nichtlineare, partielle Differentialgleichungen (in Raum und Zeit), interagieren gemäß geg. Rand- und Anfangsbedingungen

↓

Numerisches Modell

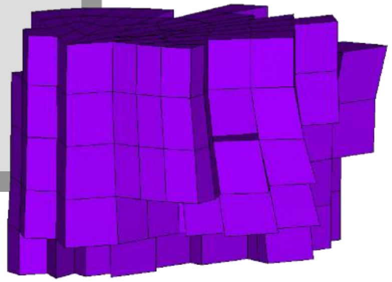
diskretisierte Version des mathematischen Modells

Grundbegriffe

Finite-Elemente-Methode

=

„Numerisches Verfahren
zur näherungsweisen Lösung von
partiellen Differentialgleichungen“



Anwendungsgebiete

Dynamik
- implizit: Modalanalyse
- explizit: transiente Vorgänge (Crash)
- Poro- und Viskoelastische Vorgänge (Kriechen, Relaxation)

Statik, Festigkeit

Linear:
- Spannungen, Dehnungen

Nichtlinear:
- Kontakt, Reibung
- Materialien
- Dehnungen, Verschiebungen
- Plastizität, Verfestigung
- Ermüdung, Bruchmechanik
- Gestaltoptimierung

FEM

Akustik

Strömungen

Wärmefluss, Diffusion

Elektromagnetische Felder

Vorteile

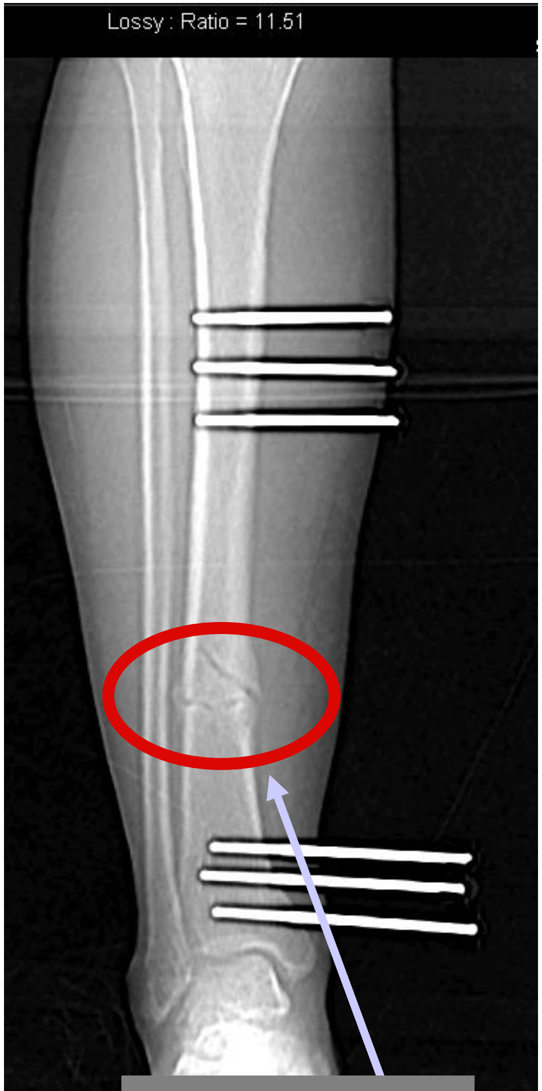
- Einsparung von In-vivo- und In-vitro-Versuchen
- Detaillierte Untersuchung der (mechanischen) Zusammenhänge
- Untersuchung des Einflusses einzelner Parameter
- Einfache Darstellung von Ergebnissen

Nachteile / Probleme

- FE-Analysen führen fast immer zu Ergebnissen (egal ob richtig oder falsch)
- Aufwand für FE-Analysen wird häufig unterschätzt
- Aussagekraft bzw. Gültigkeit von FE-Analysen wird häufig überschätzt

- Anwendungsgebiete aus der Biomechanik
- Aufbau einer FE-Analyse (praktisch / theoretisch)
- Modellverifikation und -validierung

Anwendungsgebiete



Biegekeilfraktur

Hintergrund

Frakturheilung hängt stark von der interfragmentären Bewegung ab

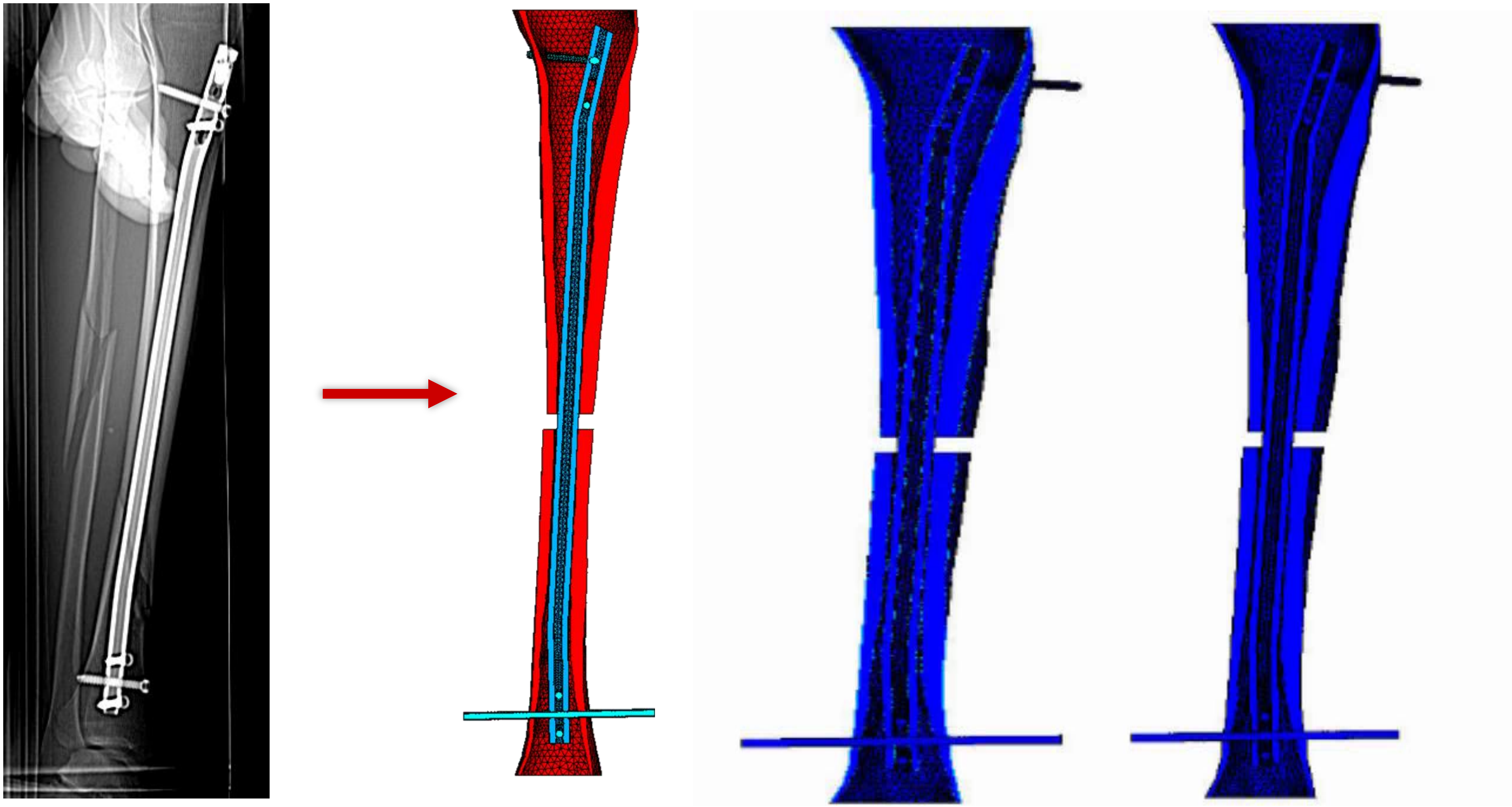
Fragestellung

- Wie groß darf die interfragmentäre Bewegung sein, damit keine Pseudarthrose entsteht?
- Wie stark darf das Implantat belastet sein, damit ein Versagen ausgeschlossen werden kann? (z.B. Pinbruch)

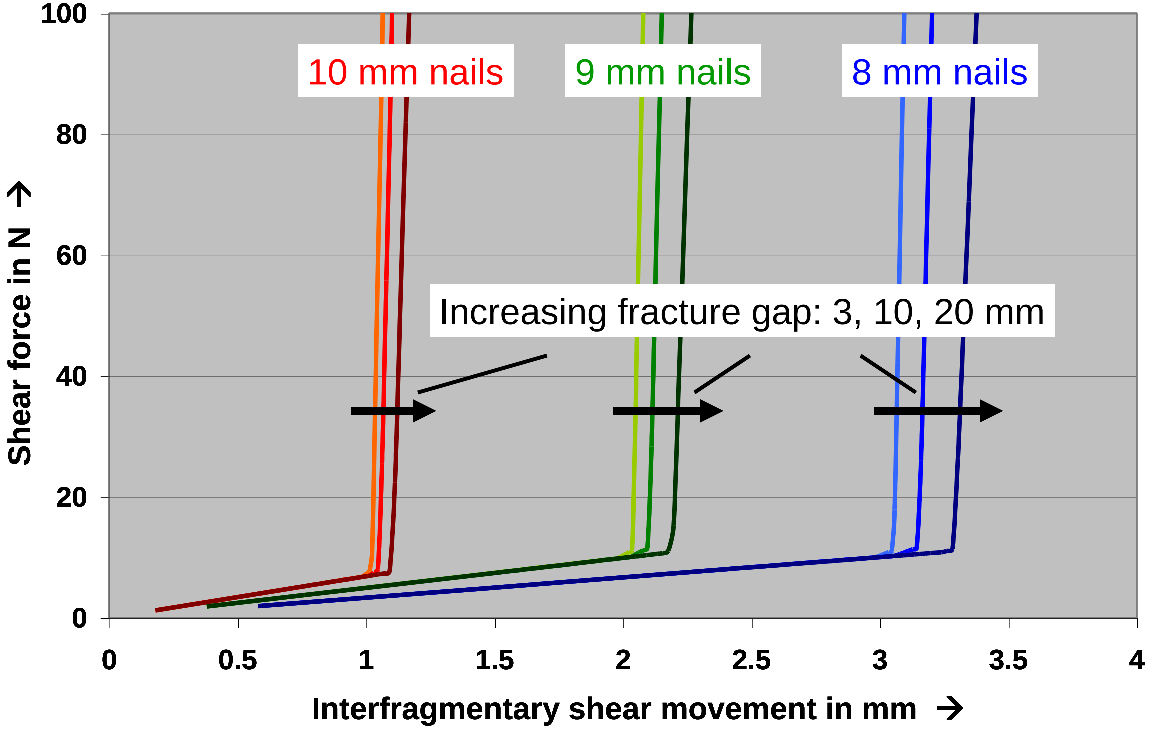
Welche Faktoren bestimmen die Stabilität einer Fixateur-externe-Osteosynthese?

