CT image acquisition for implant modeling in bone biomechanics

Thomas Gross\textsuperscript{1}, Christian Gusenbauer\textsuperscript{2}, Alexander Synek\textsuperscript{1}, Dieter Pahr\textsuperscript{1}

\textsuperscript{1}Institute of Lightweight Design and Structural Biomechanics, Vienna University of Technology, Vienna, Austria, e-mail: tgross@ilsb.tuwien.ac.at

\textsuperscript{2}University of Applied Sciences Upper Austria, Wels Campus, Wels, Austria, e-mail christian.gusenbauer@fh-wels.at

Abstract
CT based finite element modeling allows to virtually test the rigidity of bone implant systems in a variety of loading cases. However, CT image acquisition is challenging due to a combination of biological tissue and titanium implant. Using an industrial micro-CT system, we were able to obtain high quality ex vivo scan data, which allowed us to accurately describe the geometry of the implant as well as the bone macro- and microstructure. Based on the CT scan a finite element model was generated and the implant was tested virtually in a physiological loading case.

Keywords: micro computed tomography, bone, implant, biomechanics, finite element method

1 Introduction
Traumatic bone fractures with segmental defects are a challenging problem in clinical surgery. To obtain an optimal healing process a stable but flexible fixation of the fractured bone is required. Despite good results in clinical practice, the mechanics behind volar plate osteosynthesis (see Fig. 1) is not fully understood. To study this system, CT image based finite element (FE) models can be used to virtually test the rigidity of the implant. However, the combination of biological tissue and titanium implant leads to challenges in CT image acquisition due to metal artefacts, image noise and lack of contrast between the air and bone phase. In this work, a high resolution CT image of a distal radius osteosynthesis was acquired ex vivo. Based on the CT scan, a finite element framework for virtual testing of the implant is proposed.

Figure 1: Clinical radiograph of a human distal radius fracture stabilized with a volar radius plate on the left; CT image based osteosynthesis model used to investigate the rigidity of the radius implant on the right.
2 Methods
One distal radius with a titanium volar plate implant was non-destructively investigated with X-ray computed tomography (CT) using an isotropic resolution of 40 µm. The scans have been performed with an industrial cone beam RayScan 250E CT-device, equipped with a Viscom 225 kV micro-focus X-ray tube with a minimal focal spot size of about 8 µm and a Perkin Elmer flat panel detector with 2048x2048 pixels. The scan parameters have been adjusted in order to ensure a minimum of image blurring caused by the focal spot of the X-ray tube. Therefore the electric power has been set in accordance with the chosen voxel size of 40 µm. Based on the CT scan (Fig 2 a.) a coarse tetrahedral finite element mesh was generated by means of an automated meshing algorithm. Each element was assigned a homogenized elastoplastic material behavior, dependent on the underlying bone microstructure [1]. The mechanical behavior of the influence zone surrounding the implanted titanium screw was calibrated using the results of a more detailed micro mechanical finite element model (Fig 2c.). The full implant was then virtually tested in a physiological loading case representing an axial loading of the forearm (Fig 2b.).

Figure 2: a) Sagittal slice of an ex vivo CT scan from a distal radius volar plate osteosynthesis; b) homogenized FE model of a fractured distal radius bone stabilized with a volar plate and subjected to axial loading. c) micromechanical FE model of the bone screw influence region.

3 Discussion
The CT scan could accurately image the medical implant as well as the bone macro- and microstructure (Fig 2a). Typical metal artefacts were kept at a minimum by using an adequate Cu pre-filter material, which allowed a detailed modeling of the bone/screw interface. First, a micro-mechanical FE model was used to obtain the mechanical behavior of the trabecular bone region around the titanium screw. Then, the rigidity of the full implant system was tested by means of a homogenized finite element model, incorporating the findings from the micrometer scale. Analysis of a physiological loading case showed good agreement with values found in experimental osteosynthesis tests [2]. Once validated, this modeling framework will allow to adress clinically relevant questions and therefore help to improve the quality of future bone fracture treatment.

References
[1] Pahr D.H., Zysset, P.K., From high-resolution CT data to finite element models, Computer Methods in Biomechanics and Biomedical Engineering, 12, 45–57, 2009