CBU 2013

Symposium on Computational Biomechanics in Ulm





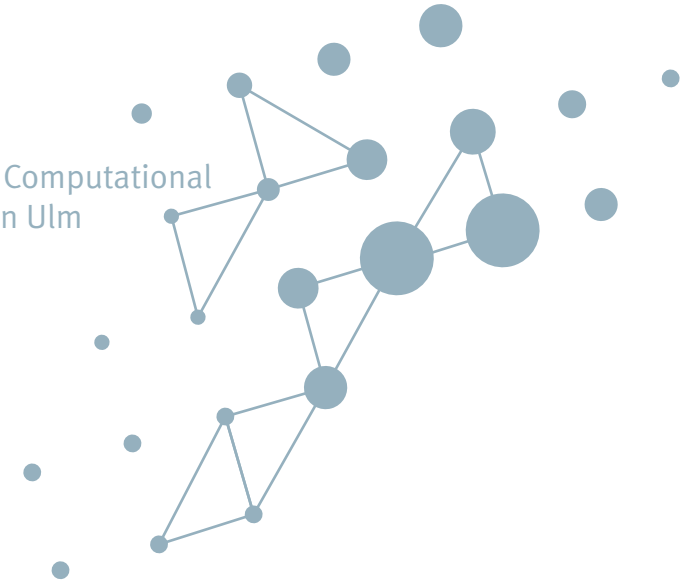


18th International Symposium on Computational Biomechanics in Ulm

May 13 - 14, 2013

CBU 2013

Symposium on Computational Biomechanics in Ulm



PROCEEDINGS AND SYMPOSIUM GUIDE

WELCOME NOTE



It is our pleasure to welcome you to the

18th International Symposium on Computational Biomechanics in Ulm.

CBU 2013

Special Feature: "Musculoskeletal Research"
Our upcoming symposium will be characterized by two special sessions, both focusing on musculoskeletal research. Both sessions will be introduced and chaired by internationally well known experts in their fields:

- Prof. Rasmussen from Aalborg University
- Prof. Röhrle from the University of Stuttgart



Prof. Rasmussen



Prof. Röhrle

"Biomechanics Spring" in Ulm 2013
This year the CBU 2013 will be held earlier than it used to be in the past. We took the opportunity to place it as a satellite to the DGfB Meeting (German Society of Biomechanics) which has been organized by Prof. Dürselen in Ulm (Neu-Ulm) this year. Don't miss the chance and have a look to that subsequent meeting just on the other side of the Danube river.

New: "Introductory Course"
Another novelty, the *Introductory Course*, is scheduled on Wednesday morning. Young researchers and not-yet experts from both meetings will get a practical introduction into standard computing tools widely used in biomechanics research.



We would like to thank all participants for their enrichment of our symposium by vivid discussions or even presentations. Special thanks to our sponsors (see next page).

We wish us two interesting days and fruitful talks here in Ulm.

The CBU 2013 Committee: Martin Geiger,
Bernd Lapatki,
Jürgen Salk,
Hans-Joachim Wilke,
Ulrich Simon

CBU 2013 COMMITTEE

The 18th International Symposium on Computational Biomechanics in Ulm is a joint collaboration organized and hosted by:

Department of Orthodontics (KFO), University Medical Center Ulm

Martin Geiger
Bernd Lapatki

Scientific Computing Centre Ulm (uzwr), University of Ulm

Ulrich Simon

Communication and Information Center (kiz), University of Ulm

Jürgen Salk

Institute of Orthopaedic Research and Biomechanics (UFB), University of Ulm

Hans-Joachim Wilke

ACKNOWLEDGEMENTS

The CBU 2013 conference cannot be financed by registration fees only. This is why financial support of industry and other related organisations is highly appreciated. The CBU 2013 Committee wishes to thank for their support to this symposium:

CADFEM
Simpleware
ulrich medical

GENERAL INFORMATION

Contact Details

E-Mail: cbu.office@uni-ulm.de
WWW: http://www.uni-ulm.de/cbu

Venue

The 18th Symposium on Computational Biomechanics in Ulm will take place at Villa Eberhardt in Ulm, Germany:

Villa Eberhardt
Heidenheimer Straße 80
89075 Ulm

On-site Registration

Upon arrival please check in at the registration desk. You are kindly requested to pay open registration fees cash on-site. There will be no possibility to pay the symposium fees on location by credit card.

Badges

Upon registration you will receive a personal badge. You are kindly requested to wear your badge when attending the scientific sessions and/or social events. At the end of the symposium, please return your badge holder to the registration desk.

Certificates

Certificates of attendance will be available at the registration desk as of Tuesday morning, 14 May.

Official Language

The language of the symposium sessions will be English.

Dress

Dress code will be informal for all symposium events.

Exhibition

The exhibition area will be on the ground floor of Villa Eberhardt next to the main symposium hall.

Wireless Internet

Internet access will be available via Eduroam service. We will also be providing guest wireless services to attendees without Eduroam support at their home institutions.

Catering

Coffee and snacks will be served during coffee breaks in the foyer at the ground floor. Lunch buffet will be available on both conference days during lunch breaks.

Social Event

In order to conclude the first day on Monday, 13 May, all attendees are invited to participate in a guided city tour and to enjoy a relaxed evening at the traditional swabian restaurant and historical brewery “Drei Kannen”.

SCIENTIFIC PROGRAM

Monday, 13 May

09:45 – 10:30	*Registration & Coffee*	
10:30 – 10:40	**Welcome Address**	
	Session I: Orthodontics & Orthopedics *Chair: Bernd Lapatki, Tim Wehner*	
10:40 – 11:00	Centre of Resistance of Constructed Tooth Models of Various Morphologies *M.E. Geiger, F. Schmidt, B.G. Lapatki*	*Martin Geiger*, Ulm
11:00 – 11:20	A New Bone Implant Concept: AMPLIFIX - First Numerical Results of Dynamic Osseointegration Simulation *L. Engelhardt, F. Niemeyer, F. Heuer, F. Trautwein, L. Bartolini, U. Simon*	*Lucas Engelhardt*, Ulm
11:20 – 11:40	Optimization of Bone Scaffold Structures to Match the Elastic Properties of Human Cortical Bone – a Numerical *A. Wolf, J. Wieding, R. Bader*	*Jan Wieding*, Rostock
11:40 – 12:00	Determining Tissue Material Properties in Fracture Calluses using ESPI and FEA *M. Steiner, L. Claes, A. Ignatius, U. Simon, T. Wehner*	*Malte Steiner*, Ulm
12:00 – 12:20	METAmorph: An Application-Agnostic Simulation Framework and its Application to Bone Healing *F. Niemeyer, L. Claes, U. Simon*	*F. Niemeyer*, Ulm
12:30 – 13:30	*Lunch*	
	Session II: Musculoskeletal Systems *Chair: John Rasmussen, Ulrich Simon*	
13:30 – 14:20	**Keynote:** Musculoskeletal modeling as a design tool *John Rasmussen*	*John Rasmussen*, Aalborg
14:20 – 14:40	Use of OpenSim and AnyBody modelling software for dynamic simulation of the human masticatory system *M. Cadova*	*Michala Cadova,* Zurich
14:40 – 15:00	Wrist protectors in snowboarding – Combined MBS and FEM simulation to analyze the wrist loading *K. Rauner, L. Karstensen, S. Lehner, F. Michel, V. Senner*	*Kilian Rauner,* Munich
15:00 – 15:20	A Finite Element Updating method for in vivo identification of elastic properties of the human aortic wall based on full field displacement measurement by 3D ultrasound speckle tracking *A. Wittek, K. Karatolios, R. Moosdorf, S. Vogt, C. Blase*	*Andreas Wittek*, Frankfurt a.M.
15:30 – 16:00	*Coffee Break*	

Session III: Muscle and Soft Tissues
Chair: Oliver Röhrle, Martin Geiger

16:00 – 16:50	**Keynote:** Multi-Scale Skeletal Muscle Modeling *O. Röhrle, T. Heidlauf, M. Sprenger, E. Ramasamy*	*Oliver Röhrle*, Stuttgart
16:50 – 17:10	A first Attempt to Model Contraction in a Straited Muscle by a Constitutive Model with Data from an In-Vivo Experiment *G. Benderoth, C. Clemen, I. Kernhof, G. Silber*	*Günther Benderoth*, Frankfurt a.M.
17:10 – 17:30	Skeletal Muscle Contraction – Experiment and Theory *M. Sturmat, C. Weichert, K. Leichsenring, T. Siebert, R. Blickhan, M. Böl*	*Maike Sturmat*, Braunschweig
17:30 – 17:50	Determination of Viscoelastic Properties of Meniscal Tissue Using both Experiment and Simulation *A.M. Seitz, F. Galbusera, C. Krais, A. Ignatius, L. Dürselen*	*Andreas Seitz,* Ulm

19:00 – 20:00 **Guided City Tour**

20:00 **Dinner**

Tuesday, 14 May

Session IV: Spine
Chair: Fabio Galbusera, Frank Niemeyer

09:00 – 09:20	Inverse Prediction of Transport Properties of the Intervertebral Disc *F. Galbusera, M. Tibiletti, M. Brayda-Bruno, C. Neidlinger-Wilke, H.-J. Wilke*	*Fabio Galbusera*, Ulm
09:20 – 09:40	Numerical Analysis to Investigate the Value of Standards for Preclinical Evaluation of Posterior Spinal Implants *L. La Barbera, T. Villa, F. Galbusera*	*Luigi La Barbera*, Milano
09:40 – 10:00	The Proposition of a Numerical Model of the Experimental Set-up for Testing Bone-Screw Interface with Use of the Foam Vertebra Model *B. Nowak*	*Bartosz Nowak*, Ulm
10:00 – 10:20	InstantSpine: An Open Source Spine Model Generator *F. Niemeyer, H. Schmitt, A. Ignatius, H.-J. Wilke*	*Frank Niemeyer* Ulm
10:30 – 11:00	*Coffee Break*	

Session V: Computing, CFD & Industrial
Chair: Martin Geiger, Jürgen Salk

11:00 – 11:20	Optimal Control in Implant Shape Design *L. Lubkoll, A. Schiela, M. Weiser, S. Zachow*	*Lars Lubkoll*, Berlin
11:20 – 11:40	Uncertainty Quantification using Stochastic Collocation Method and Application in a Model of Mineralized Turkey Leg Tendon *A. Gerisch, B. Schieche, S. Tiburtius, K. Raum, J. Lang*	*Alf Gerisch*, Darmstadt
11:40 – 12:00	CFD Approach to Bile Flow Problems Solution *A.G. Kuchumov, Y.I. Nyashin, V.A. Samartsev*	*Alex Kuchumov*, Perm
12:00 – 12:20	Patient-Specific Simulation: Bridging the Gap between Research and Clinical Application *A. Nolte*	*Alexander Nolte,* CADFEM Grafing
12:30 – 13:30	*Lunch*	

13:30 – 15:00 **Session VI: Round Table Discussion**
Panel of experts: John Rasmussen, Oliver Röhrle, Jan Wieding et al.
Chair: N.N.
Experts discussion on current issues in Computational Biomechanics
Participants are encouraged to suggest topics and actively contribute to discussions

15:00 – 15:30 *Coffee Break & Farewell*

Please also note:

Wednesday, 15 May

09:00 – 13:00 **Introductory Course: Computational Biomechanics**
Introduction to ANSYS and AnyBody
Language: German
Limited number of participants

I. ORTHODONTICS & ORTHOPEDICS

Chair: Bernd Lapatki, Tim Wehner

Centre of Resistance of Constructed Tooth Models of various Morphologies

Martin E. Geiger[*], *Falko Schmidt, Bernd G. Lapatki*

Department of Orthodontics, University of Ulm,
http://www.uniklinik-ulm.de, Germany
*) martin.geiger@uni-ulm.de

Introduction

For precise control of tooth movement the center of resistance (CRE) of each individual tooth is important. The CRE is the point through which the line of action of a pure force has to pass in order to achieve a pure translation. Correspondingly, the quantitative moment-to-force (M/F) ratio necessary for pure translation corresponds to the vertical distance between the force application point (FA point) and the CRE [1]. Previous studies did not systematically investigate the influence of the morphology of the tooth root. Solving the optimization problem for designed 3D finite-element models (FE models) allows for performing detailed parametric studies to examine the influences of full and partial root fixation via the periodontal ligament (PDL) under different load directions on the location of the CRE.

Materials and Methods

The modeling and optimization was carried out according to a previous study with FE models based on human teeth [2]. To represent the dependence of the position of the CRE on various morphological parameters systematically the current study uses designed schematic 3D FE models with (in bucco-lingual direction) extruded profiles of half of an ellipse of the tooth root (Figure 1). The PDL was modeled as a homogeneous, linear elastic layer of constant thickness. The root was fixed by the PDL (i.e., embedded) either a) completely, or b) only at its mesial plus distal sides or c) buccal plus lingual sides. The PDL`s outer surface was fixed as boundary condition. A constant force (F) of 1 N was applied to the tooth model at the estimated center of a bracket base (i.e., the FA point). As the FA point has a certain distance to the CRE, an initial pure translation of the tooth model requires precisely coordinated counter rotation moments (M). To estimate the 3D location of the CRE in our study the counter tip and counter rotation moments have been optimized for obtaining minimal residual rotations during load application. The location of the CRE was normalized relative to the root height defined as the distance from the gingival margin (0%) to the apex (100%). The objective function of the optimization procedure was the summation of reaction moments of the tooth model in the FA point. Additionally, the movement normal to the translational force had to be minimal. The optimization problem has been solved using OPTISLANG (Dynardo GmbH, Weimar) which has a NLPQLP algorithm [3] implemented, and the Abaqus FE-solver (Dassault Systèmes Simulia Corp., Rhode Island, USA).

Results

The mean value of the unwanted tipping after optimization was negligible. The displacement difference in load direction between crown and apex was at least by a factor 1000 smaller

than the translation for the whole tooth. The displacement difference between the mesio-distal contact points of the model teeth was at least by a factor 10 000 smaller.
In the distal load case (black squares in Fig. 1) the CRE was located at 49,6% of the root height for the completely embedded root as well as for embedding at mesial plus distal root surfaces only. This value was relatively constant when profile lengths were varied between 5, 10, 15 and 20 mm. Distal loading and buccal plus lingual root embedding resulted in a CRE location at 42.4% of the root height for all profile lengths.

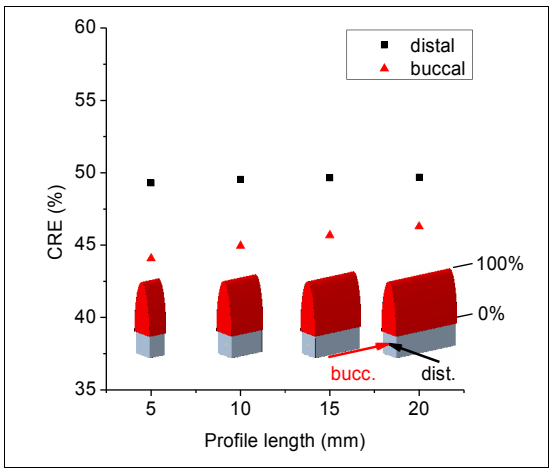


Figure 1: Designed tooth models with calculated CRE for distal (black squares) and buccal load (red triangles). The distal load itself is represented by a black arrow and the buccal load by a red one. These tooth models have a root completely surrounded and fixed respectively via the PDL (red). Design variable is the length of the profile (5, 10, 15 and 20 mm) in buccal direction. The root height of the tooth model is 14 mm with a crown height of 6 mm.

In the buccal load case (red triangles in Fig. 1) the CRE of the completely embedded root was located at 44.1% of the root height, i.e. ca. 6% more gingivally than in the distal load case. Moreover, in this load case the CRE location showed a shift of 2.2% in apical direction when the profile length was increased from 5 mm to 20 mm (Fig. 1). Additionally, under buccal load the mesially plus distally embedded root showed a CRE location of 55.2%. In contrast, the buccal plus lingual embedding of the root led to a CRE of 42.4%.

Discussion

The effects of the root morphology are dominated by the root projection on the area perpendicular to the translation. In the distally directed load this was shown by the constant CRE location (ca. 50% corresponding to the centroid of a rectangular surface) for the completely and mesially plus distally embedded root. Correspondingly, in the buccally directed load this conclusion is supported by the CRE of 42.4% (corresponding to location of the centroid of half of an ellipse) for the buccally plus lingually embedded root. These results may improve understanding of morphologic influencing factors on the location of the CRE.

