Industry Sector	RTD Thematic Area	Date
Biomedical		Nov.-13-2001

Design and Analysis of Orthopedic Implants by means of FE Simulation

Dr Yasar Deger

Sulzer Markets & Technology Ltd

Sulzer Innotec / Structural Mechanics

8401 Winterthur / Switzerland

Industry Sector	RTD Thematic Area	Date
Biomedical		Nov.-13-2001

Abstract:

Sulzer Orthopedics develops medical implants with close cooperation and technical support of Sulzer Innotec. During the last years finite element analyses are used for development and optimisation of mainly three orthopedic product groups: knee and spine implants, artificial hip joints. The use of CAE techniques gives insight to load mechanisms, material behavior and response of implants and the bone. This presentation gives an overview on the computer-aided bio-mechanical modelling and analysis techniques recently applied by Sulzer Innotec.

An explicit finite element analysis is used for simulation of a knee joint testing facility. The combination of rigid body kinematics (femoral part) and elastic components (tibia part) allows for stress/strain distributions, contact zones, contact pressures and reaction forces during the gait cycle supporting the test results. Numerical results are given for coarse and fine meshes.

In-vitro material tests were carried out to establish a more accurate spine model, specifically for the mechanical behaviour of the intervertebral disk. This new material model gives detailed information of spine mechanics and supports the development of new products that help to reduce back pain.

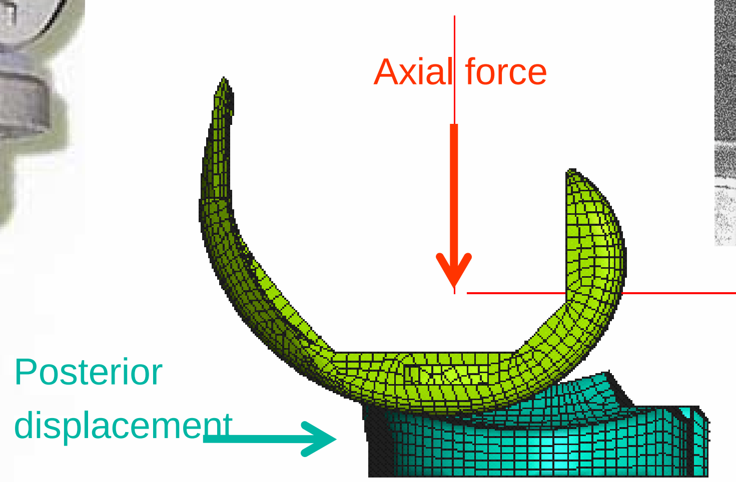
For the design and analysis of tribological concepts and contact conditions in artificial hip joints, the interface of the ball head and the hip cup is modelled with a powerful two-dimensional model. The contact conditions are governed by the stiffness and structural response of the artificial hip cups. Here axisymmetrical elements are used, with unsymmetrical loading by Fourier series expansion. This approach allows very efficient parameter studies and furtherly can enable the use of commercially available computational optimisation tools.

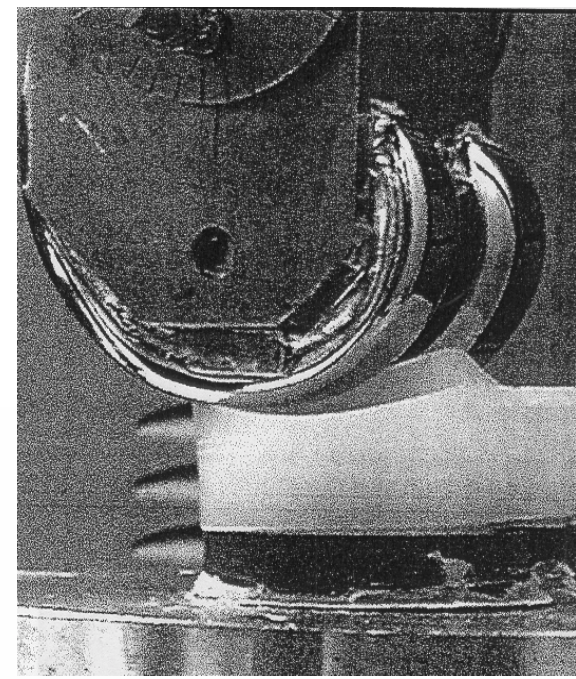
State of the art of finite element analysis in biomechanical applications

- **Simulation of Stanmore knee test, using explicite time integration**
- **Advanced material modelling and FEA of the human spine**
- **Design and analysis of an artificial hip joint system**

Simulation of knee implants

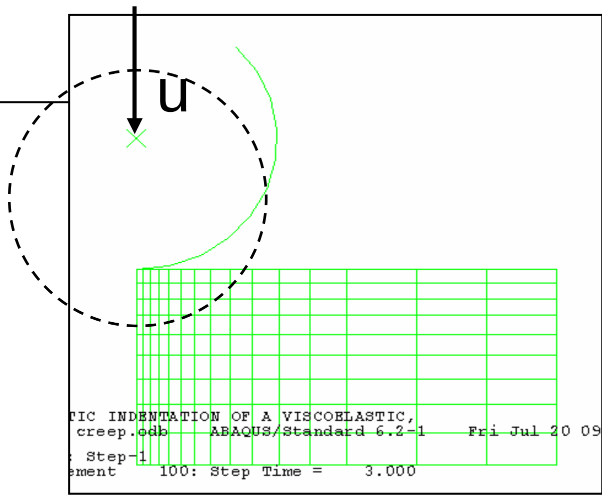
Laxity tests →

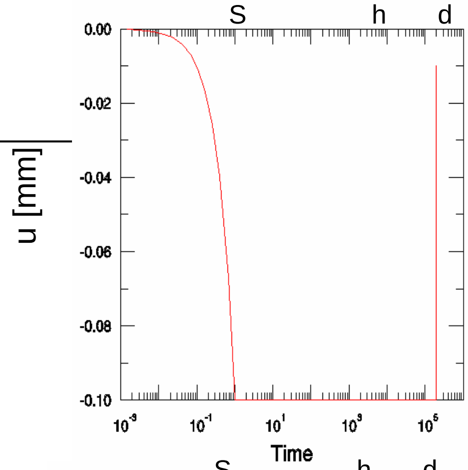




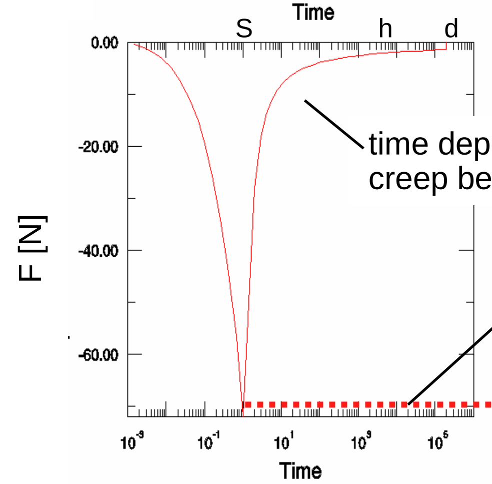
Simulation of knee implants

Modelling of nonlinear material behaviour





Input: displacement



Output: reaction force / load

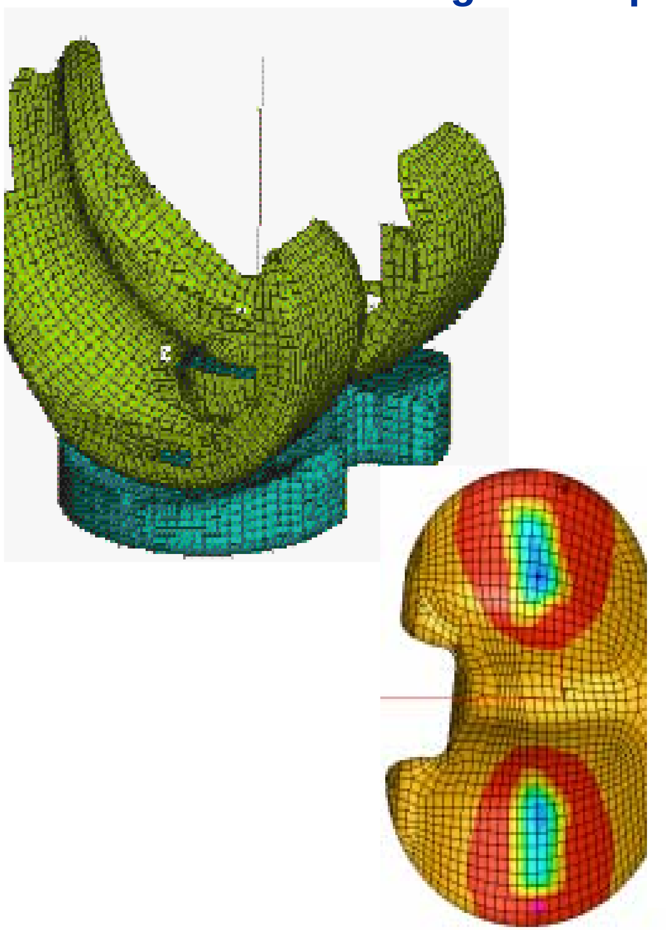
Abaqus material model:

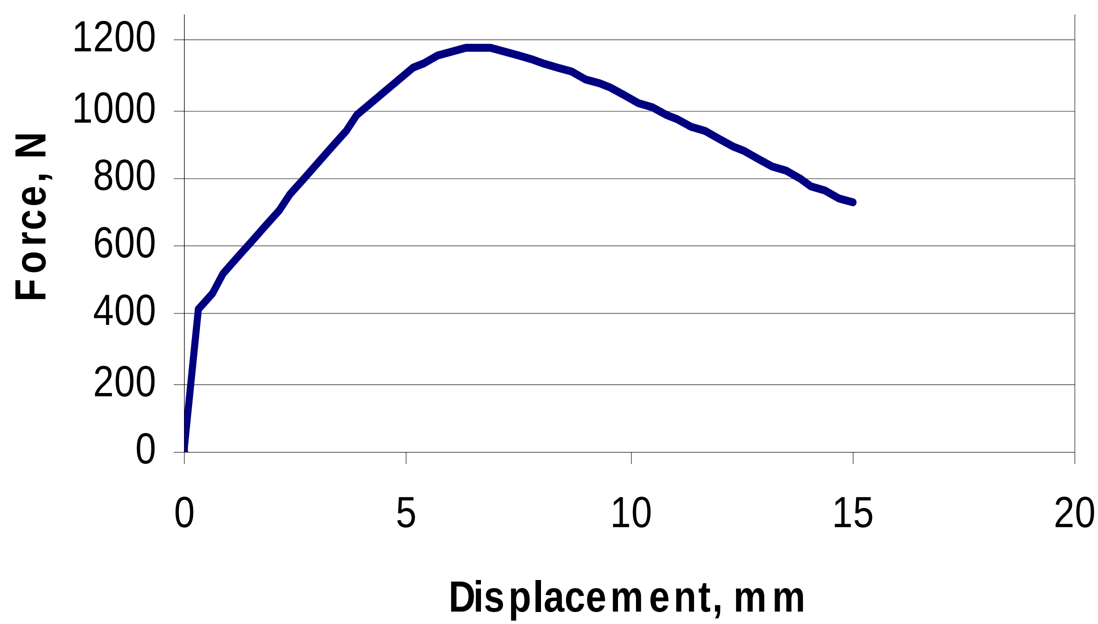
$$\dot{\varepsilon} = A \sigma^{n} t^{m}$$

Parameter A, n, m

Simulation of knee implants

FE modelling of complete Posterior laxity test (femur at 0°)





Simulation of knee implants

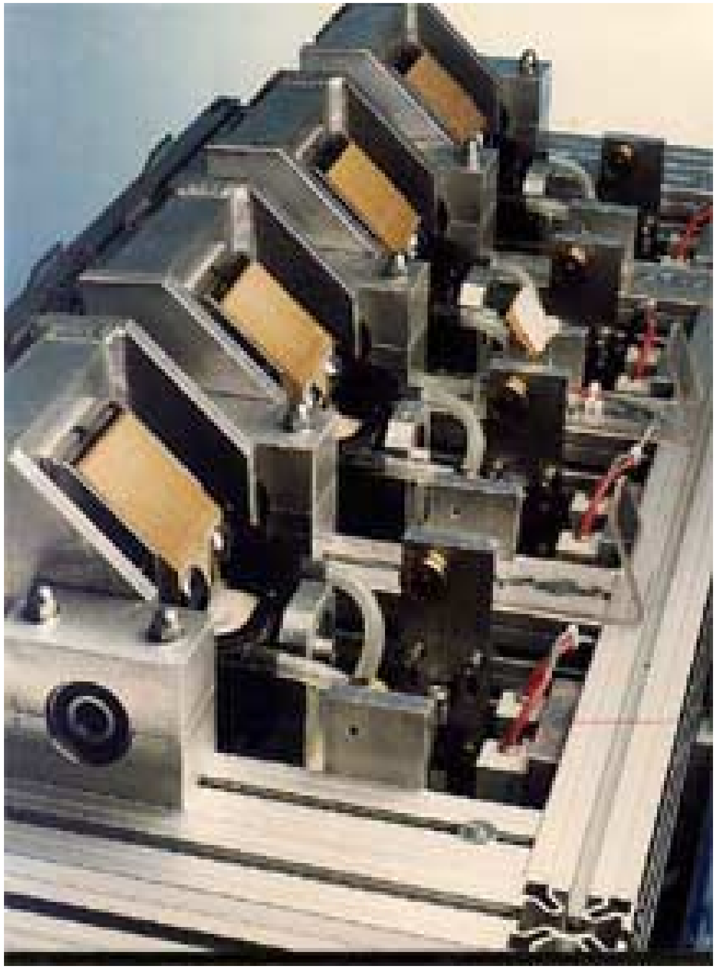
Stanmore knee simulator

Reproduction of in-vivo loading (millions of gait cycles):

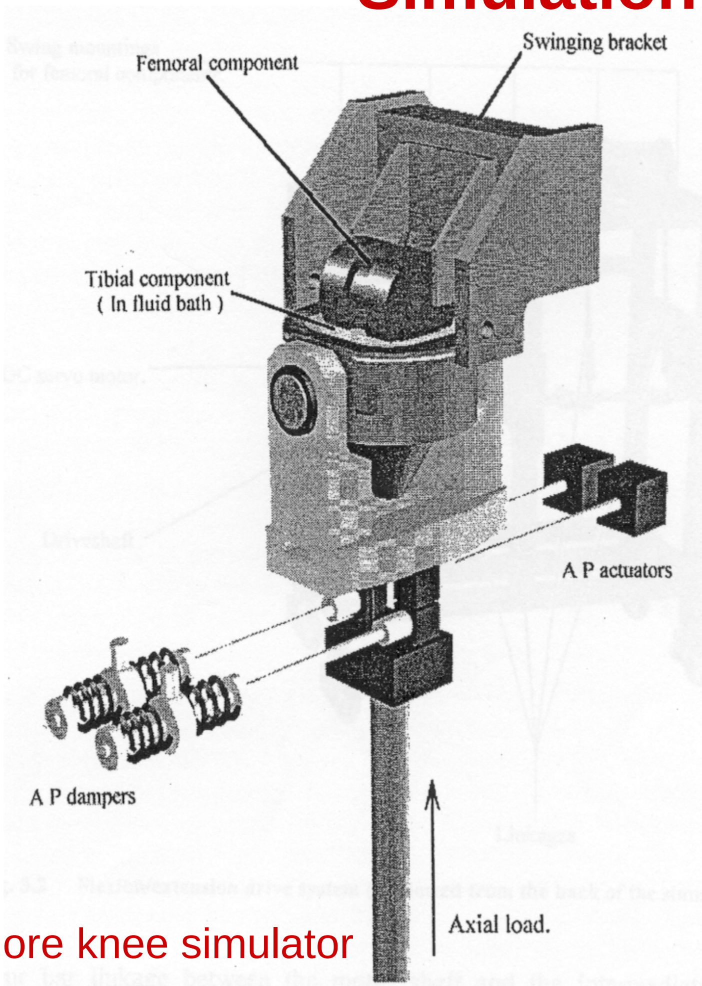
- Applied forces
- Applied constraints

Test of the performance of knee joints:

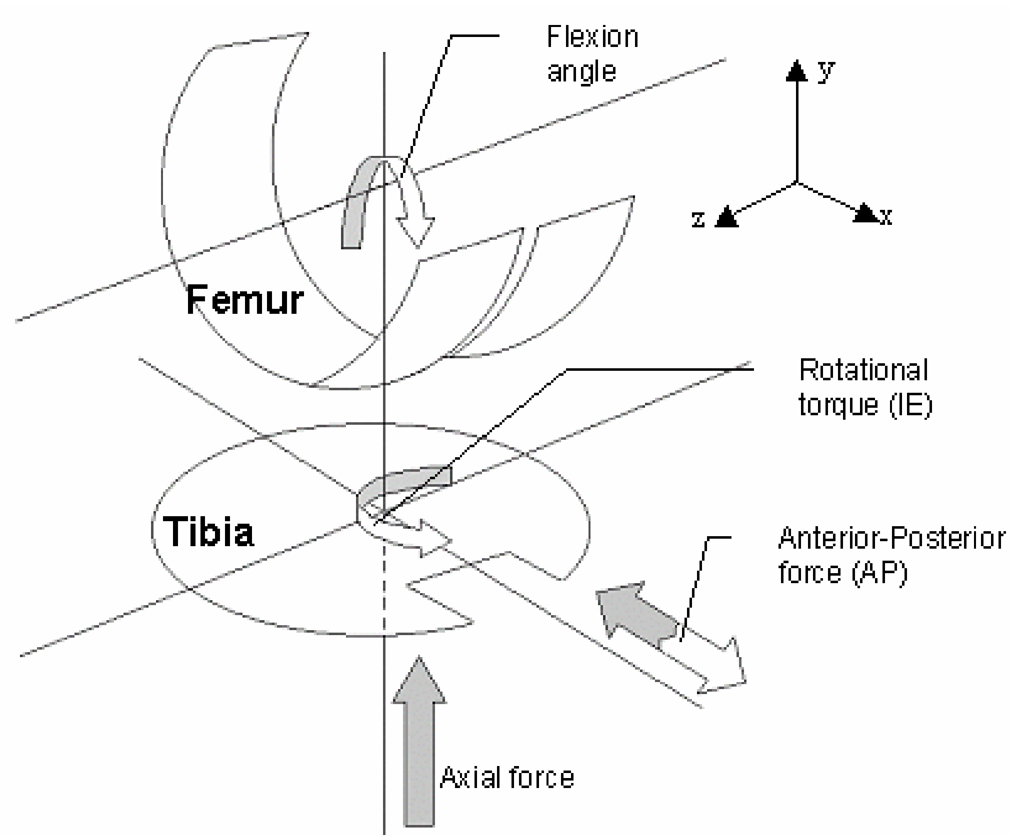
- Evaluation of the kinematics
- Evaluation of the wear



Simulation of knee implants

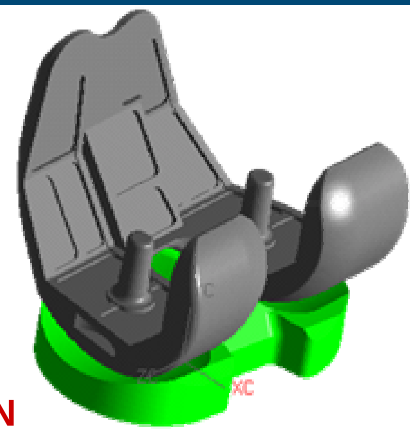


Stanmore knee simulator



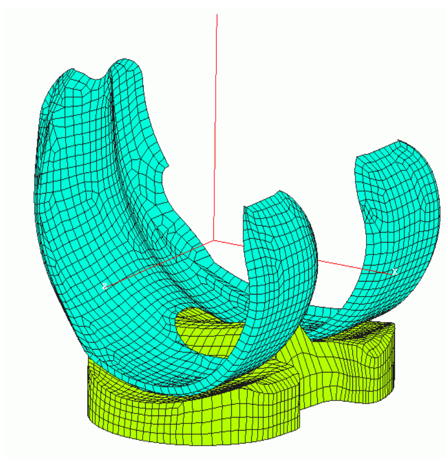
Solution strategy

- **CAD Geometry created in UG**



- **Geometry meshed in MSC.PATRAN**

- **Complete FEA input by defining material properties, boundary conditions, contact parameters in PAM/Generis**



Unigraphics Solutions™ — CAD

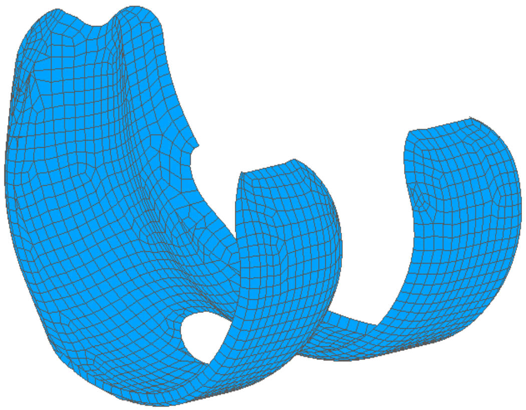
MSC Patran — Pre-processing

PAM — Pre-processing, Solver, Post-processing

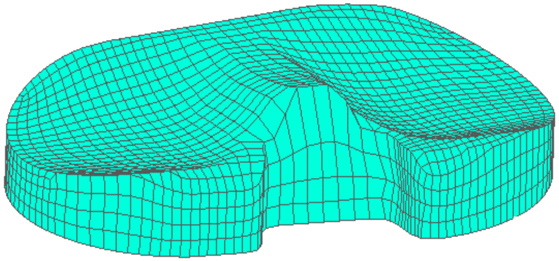
Simulation of knee implants

Set-up of the FE model

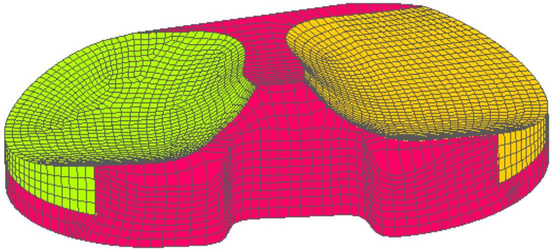
- Femural part: modelled with 4 noded QUAD rigid elements (2100 elements)
- Tibial insert: modelled as 8-noded hexahedral deformable elements (Elastic/Plastic):
  - Coarse mesh: 4700 elements
  - local fine mesh: 12500 elements
- contact conditions between the tibial insert and the femural part ($\mu = 0.07$),
- use of a special smooth contact algorithm available in PAM-Crash



Femur component
2107 shell elements
*2*2 mm*



Tibia component (coarse mesh)
4740 solid elements
*2*2*2 mm*



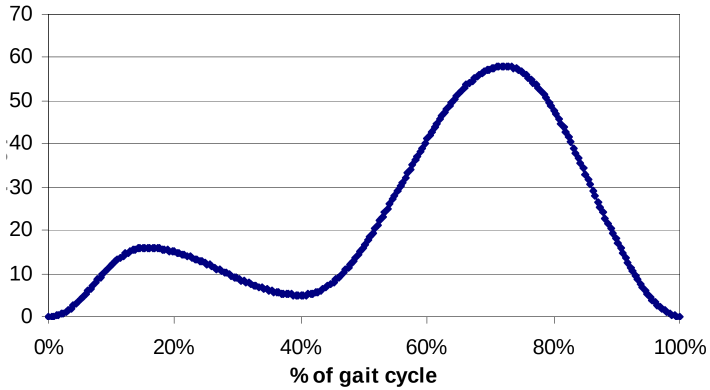
Tibia component (local fine mesh)
12508 solid elements
*2*2*2 mm (dish)*
*1.2*1.2*1.2 mm (meniscus)*

Simulation of knee implants

Prediction of the kinematics of the tibial insert

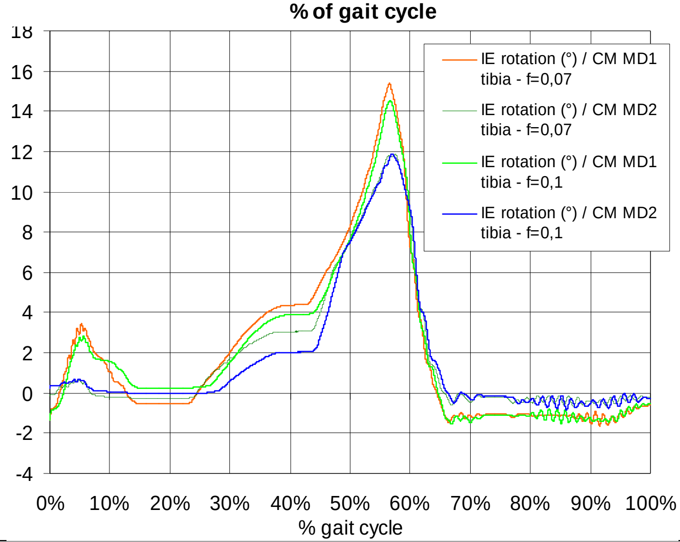
Inputdata:

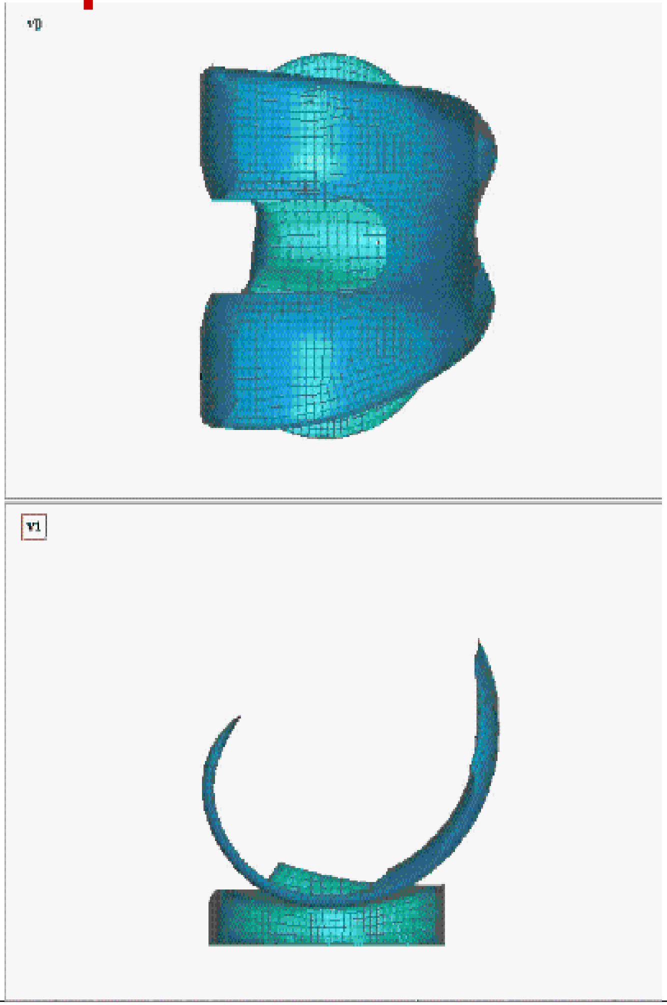
femur flexion



Outputdata:

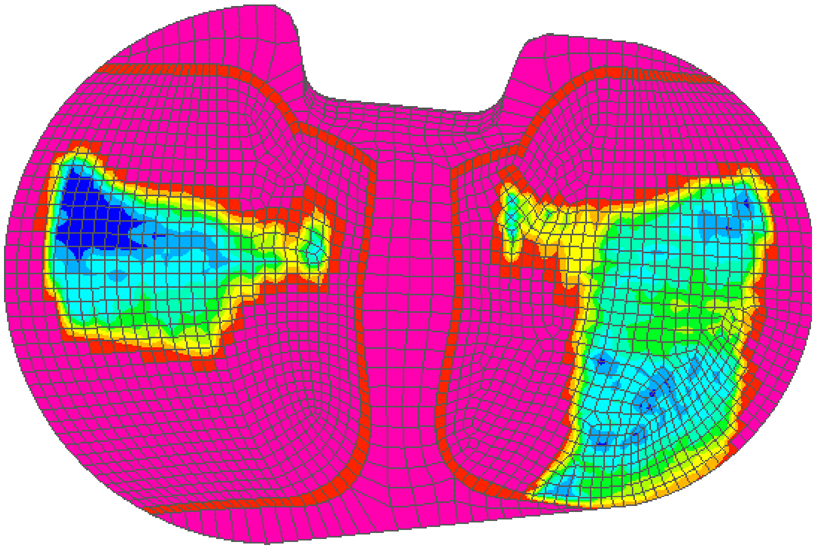
tibia rotation

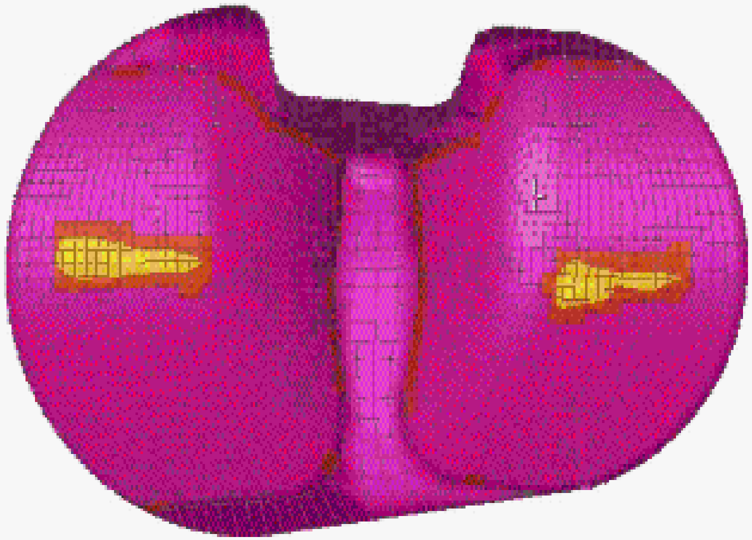




Simulation of knee implants

Prediction of the stresses acting on tibial insert



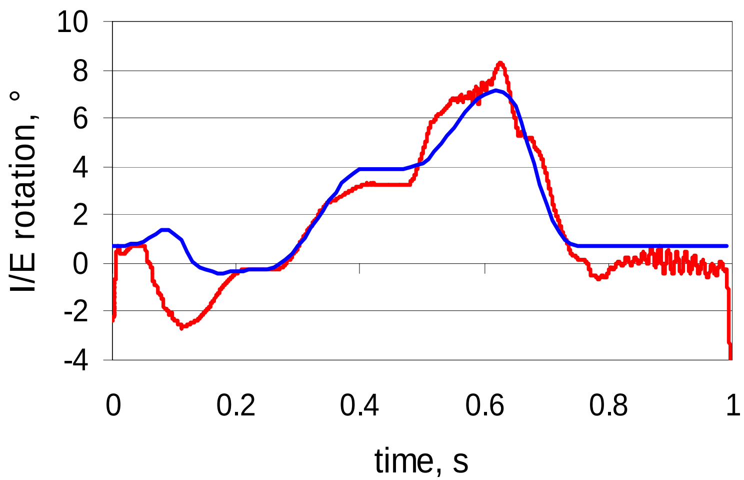


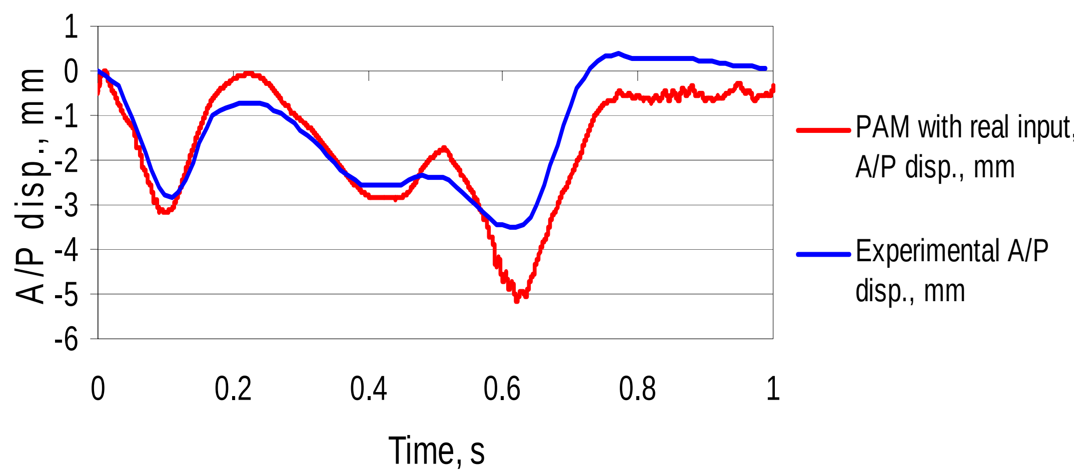
Time histories for contact zone pressure, slip and shear stresses → significant maxima

