FEM analysis of porous titanium bone scaffolds

W Hoffmann[1,2], S Fabbri[2], R Schumacher[2], S. Zimmermann[2], M de Wild[2]

[1] Institute for Surgical Research and Hospital Management, University Hospital Basel, Basel, Switzerland [2] University of Applied Sciences Northwestern Switzerland, School of Life Sciences, Institute for Medical and Analytical Technologies, Gruendenstrasse 40, CH-4132 Muttenz Switzerland, Waldemar.Hoffmann@fhnw.ch

INTRODUCTION: Porous implants are effective clinical choices for the surgical reconstruction of large bone defects [1]. The porosity has two advantages: Firstly, the stiffness can be adapted to the biomechanical loading situation and secondly, the interconnected pores allow vascularisation and therefore support osseointegration. The additive manufacturing method selective laser melting (SLM) allows the physical realization of complex open-porous titanium structures designed in a three-dimensional CAD file [2]. When constructing such a synthetic scaffold, its reaction upon external load has to be considered as a function of the macroscopic lattice geometry in order to optimize fatigue resistance, osseointegration and reduce stress shielding. The resulting structural and mechanical responses like deformation, stiffness and stress distribution are simulated with the numerical technique of finite element method (FEM) [3].

METHODS: The designed three-dimensional lattice consists of translational multiplications of rhombi-dodecahedral (*rdh*) unit cells filling out the entire implant volume, see fig. 1. In order to simulate the mechanical properties as a function of various lattice parameters (strut thickness, cell height, porosity), the scaffold geometry is based on a completely parameterized SOLIDWORKS™ CAD model. Using COMSOL™ Multiphysics 4.3, the CAD geometry and the FEM model is linked associatively. The elasto-plastic material properties of SLM processed titanium are derived from unidirectional static compression tests (Zwick/Roell Allround Line 100 kN, LaserXtens, testXpert II V3.4) [4]. In accordance to typical clinical situations, the applied load is defined as a compressive load. In the FEM simulation, a prescribed displacement is applied. The geometry is discretized by meshing with tetrahedral volume elements, see fig. 1. The elements are refined in the proximity of the geometrical nodes and are adapted to the minimal strut size by the mesh generator. The deformation and von Mises stress distribution of the biomaterial is calculated using the COMSOL™ solver.

RESULTS: As expected, the induced stress is maximised in the nodes, see fig. 2. From the arising reaction forces, the stiffness of the entire scaffolds can be derived in dependence of the parameterized dimensions.

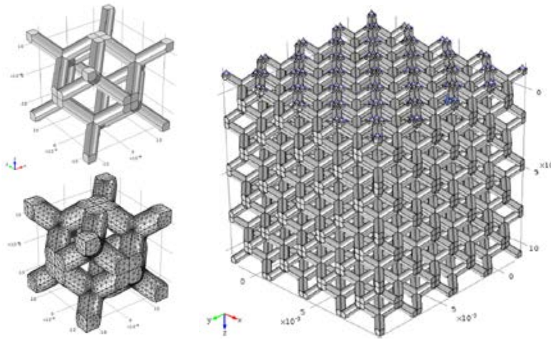

Fig. 1: Rhombi-dodecahedral unit cell, and cubic scaffold consisting of 4^3 rdh unit cells.

DISCUSSION & CONCLUSIONS: A numerical FEM simulation using the COMSOL™ software enables the characterization of the elasto-plastic behaviour of artificial porous structures under unidirectional compressive loadbearing. The stiffness as well as von Mises stress distribution can be evaluated in respect of unit-cell design.

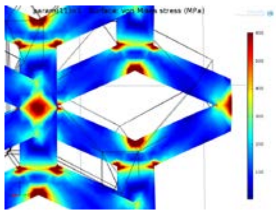

Fig. 2: Details of the deformation and the von Mises stress distribution in the nodes.

REFERENCES: [1] O. Hasart et al. (2010), *Oper Orthop Traumatol.* **22**(3):268-77. [2] T. Bormann et al. (2012), *Journal of Materials Engineering and Performance*, **21**(12), 2519-2524. [3] R. Singh et al. (2010), *Acta Biomaterialia* **6**;2342–2351. [4] P. Lamprecht (2012), bachelor thesis.

ACKNOWLEDGEMENTS: This project was supported by the HLS research fund.