References

[1] Hocevar RA. Understanding, planning, and managing tooth movement: orthodontic force system theory. 1981, Am J Orthod. **80**(5),457–77.

[2] Geiger ME, Lapatki BG. Bestimmung des Widerstandszentrums einzelner Zähne auf Basis von 2D- und 3D-röntgenologischen Daten. 2013, J Orofac Orthop. accepted.

[3] Schittkowski K. A robust implementation of a sequential quadratic programming algorithm with successive error restoration. 2011, Optim Lett. **5**(2),283–96.

A New Bone Implant Concept: AMPLIFIX
—
First Numerical Results of Dynamic Osseointegration Simulation

L. Engelhardt[a], *F. Niemeyer*[a], *F. Heuer*[b], *F. Trautwein*[b], *L. Bartolini*[a], *U. Simon**[a]

a) Scientific Computing Centre Ulm, Ulm University, www.uni-ulm.de/uzwr, Germany
b) ACES Ingenieurgesellschaft mbH, Filderstadt, www.acesgmbh.de, Germany
*) ulrich.simon@uni-ulm.de (corresponding author)

Introduction

Spinal fusion implants (cages), which are used after disc replacement, should achieve sufficient primary stability as well as long term stability. However, both objectives are often in conflict to each other: Stiff implants are able to preserve the intervertebral distance against musculoskeletal loadings, but the mechanical stimulus around them is often too low to stimulate a sufficient bony fusion between the vertebral bodies.

The new AMPLIFIX concept consists of a relatively stiff central body surrounded by lamellas, which are in an alternating contact to the top and bottom disc plate. This design is able to amplify the very small movements of the lamella into sufficiently high shear strains in the interlamellar spaces. With this concept we hope to get bone formation and bony fusion being stimulated around the beside

It was the aim of this project to evaluate the new concept using a numerical simulation before any experimental study. Therefore, an existing fracture healing model [1] should be adopted to simulate the bone fusion process. Specifically we wanted to know if the strain amplification concept would really help to let the bone tissue bridge the gap in situations where it would not for simple implants without any lamella. In more detailed we also wanted to evaluate the influence of the interlamellar gap size onto the bone formation.

Materials and Methods

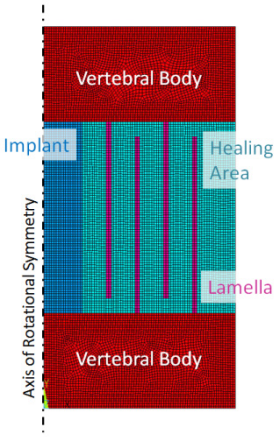


Figure 1: Axisymmetric FE bone fusion model including the AMPLIFIX lamella concept.

Briefly, the existing callus healing model [1] consists of an iterative simulation over time. In each iteration step the FE model calculated the local mechanical stimuli (dilatational and distortional strain). These stimuli together with the current tissue composition and local blood

supply were used as an input for a fuzzy controller. Tissue differentiation and development was simulated by a set of fuzzy rules representing intramembranous and endochondral ossification, chondrogenesis, calcification, tissue destruction due to overloading as well as revascularization. The fusion model consists of a stiff central part of the implant (titanium) with four lamella connected to either the top or bottom vertebral end plate in alternating order. The area between the vertebral bodies including the interlamellar spaces is defined to be the healing area, where bone formation can potentially be predicted; it starts as soft connective tissue. Mechanical stimuli were calculated based on a pure axial compressive load of 450 N.

Results

Bone formation started at both end plates and was growing into the central area. In the model with the standard implant without any lamella bone formation stopped before the half of the gap was bridged. In contrast, all AMPLIFIX implants predicted bone fusion. The area of new bone formation was maximal in the model with the larger interlamellar spaces (Fig. 2, right).

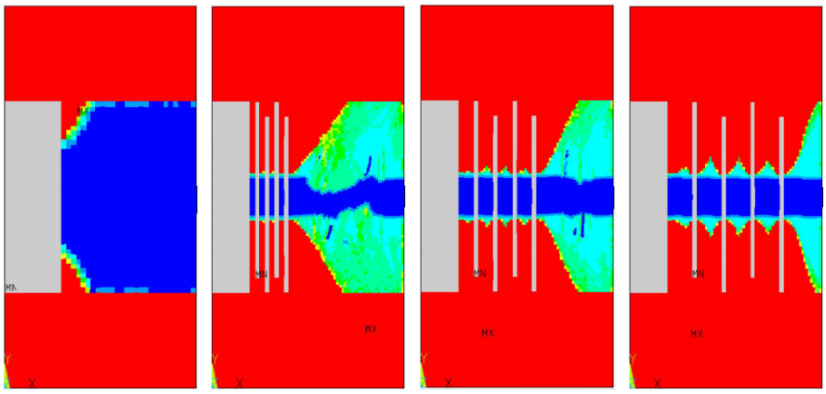


Figure 2: Simulation results: Bone formation (from 0% = dark blue to 100% = red) after 130 iteration steps in (from left to right) a standard implant, AMPLIFIX implants with small to large interlamellar spaces.

Discussion

Assuming the validity of the model these simulations corroborated the AMPLIFIX principle. However, further experimental studies are necessary. In addition, the results of the simulations including the parameter variations will be very helpful for the detailed planning of the experimental implant designs.

Acknowledgments

The work was supported by the German government (*Bundesministerium für Bildung und Forschung,* BMBF, FKZ13EZ1207B).

References

[1] Simon, U., Augat, P., Utz, M., and Claes, L. (2011) "A numerical model of the fracture healing process that describes tissue development and revascularization", Computer Methods in Biomechanics and Biomedical Engineering 14(1), 79-93.

Optimization of bone scaffold structures to match the elastic properties of human cortical bone – a numerical approach

Andreas Wolf[a], *Jan Wieding*[a*], *Rainer Bader*[a]

a) Biomechanics and Implant Technology Research Laboratory, Department of Orthopaedics, University Medicine Rostock, Germany, www.forbiomit.med.uni-rostock.de,
*) jan.wieding@med.uni-rostock.de

Introduction

Scaffolds shall satisfy various requirements for regeneration within large segmental bone defects, which can be caused by tumour, severe fracture, infections or implant loosening. In order to be qualified as a bone scaffold, mechanical as well as biological aspects have to be considered. On the one hand, scaffolds must exhibit a sufficient mechanical stability for load-bearing areas. On the other hand an open-porous structure with an interconnecting pore system has to be implemented for sufficient bone ingrowth [1]. Furthermore, the elastic properties shall be adapted to those of the surrounding bone tissue in order to avoid stress shielding [2].

Nowadays, additive manufacturing (AM) techniques like electron beam melting (EBM) or selective laser melting (SLM) offer a wide range of structural design possibilities and can subsequently cope with the requirements for the scaffolds [3]. As there are nearly no limits for the fabrication, the mechanical properties has to be optimized for the application area [4-6].

Therefore, the aim of our study was to utilize an optimization approach to fit the mechanical properties of cortical bone and offer an open-porous structure for sufficient bone ingrowth.

Materials and Methods

In general, two different open-porous titanium scaffolds with varying structural designs have been investigated, as a linear pattern of one unit-cell (Figure 1).



Figure 1: Parameterized models for the optimization approach consisting of 3x3x3 unit-cells. Cubic structure (left) and diagonal orientated struts (right).

All designs have been generated as fully parameterized models with the Finite Element software package Abaqus (version 6.12-EF, Dassault Systèmes, Vélizy-Villacoublay, France). Struts have been modeled as beams featuring an analytical cross-section. Each segment was meshed with at least 15 elements. Parameters for the structural design were strut diameter and size of the unit-cell. These parameters were implemented into the software Isight (version 5.7,

Dassault Systèmes, Vélizy-Villacoublay, France), which is fully compatible with Abaqus and offers a range of design exploration and optimization algorithms.
A *Sequential Quadratic Programming* (SQP) procedure was used as optimization algorithm. Under uniaxial compression the parameters have been optimized to receive a structural modulus of 15.0 GPa as a target value. Highest possible pore size was a further side condition of the optimization procedure. Parameters have been varied in the range of 0.2 and 1.5 mm for the strut diameter and 0.4 and 5.0 mm for the size of the uni-cell.
Furthermore, bending and torsional testing was additionally applied to scaffolds with optimized structure as derived from compression testing.

Results

Table 1 summarizes the results of the optimization procedure for the structural parameters (strut diameter, size of the unit-cell and pore size) as well as for the mechanical properties (elastic, bending and torsional modulus). Although both designs have been optimized for the elastic modulus under compression (i.e. 15.0 GPa), their behavior under bending and torsion is totally different.

Table 1: Structural parameters and mechanical properties for the two investigated scaffold designs (cubic vs. diagonal) after optimization procedure

Design	*Strut diameter [mm]*	*Size of unit-cell [mm]*	*Pore size [µm]*	*Elastic modulus [GPa]*	*Bending modulus [GPa]*	*Torsional modulus [GPa]*
Cubic	0.25	0.80	550	**15.0**	8.0	0.8
Diagonal	0.76	2.11	420	**15.0**	19.0	24.1

Discussion

In this study we could show the accurate adaptation of the mechanical properties for titanium open-porous scaffolds to cortical bone by using a parametric optimization approach. Especially for bone scaffolds their fabrication with defined mechanical properties is required [5,6]. Hence, optimization tools should always be used to ensure the best possible adaptation especially since the manufacturing process will be able to produce even components with complex geometry. Furthermore, our numerical results clearly demonstrate that not only uniaxial testing but also bending and torsion should be investigated in order to characterize the mechanical properties.
For further application the validation of the numerical data by experimental testing has to be performed. Furthermore, *in vitro* testing should be conducted to show the ability of bone growth into the open-porous structure.

References

[1] Karageorgiou, V. et al. 2005, Biomaterials **26** (27), p. 5474-91
[2] Niinomi, M. et al. 2011, Int J Biomater **2011**, p. 1-10
[3] Wieding, J. et al. 2012, Mater **5** (8), p. 1336-47
[4] Almeida, H. D. et al. 2010, Med Eng & Phys **32** (7), p. 775-82
[5] Adachi, T. et al. 2006, Biomaterials **27** (21), p. 3964-72
[6] Hollister, S. J. et al. 2002, Biomaterials **23** (20), p. 4095-103

Determining Tissue Material Properties in Fracture Calluses using ESPI and FEA

M. Steiner[a*], *L. Claes*[a], *A. Ignatius*[a], *U. Simon*[b], *T. Wehner*[a]

a) Institute of Orthopaedic Research and Biomechanics, Center of Musculoskeletal Research, Ulm University, Helmholtzstrasse 14, 89081 Ulm, Germany
b) Scientific Computing Centre Ulm, Ulm University, Helmholtzstrasse 20, 89081 Ulm, Germany
*)malte.steiner@uni-ulm.de

Introduction

Numerical simulation of fracture healing processes is dependent on reliable material properties of the involved tissues. Published material parameters values show a wide range [1], depending on the heterogeneity of the material and on the determination methods (e.g. nanoindentation [2-3] or material testing [4]). Electronic speckle pattern interferometry (ESPI) is able to detect local mechanical deformation in high resolutions from in vitro specimens [5-7]. This study presents a method to obtain material parameters for the ovine fracture callus tissues by comparison of ESPI deformation data presented by Bottlang et al. [7], with finite element analysis (FEA) data of the same setup. The target conformity between FE and ESPI data is achieved by applying an optimization procedure to material parameters of the involved tissues.

Materials and Methods

We examined two mid-sagittal slices of ovine fracture callus harvested eight weeks after tibial osteotomy having been stabilized by an external fixator, described in detail by Bottlang et al. [7]. Slices were subjected to unconfined quasi static axial compression of 5N. An ESPI sensor captured displacements from incremental measurement points over a region of interest (ROI) on the surfaces of the callus slices. These in vitro experiments were simulated using finite element (FE) models in ANSYS (ANSYS Inc.). Strain data was used for automatic tissue identification for each element into the three tissues woven bone, cartilage and connective tissue. The cortices were identified manually using contact radiography and histology data. In a two-step parameter analysis procedure, we used the optislang software package (DYNARDO GmbH) to determine the material properties. Therefore in the first step, a parameter sensitivity analysis quantified the influence of single tissues on the mechanical behavior of the callus specimens. In the second step, material properties (i.e. Young's moduli and Poisson's ratios) for the most dominant material of each callus specimen were determined through a parameter sampling procedure minimizing the mean local deviations between the simulated (FE) and measured (ESPI) equivalent element strains ε:

$$G(\varepsilon.FE_j) = \Delta\bar{\varepsilon} = \frac{1}{m}\sum_{j=1}^{m}\left|\varepsilon.FE_j - \varepsilon.ESPI_j\right|, \qquad (1)$$

where j is the element index and m is the total number of elements.

Results

In the first step, for the specimen with higher axial stiffness woven bone was quantitatively detected as most influencing tissue, whereas for the specimen with lower axial stiffness, cartilage was most determining. In the second step the median values for Young's modulus and Poisson's ratio of woven bone were found to be 1881 MPa and 0.327, and for cartilage 16 MPa and 0.303, respectively. Figure 1 shows the comparison between ESPI-results and FE results.

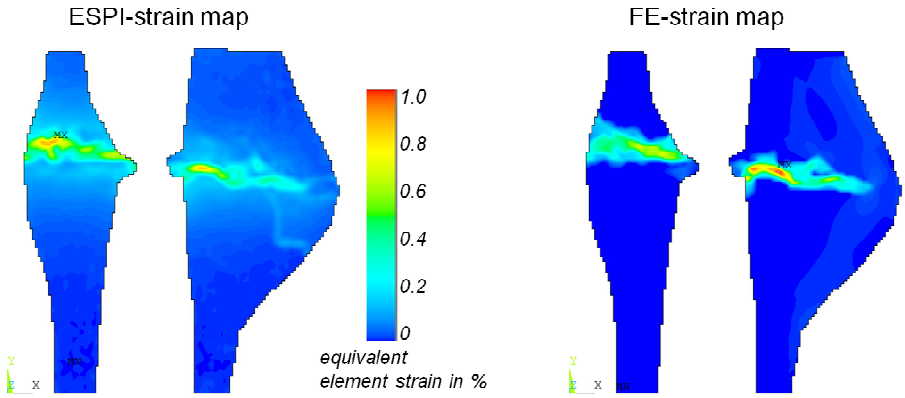


Figure 1: Exemplary strain contour plots calculated from ESPI experimental (left) and FE simulation data (right) after determination of reasonable material properties. Adapted from Steiner et al. [8].

Discussion

The advantage of combining ESPI experimental data with FE simulation of the same setup is to determine material properties of tissues still embedded in the fracture callus structure in a high local resolution and nevertheless comprising the global condition of the whole specimen. The results for the material properties are able to refine pre-assumed ranges from the literature [1]. For FE simulation of fracture healing processes, these callus-specific values might be more appropriate than previously applied more general values. There is huge potential for this method since it allows assigning reliable material properties to numerical models of different biological processes.

References

[1] Isaksson, H. et al. 2009, J Biomech 42(5), p. 555-564.
[2] Leong, P. L. et al. 2008, Acta Biomaterialia 4(5), p. 1569-1575.
[3] Augat, P. et al. 1998, J Orthop Res 16(4), p. 475-481.
[4] Hori, R. Y. et al. 1982, J Biomed Mater Res 16(6), p. 911-927
[5] Kirkpatrick, S. J. et al. 1998, J Biomed Mater Res 39(3), p. 373-379
[6] Shahar, R. et al. 2007, J Biomech 40(2), p. 252-264
[7] Bottlang, M. et al. 2008, J Biomech 41(3), p. 701-705
[8] Steiner, M. et al. 2012, Med Eng Phys 34(10), p. 1521-1525.

METAmorph: An Application-Agnostic Simulation Framework and Its Application to Bone Healing

Frank Niemeyer [a*], *Lutz Claes*[b], *Ulrich Simon*[a]

a) Scientific Computing Centre Ulm (uzwr), www.uni-ulm.de/uzwr, Germany
b) Institute for Orthopaedic Research and Biomechanics (UFB), www.biomechanics.de, Germany
*) frank.niemeyer@uni-ulm.de

Computational Science and Software Engineering

Let's face it: The quality of most custom-written software for research purposes leaves a lot to be desired. Approaching deadlines encourage a "quick-and-dirty" style of development, which at first does not seem to be a big deal, as most software for research is intended to be short-lived and to be replaced by entirely new and different implementations soon anyway. Yet, this pious hope rarely becomes true and most "throw-away" software does not get thrown away at all, because in reality hardly anyone can afford to start from scratch. Thus, over time, the initial software artifact continues to evolve organically until it has become a completely unmaintainable mess. We are no less guilty of producing low-quality software than anybody else who has ever written software for research. The implementation of our fracture healing simulation, essentially a conglomerate of APDL, plain C and a few shell scripts, exhibits all those problems mentioned before [1]. A few years ago, we started to extend our model to cover distraction osteogenesis as well. Instead of again bolting some additional balconies to our house of cards, we took the chance to implement our simulation anew from scratch, but this time by using clean modularization and with scalability and extensibility in mind.