Simulation of knee implants

Prediction of the kinematics of the tibial insert

PAM vs experimental data: I/E rotations





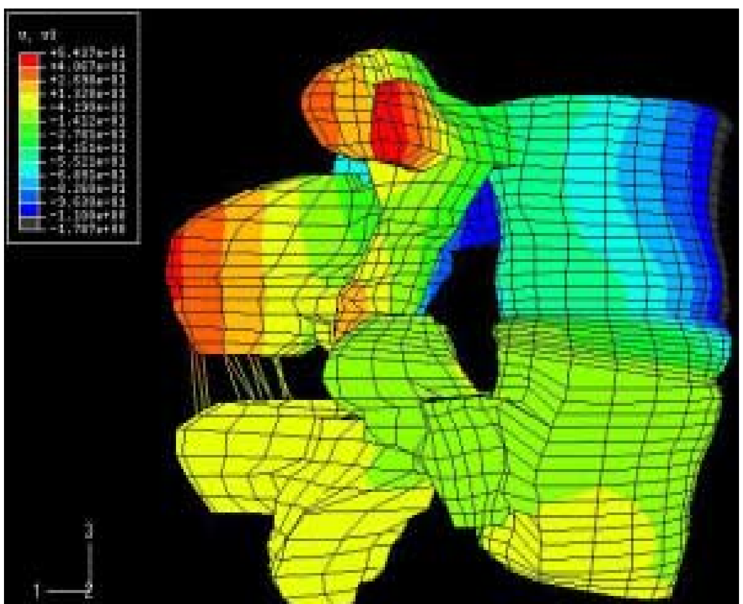
FEA of soft tissues in the human spine

Ligament modelling

- Ligament alignment
- Stress-strain curves

Intervertebral disc modelling

- Experimental investigation
- Anisotropic material model



Unigraphics Solutions™ — QuickShape, Polygonization

↓

MSC Patran — Pre-processing

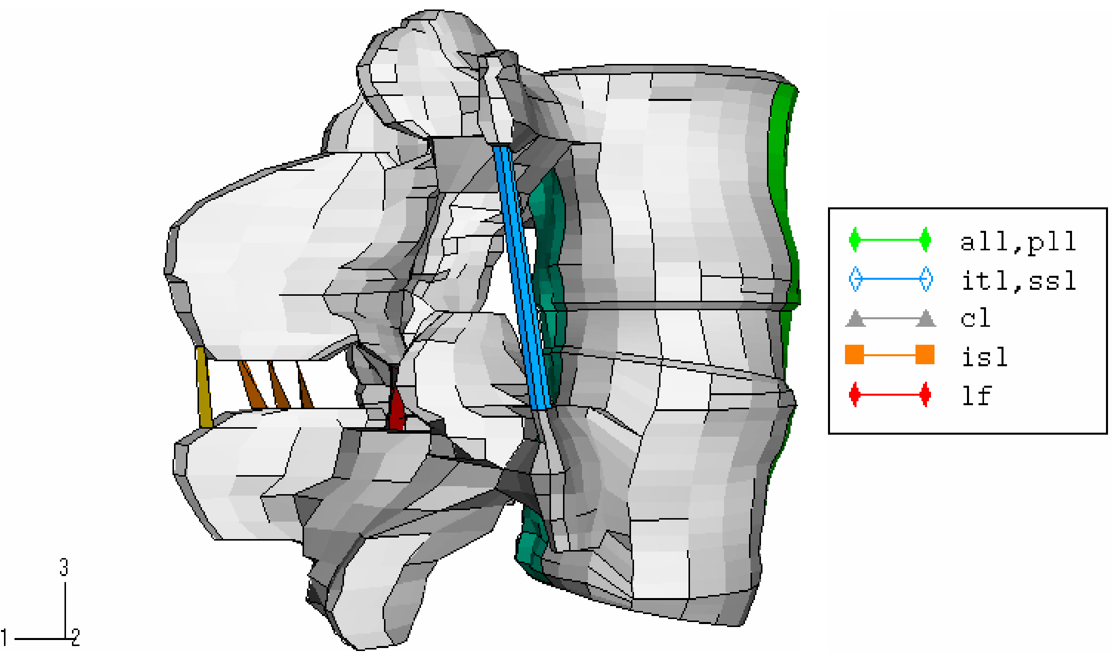
↓

ABAQUS — Material modeling, Solver, Post-processing

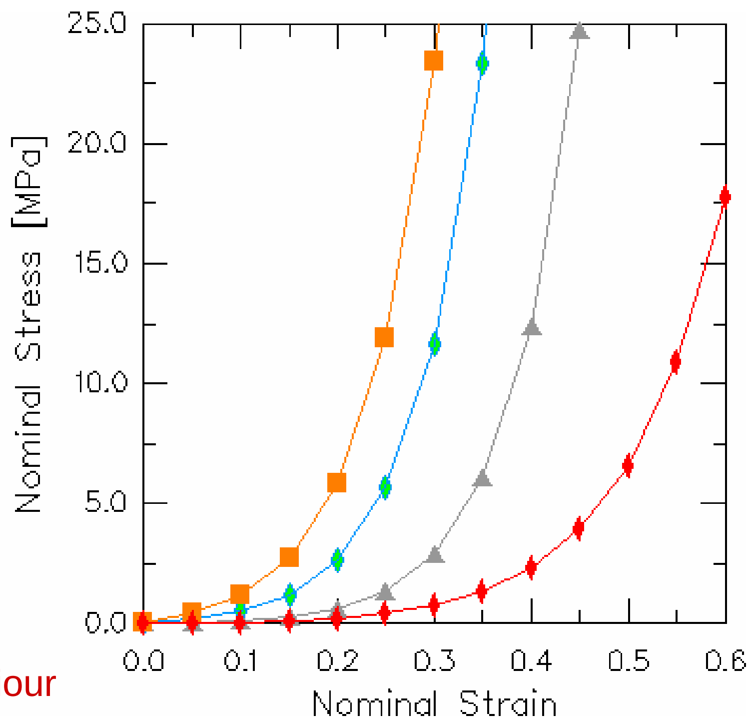
FEA of soft tissues in the human spine

FE modelling of spinal ligaments

Alignment of spinal ligaments



Stress-strain curves for FEA



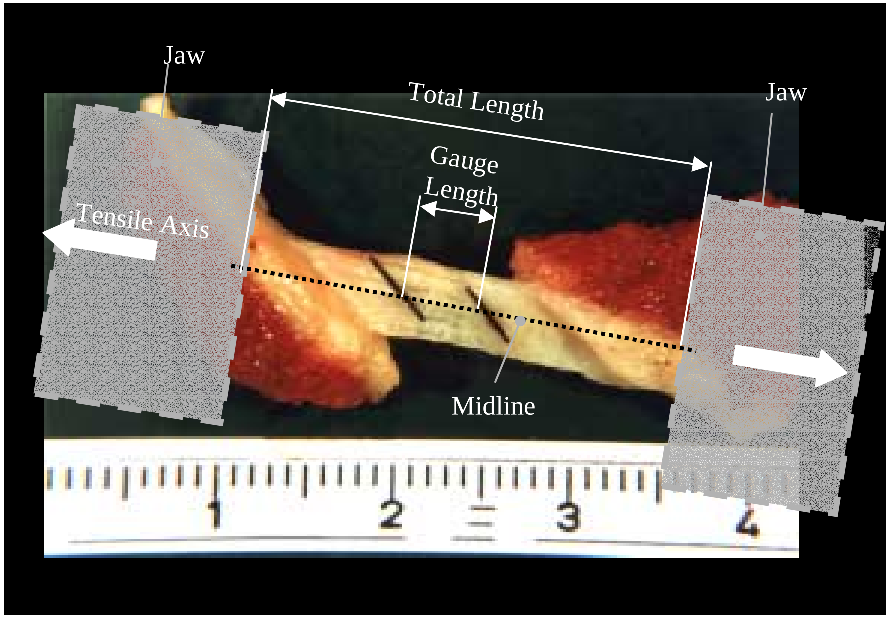
Remarkable hyperelastic behaviour

FEA of soft tissues in the human spine

FE modelling of intervertebral discs

Experimental investigations of human disc body units

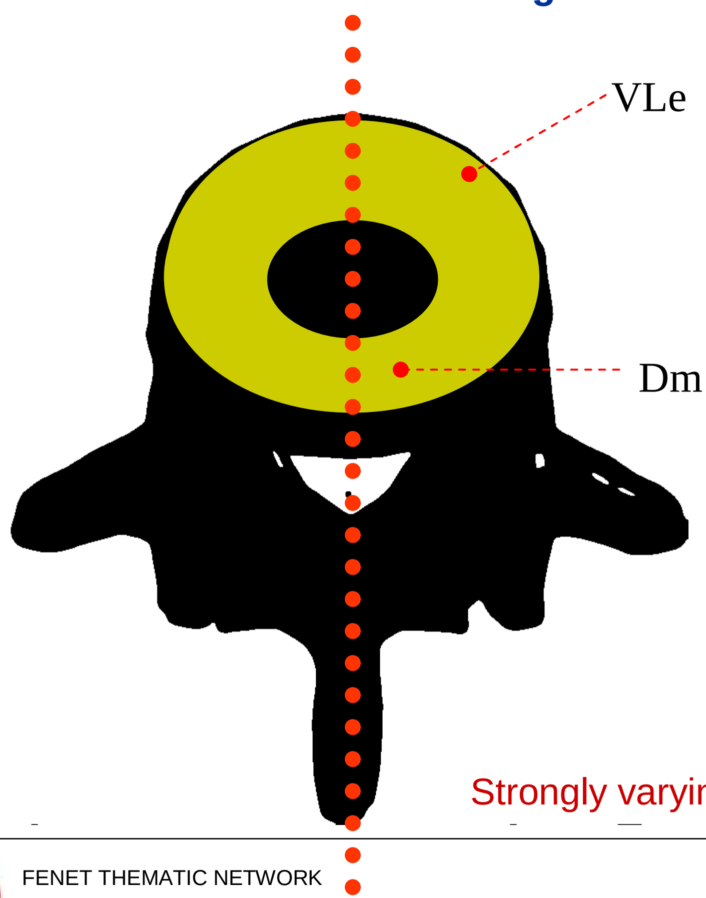
- In vitro investigation of geometric BCs and material properties
- Quasistatic uniaxial tension tests on single lamellae