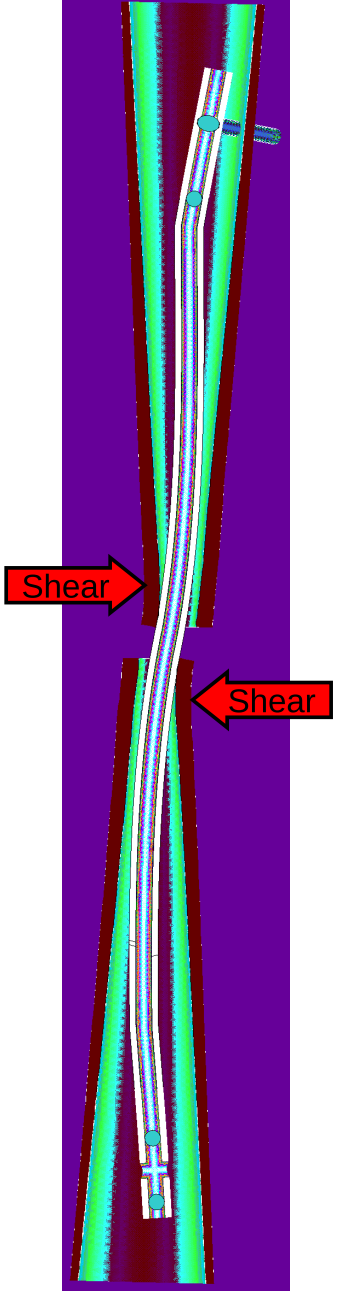
Anwendungsgebiete

Scherung Biegung

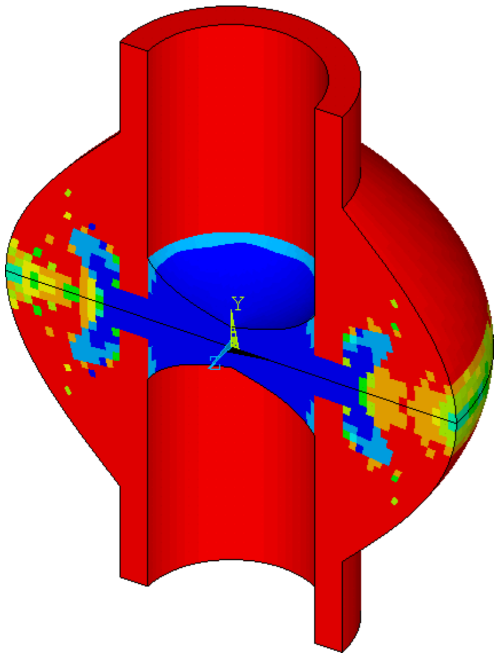


Anwendungsgebiete

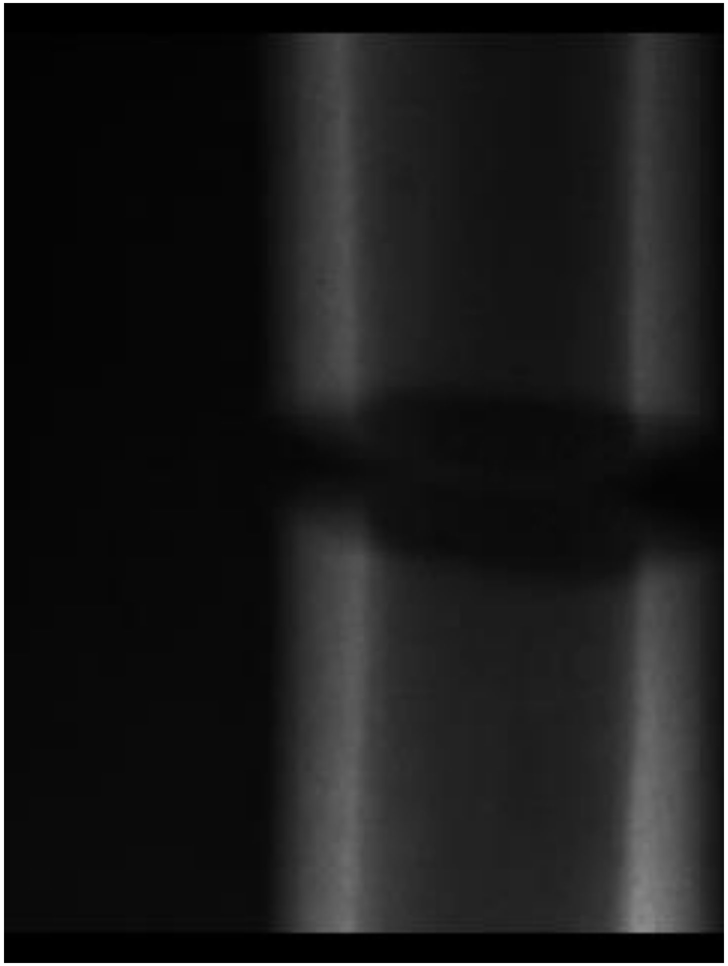




Anwendungsgebiete

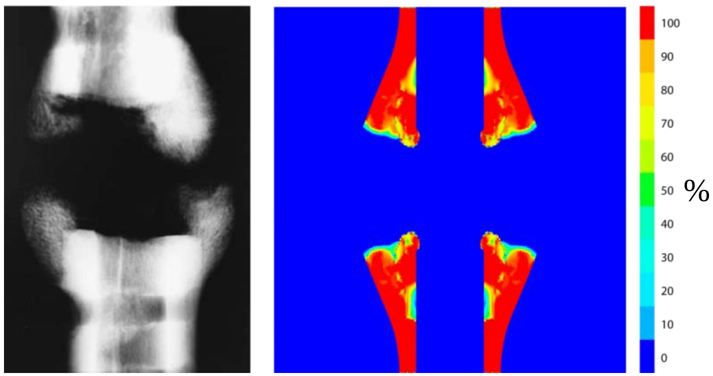


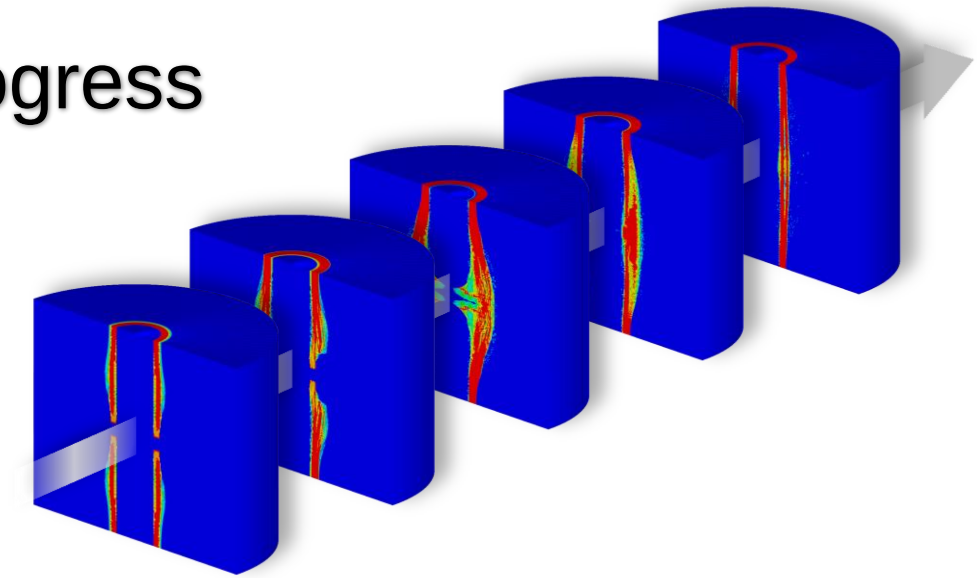
Wirklichkeit

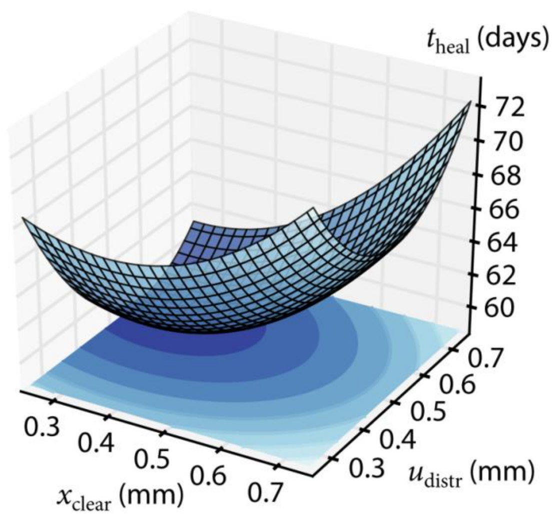


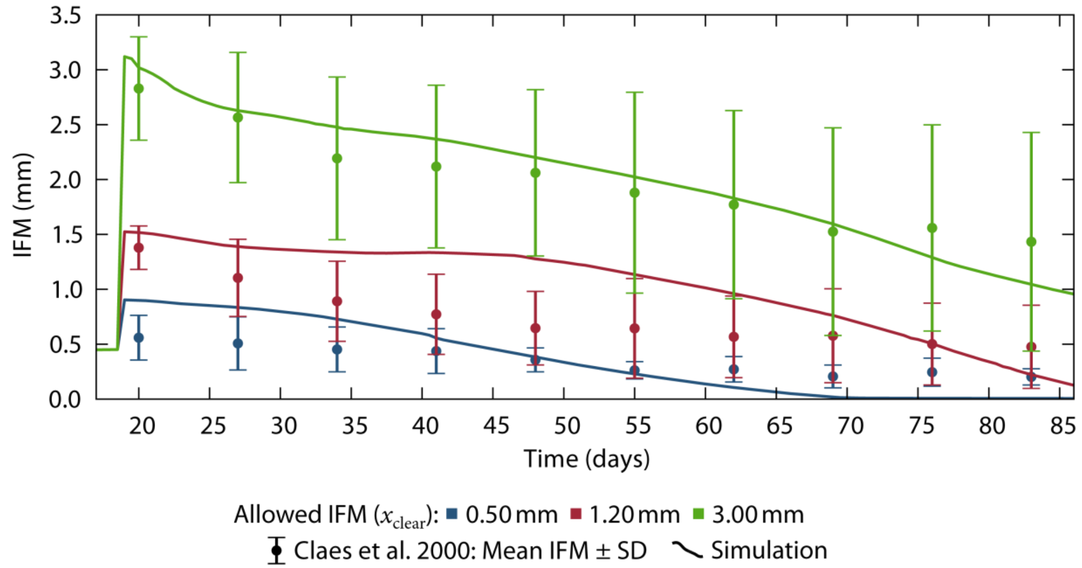
Modell

Predicting Healing Progress





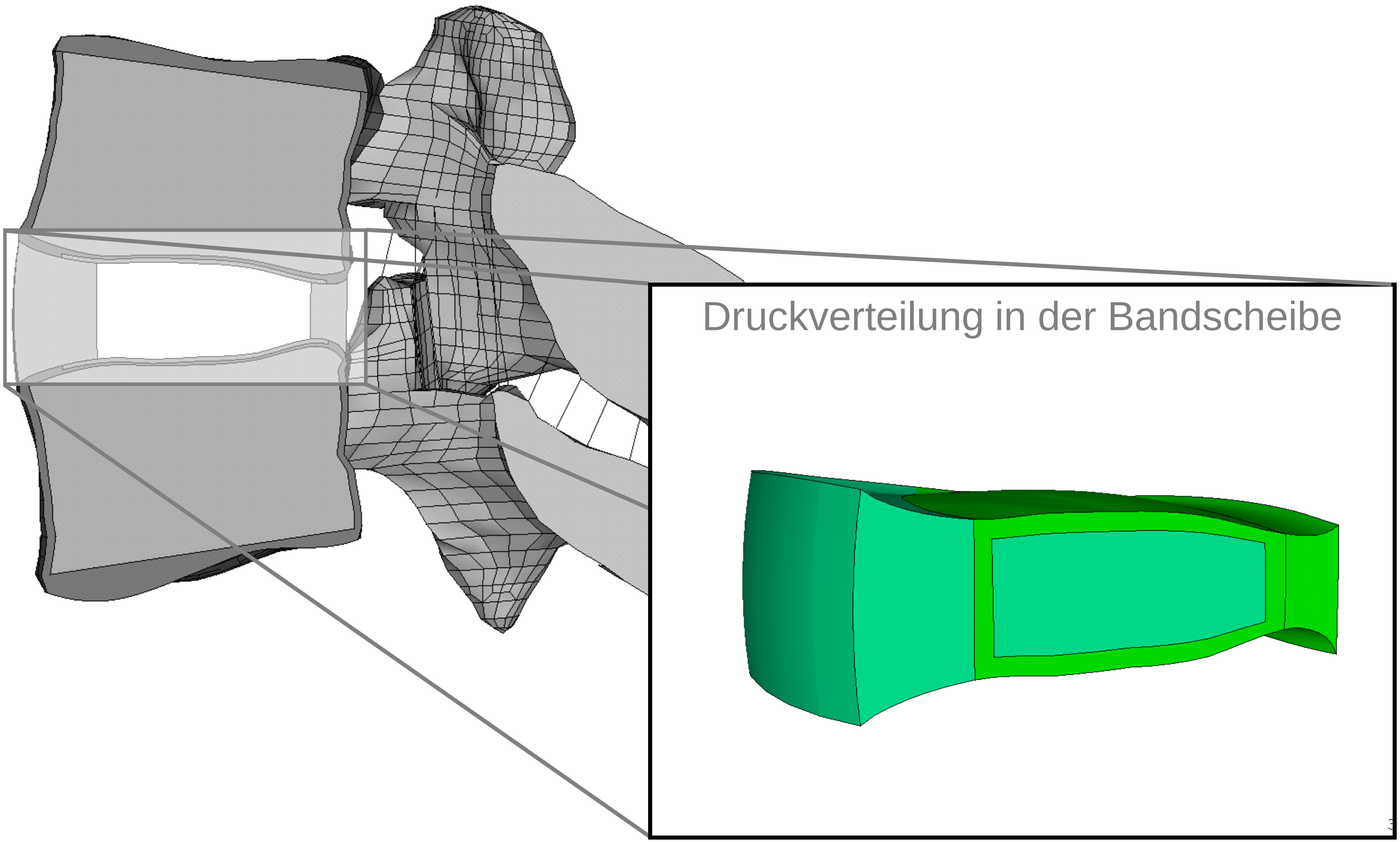




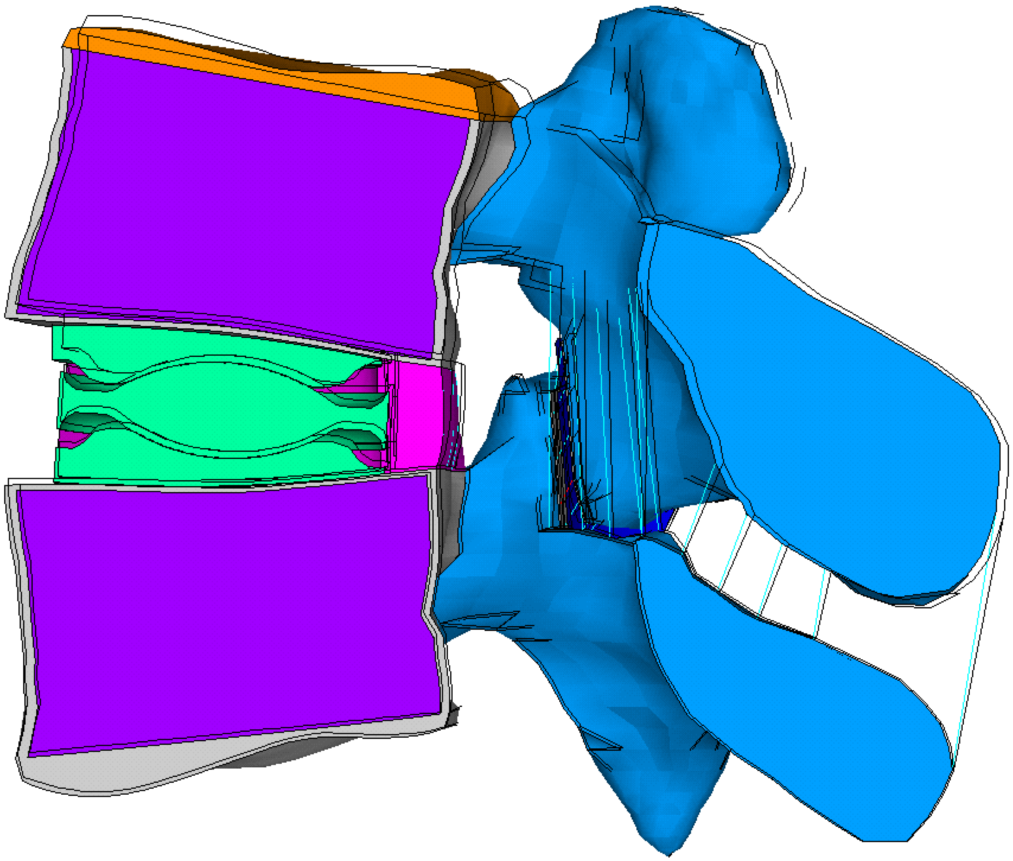
universität ulm zmfu

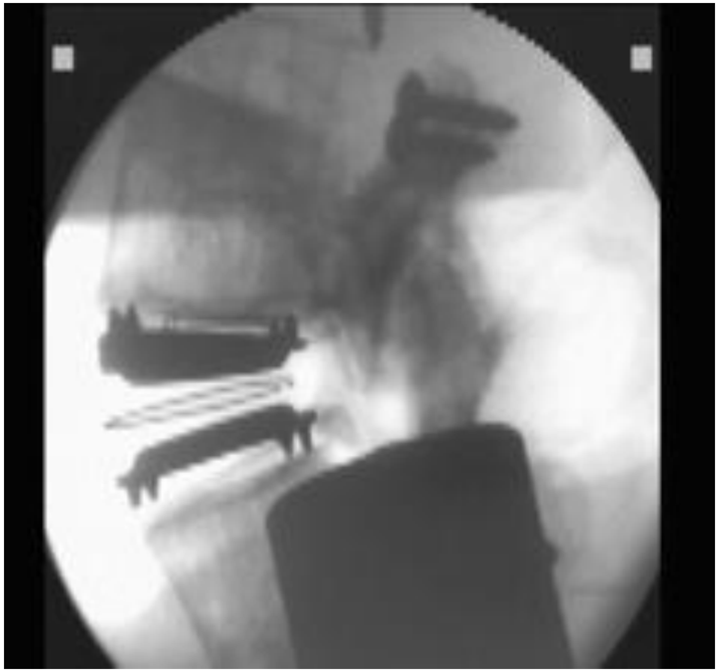
www.biomechanics.de

Anwendungsgebiete

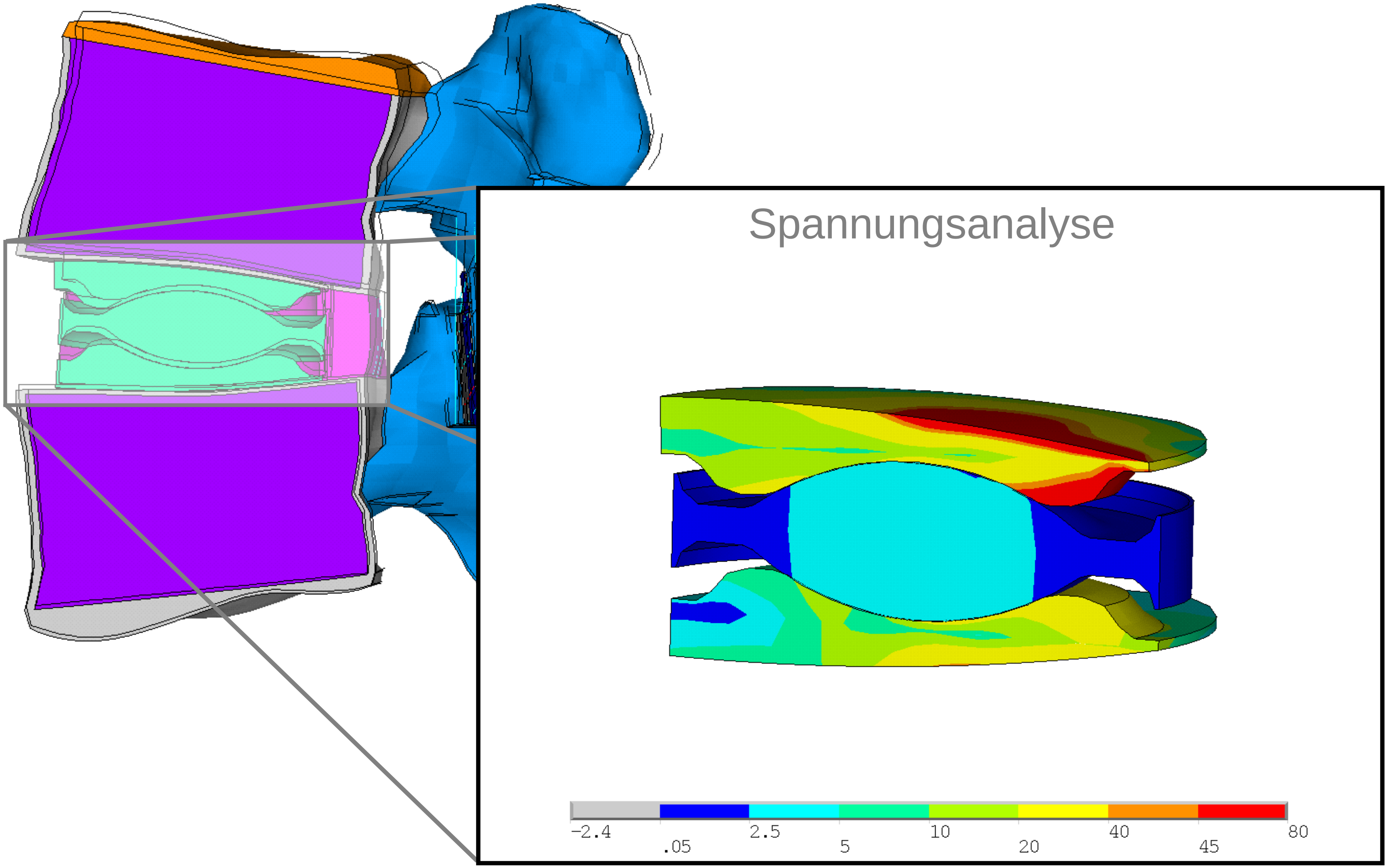


Anwendungsgebiete





Anwendungsgebiete



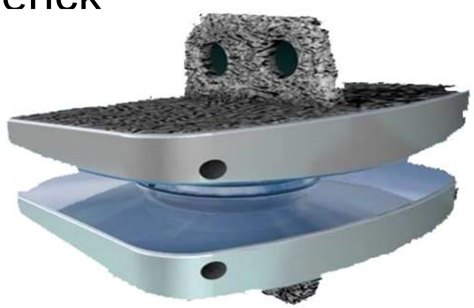
Anwendungsgebiete

constrained

ProDisc®-L

Synthes Spine; West Chester, PA, USA

Maverick™



Medtronic; Minneapolis, MN, USA