Mile-High Overview of METAmorph

Simulations that solve an IVP (initial value problem) iteratively all share a common structure: First, a data structure describing the initial state of the simulated system is initialized. This data structure is then repeatedly pushed through a series of state transformations until some termination criterion is met. Realizing this underlying pattern allowed us to build a unifying software infrastructure, which can serve as a basis for modularized, maintainable bone healing simulations of any kind. To assemble custom simulations, METAmorph utilizes a specialized, very light-weight service locator to find and instantiate user-supplied components.

Building Simulations Using METAmorph

Implementing a simulation using our framework always involves the definition of a state model, which represents the simulation domain (Fig. 1). The next step is concerned with the creation of appropriate transformation plugins. Applied to a given state representation, those transformations might either modify the current state or yield a new state object. Each transformation plugin should perform one single, very specific task and should be as independent as possible from other components to ensure future extensibility. A textual configuration file specifies which state type and what transformation plugins should be instantiated and in

which order and how often they should be applied to the initial state. METAmorph can also host Python scripts to model more complicated control flows (e.g. switching between different transformations based on some condition).

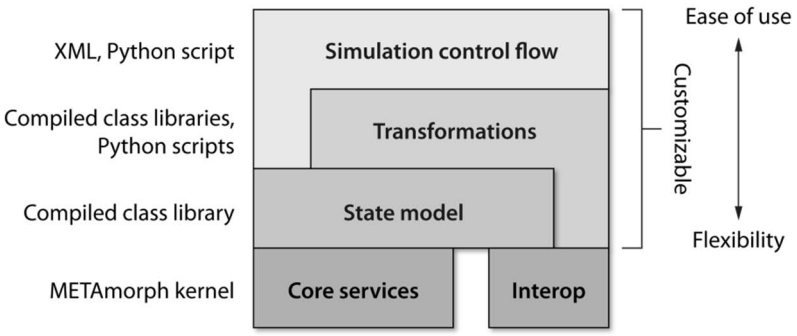


Figure 1: Layered architecture of METAmorph

Simulating Bone Healing

Porting our bone healing simulation to METAmorph proved to be straightforward for the most part. The APDL-part of the simulation could be readily translated to corresponding transformation modules written in C# and F#, albeit now featuring proper modularization and encapsulation, not to mention static type checking and performance enhancements by several orders of magnitude. We converted the fuzzy logic controller, originally a stand-alone C binary, to a native static library, which we now call from a managed wrapper. If we eventually decided to move to a different fuzzy logic implementation, we could simply replace the native C library with a new solution. Importing FE models and interacting (e.g. running FEAs, importing results) with different FE packages, in particular ANSYS, turned out to be more challenging. We therefore introduced an additional abstraction layer for translating between an internal, completely generic representation of an FE model and the specific representations required by external FE packages. This way we can largely circumvent vendor lock-in; switching to a different FE package or library merely requires a corresponding translation plugin. The rest of the simulation remains unaffected. In the end, we were able to reproduce the results of the original implementation of our bone healing model with the new implementation without any noticeable differences.

Did It Pay Off?

Unfortunately and contrary to our initial plan, we did *not* get the chance to implement the distraction osteogenesis model on top of METAmorph and the already existing bone healing plugins. Implementing distraction osteogenesis would require some further extensions, in particular to the ANSYS interoperability component, but deadlines pressed us to go with the ugly, yet quick solution once more, despite knowing that we will have to pay for this decision rather sooner than later. While anticipating future requirements may be hard, even a little bit more planning can mean a day and night difference in terms of software quality compared to pure ad hoc solutions, as we learned when re-implementing our bone healing simulation. METAmorph will be available for download on our homepage (http://www.uni-ulm.de/uzwr).

Acknowledgments

This work was supported by the German Research Foundation (DFG CL77/14).

References

[1] Simon, U. et al. 2010, Comput Method Biomec **14**(1), p. 79 – 93

II. MUSCULOSKELETAL SYSTEMS

Chair: John Rasmussen, Ulrich Simon

Musculoskeletal Modeling as a Design Tool

John Rasmussen[a*]

a)Aalborg University, www.m-tech.aau.dk, Denmark
*) jr@m-tech.aau.dk

Introduction

Musculoskeletal modeling has come a long way in the past decade from simple models comprising a few muscles to anatomically realistic models in three dimensions performing complex motion patterns. In this sense, the technology's evolution is similar to what has happened to other simulation technologies such as finite element simulation for field problems, albeit a few decades later.

Initially, musculoskeletal simulation appears to be a technology that provides information about the internal working of the human body, properties that are difficult or impossible to measure in-vivo [1], and as such they are important research tools.

However, in a longer perspective, Computer-Aided Engineering (CAE) technologies have attained their full potential only when they are used to design and improve products. Similarly,musculoskeletal modeling is likely to make its principal impact when the information they provide about the human body will allow the development of better products in the widest sense of the word, i.e. better treatments, better medical devices, better industrial products, better workplaces, etc.

This paper reviews the first steps in this direction for musculoskeletal simulation by way of examples and attempts to draw conclusions pertaining to the future development of the field.

Materials and Methods

Previous and current examples of the use of musculoskeletal modeling as CAE tools are reviewed and some of the key developments that enable this use and possible remaining shortcomings are pointed out. Based on this information, conclusions are made on remaining technical and scientific challenges.All examples are made with the AnyBody Modeling System [2].

Since possible applications are extremely diverse, we shall address the targets of development on a scale ranging from inside to outside-the-body products, where medical applications such a surgical planning or design of joint prostheses are inside applications. Development of orthotics is close to the body, while a wheelchair, a bicycle or a workplace would be further from the body.

Results

Inside-the-body applications

Minimally Invasive Spine Surgery (MISS) is a technique that is gradually replacing Traditional Open Spine Surgery (TOSS) for rigidification of spinal joints and implantation of disk replacements. The approach involved in TOSS sacrifices the vast majority of muscle fascicles attached to the involved vertebrae. The precise biomechanical consequence of this has not been fully recognized but was investigated by Rasmussen and Rasmussen [3] using a

model of normal gait (Fig. 1). The model revealed that TOSS, due to the sacrificed muscles, increases the load on the remaining muscular system compared with MISS, in particular for the higher approaches, i.e. L3L4, while the there was no difference in the end result for the L5S1 case.

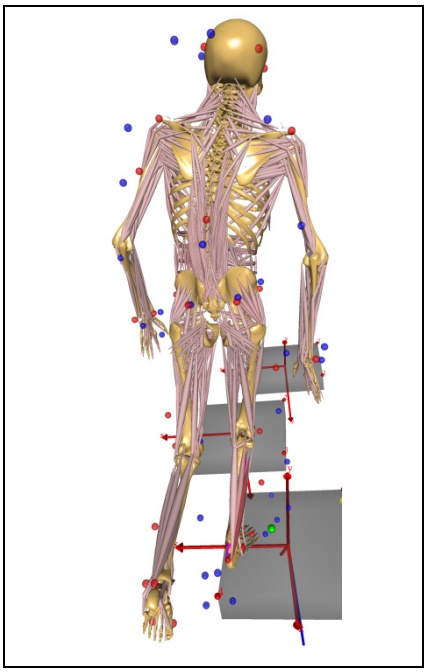


Figure 1: Full-body gait model for the MISS investigation.

This investigation highlighted the need for musculoskeletal models that match the investigated case in terms of muscular resolution. In the present case, the fascicles of the lumbar vertebrae were modeled according to de Zee et al. [4], which allowed the consideration of the effect of sacrificing uni-articular, bi-articular and multi-articular fascicles of the muscles in the involved region, which is the reason for the complex progression of the effect of the procedure to vertebrae beyond the near surgical location.

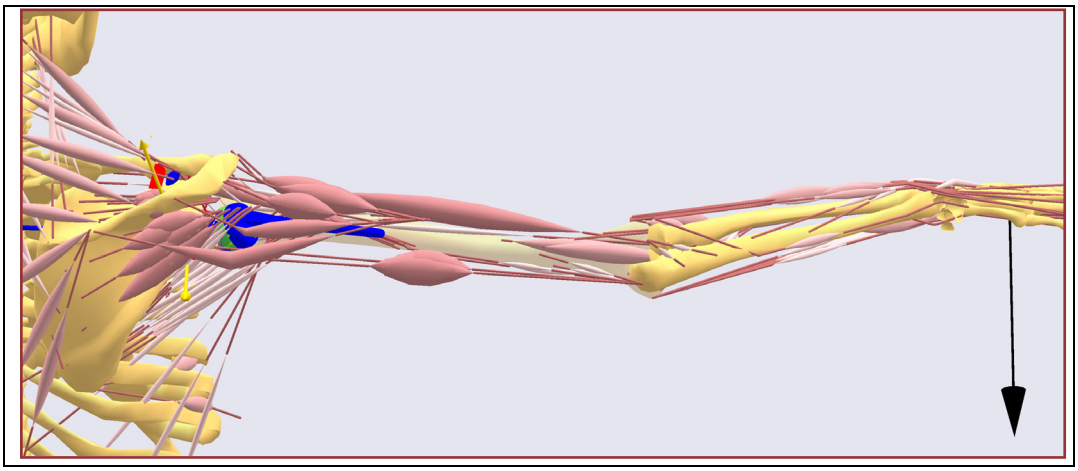


Figure 2: Model of an implanted reverse shoulder prosthesis. The model performs abduction against the self-weight of the arm and an additional load of 1 kg attached to the hand.

Reverse shoulder prostheses (Fig. 2) are a salvage solution for end-stage cuff tear patients with poor biomechanical function. Patients typically have problems with activities of daily living due to lack of muscle strength, i.e. the rotator cuff muscles supraspinatus, infraspinatus, subscapularis and teres minor are deficient. The advantage of a reverse prosthesis is that it enables a larger moment arm for the remaining abductors, primarily the deltoid. However, the biomechanical function of the shoulder joint is complex and the prosthesis design as well as the implantation involves the setting of several parameters that influence the patient's function. To fully understand this set of problems, Rasmussen et al. [5] created a parametric model of the DePuy DELTA XTEND® reverse shoulder prosthesis and investigated the importance of the location of the proximal part of the prosthesis called the glenosphere. During implantation, the glenosphere can be placed more or less laterally (lateralization) and more or less inferiorly (inferiorization) on the scapula.

The model revealed that the patient's biomechanical outcome, i.e. the ability to abduct the arm, depends much on the inferiorization and less on the lateralization, where inferiorization of more than 6 mm compared to the anatomical position makes it possible for the model to perform the full 90 degree abduction.

In terms of musculoskeletal simulation, the finding from this work was that the ability to use parametric models can provide valuable insight for designers of orthopaedic implants and orthopedics in general. Even if the model does not represent a particular patient, the trends revealed by the parameter variations provide experience that may be valid across patients.

Near-body applications

The field of orthotics covers a comprehensive spectrum of products that guide, support and reinforce particular elements of the biomechanics of the human body, ranging from sports tape over shoe insoles to joint braces and exoskeletons. The common denominator of these products is that they represent attempts to influence the biomechanics of the body.

One obvious use of orthotics is to support weak or injured joints. The forces these devices can apply from outside the body are usually small compared to the internal forces they seek to influence. However, overuse injuries can occur due to small loads if these occur often enough, and therefore the relatively small force offset of an orthosis may be sufficient to restore homoestasis in a joint.

A common type of orthotocs is the knee brace, which is successful in many sports contexts for providing stability or relieving sore tissues. An obvious use is for the initial stages of knee osteoarthritis, where the combination of force and movement in the knee causes pain and further degenerates the interacting surfaces.

It is difficult to obtain a directly unloading effect of the knee by a joint-separating force from an orthosis. The reason is that the soft tissues, to which the orthosis is anchored on the outside, do not provide enough stiffness. However, it is quite easy to design an orthosis that provides a passive knee moment. If we furthermore observe that the compressive force in the knee is due to muscle forces in addition to the body weight, then we have the opportunity to influence the knee compressive force by augmentation of the muscle-generated moment by an elastic moment from the orthosis.

Fig. 3 shows a squatting model starting with a deep knee bend of 120 degrees and ending in a standing posture. The simulated knee compression forces are shown for the baseline situation and for the case of equipping the right knee with an elastic brace providing 0.35 Nm/degree elastic reaction moment against flexion. As evident from the figure, the knee brace reduces the knee reaction force by roughly 25% and may thus be a suitable orthosis for patients with initial-stage knee osteoarthritis.

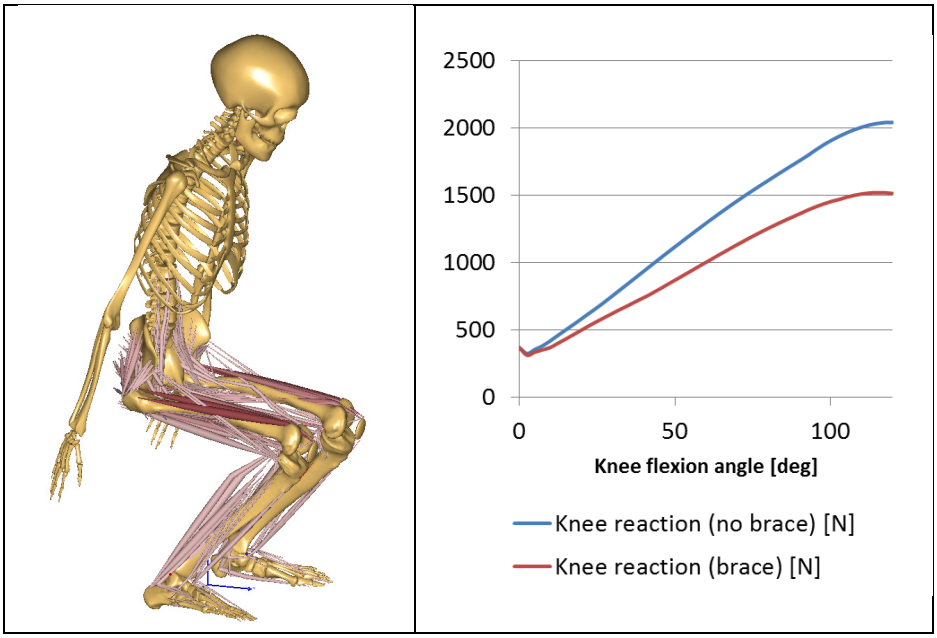


Figure 3: The squatting model (left) and the computed knee joint reactions without and with the knee brace.

Outside-the-body applications

In this category are applications such as design of sports equipment, exercise machines, tools, furniture and workplaces. The latter category of applications has also traditionally been targeted by digital manikin software. These are relatively simple and mostly kinematic models that can be scaled anthropometrically to check size compatibility and accessibility. Well-known products in this category are Jack®, Delmia® and Ramsis®.

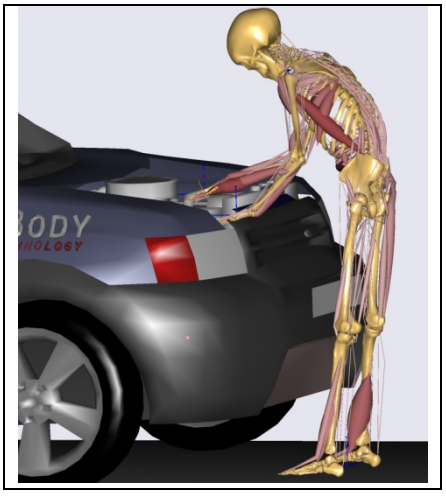


Figure 4: Investigating the ergonomics of a mounting operation using a musculoskeletal model.

Due to their relative simplicity, digital manikins are not suited for kinetic investigations such as assessment of joint forces or muscular efforts in different working situations. Fig. 4 shows a typical such example of a mounting operation in the engine compartment of a vehicle. Musculoskeletal models can handle such cases but will need to function in the same context as digital manikins do, i.e. inside CAD systems in which the workplaces, factory layouts, assembly lines and such can be modeled. Furthermore, the ability to easily handle closed kinematic chains is crucial to this type of applications because most operations involve multiple limbs in simultaneous contact with the environment and work pieces.

