PREDICTION OF OVERLOADING FAILURE OF OSTEOSYNTHESIS PLATES USING VALIDATED FINITE ELEMENT SIMULATIONS

Dominic Mischler (1), Alessia Valenti (1,2), Peter Varga (1)

1. AO Research Institute Davos, Switzerland; 2. University of Bern, Switzerland

Introduction

The role of osteosynthesis is to maintain fracture reduction until bone healing is achieved [1]. However, plastic plate bending via overloading is a common failure mode in midshaft fractures, ranging up to 17% and often requiring reoperation [2,3]. Plate bending was also observed in a recent ovine study [4] for six out of eight animals. The aims of this study were to replicate the in-vivo situation in a cadaveric experiment and to validate a finite element (FE) simulation to predict plastic plate bending.

Methods

An established ovine tibial osteotomy model fixed with locking plates [4] was replicated in-vitro in six cadaveric bones (Figure 1). Two implant materials (titanium or steel) and three fracture gap sizes (30, 60, 80 mm) were used to investigate the bending behaviour of the fixation construct biomechanically. Axial alignment of the proximal and distal ends of the bones was ensured using a laser-guided embedding station. Axial compression testing until plastic deformation of the plate was performed (Instron 5866). Construct displacement was measured with Aramis SRX (GOM) to eliminate potential slippage errors or test setup compliance (Figure 1b).

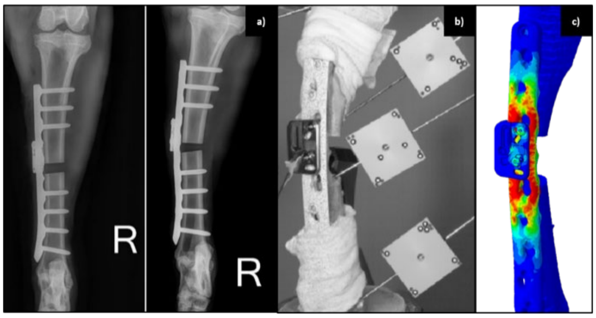

Figure 1: Plastic plate deformation occurring in vivo (a) [4], replicated experimentally in vitro (b), and simulated with FE modelling (c).

CT image-based specimen-specific FE models were created for each of the six constructs using Simpleware (Synopsis). Elastic and plastic implant material properties of titanium and steel were evaluated via uniaxial tensile testing of dog bone shaped samples extracted out of implant raw material profiles and incorporated into the models. The boundary and loading conditions were set to mimic the cadaveric experiment. The simulations were performed in Abaqus (SIMULIA). Stiffness, yield, and maximum loads were determined and compared between the experiment and FE models.

Results

Implant material properties determined for steel and titanium resulted in Young's moduli and yield stresses of 172.1 GPa and 887.7 MPa, and 104.3 GPa and 758.6 MPa, respectively. Yield and maximum loads of the constructs ranged between 469-491 N and 652-683 N, and between 759-995 N and 1252-1600 N for steel and titanium fixations, respectively. The FE models were able to predict the experimental results with high accuracy for stiffness (R^2=0.96), yield (R^2=0.97, Figure 2), and ultimate load (R^2=0.97).

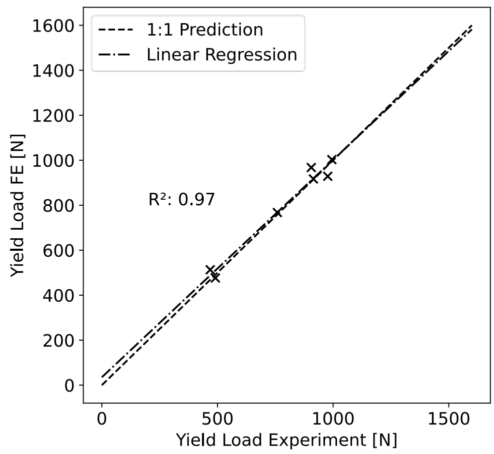

Figure 2: Linear regression analysis of yield loads between experiments and FE models.

Discussion

This study demonstrated the ability of FE simulations to accurately predict plastic plate bending in osteosynthesis constructs. As the construct behavior was predominantly driven by the implant, knowing the underlying material properties was vital. In future, these validated FE models could be used to predict the subject-specific load bearing capacity of osteosyntheses in preclinical or clinical studies.

References

1. Perren et al., Acta Chir. Orthop. Traum. Cech., 82, 2015.
2. Alzahrani et al., J Orthop Traumatol, 19:8, 2018.
3. Henderson et al., J Orthop Trauma, 1:S8-14, 2011.
4. Windolf et al., Medicina, 7:858, 2022.

Acknowledgements

This study was supported by the AO Foundation via the AOTRAUMA Network (Grant No.: AR2021_03).