semi-constrained

Active®-L

B.Braun Aesculap; Melsungen

unconstrained

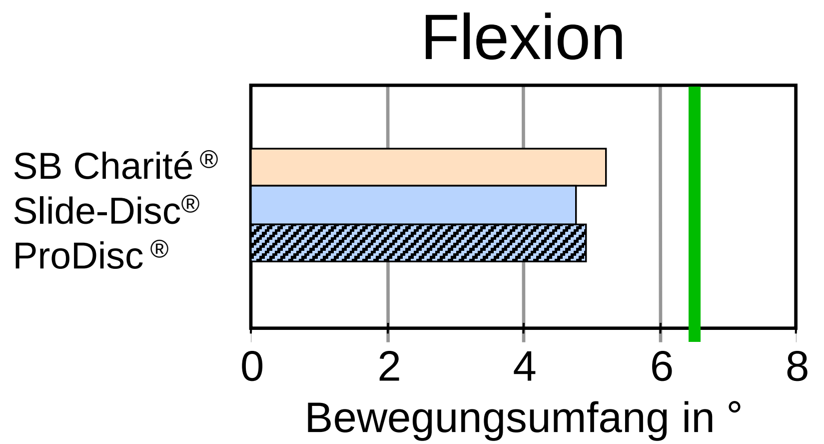
SB Charité® III

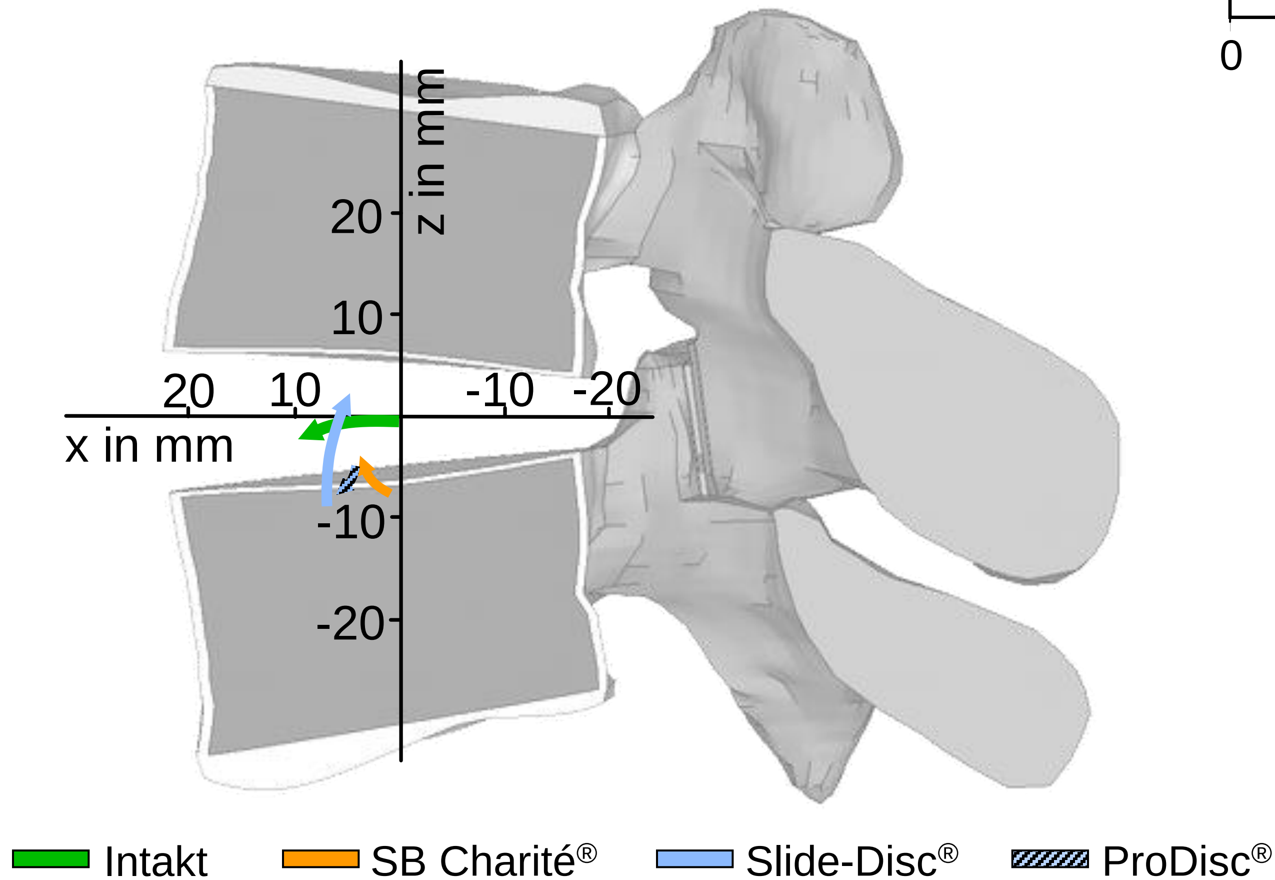
Depuy Spine; Raynham, MA, USA

Mobidisc®

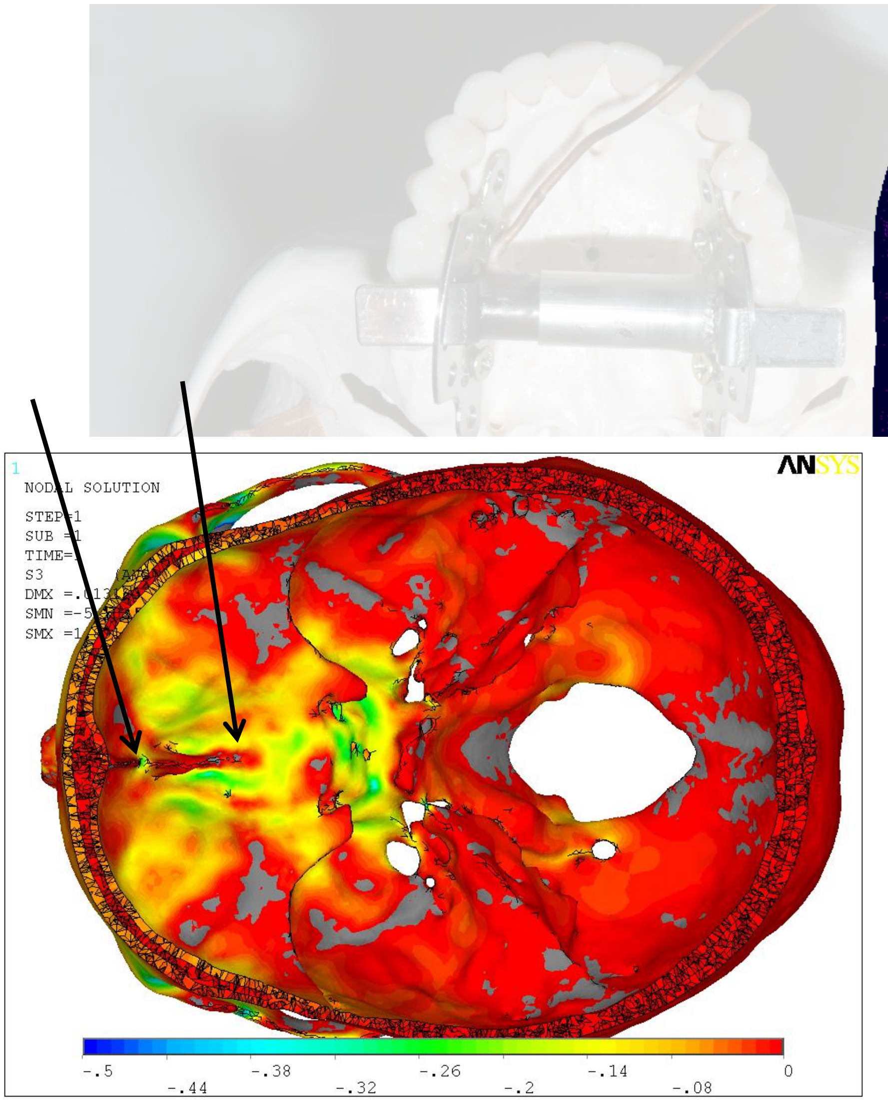
LDR médical; Troyes, Frankreich

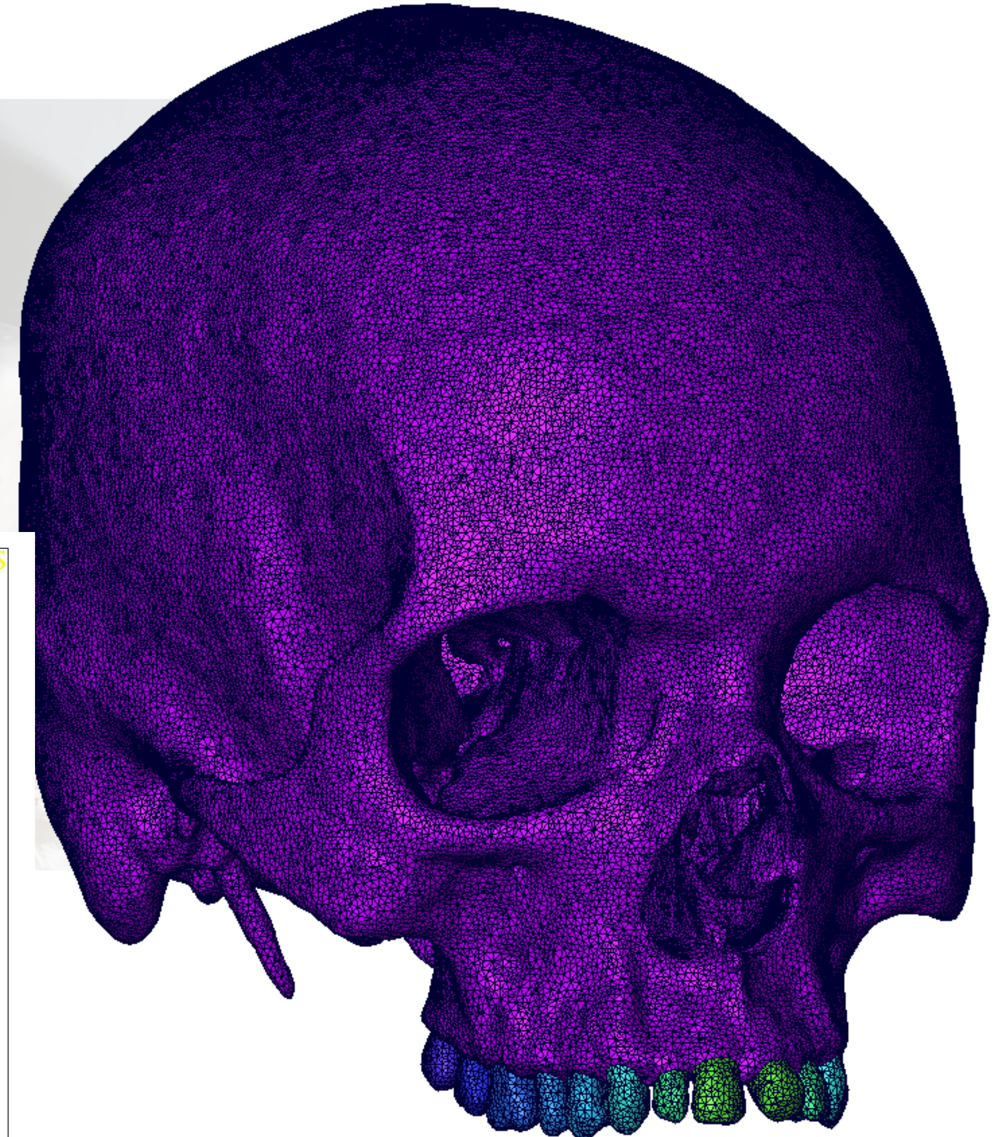
Anwendungsgebiete





Anwendungsgebiete





Zusammenfassung

Was macht die FEM erforderlich in der Biomechanik?

- Erlangung eines besseren Verständnisses von biomechanischen Abläufen individueller gesunder und degenerierter Strukturen.

- Erlangung eines besseren Verständnisses von pathologischen Vorgängen

- Parameterstudien: Untersuchung des Einflusses einzelner Parameter (z.B. Variation des Marknagelradiuses zur Versorgung von Brüchen an einem Röhrenknochen)

- Simulation/Vorhersage von Operationsergebnissen mit einem Computermodell