Discussion

From the aforementioned results, a number of conclusions can be derived pertaining to different properties that may significantly influence the future use of musculoskeletal modeling:

- Anatomical fidelity is important and the development of more detailed models is likely an open-ended process. Especially for planning of surgical procedures and design of joint and trauma implants, the detailed modeling of the immediate anatomical surroundings is crucial.
- As with any modeling for design purposes, the ability to create parametric models and thereby systematically investigate a large variety of design options is important to the success of the musculoskeletal models. The importance of parametric models was realized by the Computer-Aided Design (CAD) community already in the 1980'ies and caused a paradigm shift in CAD technology, which extended the use of CAD from early adopters like the aerospace and automotive sectors to the industry in general.
- Model must be extensible to allow addition of internal as well as external devices like the knee brace to the model. Models that are closed or can only be modified with difficulty have little value for many investigations.
- Musculoskeletal models must work in the context of the investigation they are being used for. In terms of ergonomic product design and workplace design, the integration of the technology into CAD environments is necessary to make the technology available to designers. The related technology of digital manikins has come a long way in this respect and much can be learned from their integration into CAD environments.

With these considerations, musculoskeletal modeling should come into widespread use and open an entirely new range of applications of Computer-Aided Design.

References

[1] Erdemir, A. et al. 2007, Clin.Biom.**22**, p. 131-54
[2] Damsgaard, M. et al. 2006. SimulationModelling Theory and Practice, **14**(8), p. 1100-11.
[3] Rasmussen, J. and Rasmussen, S. Proc. of the XXIV Congress of the ISB (To appear).
[4] de Zee et al. 2007. J. Biomch, **40**(6), p. 1219-27.
[5] Rasmussen, J. at al. 2009. 17th Annual Symposium on Computational Methods in Orthopaedic Biomechanics, Las-Vegas, Nevada.

Use of OpenSim and AnyBody modelling software for dynamic simulation of the human masticatory system

Michala Cadova[a,b*]

a) Clinic of Masticatory Disorders, Removable Prosthodontics, Geriatric and Special Care Dentistry, University of Zürich, http://www.zzm.uzh.ch, Switzerland
b) Laboratory of Biomechanics, Faculty of Mechanical Engineering, Czech Technical University in Prague, http://www.biomechanics.cz, Czech Republic
*) Michala.Cadova@zzm.uzh.ch

Introduction

Dynamic mathematical modelling is an invaluable method that helps understand the biomechanics of the anatomically and functionally complex muscle systems, such as for instance the masticatory system. Musculoskeletal models provide insight into variables which are difficult, or even impossible, to measure directly; such as joint loads and muscle forces [1]. Individual input parameters of a model can be modified to understand their influence on the function.

Among the large family of musculoskeletal models of different parts of the human body, only few mathematical models of the temporomandibular joint (TMJ) exist. One reason for this is that TMJ has been mostly regarded as a small, non-load-bearing, and therefore unimportant, joint. However, the impact of a diseased TMJ on individuals and subsequently on the society is enormous [2, 3].

The OpenSim software is an open-source simulation environment, developed to better integrate dynamic simulations into the field of movement science and to share the simulation technology and models [4]. The AnyBody Modeling System is a commercial musculoskeletal modelling system for biomechanical simulations analysing reactions within the human body or between the human body and the environment.

To date, no model for the analysis of chewing and clenching tasks has been developed in OpenSim. On the other hand, in AnyBody there exists a validated model of the mandible (4 degrees of freedom) used to analyse chewing motion [5].

The objective of this study is to present the suitability of both software for the analysis of the mandibular motion, and estimation of the masticatory muscles and joint forces. The ease of use of the software is emphasized, as well. The aim is, for now, not to compare exact resultant values, but rather to focus on the way of creating the model and the way leading to the desired results.

Materials and Methods

Model for the simulation in OpenSim was adopted from the Artysinth modelling software [6] (geometry and physiological muscle parameters). Model for AnyBody was adopted from de Zee [5]. The movement of the mandible (open-close movement) with respect to fixed head was recorded at the Dental Clinic (University of Zürich, Switzerland). Using our software the movement of both condyles and the incisor point was extracted and subsequently used as the input kinematics for the mandible movement (Fig. 1). Clenching force (between

left first premolars) preceding the beginning of the mouth opening was measured using a one-dimensional piezoelectric force transducer (type 9001, Kistler AG, CH-8408 Winterthur, Switzerland).

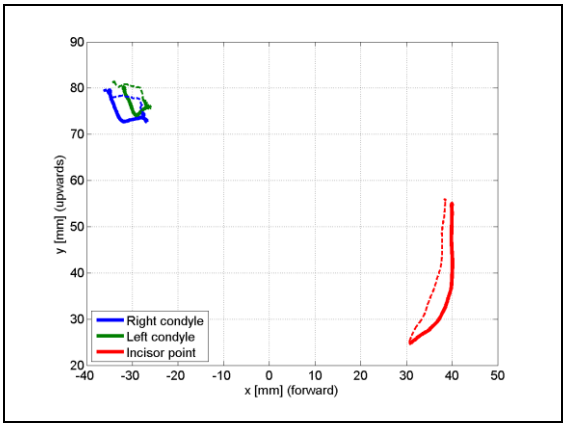


Figure 1: Movement of the condyles and the incisor point viewed in the sagittal plane. Thick line corresponds to the first half of the cycle (opening), thin dashed line to the second half of the cycle (closing).

Results and Discussion

Both software give as output the muscle forces and the joint forces. However, there is difference in the computational procedure. OpenSim uses two steps: inverse dynamics (to compute the joint forces) and static optimisation (to compute the muscle forces). AnyBody uses the inverse dynamics together with the optimisation (both, muscle and joint forces are unknown at the same time). That is also the reason why AnyBody solves better the tasks with closed loops (as the mandible-skull system) than OpenSim. Results from OpenSim did not fulfil the assumptions of the trend of the muscle forces, partly due to the difficulties that OpenSim has when solving the closed loop system.

For a beginner OpenSim is easier to use, thanks to its user interface. However, it partially works as a “black box” and therefore the interpretation of the results can be difficult. For AnyBody one need to get used to the anyscript “programming” language to be able to create the model. But user has high control of individual parts of the model. Moreover, AnyBody comes with a full body model containing most bones, joints and muscles in the human physiognomy.

For the analysis of the dynamic tasks of the masticatory system I would definitely recommend the AnyBody software.

Acknowledgments

This work was supported by the LLP-Erasmus Programme, Grant Agency of the Czech Technical University in Prague (Czech Republic), grant No. GS10/247/OHK2/3T/12, and by the Standard financial plan of the University of Zürich (Switzerland).

References

[1] Peck, C. C. et al. 2007, ARCH ORAL BIOL **52**(4), p. 300-304
[2] Nickel, J. C. et al. 2009, ORTHOD CRANIOFAC RES **12(**3), p. 159-167
[3] Spilker, R. L. et al. 2009, ANN BIOMED ENG **37**(6), p. 1152-1164
[4] Delp, S. L. et al. 2007, IEEE BIO-MED ENG **54**(11), p. 1940-1950
[5] De Zee, M. et al. 2007, J BIOMECH **40**(6), p. 1192-1201
[6] http://www.artisynth.org [cite 20.8.2012]

Wrist protectors in snowboarding – Combined MBS and FEM simulation to analyze the wrist loading

Kilian Rauner[a*], *Lennart Karstensen*[a], *Stefan Lehner*[a], *Frank Michel*[b], *Veit Senner*[a]

a) Technical University of Munich, institute of Ergonomics, division of Sports Equipment and Materials
b) BFU – Swiss council for Accident Prevention, Bern, Switzerland
*) kilian.rauner@mytum.de

Introduction

The wrist is the dominant injury region among snowboarders [1]. Although different studies show, that wrist-guards can decrease the injury risk, there are no standards, which contain adequate tests to determine the effectiveness of such protective gear [2].
The overall aim of this study is to acquire further knowledge about the functional requirements of wrist-guards for children and adolescents, since this age group has the highest risk of suffering wrist injuries [1]. In the course of this aim, a flexible radius model for usage in a MBS model of the human upper extremity was developed as well as a MBS model simulating the fall of a snowboarder onto his outstretched hand.

Materials and Methods

Model 1: The source of the radius is the MBS-Model of the human arm, which is built with SIMPACK 9.0 (SIMPACK AG, Wessling, Germany). For the flexible modeling the FEM Model is created with Abaqus v.6.10 (Dasault Systèmes, Vélizy-Villacoublay, France). To transform the SIMPACK Markers of the MBS body reference system of the radius into the DICOM reference system of the radius a coordinate transformation script was written with the open source software Scilab (Scilab Enterprises, Orsay Cedex, France). Since the radius file contains only surface data, the different bone structures are modeled in a simplified way. Also it is not possible to use non-linear material behavior in an MBS-simulation. Therefore isotropic, linear materials are used for each the cortical bone and the trabecular bone. The mechanical properties are taken from literature:

Table 1: Mechanical properties of the bone materials

	Cortical	***Trabecular***
Young's Modulus in GPa	12 [3]	0.05 [6]
Poisson's ratio	0,3 [3]	0,181 [7]
Density in mg/cm³	1251 [4]	227[4]
Mean thickness in mm	2,9[5]	-

The analysis type is modal analysis. To model the different bones a composite layup is designed within which the cortical bone covers the whole radius as a shell and the trabecular bone is fills the distal and proximal end. For validation of the created radius a model in SIMPACK is built which refers to the experimental test of Varga et al. [3]. This test is replicating a Colles fracture by fixing the proximal end up to the $^1/_3$ distal site and applying a force to the distal end with 2° to 5° palmar inclination. It is derived from the test protocol developed by Muller et al. [8].

Model 2: This is the MBS SIMPACK model, where a snowboarder falls from 1 m height on his right hand. The right hand, lower arm, upper arm and the torso are modeled. The relative movement of the soft tissue is considered by so called wobbling masses [9]. The joint force elements have been tested and altered to achieve realistic results. Finally a model of a wrist protector has been added and the occurring forces have been compared.

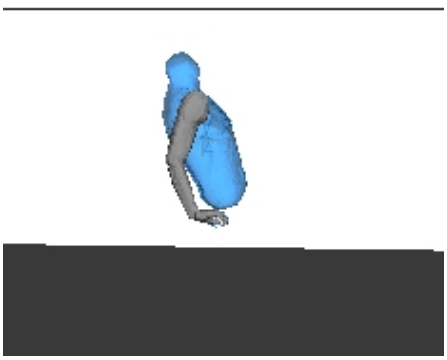


Figure 1: MBS model of the falling snowboarder

Results

Model 1: The verification of the FEM model shows a deformation of 0.4 mm resulting from a applied force of 1800 N. The result sensitivity to parameter variation shows is high to variation of the Young's modulus of cortical bone, as well as the variation of the shell thickness of cortical bone. The mechanical properties meet the original ones.

Model 2: By precisely adapting the joints unnatural peaks of forces are smoothened and realistic results are seen. Important factor was the adaption of the damping factor to absorb energy of the system. By adapting the idealized wrist protector model the torque in the wrist is reduced by 40% for an adult man and by 27% for a 12-year-old boy.

Discussion

The FEM model shows good results in comparison with literature but, with respect to its sensitivity towards cortical thickness, it has to be evaluated if a more detailed thickness profile for the cortical bone is necessary. The MBS model shows realistic impact forces, which suggest a fracture of the radius. In the future an implementation of the FEM Radius will show, if there are huge differences between using a stiff radius or using a flexible one.

References

[1] BfU – Swiss Council for Accident Prevention, statistics on transport of persons injured in snowsports 2011, p. 15
[2] Brügger, O. 2004, BfU-Report 54
[3] Varga, P. et al. 2009, J. of Biomechanics 42 (11), p. 1726-31
[4] Wilks, D. C. et al. 2009, Bone 45 (1), p. 91-7
[5] Bosisio, M. R. et al. 2007, J. of Biomechanics 40 (9), p. 2022-8
[6] Gibson, L. J. 1985, J. of Biomechanics 18 (5), p. 317-328
[7] Rincón-Kohli, L. et al. 2009, Biomech. and Modell. in Mechanobio. 8 (3), p. 195-208
[8] Muller, M. E. et al. 2003, Osteoporosis Int. 14 (4), p. 345-352
[9] Gruber, K. et al. 1998 J Biomech. 31 (5), p. 439-44

III. MUSCLE AND SOFT TISSUES

Chair: Oliver Röhrle, Martin Geiger

Multi-Scale Skeletal Muscle Modeling

Oliver Röhrle,[a,b,c,*], *Thomas Heidlauf*[a,b], *Michael Sprenger*[a,b], *Ellankavi Ramasamy*[c]

a)Institute of Applied Mechanics (CE), University of Stuttgart, Germany,
http://www.mechbau.uni-stuttgart.de/ls2/jrg
b) Cluster of Excellence for Simulation Technology, University of Stuttgart, Germany
www.simtech.uni-stuttgart.de
c) Virtual Orthopedic Lab, Biomechtronic Systems, Fraunhofer Institute IPA, Stuttgart, Germany
*) E-mail: roehrle@simtech.uni-stuttgart.de

Abstract

In computational neurophysiology and continuum biomechanics, researchers focus on modeling of the neuromuscular system using models of different detail and complexity. Neurophysiological models describe how the motoneuron pool in the central nervous system (CNS) operates as an ensemble to control the force that is exerted by a skeletal muscle. To vary this force, the CNS has two options: (i) recruitment, i.e. alter the number of active motor units, and (ii) rate coding, i.e. changing the frequency of electrical impulses driving the motor units. Muscle force generation in neurophysiological models is based on simple analytical equations that do not involve any spatial components. Continuum-biomechanical models of skeletal muscle[1], on the other hand, focus on the generation of force. Geometrical aspects, such as complex fiber architectures or local motor unit distributions, can be adequately represented in three-dimensional continuum-biomechanical models[2]. However, recruitment mechanisms in such continuum-biomechanical models are either heavily simplified[3] or completely neglected. This talk will focus on how both fields can be merged to model the electro-mechanical fundamentals of skeletal muscle recruitment mechanisms and how these aspects can be extended to a multi-muscle framework including for musculoskeletal systems.

Acknowledgments

The authorswould like to thank the German Research Foundation(DFG) for financial support of the project within the Clusterof Excellence in Simulation Technology (EXC 310/1) at the University and the European Research Council (ERC) under the European Union's Seventh Framework Programme (FP7/2007-2013), ERC Starting Grant “LEAD” under grant agreement n° 306757.

References

[1] Röhrle, O. and A.J. Pullan, Journal of Biomechanics, 2007. 40(15): p. 3363-3372.
[2] Röhrle, O., Computing in Science & Engineering, 2010. 12(6): p. 48-57.
[3] Röhrle, O., J.B. Davidson, and A.J. Pullan, Front Physiol, 2012. 3: p. 358.

A first attempt to model contraction in a straited muscle by a constitutive model with data from an in vivo experiment.

G. Benderoth*, C. Clemen, I. Kernhof, G. Silber
University of Applied Sciences
Institute of Material Science
www.fh-frankfurt.de
* *benderot@fb2.fh-frankfurt.de*

Introduction

To model the mechanical behavior of biological tissue by constitutive models is usually not different from modeling the behavior of none biological materials. There is only one exception, the muscle. Beside all the passive properties there is the capability of this tissue to contract intentionally or none intentionally. This contraction is either a 3- or 2dimensional contraction for smooth muscles or an 1dimensional for straited muscles. To model the muscle one needs a mathematical description, in this case a constitutive model, and an appropriate experiment to parametrize such a model. For a straited muscle a 2dimensional loading experiment is needed, because of at least a transverse isotropy of the muscle.

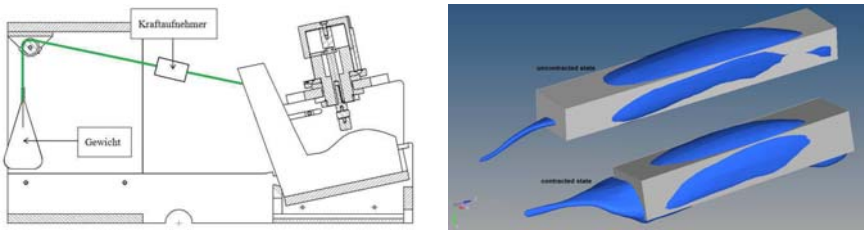


Figure 1: The load experiment device (left), Box approximation of a m. biceps in a contracted and none contracted state (right)

Experiment

As given in figure 1 there is a holding load in the contraction direction and a loading in the direction perpendicular to this direction to evaluate the passive properties. All experiments were made in vivo at the biceps of male test persons. To be sure, that there is no intentional contraction performed by the test person, an ANOVA was made to test the significance of the reaction force of the muscle to the intrusion of the indenter in dependence of the intrusion itself and the holding load. There was a high significance determined for interrelation between the reaction force of the muscle to the holding force, the intrusion of the indenter and their interrelation. A multivariate correlation model (see figure 2) for these variables shows a coefficient of determination of 0.95. Thus 95 % of the relation is described by the model.