- Material testing in (visco)elastic regime with final rupture test

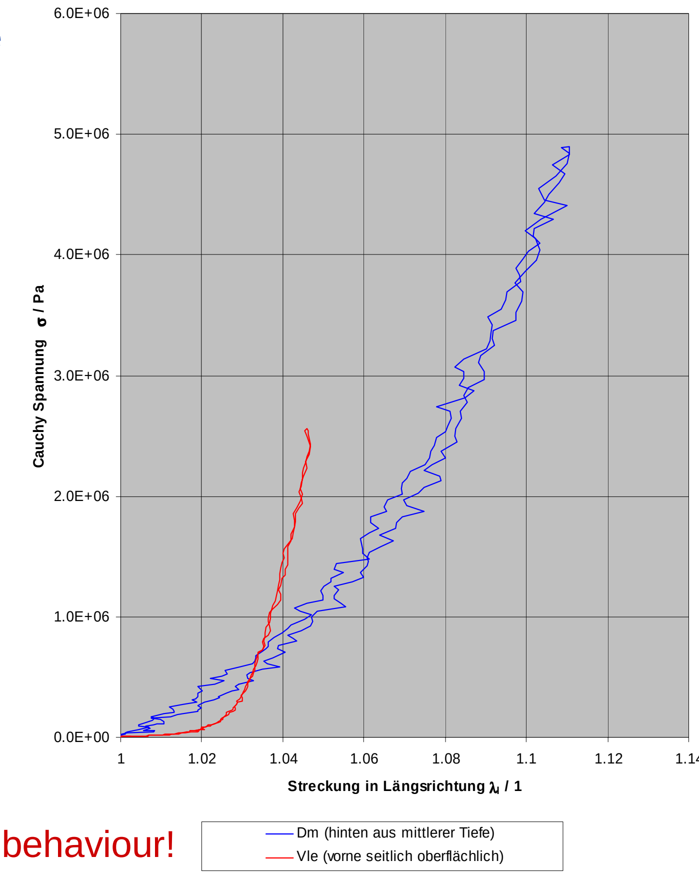


FEA of soft tissues in the human spine

FE modelling of intervertebral discs

Stress-strain relations of single annulus lammellae





Strongly varying hyperelastic behaviour!

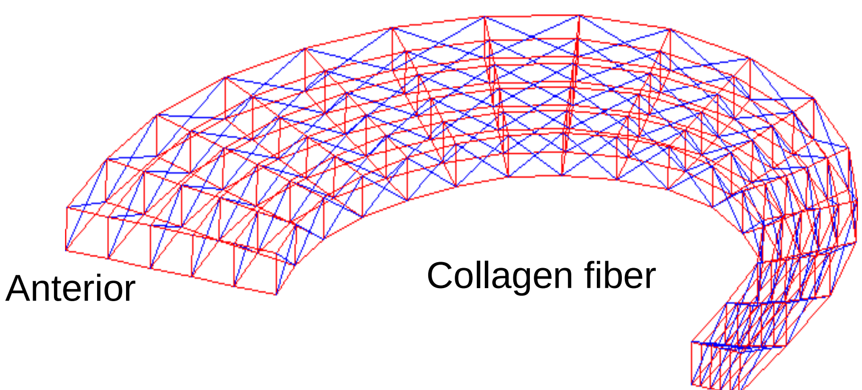
FEA of soft tissues in the human spine

FE modelling of intervertebral discs

Constitutive models for annulus tissue

Discrete fiber model

Discrete collagen fiber arrangement



Old model

Continuous fiber model

Continuous collagen fiber arrangement

Free-energy function (elastic potential):

$$\Psi = \Psi(\mathbf{C},\ldots) = \Psi_{iso} + \Psi_{aniso}$$

with

$$\Psi_{iso} = c_1(I_1 - 3) \quad \textbf{(Neo-Hookean)}$$

$$\Psi_{aniso} = 0.5\, k_1\, k_2 (e^{[k_2(I_4-1)^2]} - 1)$$

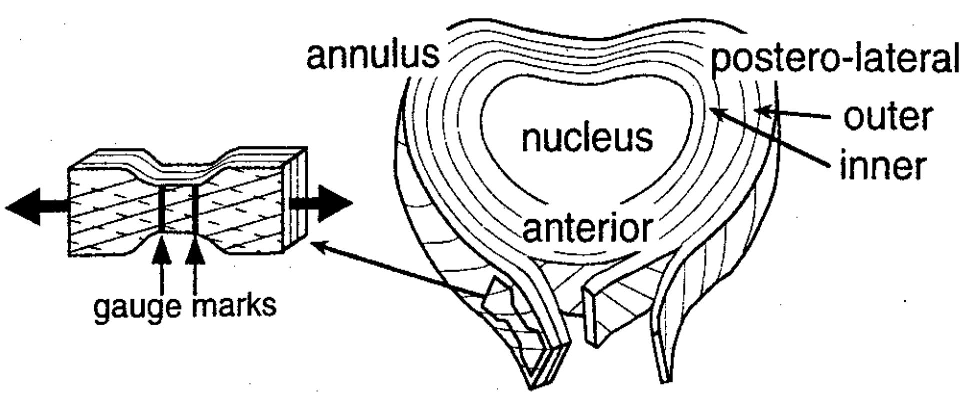
New model

FEA of soft tissues in the human spine

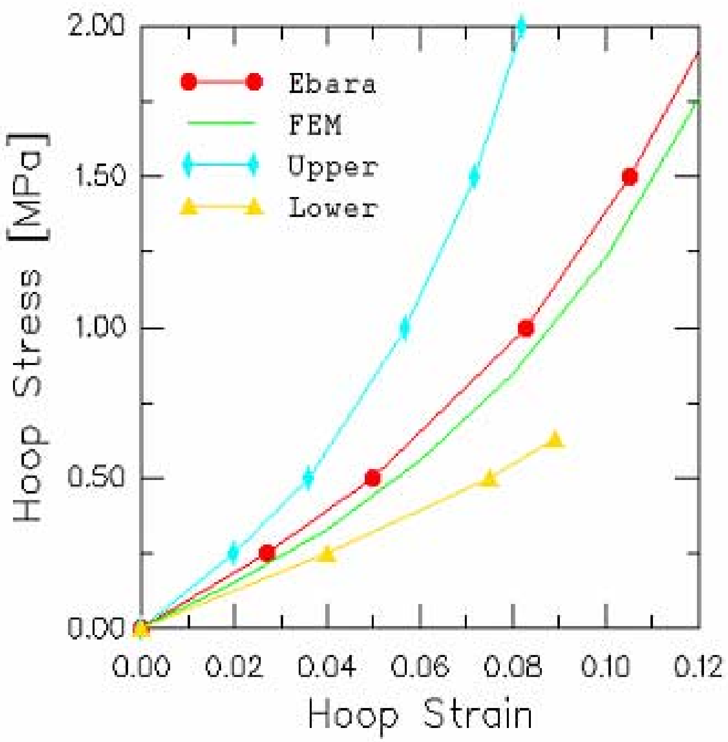
FE modelling of intervertebral discs

Comparison with experimental results from literature

Multi-layer annulus specimens
(Ebara et al. 1996)