- Entwicklung und Optimierung von Implantaten

- Anwendungsgebiete aus der Biomechanik
- Ablauf einer FE-Analyse
- Solution (Was macht der SOLVER ?)
- Modellverifikation und -validierung

Allgemeiner Ablauf einer FE-Analyse

1. Pre-Processor

- Geometrie
- Diskretisierung
- Werkstoff
- Last / Randbedingungen

2. Lösungsphase (Solution)

- Der Computer rechnet: Aufstellen und lösen des Gleichungssystems

3. Post-Processor

- Ergebnisse darstellen

Schritt 1: Preprocessor

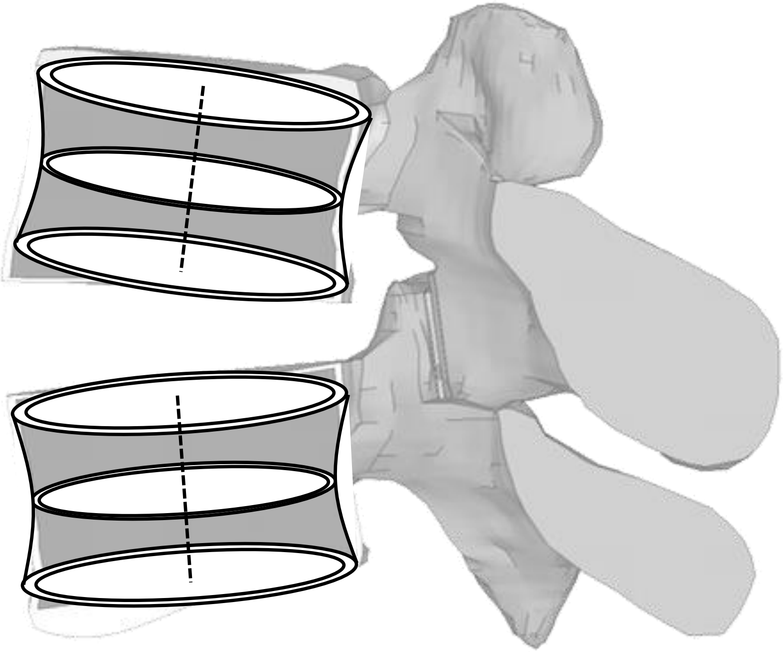
1.1 Geometrie erzeugen im FE-Programm

- Mit booleschen Operationen: Addition, Subtraktion von Grundvolumina
  - Schnell
- Nicht für komplexe Geometrien

- Botom-up-Methode: Punkte, Linien, Flächen, Volumen
  - Beste Kontrolle, Hexaedernetze
  - Langsam, manuelle Eingriffe