Model

A constitutive model from literature [1] was taken fulfilling the properties of polyconvexity, coercitivity and transverse isotropy (see equation (1)). For a first analysis a deformation in principle coordinates and in-compressibility was

assumed. The needed Lagrange-Multiplicand for the in-compressibility was calculated by setting the stress in 3direction zero.

$$\begin{aligned} \Psi &= \frac{\mu}{4}\left\{\frac{1}{\alpha}\left[e^{\alpha(I-1)}-1\right]+\frac{1}{\beta}\left[e^{\beta(K-1)}-1\right]\right\} \\ &\text{with the invariants} \\ I &= \frac{w_0}{3}\,\mathrm{Sp}\,\boldsymbol{C}+(w_p+w_a)\,\mathrm{Sp}\,(\boldsymbol{C}\cdot\boldsymbol{M}) \\ K &= \frac{w_0}{3}\,\mathrm{Sp}\,\boldsymbol{C}^{-1}+w_p\,\mathrm{Sp}\,(\boldsymbol{C}^{-1}\cdot\boldsymbol{M}) \end{aligned} \tag{1}$$

Results

To determine the passive parameters of (1) muscles experiments in none contraction state were used. The use optimization routine was Simulated Annealing. Results are shown in figure 2. The coefficient of determination was 0.986. With these parameters a load function $w_a(hl)$ determined by isometric loading part of the experiment was obtained. This load function is shown in figure 2.

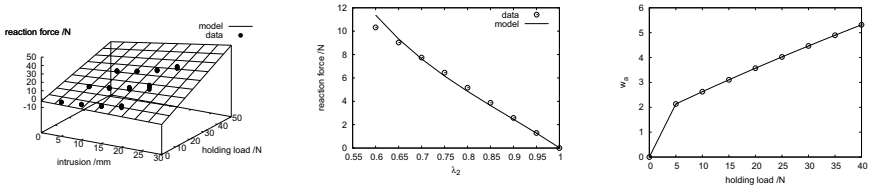


Figure 2: Results of the regression model (left), the parameter identification parameters in the none contracted state (center) and the holding load function (right)

Discussion

The analysis has shown that with an adequate experiment the additional stress generated by contraction in a straighted muscle can be modeled by a constitutive model in dependence of a holding load. This model can now be included in a human body model to predict the mechanical behavior of a muscular skeleton system more precisely.

References

[1] A. E. Ehret, M. Böl, and M. Itskov. A continuum constitutive model for the active behaviour of skeletal muscle. *Journal of the Mechanics and Physics of Solids*, 59:625–636, 2011.

SKELETAL MUSCLE CONTRACTION – EXPERIMENT AND THEORY

Maike Sturmat[a], *Christine Weichert*[a], *Kay Leichsenring*[a,b], *Tobias Siebert*[b], *Reinhard Blickhan*[c], *Markus Böl*[a]

[a] Institute of Solid Mechanics, TU Braunschweig, 38106 Braunschweig, Germany
[b] Department of Sport and Motion Science, University of Stuttgart, 70569 Stuttgart, Germany
[c] Institute of Motion Science, Friedrich-Schiller University Jena, 07749 Jena, Germany

Introduction

The description of active muscle tissues as well as the determination of their mechanical properties during contraction are of high interest and require complex numerical and experimental techniques. Active muscle experiments are essential to improve previously investigations on activated skeletal muscles [Tang et al., 2007, Böl et al., 2011a] by considering additional to the generated force and length characteristics the three-dimensional deformable muscle shape. In doing so, an optical measurement technique is proposed to obtain a convenient and comprehensive data set for muscle model validation. In the present contribution, an approach is presented, where a recently developed numerical muscle model [Böl et al., 2011b, Ehret et al., 2011] is validated by consideration the locomotive muscle geometry, the fibre alignment, and also the generated force characteristics [Böl et al., 2013].

Materials and Methods

An optical measurement method is developed to measure the three-dimensional shape of a muscle during the contraction process. The measurements are carried out *ex situ* on isolated rabbit soleus muscles. In this way an immediate determination of forces and velocities can be ensured as well as the deformation state. The recorded surface is marked by a special coating technology. Thus, the sensors have the ability to detect the various facets and a closed three-dimensional muscle model separated in muscle tissue and tendon/aponeurosis complex can be reconstructed from different recordings, see Figure 1. In a second step, the transversal isotropic muscle tissue behaviour is described by a hyperelastic constitutive law whereas the activation of the muscle fibres could be inserting by a single parameter, see [Böl et al., 2011b, Ehret et al., 2011].

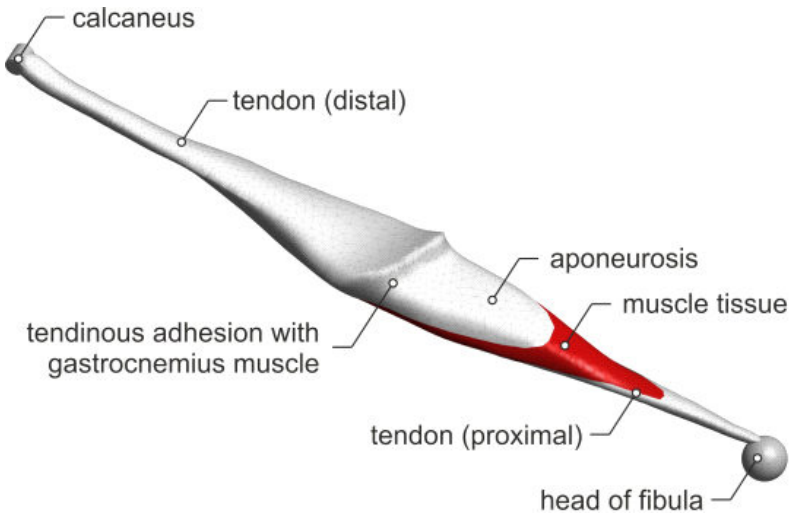


Figure 1: Exemplary illustration of a typical three-dimensional rabbit soleus muscle consisting of muscle, aponeurosis, and tendon tissues.

Results and Discussion

Figure 2 shows first results of the validation process. The black curve illustrates the generated force of an experimental isometric muscle contraction and the curve for the related simulated contraction is colored in green. The proposed model validation combines, in contrast to previous validation approaches, complex three-dimensional muscle architecture with generated forces during different contraction modes. This allows a more accurate mechanical description of muscle tissues and further to a better understanding of the interaction between muscles in a whole muscle package. Due to the consideration of biological tissue, results underline the ability of the proposed approach.

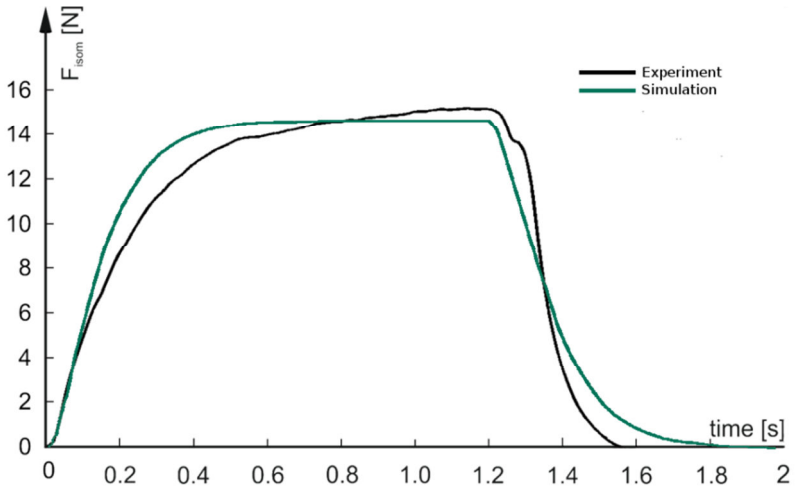


Figure 3: Generated force curves of an isometric muscle contraction. The black curve denotes the experiment and the green curve the simulated muscle contraction.

Acknowledgments

Partial support for this research was provided by the Deutsche Forschungsgemeinschaft (DFG) under Grants BO 3091/4-1 and SI 841/3-1.

References

Tang C.Y., Tsui CP., Stojanovic B., Kojic M., Finite element modelling of skeletal muscles coupled with fatigue. International Journal of Mechanical Science, 49:1179-91, 2007.

Böl M., Sturmat M., Weichert C., Kober C., A new approach for the validation of skeletal muscle modelling using MRI data. Computational Mechanics, 47:591-601, 2011.

Böl M., Weikert R., Weichert C., A coupled electromechanical model for the excitation-dependent contraction of skeletal muscle. Journal of the Mechanical Behavior of Biomedical Materials, 4:1299-1310, 2011b.

Ehret A.E., Böl M., Itskov M., A continuum constitutive model for the active behaviour of skeletal muscle. Journal of the Mechanics and Physics of Solids, 59:625-636, 2011.

Böl M., Leichsenring K., Weichert C., Sturmat M., Schenk P., Blickhan R., Siebert T., Three-dimensional surface geometries of the rabbit soleus muscle during contraction - input for biomechanical modelling and its validation. Biomechanics and Modeling in Mechanobiology, accepted for publication, 2013.

Determination of Viscoelastic Properties of Meniscal Tissue Using both Experiment and Simulation

Andreas M. Seitz[a*], *Fabio Galbusera*[a,b], *Carina Krais*[a], *Anita Ignatius*[a], *Lutz Dürselen*[a]

a) Institute of Orthopaedic Research and Biomechanics, www.biomechanics.de, Germany
b) Instituto Ortopedico Galeazzi, www.galeazzi-gsd.it, Italy
*) andreas.seitz@uni-ulm.de

Introduction

The objective of this study was to determine the viscoelastic properties of human meniscal tissue during stress-relaxation under confined compression conditions using both *in-vitro* and FE analyses. Further, it was hypothesized that two different underlying biphasic models lead to significantly different predictions for the aggregate modulus (H_A) and permeability (k).

Materials and Methods

Coplanar plugs (Ø 4.6 mm), obtained from the anterior (AH) and posterior horn (PH), and pars intermedia (PI) of lateral and medial menisci of 25 donor knees (57.5 ± 10.5 yrs; n_{total} = 150) were exposed to stress-relaxation tests under confined compression conditions at three compression levels (ε_i =10%, 15%, 20%). Each strain level was held constant for 90 minutes to ensure equilibrium state was reached. H_A and k were determined solving a one-dimensional diffusion equation [1,2] using an iterative least square algorithm:

$$\sigma_t = \sigma_{t\to\infty} + 2 * H_A * \varepsilon_i * e^{(-(\frac{\pi}{h_0})^2 * H_A * k * t)} \quad (1)$$

where σ_t is the time-dependent stress, $\sigma_{t\to\infty}$ is the equilibrium stress, and h_0 the initial sample height. Subsequent FE modelling using the previously in-vitro determined geometries, properties, and boundary conditions resulted in 150 individual calculations. Strain-dependent permeability was introduced following the Holmes-Mow formulation [3]:

$$k = k_0 (\frac{J-\varphi_0}{1-\varphi_0})^2 e^{\frac{1}{2}M(J^2-1)} \quad (2)$$

where k_0 equals the initial permeability, J the Jacobian of the deformation gradient tensor, φ_0 initial porosity and M the strain-dependence coefficient; while H_A was calculated as:

$$H_A = \lambda + 2\mu \quad (3)$$

where λ and μ are the Lamè parameters of the drained solid structure. Parameter optimisation was then performed to solve the inverse finite element problem. Gaussian distributed data were examined using two-factor repeated measures ANOVAs with t-test post-hoc testing when significant (SPSS). Linear regression was used to check the correlations (SPSS), and $p \leq 0.05$ was considered statistically significant.

Results

Aggregate modulus and permeability calculated with the diffusion equation (Eq. 1) were statistically higher ($p \leq 0.01$) than those obtained with subsequent FE analyses (Fig. 1).

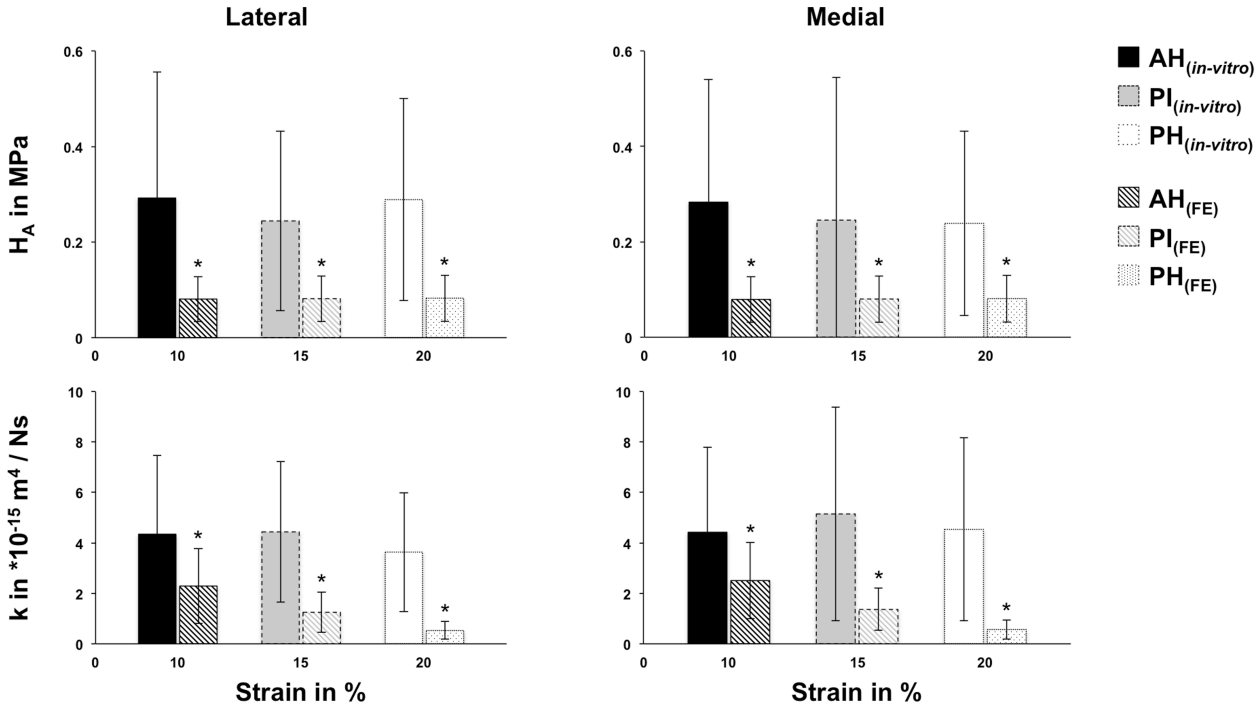


Fig. 1: Aggregate modulus (H_A) and permeability (k) values of lateral and medial menisci, divided into AH: anterior horn; PI: pars intermedia; PH: posterior horn, obtained from in-vitro investigations and FE analyses at three different strain levels (10%, 15%, 20%) (Mean values ± SD; n = 25; *p<0.05).

Discussion

To the authors knowledge this is the first study investigating the stress-relaxation of human menisci under confined compression. Despite the results of the two approaches being statistically different, they show adequate values when compared with literature. Thus our hypothesis could be confirmed. The results might contribute to a better understanding of the complex nature of meniscal tissue and might also have an impact on the design of future meniscal substitutes. Furthermore, it implicates that when comparing viscoelastic properties of meniscal tissue, both the test conditions and the underlying model should be considered.

Acknowledgments

This study was funded by the German Research Foundation (DFG: DU254/5-2).