Stress-strain curves

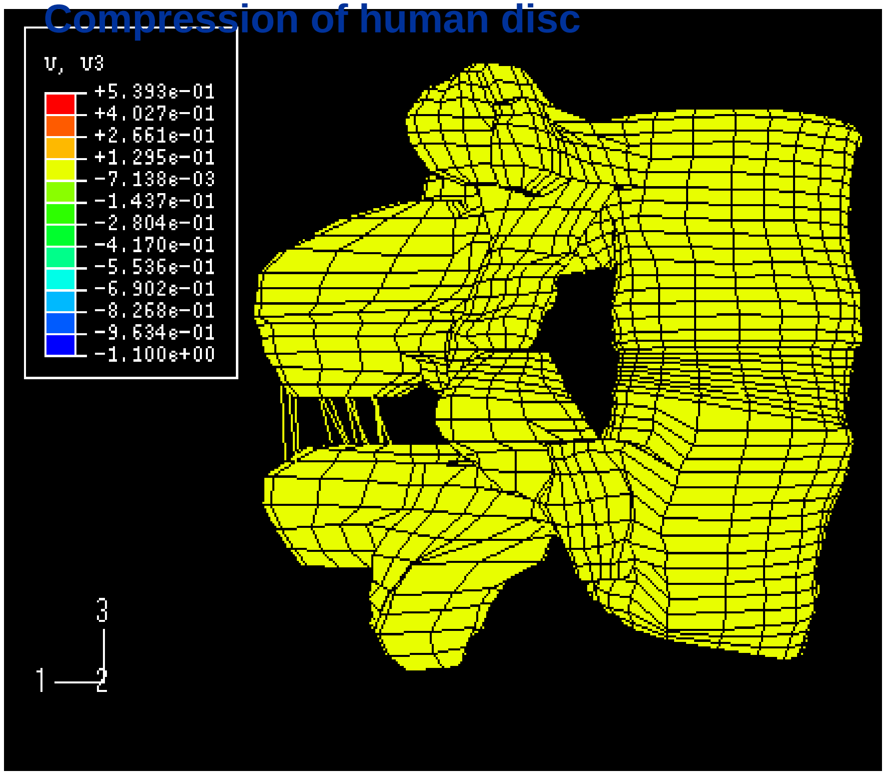


FEA of soft tissues in the human spine

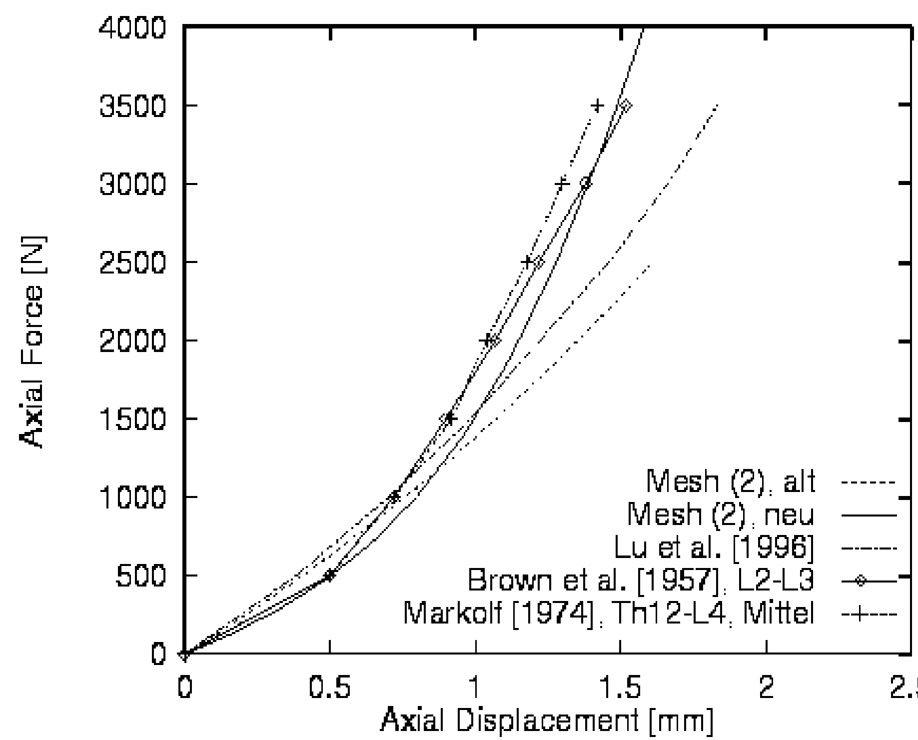
FE modelling of intervertebral discs

Comparison with experimental results from literature

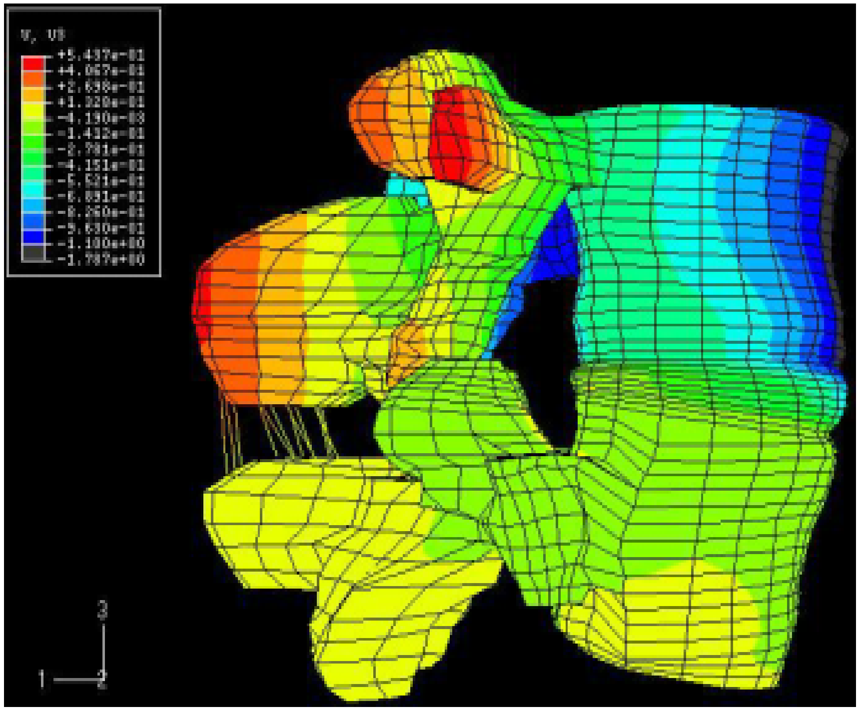
Compression of human disc

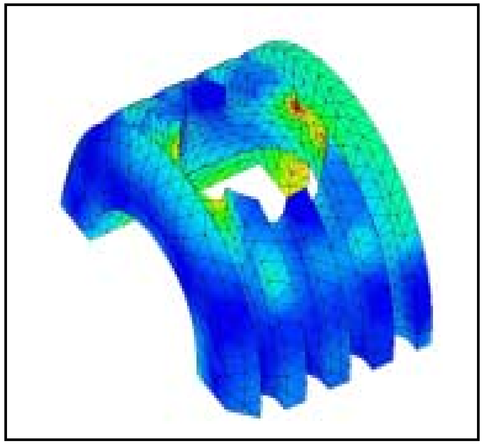


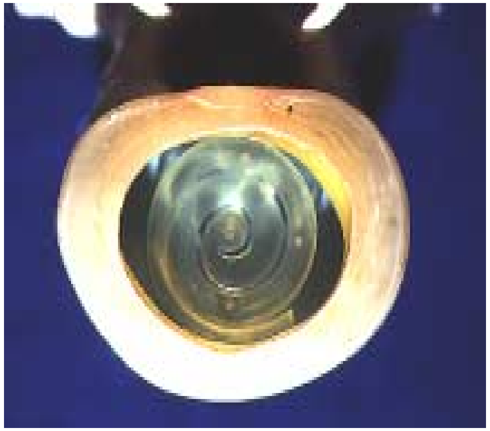
Load-displacement curves



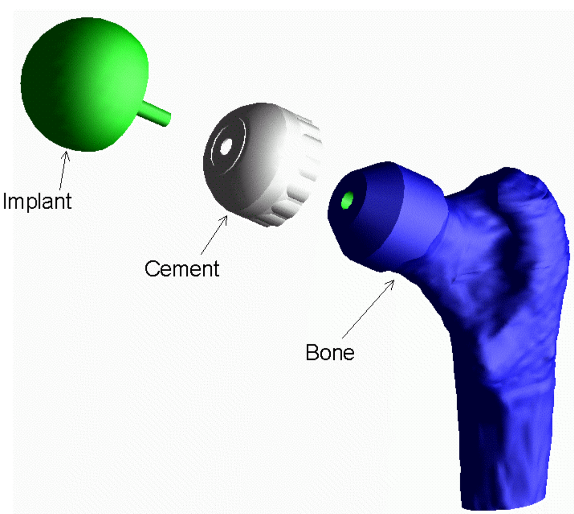
FEA of spine implants

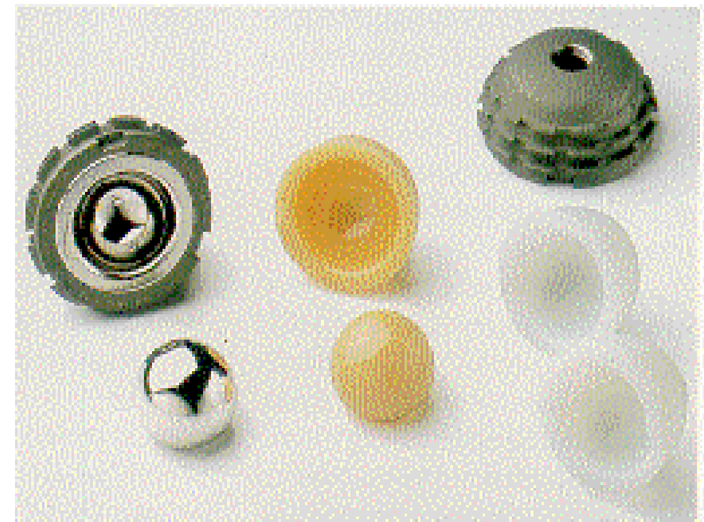






FEA of artificial hip joints





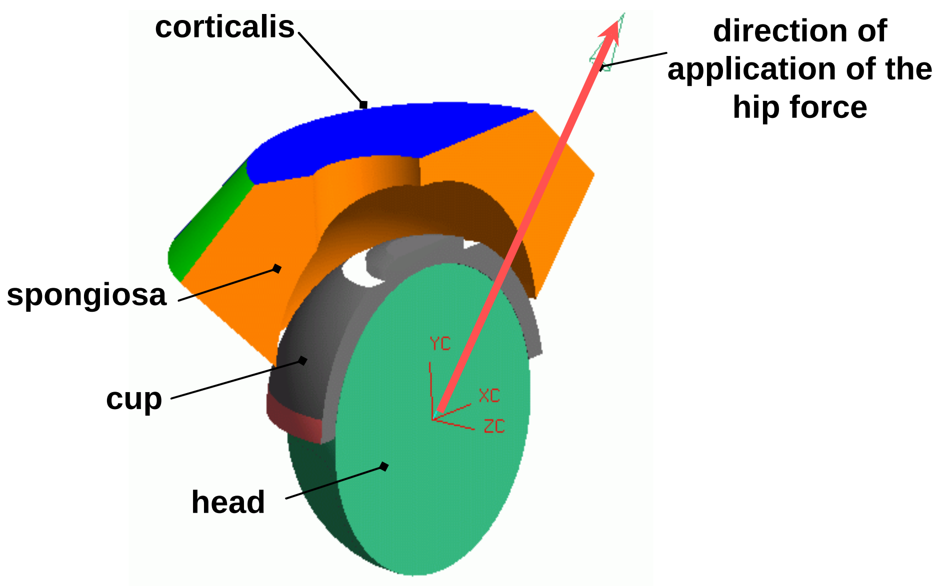
FEA of artificial hip joints

Optimization of implant geometry