- Import von fertigen Netzen z.B. aus AMIRA (Segmentierung von CT-Daten)
  - Komplexe Geometrien
  - Keine Hexaedernetze, manuelle Eingriffe

- Direkte Erzeugung der Elemente: Voxelmodelle
  - Vollautomatisch aus CT-Daten
  - Keine gekrümmten, glatten Oberflächen, kein Kontakt



Schritt 1: Preprocessor

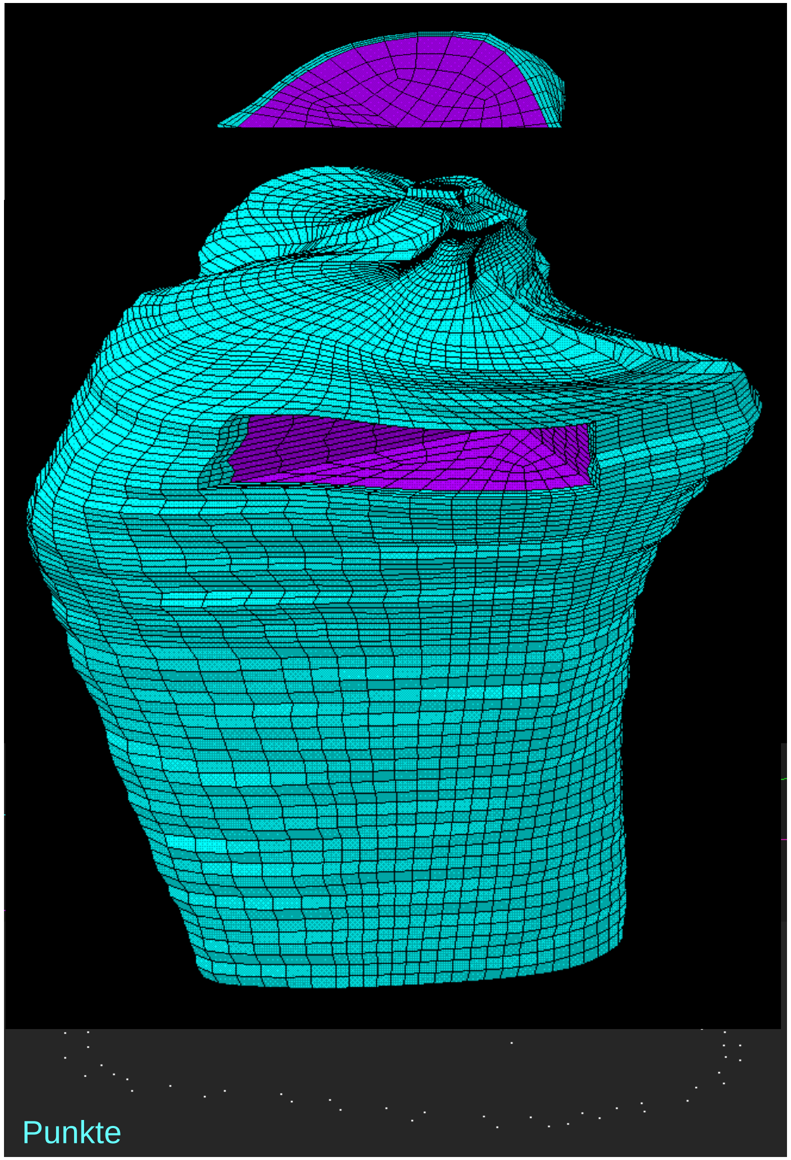
1.1 Geometrie erzeugen

- Mit Boolschen Operationen: Addition, Subtraktion von Grundvolumina
  - Schnell
- Nicht für komplexe Geometrien

- Bottom-up-Methode: Punkte, Linien, Flächen, Volumen
  - Beste Kontrolle, Hexaedernetze
  - Langsam, manuelle Eingriffe

- Import von fertigen Netzen z.B. aus AMIRA (Segmentierung von CT-Daten)
  - Komplexe Geometrien
  - Keine Hexaedernetze, manuelle Eingriffe

- Direkte Erzeugung der Elemente: Voxelmodelle
  - Vollautomatisch aus CT-Daten
  - Keine gekrümmten, glatten Oberflächen, kein Kontakt