References

[1] Frank et al. 1987, J Biomech. **20**, p. 615-27
[2] Mow et al. 1980, J Biomech Eng. **102**, p. 73-84
[3] Holmes et al. 1990, J Biomech. **23**, p. 1145-56

IV. SPINE

Chair: Fabio Galbusera, Frank Niemeyer

Inverse Prediction of Transport Properties of the Intervertebral Disc

Fabio Galbusera[a,b*], *Marta Tibiletti*[b], *Marco Brayda-Bruno*[b], *Cornelia Neidlinger-Wilke*[a], *Hans-Joachim Wilke*[a]

a) Institute of Orthopedic Research and Biomechanics, Ulm University, Ulm, Germany
b) IRCCS Istituto Ortopedico Galeazzi, Milan, Italy
*) fabio.galbusera@grupposandonato.it

Introduction

Disc nutrition is believed to be based on diffusion of nutrients across the vertebral endplates and through the intervertebral disc [1]. Some studies were targeted to the investigation of the in vivo transport behavior of small solutes through the endplate zone (EPZ) and inside the intervertebral disc. Serial contrast-enhanced magnetic resonance imaging (MRI) was widely employed to this aim [2], by using nonionizing contrast agents with molecular weights lower than 1 kD, such as Gadodiamide or Gadoteridol. However, published studies were limited to semi-quantitative analysis of the T1-weighted signal intensity curve, searching for patterns which may be recognized among the patients population. In this work, a novel quantitative method based on contrast-enhanced MRI to predict the transport behavior of the contrast agent in the vertebral endplates and intervertebral discs by means of finite element optimization is presented.

Materials and Methods

Patient specific one-dimensional finite element models aimed to replicate the transport of the contrast agent (Gd-HP-DO3A) from the vertebral bodies to the nucleus pulposus were developed (Fig. 1). The transport coefficients of the vertebral structures were optimized in order to minimize the error between the Gd-HP-DO3A concentration predicted with the model and the actual concentration determined by MRI acquisitions during the first six hours after injection of the contrast agent. Simulations were run to calculate the transport properties of 68 discs from a population of 32 patients suffering from low back pain.

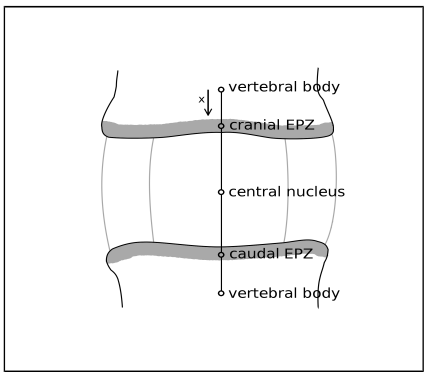


Figure 1. One-dimensional finite element model of the transport of Gd-HP-DO3A

Results

A preliminary sensitivity analysis showed a wide range of behaviors of the Gd-HP-DO3A concentration for values of the transport coefficient α ranging from 5 x 10^{-7} cm^2/s to 5 x 10^{-5}

cm^2/s. At the lower limit, a negligible concentration of the contrast agent was predicted in the nucleus pulposus during the 6 hours. For increasing values of α, the concentration of the contrast agent increased in both the EPZs and the nucleus pulposus.

The transport properties of Gd-HP-DO3A showed a high variability within the population. In general, the predicted diffusion coefficients were higher than the bulk diffusion coefficients of oxygen in water, thus showing that other chemico-physical processes may be involved in the transport of the contrast agent. Comparisons between different age groups and disc degeneration degrees did not reveal any relevant differences (Fig. 2).

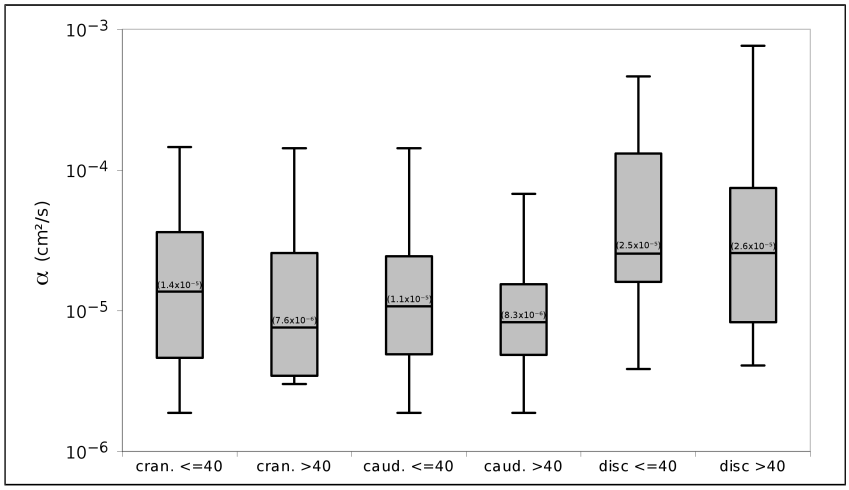


Figure 2. Gd-HP-DO3A transport coefficients (α) in the cranial and caudal EPZs and in the intervertebral disc in the population subdivided in two age groups (<= 40 and > 40 years). Median values are reported in brackets

Discussion

As transport of the contrast agent and nutrients is usually assumed to be almost purely diffusive [1], it may seem appropriate to compare the predicted values of α with the diffusion coefficients of nutrients. In general, the predicted transport coefficients were markedly higher, at least by one order of magnitude, in comparison with corresponding literature data [3]. Values of α lower than 10^{-6} cm^2/s (as should be expected if the transport of Gd-HP-DO3A was purely diffusive) would result in a negligible inflow of the contrast agent into the nucleus pulposus, in clear contrast with the MRI observations. It can be hypothesized that the predicted values include also other phenomena present in vivo and not in in vitro investigations, such as electrical interactions between the molecules and convection.

Acknowledgments

This project was funded by EU project GENODISC (HEALTH-F2-2008-201626).

References

[1] Rajasekaran, S. et al. 2007. J Magn Reson Imaging **25**(2), p. 410-18
[2] Rajasekaran, S. et al. 2004. Spine **29**(23), p. 2654-67
[3] Holm, S. et al. 1981. Connect Tissue Res **8**(2), p. 101-119

Numerical Analysis to Investigate the Value of Standards for Preclinical Evaluation of Posterior Spinal Implants

Luigi La Barbera[a,b*], *Tomaso Villa*[a,b], *Fabio Galbusera*[b,c]

a) Laboratory of Biological Structure Mechanics, www.labsmech.polimi.it, Italy
b) IRCCS Istituto Ortopedico Galeazzi, www.galeazzi-gsd.it, Italy
c) Zentrum für Muskuloskelettale Forschung Ulm, www.biomechanics.de, Germany
*) luigi.labarbera@polimi.it (corresponding author email)

Introduction

The preclinical evaluation of any orthopedic device is an important step to evaluate and assess its mechanical reliability. For posterior spinal fixators, some international standards have been recently introduced or reapproved in order to describe how to assess both static and fatigue properties of spinal implant in a vertebrectomy model [ASTM F1717-2012] or using an anterior support [ISO 12189-2008]. In general, although these standards allow for the comparison between different constructs in controlled conditions, they necessarily represent a simplification of the real boundary and loading configuration experimented in-vivo. The disagreement that exists between standard testing conditions and real in vivo loading, could be reduced introducing testing protocols that take into account a "more physiological" testing configuration. Thus, the aim of this work is to provide a better understanding of ASTM and ISO standards and investigate their value by means of finite element (FE) models validated through experimental tests.

Materials and Methods

FE models reproducing standards set-ups were built according to ISO and ASTM standards (Fig. 1). A simplified model of a spinal fixator was designed and introduced into polyethylene (PE) blocks, assuming a perfect tie and neglecting any relative motion. All materials were assumed linear elastic and isotropic. In order to validate the numerical predictions for both models, on each rods of the spinal fixators, one uniaxial linear strain gage was glued and aligned along the longitudinal axis. The constructs were then loaded cyclically on a servohydraulic testing machine (ASTM: 30-300 N, ISO: 600-2000 N). In order to evaluate the contribution of spring precompression in ISO model, the initial deformation on the rods was also considered. Mean value and standard deviation of the measurements were than calculated and compared with FEM predictions. The spinal fixator was then virtually implanted in a previously validated FE model of L2-L4 segment [1], so that the orientation of the screws and the curvature of the rods matched the morphometry of the pedicles and lumbar lordosis. Three pseudophysiological models were then built (Fig. 1).

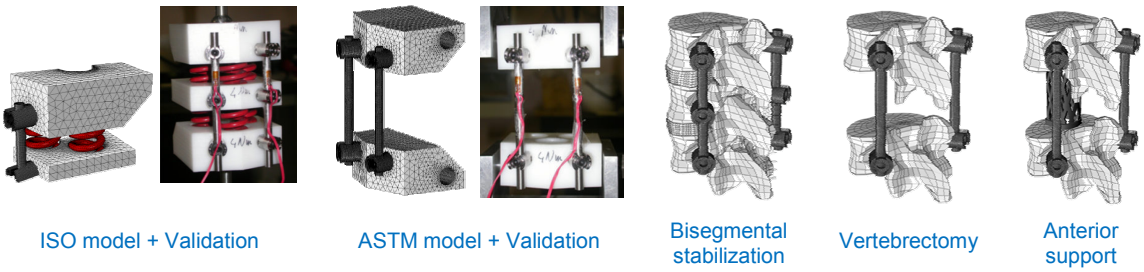


Figure 1: FE models reproducing standard configurations and pseudophysiological conditions.

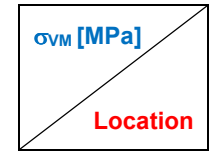
Besides the loading condition prescribed by ISO standard, different loads or moments were applied to each model (Fig. 2).

Results

For each simulation, both the location of the most stressed points in traction and the correspondent value of σ_{VM} were extrapolated (Fig. 2).



Vertical load		Flexion	Extension	Lateral Bending	Axial Rotation
2 kN	500 N	10 Nm	10 Nm	10 Nm	10 Nm
255 / P	64 / P	- / -	- / -	- / -	- / -
2420 / P	593 / P	236 / P	245 / A	- / -	265 / A
479 / P-M	124 / P-M	70 / P-M	49 / A	68 / A-L	95 / A-M
2306 / P-M	576 / P-M	243 / P-M	380 / A	80 / A-L	290 / A-M
190 / P-M	48 / P-M	24 / P-M	- / -	- / -	- / -



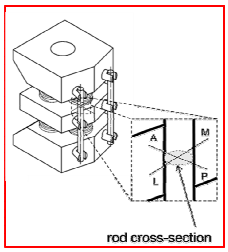


Figure 2: Loading conditions and results for each model.

Discussion

(i) In ISO configuration, the initial precompression of the springs induces an extension on the rods, reducing the final value of the Von Mises stress of about a 50% if compared to our previous model where the preload effect wasn't taken into account [2]. (ii) Both configuration and load value (2 kN) prescribed by ISO standard, tend to overstress the rods if compared to a more physiological load of 500 N applied to bisegmental stabilization model: this scenario guarantees a safety coefficient of about 2. Even superposing the contribution due to flexion (coupled to axial compression during walking [3]), the safety coefficient is about 1.3. (iii) ASTM configuration is highly severe with respect to reality. The vertical load that induces the same state of stress as in ISO configuration is approximately 215 N, while to obtain the same state of stress as in a physiological condition about 100 N should be applied. Moreover, if an anterior support is used, the stress on the rods decreases of more than one order of magnitude. (iv) Due to the natural curvature of the spine and to the orientation of the pedicles, the positions of the most stressed areas move toward a more medial position. Since these aspects are not taken into account neither in ISO nor in ASTM standard, a further investigation through a numerical parametric analysis.

References

[1] Galbusera F. et al. 2011, Medical Engineering & Physics, 33:490-496.
[2] Villa T. et al. 2013, The Spine Journal, under review (22/01/2013).
[3] Rohlmann A. et al. 1997, J. Biomechanics, 30(1):41-47.

The proposition of a numerical model of the experimental set-up for testing bone-screw interface with use of the foam vertebra model

Bartosz Nowak

ulrich medical, www.ulrichmedical.com, Germany
b.nowak@ulrichmedical.com

Introduction

Spinal systems and pedicle screws have been widely and successfully used in orthopaedics and spine surgery for more than three decades [1]. Despite their long and extensive development [3], loosening of pedicle screws takes place and appears even more often in those bones afflicted with either osteopenia or osteoporosis [6].

In this study an attempt was made to create the numerical model of the experimental set-up developed previously for testing the bone-screw interface with use of the foam vertebra model. Though the results from experimental tests included both elastic and plastic deformations of either vertebra or spinal system, the extent of this study was strictly limited to linear-elastic static calculations only, in order to account for the initial stiffness of the numerical model. Therefore, only elastic deformations in terms of global displacements were considered and their convergence to initial part of the experimental tests (elastic deformations) was discussed.

Materials and Methods

The geometry of the numerical model destined for the finite element method simulations derived directly from the 3D CAD geometry of the experimental set-up, see Fig. 1. The foam vertebra model was created as the foam model of the human vertebra made out of biomechanical materials [6], together with reproducible testing methods for determining of stability at the bone-screw interface. The experimental set-up was based on the already developed ASTM F- 1717 standard [2] and included some minor modification concerning materials usage and design dimensions, see Fig. 1.

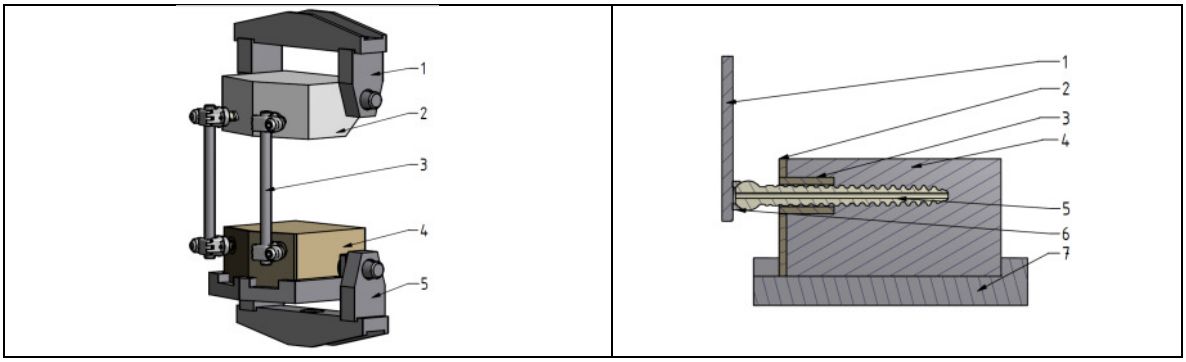


Figure 1: Geometry of the experimental set-up (left); 1 - upper holder, 2 - UHMWPE block, 3 - tangoRS spinal system (ulrich medical, Ulm, Germany), 4 - foam vertebra model, 5 - lower holder; Cross-section of the numerical model's geometry (right); 1 - rod, 2 - cortical layer of the foam vertebra, 3 - pedical canal, 4 - cancellous, 5 - pedicle screw, 6 - connecting washer, 7 - lower holder.

Results

The results obtained in the numerical simulations had been performed in Ansys Workbench 13.0 [4], and included displacements calculated for the whole geometry of the numerical model. However, within this study only the displacements of the reference point in which the loading force was placed were taken into account. This situation corresponds entirely to the experimental tests in which the displacements of the testing machine's jaw were recorded. It allowed therefore for direct comparison between numerical and experimental results. The displacements for both numerical simulations and experimental tests were gathered and presented in Tab.1.

Table 1: Comparison of numerical and experimental results.

	Displacements [mm]		***Difference [%]***
	Numerical	***Experimental***	
Normal (15 pcf)	3,03	3,42	13
Osteopenic (10 pcf)	3,22	3,55	10
Osteoporotic (5 pcf)	3,74	3,75	1

Discussion

Considering the results obtained within numerical simulations, it must be stated that the highest maximum displacements under desired testing conditions were found for models with osteoporotic bone quality (5 pcf) while the lowest for normal (15 pcf). It seems to be consistent with the experimental tests performed previously. Taking into account the results obtained within the numerical simulations along with all the assumption made upon the numerical model i.e. double symmetry, boundary and loading conditions it can be seen that the convergence to the experimental tests fell into reasonable range of 1% - 16%. The further usage of the numerical model can be anticipated for calculation with use of material models including damage and fatigue. Furthermore, the application of already developed model augmented with appropriate material model can be foreseen in testing for differences in loosening between various pedicle screws and spinal systems.

Acknowledgments

The financing was provided by the EU under the FP7 Marie Curie Action - Grant number - 238690.