- Reduced contact pressure
- Wear minimization

FE modeling

- Axisymmetric elements
- Asymmetric element deformations



Unigraphics Solutions™ — CAD

↓

MSC Patran — Pre-processing

↓

ABAQUS — Solver, Post-processing

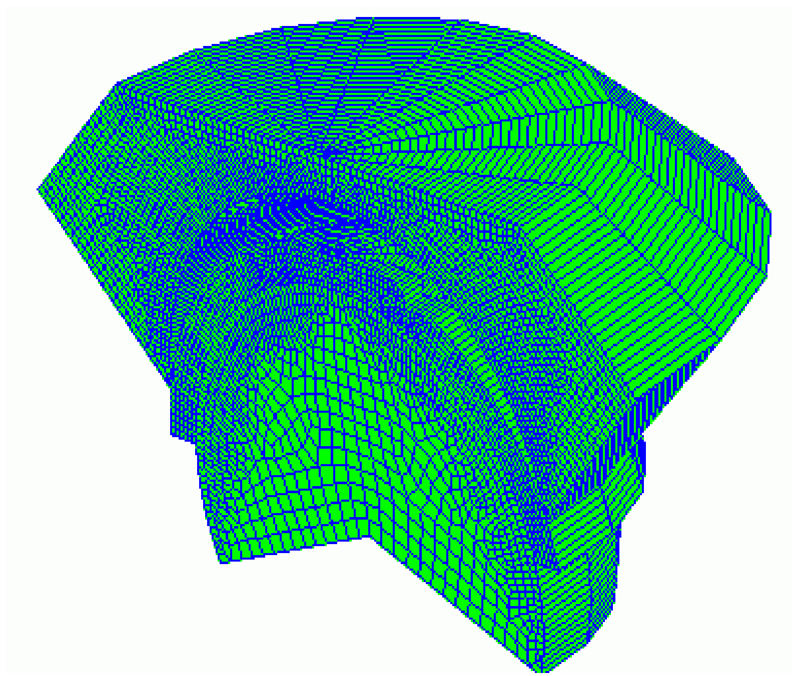
FEA of artificial hip joints

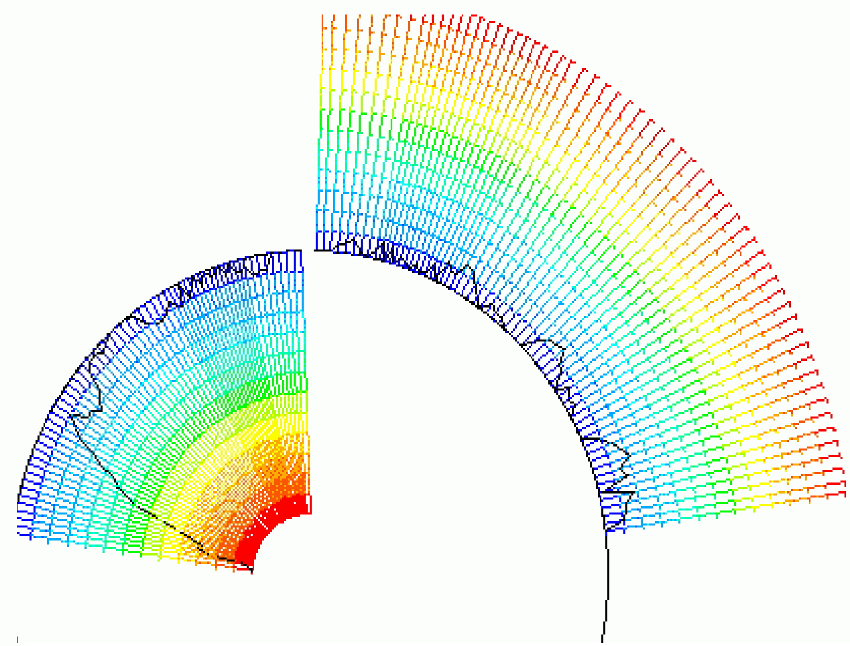
FE model

- 2D mesh with QUAD elements
- View from 0° to 180°

Results

- Radial rim clearance





FEA of artificial hip joints

Typical geometric set-up

- 60° cup rotation in REF-system
- variation of force/load conditions
  +/- 10°
- variation of geometry/replacement conditions
  +/- 15°
- total variation of structural loading
  +/- 25°