COMPUTATIONAL METHOD FOR EVALUATING FRACTURE-FIXATION STABILITY OF COMPLEX BONE FRACTURES

Simon Comtesse (1), Arvind von Keudell (2), Stephen J. Ferguson (1), Thomas Zumbrunn (1)

1. Institute for Biomechanics, ETH Zurich, Switzerland; 2. Department of Orthopaedic Geriatric Trauma Surgery and Research, Brigham and Women's Hospital, Harvard Medical School, Boston, MA, USA;

Introduction

Bone fractures cause about two million hospitalizations per year just in the US. Some of them are highly complex with complication rates up to 28% [1]. It is often unclear how the screws and plates shall be positioned for a specific fracture case and outcomes heavily depend on the surgeon's experience.

This implies that there is no objective measure, how a clinically accomplished reconstruction will perform biomechanically under loading.

Hence, our goal is to introduce a method for quantitative evaluation of fracture-fixation stability, by means of finite element analysis and musculoskeletal modelling.

Methods

Based on a pre-operative computed tomography (CT) scan, ten fractured bone fragments of a right proximal tibia were segmented and aligned to achieve adequate fracture reduction. According to the post-operative CT scan, 3D models of the used stainless-steel screws were designed and aligned to the bone fragments to reverse-engineer the clinical reconstruction.

Bone material properties for each fragment were derived from Hounsfield Units (HU) of the preoperative CT scan, based on internal density calibration with air, fat, muscle, and cortical bone [2,3]. The stainless-steel screws were modelled linear elastic with a Young's Modulus of 180 GPa and a Poisson's Ratio of 0.27.

Knee joint reaction forces and muscle forces were imported from a subject-specific musculoskeletal gait model during mid-stance phase (AnyBody Technology, A/S, Denmark) and implemented in the Finite Element Model (FEM) through pressure forces across the designated attachment surface (**Figure 1**). The distal part of the tibia was fixed in space.

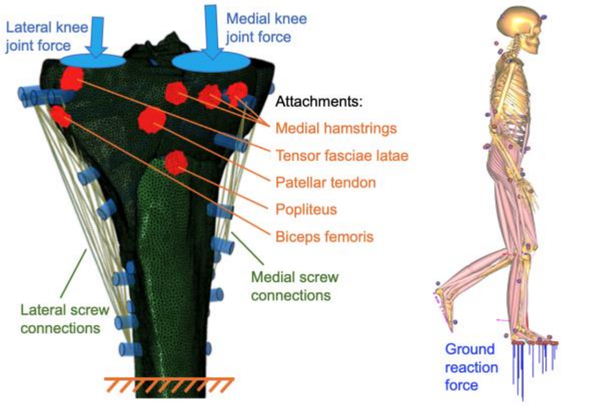

Figure 1: Setup of the FEM (left) with bone fragments (green) and screws (blue); mid-stance phase of the musculoskeletal model (right).

Respective screw heads (medial/lateral) were connected through beam elements representing the plate and the screw-bone interface was modelled with a tie constraint. A total of 994'399 linear tetrahedral elements were used, corresponding to a volume mesh density of 9.98. The Finite Element Simulation was run in Abaqus 6.14 (Simulia, Dassault Systemes, France).

Results

A maximum displacement of 1.62 mm was found at the top of the lateral fragment. The maximum von Mises stress of 423 MPa is located on the most distal screw on the lateral side (**Figure 2**).

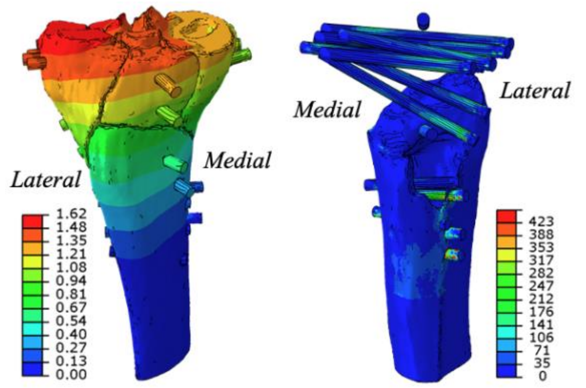

Figure 2: Displacement (left) and von Mises stress (right) of the simulated bone-screw construct.

Discussion

The method presented herein represents an objective process for the quantitative evaluation of fracture fixation stability. Our results allow prediction of bone fragment motion during daily activities, and if screws may be loaded beyond material capabilities.

After validation of the model, fragment movement could be related to fracture healing and serve as a predictive tool for clinical outcome. Possible hardware failure could be predicted by means of von Mises stresses in the screws. Furthermore, this process will enable development of patient-specific implants in the future.

References

1. Ruffolo et al, J Orthop Trauma, 29(2):85-90, 2015
2. Helgason et al, Med Eng Phys, 38(7):679-689, 2016.
3. Winsor et al, Bone, 143:115759, 2021.

Acknowledgements

Our gratitude to Ingmar Fleps, Enrico De Pieri, and Alex Baker from the Institute for Biomechanics for their support.