References

[1] Cho, W. et al. 2003. The biomechanics of pedicle screw-based instrumentation. Journal of Bone and Joint Surgery **28**, p. 1226-1254

[2] ASTM F171 2011. Standard test methods for spinal implant constructs in a vertebrectomy model,

[3] Gaines, W. S. 2000. The use of pedicle-screw internal fixation for the operative treatment of spinal disorders. Journal of Bone and Joint Surgery **82**(10), p. 1458-1476

[4] Gebhardt, C. 2011. Praxis Buch mit ANSYS Workbench. Hanser, Germany

[5] Sawbones Pacific Research Laboratories Inc. 2012, Sawbones catalog biomaterials

[6] Sanden, C.et al 2004. The significance of radiolucent zones surrounding pedicle screws. Journal of Bone and Joint Surgery **3**:457-463

InstantSpine: An Open Source Spine Model Generator

Frank Niemeyer [a*], *Hendrik Schmidt*[b], *Anita Ignatius*[c], *Hans-Joachim Wilke*[c]

a) Scientific Computing Centre Ulm (uzwr), www.uni-ulm.de/uzwr, Ulm, Germany
b) Julius Wolff Institute for Biomechanics and Musculoskeletal Regeneration, Charité, Berlin, Germany
c) Institute for Orthopaedic Research and Biomechanics (UFB), www.biomechanics.de, Ulm, Germany
*) frank.niemeyer@uni-ulm.de

Why InstantSpine

Creating finite element models of the spine is a tedious task. Building a working model of the full spine with its complex geometry from the ground up based on imaging data can easily take years. This elaborate approach yields models of high geometric fidelity that match one particular anatomy and are therefore in a sense "patient-specific." While such models have the potential to represent one specific spine in a very precise manner, they might fail miserably when used to simulate other cases. Although such models can be scaled globally to at least roughly adapt the geometry, more sophisticated adjustments are mostly impossible or require a lot of additional work. When we started to work on a model of the scoliotic spine, we realized that the aforementioned bottom-up approach would not to be able to satisfy our requirements: We needed a way to generate models of the entire spine featuring different forms and degrees of scoliosis, described by only a limited set of anatomical parameters.

What Is InstantSpine?

Instead of building a single, highly detailed model, we therefore developed a parametric model of the spine that can generate arbitrary segments of the thoracolumbar spine. The anatomical parameters of each vertebra and disc can either be specified directly, or derived from a subset of the parameters. This parametric model, implemented in APDL (ANSYS), is the first and core part of InstantSpine. We realized however, that such a model might be useful to other researchers as well and thus decided to develop a user-friendly graphical user interface that sits on top of the existing model, so that anyone could create spine models without needing to know or understand the underlying APDL code base in detail.

A Quick Tour of InstantSpine

The InstantSpine GUI is built using C#/WPF and requires .NET version 4.5 or newer. As a prerequisite, the path to ParaSpine (the parametric spine model, i.e. the APDL code) must be specified in the settings dialog. Furthermore, the user must define the directory of a working ANSYS installation. We hope to be able to remove the dependence on ANSYS in a future release of InstantSpine; for the time being, ANSYS remains mandatory, though.

The main window is divided into three sections. The first section, labeled "Anatomy," is used to list all the required anatomical parameters for the vertebra and discs. The parameter table can be either filled manually or the user can import data from plain CSV files (despite its name, InstantSpine's CSV format really is "white-space separated," i. e. the columns need to

be separated by white-space characters). InstantSpine also includes a sample file with "reasonable" parameter values, which can serve as a point of reference for own experiments.

The user can furthermore set up arbitrary dependencies between anatomy parameters by supplying a simple Python script containing a function named `update_parameters`. This function can assign any valid arithmetic expression to the different member values that represent the anatomy parameters of each vertebra-disc compound. InstantSpine includes a sample script that implements the regression model of Kunkel et al. [1–3]. As useful as it might be, note that this scripting facility opens the door for code injection attacks, because InstantSpine does and cannot check for potentially malicious side effects of the supplied script. You should therefore only execute scripts you trust.

In the "Material properties" section, the user defines the material behavior of the different structures (cortical and spongy bone, endplate, annulus, nucleus, ligaments) of the spine. In the current version, the material model to be used for each structure is fixed and cannot be changed by the user without editing the generated solver input file. We plan to make InstantSpine more flexible with respect to the selection of material models in future versions.

The last section determines the mesh quality (resolution) for different parts of the model. We currently support hexahedral meshes only. Radial mesh resolution above the annulus is currently capped at a value of eight, because the current version of the underlying parametric model imposes this arbitrary limit. The user then selects the desired segment using two slider controls (T1 – S1). Before clicking on the "Generate spine model" button, the user can choose the desired target FE solver. The current version only supports Abaqus, but we already started to work on a FEBio output filter.

Model parameterizations can be stored to and loaded from special XML files (file extension: `xism` for eXtensible InstantSpine Markup). InstantSpine provides a template parameter file, containing reasonable geometry parameters as well as some default material parameters, which we however tested only for the L3-L4 segment.

Obtaining InstantSpine

We are firm believers in the Open Science movement and we fully support the call for improved reproducibility in computational science. One important step towards improving the current situation is to *fully disclose* the methods used to obtain certain results, an aspect that is understood to be vital to any other "classic" form of science. InstantSpine is our contribution to this hopefully successful movement and may be obtained, used and modified by anyone for non-commercial purposes for free in the very near future (we will announce the release on www.biomechanics.de).

Acknowledgments

This work was supported by the German Research Foundation (Wi-1352/12-1).

References

[1] Kunkel, M.E. et al. 2010, J Anat **216**(3), pp. 320 – 328
[2] Kunkel, M.E. et al. 2011, J Anat **218**(2), pp. 191 – 201
[3] Kunkel, M.E. et al. 2011, J Anat **219**(3), pp. 375 – 387

V. COMPUTING, CFD & INDUSTRIAL

Chair: Martin Geiger, Jürgen Salk

Optimal Control in Implant Shape Design

Lars Lubkoll[a*], *Anton Schiela*[b], *Martin Weiser*[a], *Stefan Zachow*[a]

a) Zuse-Institute Berlin, www.zib.de, Germany
b) TU Berlin, www.tu-berlin.de, Germany
*) lubkoll@zib.com

Introduction

Today the design of implants is more dependent on the experience of medical scientists than on technical tools. In most cases the determination of an implant's shape is done by visually comparing CT scans with implant models and choosing one that seems to fit. This approach is very sensitive to the surgeon's skills and the geometry of the implant. Especially in the case of heavy fractures or natural deformations of the oral and maxillofacial bone structure it is often difficult to accurately predict the operation's outcome.

An automated implant design that considers the mechanical behaviour of the soft tissue would significantly improve this situation. Thus the goal of our project is the development of an algorithm that, given a desired outcome and under consideration of constraints given by the physician, computes an optimal implant shape.

Materials and Methods

The direct formulation of the shape design problem leads to a complicated MPEC (mathematical program with equality constraints) including a contact problem. For this reason we developed a formally equivalent formulation which considers the force exerted by the implant instead of its shape as design variable. This allows more natural combination with the mechanical descriptions of biological soft tissues and thus leads to a simpler control-contrained optimal control problem. The implant shape is indirectly determined by the soft tissue deformation.
In this setting constraints such as the fact that the soft tissue can glide over the implant as well as the requirement that voids between the implant soft tissue must not occur can be easily incorporated by interpreting the force exerted by the soft tissue as pressure boundary conditions.

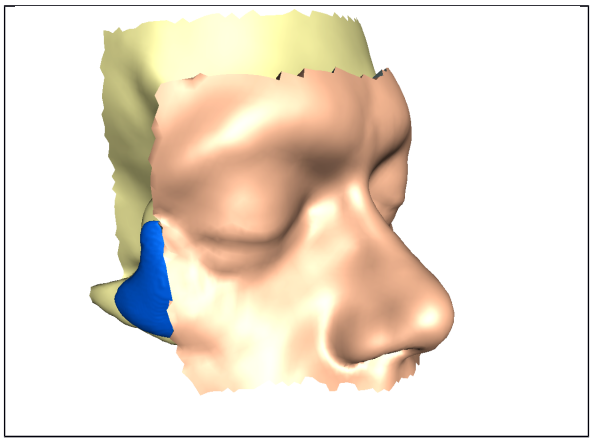


Figure 2: Zygomatic bone implant with soft tissue and bones.

Results

The proposed model not only leads to smaller domains that must be discretized. It also allows a simpler, more natural formulation than the direct formulation als MPEC and avoids the inherent difficulties of contact problems. Its advantages are supported by first proof-of-concept computations in KASKADE7 [2,4].

In order to deal with the particular challenges of the efficient discretization of complicated soft tissue geometries we developed a mesh refinement strategy that is able to cheaply reconstruct complicated soft tissue geometries from coarse grids using convex combinations of local Hermite interpolation polynomials on the grids faces. This technique also avoids strong refinements at artificial reentrant corners and thus leads to more efficient algorithms.

For the construction of efficient mesh hierarchies a goal-oriented error estimator has been developed that allows a better estimation of the influence of the discretization error on soft tissue shapes [3].

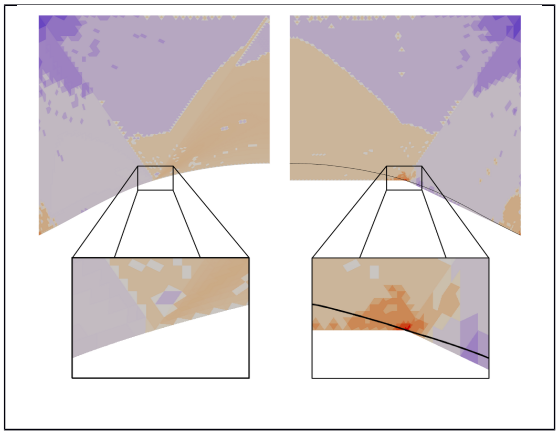


Figure 2: Hermite-Interpolation for smooth geometries avoids artifical mesh refinement at reentrant corners.

Discussion

While first computations have been realized for linear elastic constraints in general non-linear, polyconvex and compressible material models must be considered. For these problems the solution of the optimization problem is more challenging. Therefore our current work focuses on the development of an efficient affine invariant composite-step method that is based on our derived mesh-refinement and error-estimation techniques.

Acknowledgments

The authors acknowledge funding by the DFG research center MATHEON.

References

[1] Lubkoll, L. et al. 2012, ZIB-Report 12-08

[2] Weiser, M. et al. 2012, Advances in DUNE, Springer, pp. 101-112

[3] Weiser, M. 2012, Optim. Meth. Softw., pp.1-24

[4] Lubkoll, L. et al. 2011, Optimal Control in Implant Shape Design, CBU 2011 Proceedings, pp. 61-21

Uncertainty Quantification using Stochastic Collocation Method and Application in a Model of Mineralized Turkey Leg Tendon

A. Gerisch[a,*], B. Schieche[a], S. Tiburtius[a], K. Raum[b], J. Lang[a]

a) TU Darmstadt, AG Numerik und Wissenschaftliches Rechnen, www.mathematik.tu-darmstadt.de, Germany
b) Charité-Universitätsmedizin Berlin, JWI and BSRT, jwi.charite.de, Germany
*) gerisch@mathematik.tu-darmstadt.de

Introduction

Mathematical or computational models in biomechanics, as in other disciplines, often have input parameters which are subject to uncertainty in their values. Uncertainties can be seen in parameters like material coefficients, initial and boundary conditions, load functions, or the shape of spatial domains. The goal of uncertainty quantification (UQ) is to characterize the effect of this uncertainty on the output of the model. As a result, the user of a model is provided with not only the output for some particular instances of it but also with statistical information about the model output under the influence of parameter uncertainties. This additional information enhances the scientific value of the model itself.

The goal of this contribution is to present a stochastic collocation approach on adaptive sparse grids for UQ. We illustrate the method on a multi-scale model for the effective elastic properties of mineralized turkey leg tendon (MTLT) tissue [1].

Materials and Methods

In our stochastic collocation approach [2, 3] we assume that we have M continuously distributed random variables $\xi(\omega)=(\xi_1,\ldots,\xi_M)$ describing the uncertainty in the vector y of model parameters $y=\xi(\omega)$. This implies that we assume that stochastic models describing the uncertainty of model parameters are already available and that random fields, describing the uncertainty in spatially distributed parameters, have been reduced to a finite number of random variables. Consequently, the model output u is a function of the random variables, $u = f(y)$. Note that the output u may be vector-valued and may itself still depend on variables, e.g. spatial coordinates or time, of the model.

Stochastic collocation now provides a means to construct an interpolation function from the parameter space to the solution of the model. To this end, collocation points (samples from parameter space) and an interpolation procedure are necessary. In our approach we use global tensor-product Lagrange interpolation but this could be replaced with, e.g., a local spline-based interpolation. The evaluation of u for a given collocation point requires the solution of a (deterministic) model. In order to be efficient, i.e. to use only as many model simulations as necessary to achieve a certain accuracy, the collocation points must be chosen sparse and structured. Smolyak's algorithm [4] constructs such sparse grids in multiple dimensions. Furthermore, these grids are nested and hence additional collocation points can be added successively while all previously computed data is still being used in the interpolation. A hierarchical representation of the interpolation allows for adaptively selecting additional collocation points such that accuracy goals are met. Once the interpolation from parameter

space to the solution is defined, it is straightforward to evaluate statistical quantities (mean, variance,…) of the solution over the parameter space.

Results

In [1] we have developed a hierarchical model of MTLT tissue. Based on a sequence of homogenization steps, this model predicts effective elastic properties at the tissue scale. However, some of the model parameters cannot be estimated faithfully from experimental or other data and are thus subject to uncertainty. In order to investigate the effect of these uncertainties on the model output we have applied UQ following the approach described. In our analysis we included in particular the aspect ratios of inclusions like mineral platelets or pores and the distribution of mineral in the fibril bundles (intra- vs. interfibriallar). In these cases, a natural variability in the parameter can be expected which is not represented in a deterministic model. UQ can help to characterize the impact of this simplification in the model on the model output. It turned out that uncertainty in the mineral aspect ratio and in the intra- vs. interfibriallar distribution of mineral have a strong effect whereas for instance the effect of uncertainty in the shape of micro- or nano-pores is much weaker.

Discussion

The ideas behind the stochastic collocation approach on adaptive sparse grids for UQ have been presented. This approach may use an existing simulation tool for the model, i.e. the evaluation of $f(y)$, as a black box and distinguishes itself in that respect from, e.g., a stochastic finite element method. In the collocation approach, much like in Monte Carlo, independent instances of the model are simulated and allow for a straightforward use of parallel computational resources. However, as long as the parameter dependence of the model output is sufficiently smooth, the collocation approach, due to its use of adaptivity and sparse grids, outperforms Monte Carlo for moderate numbers of stochastic dimension.

Acknowledgments

This work was supported by Deutsche Forschungsgemeinschaft (DFG SPP 1420 grants GE 1894/3 and RA 1380/7).

References

[1] Tiburtius, S. et al. 2013, On the elastic properties of mineralized turkey leg tendon tissue: Multiscale model and experiment, submitted

[2] Schieche, B. 2012, Unsteady Stochastic Collocation Methods on Sparse Grids, PhD thesis, TU Darmstadt.

[3] Schieche, B. and Lang, J. 2013, Int. J. Computational Science and Engineering, in press.

[4] Novak, E. and Ritter, K. 1996, Numer. Math. **75**, p. 79-97

CFD Approach to Bile Flow Problems Solution

Alex G. Kuchumov[a*], *Yuriy I. Nyashin*[a], *Vladimir A. Samartsev*[b]

a) Perm National Research Polytechnic University, www.pstu.ru, Russia
b) Acad. E.A. Wagner Perm State Medical Academy, www.psma.ru, Russia
*) kychymov@inbox.ru (corresponding author email)

Introduction

The human biliary system includes the liver, the gallbladder, the biliary tree, the sphincters, and it is responsible for bile producing, storing, and transportation (Fig. 1). 10–15% of people in the adult Western world suffer from gallstone disease [1]. According to this, the number of the post-operative complications is also high. It's related with a fact that it is not easy to predict bile flow in normal states comparing with pathological ones. The aim of this work is the development of computer and analytical models on the basis of biomechanical experiments, clinical observations and measurements, and contemporary visualization methods for description of the bile flow in the human biliary system. On the one hand, these models may be considered as a tool for assessment of risk of the pathology; one the other hand, they may contribute to improve clinical treatment methods.

Materials and Methods

Firstly, the investigation of the bile rheology was carried out. Bile specimens were taken from the gallbladder (gallbladder bile) and biliary tree (duct bile). Rheometer *Physica MCR* 501 (*Anton Paar*, Germany) was adopted to determine the rheological characteristics of the bio-fluid. Rheological tests were conducted at a temperature of +37° C. After that, knowing the bile properties of the normal and pathological bile, two problems were solved. The first problem deals with 3D computer simulation of the normal bile flow in the human biliary system. The biliary system geometry (Fig. 2) was constructed on the basis non-invasive methods (cholangiography and ultra-sound diagnostics). The flow analysis was performed by FEM in *Ansys Workbench* [2]. The second problem deals with analytical solution of non-Newtonian pathological bile flow in the Vater's papilla duct with a stone basing on the rheological constants adopted in the result of experiment.