Schritt 1: Preprocessor

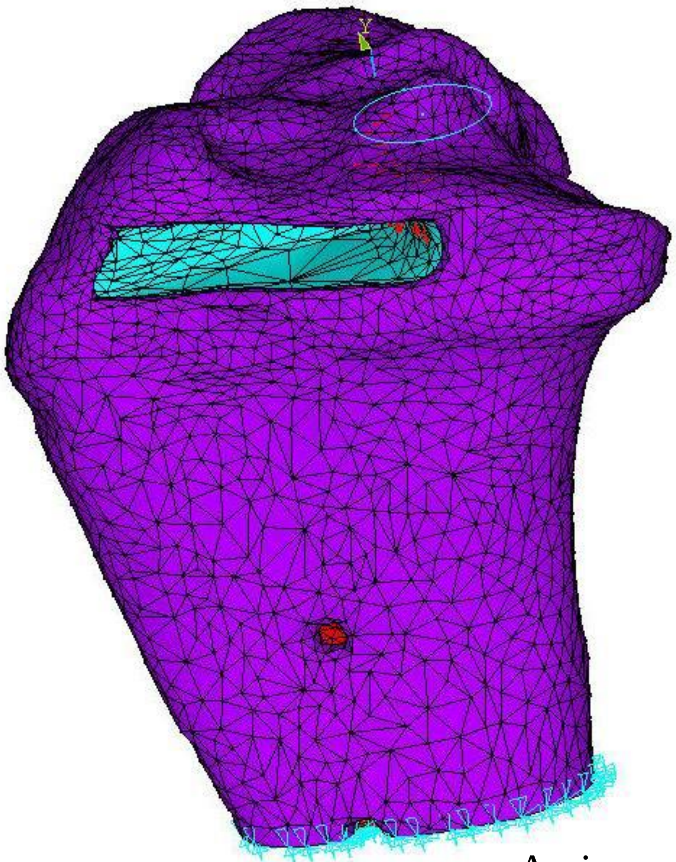
1.1 Geometrie erzeugen

- Mit Boolschen Operationen: Addition, Subtraktion von Grundvolumina
  - Schnell
- Nicht für komplexe Geometrien

- Botom-up-Methode: Punkte, Linien, Flächen, Volumen
  - Beste Kontrolle, Hexaedernetze
  - Langsam, manuelle Eingriffe

- Import von fertigen Netzen z.B. aus AMIRA (Segmentierung von CT-Daten)
  - Komplexe Geometrien
  - Keine Hexaedernetze, manuelle Eingriffe

- Direkte Erzeugung der Elemente: Voxelmodelle
  - Vollautomatisch aus CT-Daten
  - Keine gekrümmten, glatten Oberflächen, kein Kontakt



aus Amira

Schritt 1: Preprocessor

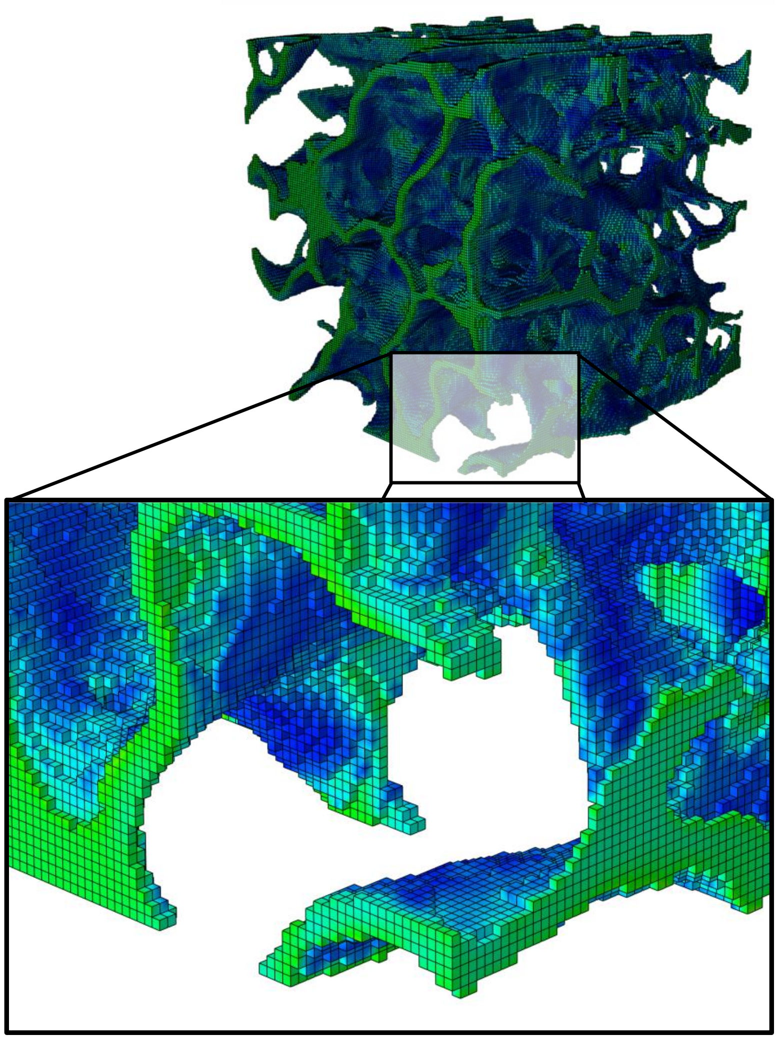
1.1 Geometrie erzeugen

- Mit Boolschen Operationen: Addition, Subtraktion von Grundvolumina
  - Schnell
- Nicht für komplexe Geometrien

- Botom-up-Methode: Punkte, Linien, Flächen, Volumen
  - Beste Kontrolle, Hexaedernetze
  - Langsam, manuelle Eingriffe

- Import von fertigen Netzen z.B. aus AMIRA (Segmentierung von CT-Daten)
  - Komplexe Geometrien
  - Keine Hexaedernetze, manuelle Eingriffe

- Direkte Erzeugung der Elemente: Voxelmodelle
  - Vollautomatisch aus CT-Daten
  - Keine gekrümmten, glatten Oberflächen, kein Kontakt



Schritt 1: Preprocessor

1.3 Werkstoffgesetze, -eigenschaften

- Linear-elastisch, isotrop: E-Modul und Querkontraktionszahl
- Nichtlinearitäten: Nichtlinear-elastisch, plastisch, Verfestigung, Ermüdung, Bruch
- Anisotropie: Transverse Isotropie (Kortikalis), Orthotropie, ...
- Mehrphasig: biphasische Materialien (poröse Materialien)

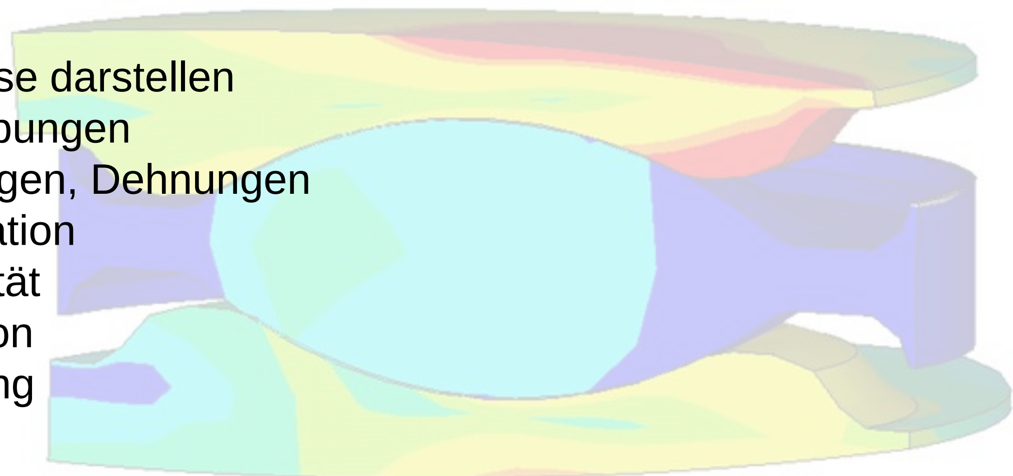
1.4 Last / Randbedingungen

- Kräfte oder Verschiebungen (Drücke, Temperaturen, ...)
- Knoten-, Linien-, Flächen-, Volumenkräfte
- Einspannungen, Symmetrien, Zwangsbedingungen

Schritt 2: Lösungsphase

- Der Computer rechnet
- Linearer Gleichungslöser, Wavefront-Löser
- Iterativer Löser für nichtlineare oder zeitabhängige Probleme

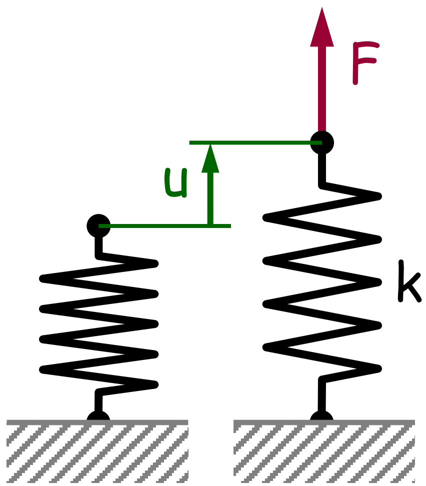
Schritt 3: Post-Processor



- Ergebnisse darstellen
- Verschiebungen
- Spannungen, Dehnungen
- Interpretation
- Plausibilität
- Verifikation
- Validierung

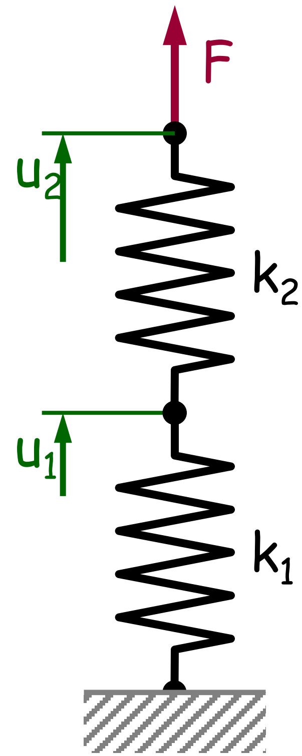
- Anwendungsgebiete aus der Biomechanik
- Aufbau einer FE-Analyse
- Solution (Was macht der SOLVER ?)
- Modellverifikation und -validierung