Moreover, we decided to make a patient-specific model of the biliary system by developed program code on the basis of MRI image processing and special growth algorithm [3].

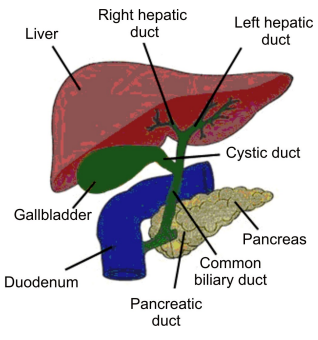


Figure 1: Biliary system scheme.

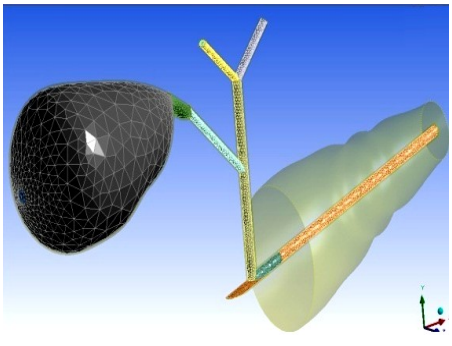


Figure 2: 3D biliary system model.



Figure 3: Bile sample.

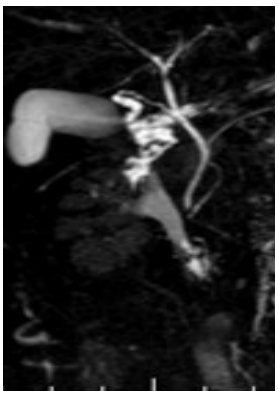


Figure 4: MRI of patient.

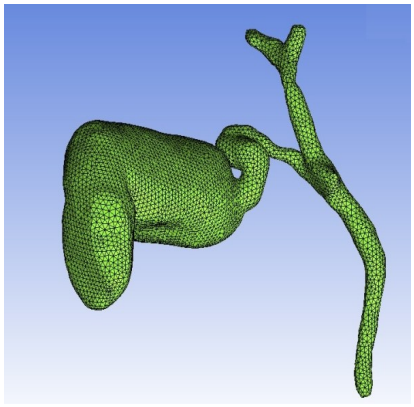


Figure 5: Patient-specific model.

Results

In the result of experiment series, it was shown that normal bile is Newtonian fluid, whereas the pathological bile is a thixotropic non-Newtonian fluid (Casson's and Carreau's equations parameters were obtained). On the basis problem related with normal bile flow, the dependences of flow resistance were obtained (velocity and pressure distributions and «pressure–flow rate» dependence). The obtained dependences play an important role in the study of the pathogenesis of gallstone formation. The results of second problem (pathological bile flow in the Vateri's papilla ampoule with a stone) solution contain a data on pressure and velocity dependences on the stone dimensions, which plays important role for surgical operation planning.

Discussion

The discussed bile flow simulations can be regarded as a virtual tool of normal and pathological bile flow assessment to predict stone formation in the gallbladder and cystic duct; moreover, the estimation of bile pressure in the Vateri's papilla with a stone is an objective measure of the effectiveness and adequacy of the biliary decompression and indications for operation on the major duodenal papilla and the common bile duct, when the pressure is high and daily bile amount is less than 35% of norm.

Acknowledgments

Authors acknowledgment V.A. Popov (Perm State National Research University) and A.P. Shestakov (Institute of Continuous Media Mechanics RAS) for discussions and help with experimental research.

References

[1] http://www.patient.co.uk
[2] http://www.ansys.com
[3] Del Fresno, M. et al. 2009, Comput. Med. Imag. Graph. **33**(5), p. 369-76

SPONSORS AND EXHIBITORS

Produkte, Service & Wissen

CADFEM – Mehr als Software!

CAE-Simulation in der Produktentwicklung eröffnet ungeahnte Möglichkeiten: Außer reduzierten Kosten und Entwicklungszeiten ist sie der Schlüssel für wegweisende Innovationen und die spürbare Erhöhung der Qualität neuer Produkte. CADFEM liefert dafür nicht nur die passende Software, sondern alles, was über den Erfolg von CAE-Simulationen entscheidet: Ausgesuchte Produkte bis hin zu kompletten IT-Lösungen. Kompetente Service-Leistungen wie Beratung, Schulung, Support und Engineering. Das neueste Wissen. Alles aus einer Hand.

Mit mehr als 25 Jahren Praxiserfahrung und dem Know-how von über 100 Simulationsingenieuren positioniert sich das mittelständische Familienunternehmen CADFEM als starker Partner von Industrie, Forschung und Lehre für die numerische Simulation in der Produktentwicklung.

In enger Partnerschaft mit der ANSYS Germany GmbH, dem ANSYS Competence Center CFD, unterstützt CADFEM als ANSYS Competence Center FEM in Zentraleuropa den führenden unabhängigen Hersteller von Simulationssoftware, ANSYS, Inc.. Dabei versteht sich CADFEM als Brücke zwischen dem Hersteller ANSYS, Inc. und dem Kunden: CADFEM Kunden erhalten ANSYS Simulationstechnologie so, dass sie sicher und ergebnisorientiert eingesetzt werden kann.

CADFEM ist Systemhaus und Ingenieurdienstleister in einem. Das ist die Basis für eine kompetente Unterstützung der Kunden mit CAE-Produkten, CAE-Services und CAE-Wissen. Aus einer Hand, abgestimmt auf individuelle Anforderungen, und vom Einstieg über den Ausbau der Simulation bis hin zur Umsetzung von Speziallösungen.

CADFEM®

Bild: Shutterstock.com/fotographie1980

InfoAnsprechpartner bei CADFEM

Deutschland | CADFEM GmbH

Software
Dr.-Ing. Volker Bäumer
Tel. +49 (0) 80 92-70 05-51
vbaeumer@cadfem.de

Seminare
Thomas Nelson
Tel. +49 (0) 80 92-70 05-47
tnelson@cadfem.de

Berechnung im Auftrag
Dr.-Ing. Marold Moosrainer
Tel. +49 (0) 80 92-70 05-45
mmoosrainer@cadfem.de

IT-Lösungen
Manfred Bayerl
Tel. +49 (0) 80 92-70 05-39
mbayerl@cadfem.de

Software Customization
Stefan Gotthold
Tel. +49 (0)30-475 96 66-24
sgotthold@cadfem.de

Hochschulen und Forschung
Dr.-Ing. Cord Steinbeck-Behrens
Tel. +49 (0) 511-39 06 03-17
csb@cadfem.de

Schweiz | CADFEM (Suisse) AG

Software
Markus Dutly
Tel. +41 (0) 52-3 68 01-02
markus.dutly@cadfem.ch

Seminare
Fabian Brändler
Tel. +41 (0) 32-6 75 80-72
fabian.bruendler@cadfem.ch

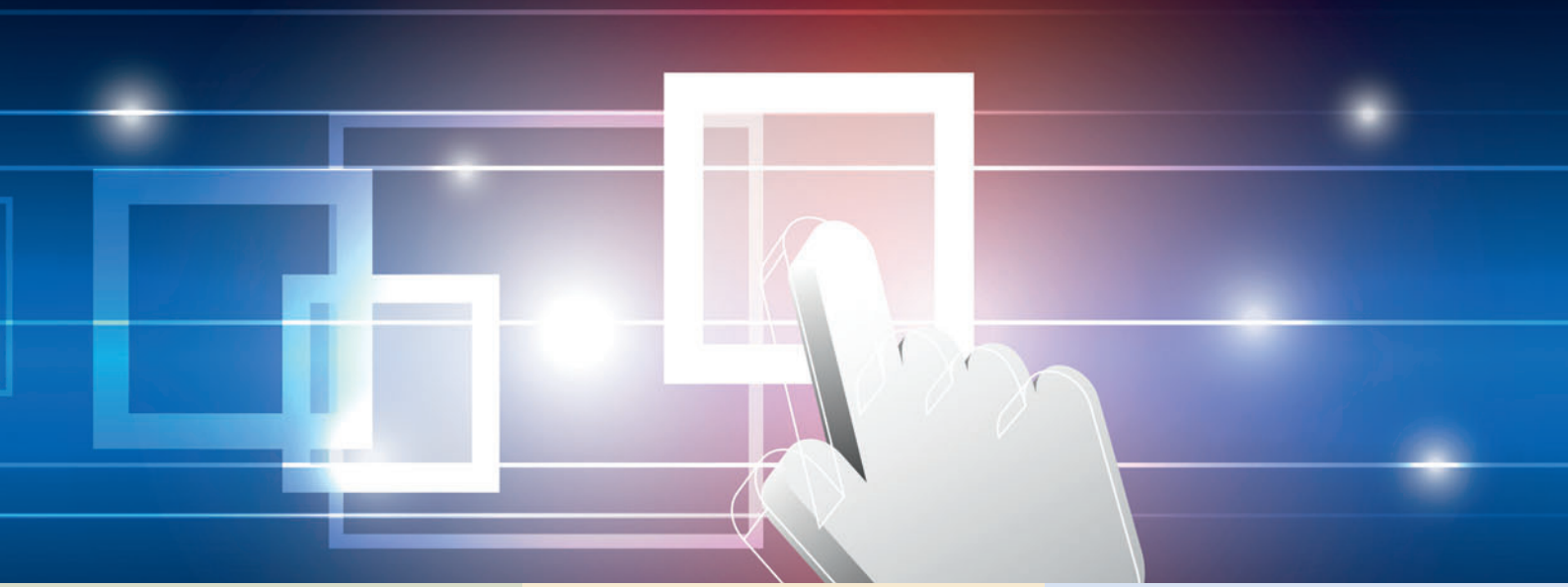
Berechnung im Auftrag
Dr.-Ing. Davide Valtorta
Tel. +41 (0) 52-3 68 01-01
davide.valtorta@cadfem.ch

Österreich | CADFEM (Austria) GmbH

Software
Alexander Dopf
Tel. +43 (0)1-5 87 70 73-14
alexander.dopf@cadfem.at

Seminare
Dr.-Ing. Bernhard Hößl
Tel. +43 (0)1-5 87 70 73-11
bernhard.hoessl@cadfem.at

Berechnung im Auftrag
Christoph Schlegel
Tel. +43 (0)1-5 87 70 73-12
christoph.schlegel@cadfem.at



Produkte

Sie erhalten von uns ein komplettes Portfolio von CAE-Software und IT-Lösungen führender Technologieanbieter. Von Einzelkomponenten bis hin zur individuellen, sofort einsetzbaren Komplettlösung.

ANSYS Software

Als ANSYS Competence Center FEM betreuen wir die Anwender in Zentraleuropa und sind bereits seit 1985 enger Partner von ANSYS, dem weltgrößten Anbieter von Simulationssoftware.
Mehr ab Seite 38
www.cadfem.de/ansys

Komplementäre CAE-Software

Unser Software-Portfolio deckt auch weiterführende Anforderungen ab, z. B. mit Produkten wie LS-DYNA, optiSLang, FTI FormingSuite, AnyBody, Diffpack, ESAComp sowie der CADFEM ihf Toolbox aus den CADFEM ANSYS Extensions.
Mehr auf Seite 37
www.cadfem.de/software

eCADFEM – CAE-Software on Demand

Mit eCADFEM nutzen Sie Simulationsprogramme besonders flexibel. Nur wenn Sie sie brauchen und solange Sie wollen. Eine wirtschaftliche Lösung, die schon mehr als 750 Kunden überzeugt hat.
www.ecadfem.com

CAE-Hardware & IT-Lösungen

Auspacken, einschalten, loslegen. Das können Sie mit unseren Hardware-Lösungen. Denn wir haben Ihr System bereits installiert, vorkonfiguriert und getestet. Auf Wunsch erhalten Sie auch umfassende Betreuung im Rahmen eines Service Level Agreements.
Mehr ab Seite 12
www.cadfem.de/hardware

Service

Damit Sie alle Vorteile der CAE-Simulation ausschöpfen können, bieten wir Ihnen umfassende Dienstleistungen. Von der Spezifikation von Simulationsaufgaben über die Auftragsberechnung bis hin zur Automatisierung Ihrer Simulationsprozesse.

Schulungen + Infotage

Mit regelmäßigen Seminaren und Informationstagen (über 8000 Teilnehmertage/Jahr) zu zahlreichen CAE-Themen hält Sie CADFEM auf dem neuesten Stand der Entwicklung. Denn aktuelles Know-how bringt Sie und Ihr Unternehmen weiter.
Mehr auf Seite 9 und 10
www.cadfem.de/seminare
www.cadfem.de/infotage

Auftragssimulation

Gerne unterstützen wir Sie, indem wir komplette Auftragsberechnungen einschließlich Nachweisführung, Ergebnisbewertung und auf Wunsch Know-how-Transfer für Sie durchführen.
Mehr ab Seite 22
www.cadfem.de/consulting

Support

Mehr als 30 CADFEM Ingenieure unterstützen Sie bei Bedarf im Tagesgeschäft. Etwa beim Prüfen und Bewerten Ihrer FEM-Modelle oder Workflows, damit Sie möglichst wenig Zeit verlieren und sich auf Ihre Ergebnisse verlassen können.
www.cadfem.de/support

Prozessautomatisierung + Customization

Wir etablieren unternehmensweite Standard-Workflows, mit denen selbst Nicht-FEM-Experten Routineaufgaben meistern. Ihre Vorteile: Reibungslose Abläufe und verlässliche Ergebnisqualität.
www.cadfem.de/customization

Wissen

Wir pflegen den Dialog mit Wirtschaft und Forschung, haben umfassende Weiterbildungsangebote, unterstützen lebenslanges Lernen und fördern den Erfahrungsaustausch unter Simulationsexperten.

CADFEM Users' Meeting

In Kooperation mit ANSYS richten wir die ANSYS Conference & das CADFEM Users' Meeting aus. Die Fachkonferenz mit jährlich mehr als 800 Teilnehmern vermittelt neuestes Wissen für Simulationsanwender.
Mehr ab Seite 2
www.usersmeeting.com

CADFEM esocaet

esocaet (European School of Computer Aided Engineering Technology) steht für CAE-Weiterbildung vom Seminar bis zum Masterstudium. Eine ideale Basis für die Planung oder den Ausbau von Karrierewünschen im CAE-Bereich.
Mehr ab Seite 48
www.esocaet.com

CADFEM Fachmedien

Das CADFEM Journal (das Sie gerade vor sich haben) ist unsere Kundenzeitschrift, mit der sich auch Simulationsinteressierte außerhalb unseres Kundenkreises über aktuelle Themen informieren. Online hat sich das CAE-Wiki als Nachschlageplattform etabliert.
www.cadfem.de/journal
www.cae-wiki.de

TechNet Alliance

Als Mitglied der TechNet Alliance hat CADFEM Zugang zu rund 1.000 CAE-Experten. Das globale Netzwerk zählt etwa 50 Unternehmen der Simulationsbranche aus mehr als 20 Ländern.
www.technet-alliance.com

Faszination Medizintechnik

Mit Begeisterung arbeiten wir mit Medizinern zusammen, um die Gesundheit von Menschen zu verbessern oder wiederherzustellen.

Besuchen Sie uns auf www.ulrichmedical.com









Wir sind Spezialisten in vier Produktbereichen:

- Implantatsysteme für die Wirbelsäule
- CT/MRT-Kontrastmittelinjektoren
- Blutsperregeräte
- Chirurgische Instrumente

Neue Produkte entwickeln wir stets im Bewusstsein, dass es Produkte für Menschen sind.

ulrich GmbH & Co. KG
Buchbrunnenweg 12
89081 Ulm
info@ulrichmedical.com
www.ulrichmedical.com

100 Jahre ulrich medical

Imprint

Publisher:

CBU Organizing Committee:
Martin Geiger[a]
Bernd Lapatki[a]
Jürgen Salk[b]
Ulrich Simon[c]
Hans-Joachim Wilke[d]

a) Department of Orthodontics (KFO), University Medical Center Ulm
b) Communication and Information Center (kiz), University of Ulm
c) Scientific Computing Centre Ulm (uzwr), University of Ulm
d) Institute of Orthopaedic Research and Biomechanics (UFB), University of Ulm

E-Mail: cbu.office@uni-ulm.de
WWW: http://www.uni-ulm.de/misc/cbu

Front page design:

Sabine Geller, Communication and Information Center (kiz), University of Ulm

Print:

Communication and Information Center (kiz)
University of Ulm
Albert-Einstein-Allee 11
89081 Ulm