FE Explanation on one slide



$$k \cdot u = F$$

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



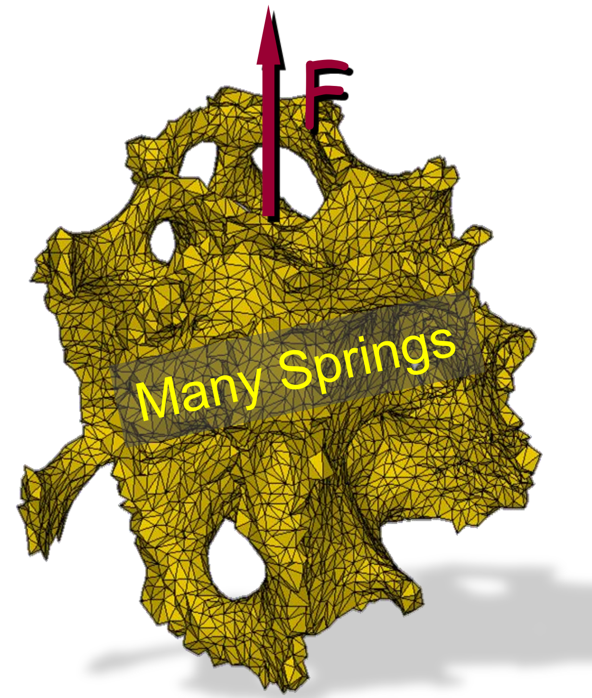
$$k_1 u_1 = k_2 (u_2 - u_1)$$

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

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

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

$\Leftrightarrow$



FE-Software

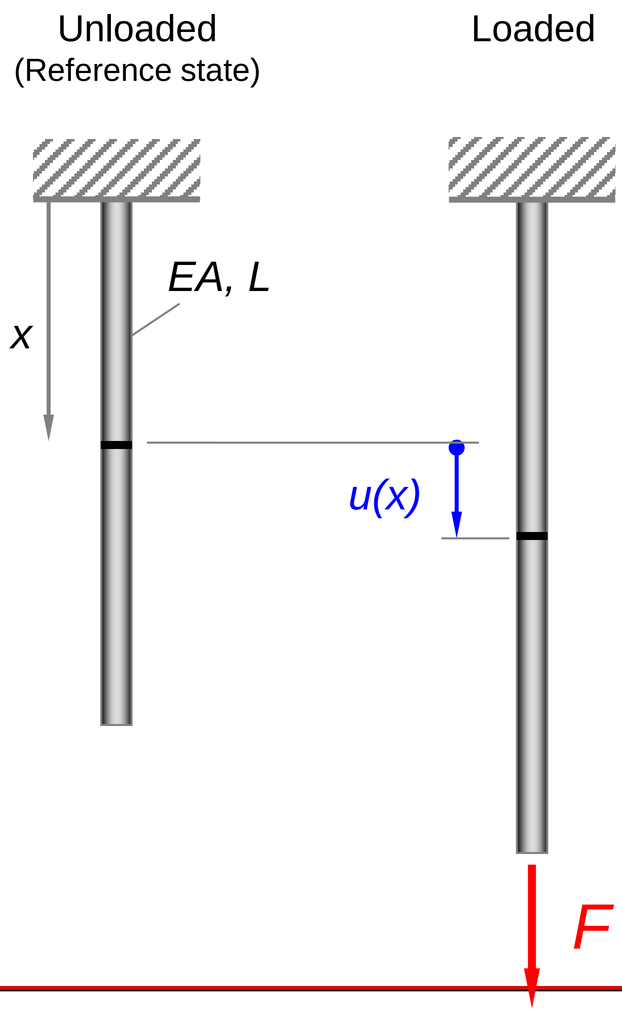
$$\underline{\underline{K}} \cdot \underline{u} = \underline{F}$$

FE-Software

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

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

Example: Tensile Rod



Given:

Rod with …

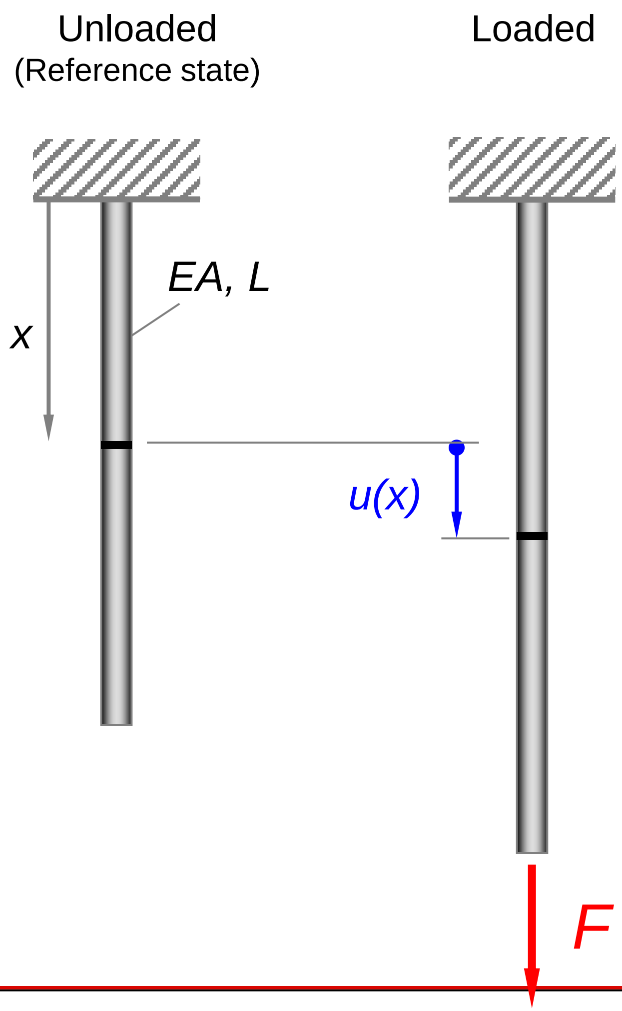
- Length *L*
- Cross-section *A* (constant)
- E-modulus *E (*constant)
- Force *F* (axial)
- Upper end fixed

To determine:

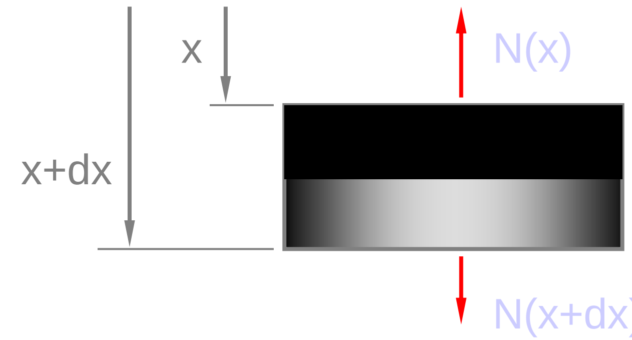
Deformation of the loaded rod:

Displacement function *u(x)*

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



Differential Element
(infinitesimale Higth dx)



Generate the Differential Equation

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

$$\boxed{\Rightarrow \text{DGl:} \quad (EA\,u')' = 0}$$

If EA = const then

$$\boxed{u'' = 0}$$

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

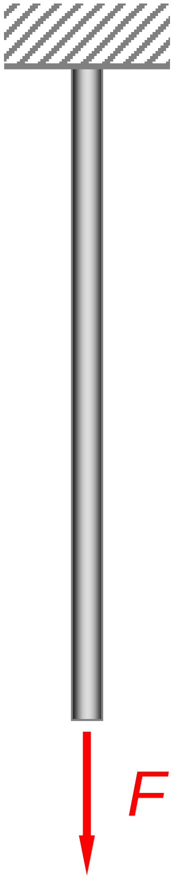
Top (Fixation): $u(0) = 0 \Rightarrow C_2 = 0$

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

$\Rightarrow C_1 = F/EA$

Adjusted Solution

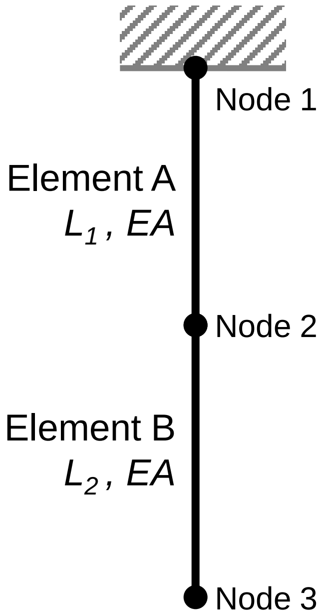
$$\mathbf{u(x) = (F/EA)*x}$$



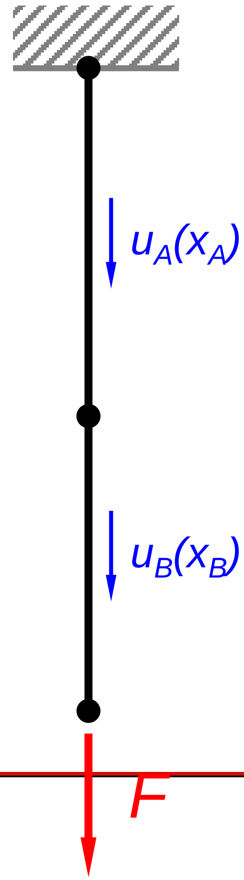
B) Solution with FEM

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

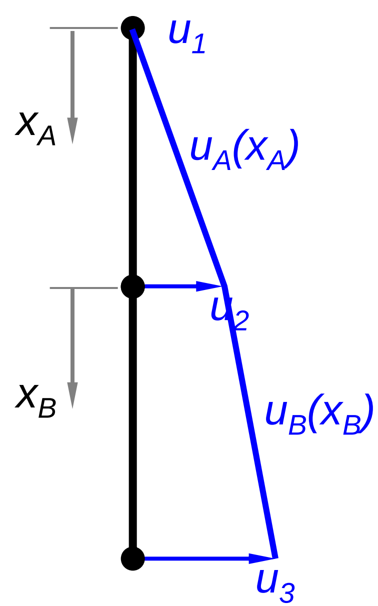
Unloaded:
(Reference condition)



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:

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

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

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:

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

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:

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

[redacted]

Or in short:

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

$\underline{\underline{K}}$ - 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 …

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

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.

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

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

Strains

$$\sigma_A(x_A) = E\varepsilon_A(x_A)$$

$$\sigma_B(x_B) = E\varepsilon_B(x_B)$$

Stresses

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.

Zusammenfassung

1. Elementsteifigkeitsmatrizen $\underline{\underline{K}}^e$ bestimmen
2. Gesamtsteifigkeitsmatrix $\underline{\underline{K}}$ bestimmen
3. Einbau der geometrischen Randbedingungen
4. Auflösen des Gleichungssystems $\underline{\underline{K}} \cdot \underline{u} = \underline{F}$ nach unbekannten Verschiebungen und Reaktionskräften
5. Innere Kräfte der Elemente bestimmen

Informationsquellen

Bernd Klein FEM – Grundlagen und Anwendungen der Finite-Elemente-Methode. Vieweg Verlagsgesellschaft; Auflage 4, 2000, ISBN 3-528-35125-X.

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/

- Anwendungsgebiete aus der Biomechanik
- Aufbau einer FE-Analyse
- Solution (Was macht der SOLVER ?)
- Verifikation und Validierung

Marco Viceconti, Clinical Biomech, 2005

“I have a question and I’m sure you won’ t like it:

How did you verify and validate your finite element model?”

Available online at www.sciencedirect.com

SCIENCE @ DIRECT®

ELSEVIER

Clinical Biomechanics 20 (2005) 451–454

CLINICAL BIOMECHANICS

www.elsevier.com/locate/clinbiomech

Editorial

Extracting clinically relevant data from finite element simulations

1. Introduction

The present editorial is the conclusion of a consensus

sional community, and also the rather simple materials that are considered in most industry applications. In biomechanics and related research our biggest challenge

Verification and validation

physical problem

VALIDATION ↓ Model abstraction and assumptions

constitutive equations (PDEs) and boundary conditions

VERIFICATION ↓ FE discretization

algebraic equations and boundary conditions

VERIFICATION ↓ Numerical solution

Solving the unknowns

Definitions

Verification
the process of determining that a computational model accurately represents the underlying mathematical model and its solution

("solving the equations right" - mathematics)

Validation
the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model

("solving the right equations" - physics)

(ASME Committee for Verification and Validation in Computational Solid Mechanics)

Verification

Step 1: code verification (solution of the discretized equations)

A verified code yields the correct solution to **benchmark problems** of known solution

check the computer code for:

- inadequate iterative convergence
- programming bugs
- lack of conservation (mass, momentum, ...)
- number round-off (single precision, double precision)
- ...

especially when developing the simulation code in-house!!!

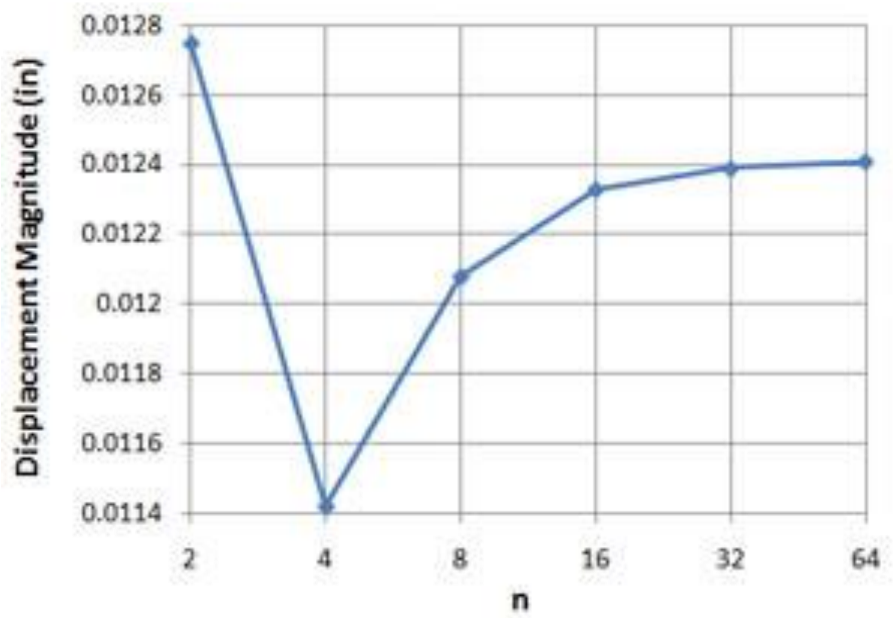
A verified code is not necessarily guaranteed to accurately represent complex biomechanical problems (this is the domain of *validation*)

Verification

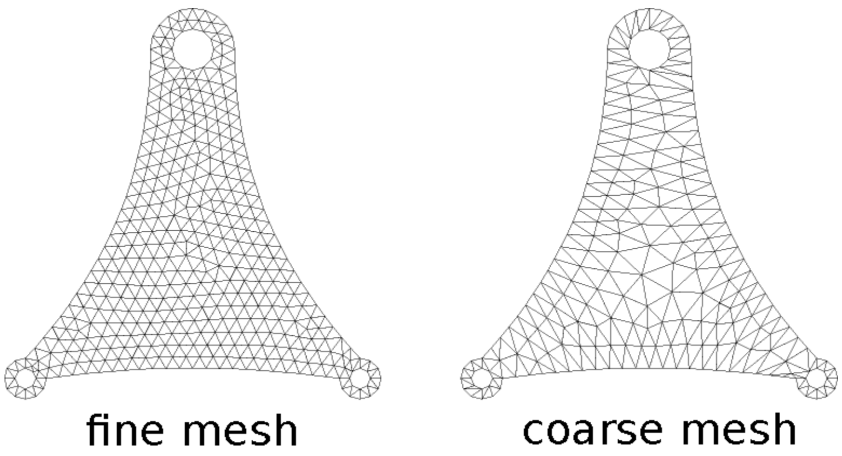
Step 2: calculation verification (correctness of the problem discretization)

Verification of the errors arising from the discretization of the problem

Mesh convergence: solution leads to an asymptote with increasing mesh density



(http://usa.autodesk.com)



(ANSYS 11.0 Documentation)

Validation

Direct validation	**Indirect validation**
purposely designed in vitro or in vivo experiments and measurements	based on literature data, clinical studies (no control by the user)

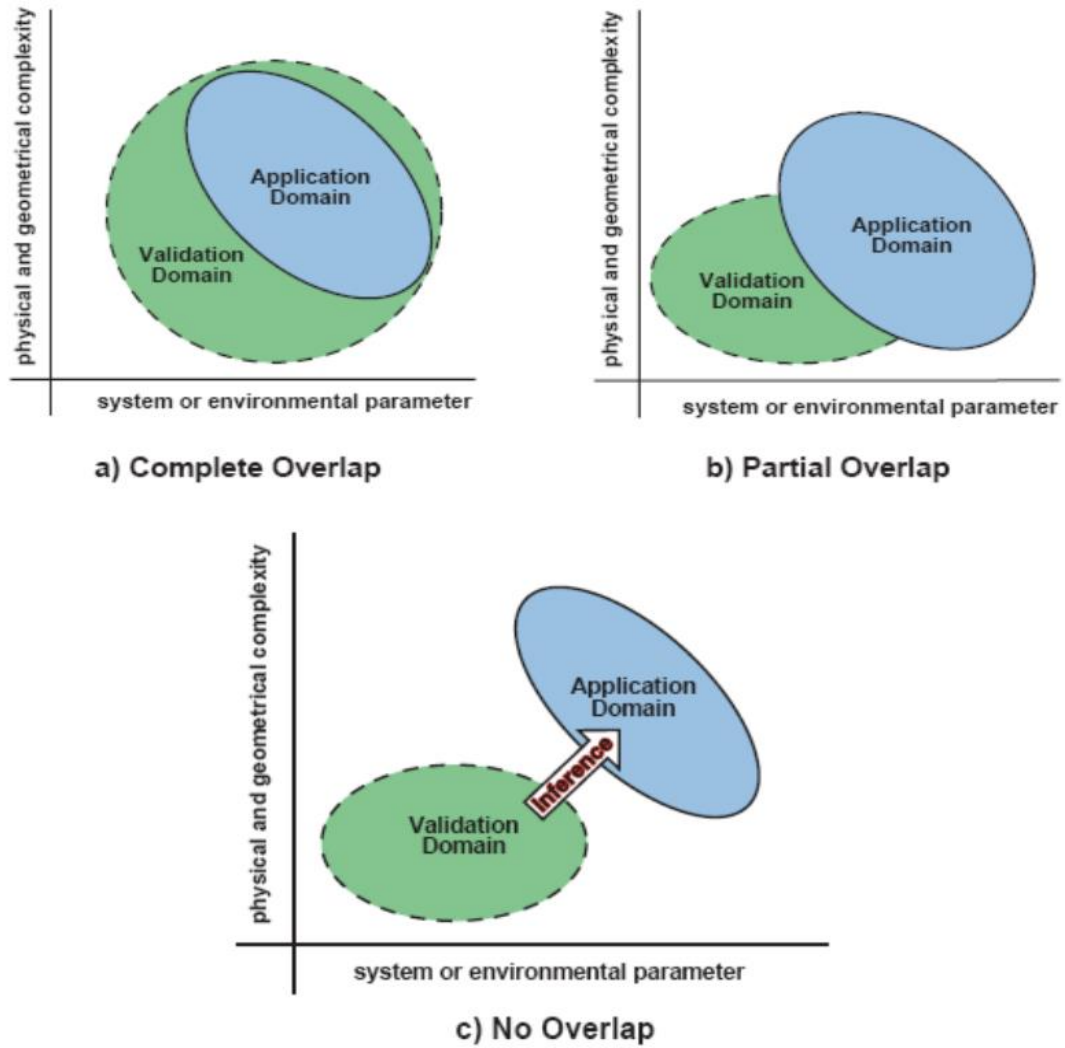
Problems with in vitro experiments and validation

Are the experiments well representing the in-vivo conditions?

Is it really necessary to mimic the in-vivo conditions for validation, or is a simple experiment enough?

(a model which is not able to replicate a simple experiment is probably not better in a complex case)

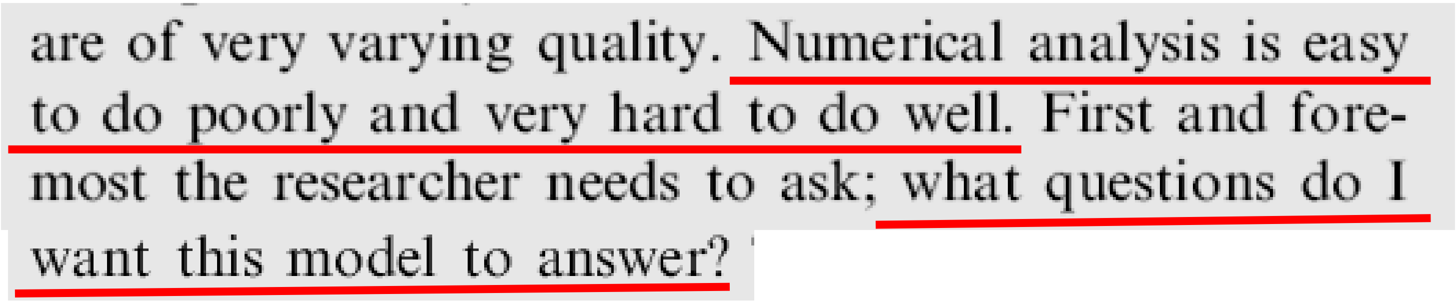
Validation and application domains



Viceconti, 2005: Anforderungen

Level 0	• Programmreliabilität	⇒ **Technical journal**
Level 1	• Modelselektion • Modellverifikation	⇒ **Journals interested in theoretical speculations**
Level 2	• Sensitivitätsanalyse • Intersubjektvariabilität	⇒ **Applied biomechanics research**
Level 3	• Validation	⇒ **Between biomechanical and clinical research**
Level 4	• Risk-benefit analysis • Prospektive Studie	⇒ **Clinical journals**

Viceconti, 2005: Conclusion


are of very varying quality. Numerical analysis is easy to do poorly and very hard to do well. First and foremost the researcher needs to ask; what questions do I want this model to answer?

Zusammenfassung

Lehrziele

Die Studierenden sollen …

- die Bedeutung der FEM für die Biomechanik anhand einzelner Beispiele benennen können;
- Vorteile und Nachteile gegenüber (i) experimentellen sowie (analytischen) Methoden angeben können;
- die generellen Arbeitsschritte einer FEA auflisten können;
- die grundlegenden Ideen und Annahmen der FE-Theorie kennen;
- die Faktoren, die die Güte einer FEA beeinflussen, benennen können;
- und die grundsätzlichen Limitation von Modellen beurteilen können, sowie die daraus immerwährend gegebene Notwendigkeit von Verifikation und Validierungen ableiten können.

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