A new SEM in-situ fatigue testing apparatus and its application for evaluation of fatigue damage at higher load cycles

Johannes Reiser [1,a], Bernd Maier [1,b], Hans-Peter Gänser [2,c],
Christoph Guster [1,d], Reinhard Pippan [3,e]

[1] Montanuniversität Leoben, Chair of Mechanical Engineering, Franz Josef Straße 18,
8700 Leoben, Austria,

[2] MCL Material Center Leoben Forschungs GmbH, Roseggerstraße 12, 8700 Leoben, Austria

[3] Erich Schmid Institute of Materials Science, Austrian Academy of Sciences, Jahnstraße 12,
8700 Leoben, Austria

[a] johannes.reiser@unileoben.ac.at, [b] bernd.maier@unileoben.ac.at, [c] hans-peter.gaenser@mcl.at,

[d] christoph.guster@unileoben.ac.at, [e] reinhard.pippan@oeaw.ac.at

Keywords: Fatigue, local deformation measurement, crystal plasticity

Abstract. The aim of this work is the investigation of damage evolution of materials under single step and variable amplitude loading in tests and corresponding simulations. Approaches for fatigue lifetime and fatigue limit prediction at the mesoscopic scale incorporate microstructure and grain features. This micromechanical approach coupled with polycrystalline plasticity should be able to take into account the contribution of grain orientation, grain shape, crystallographic orientation, as well as the influence of material defects. For the purpose of studying local plastic strain accumulation and therefore fatigue behavior at higher load cycles a new test rig for SEM in-situ tension-compression testing was developed by using finite element simulation coupled with topology optimization. Oxygen free high conductivity (OFHC) polycrystalline copper material with nearly uniform grain size distribution and no preferred grain orientation was chosen for the experimental investigations within this work. For investigation of local strain evolution during the test run digital image correlation was used. A simulation model of the chosen OFHC polycrystalline copper was set up. The model incorporates crystal plasticity with randomly distributed grain orientation and nearly uniform grain size distribution to match the properties of the tested specimens. Due to the high purity of OFHC copper the influence of grain boundaries can be neglected in the first phase of simulation. First test and simulation results are shown.

Introduction

In mechanical engineering nowadays the main goal is to design lightweight parts with knowledge of their lifetime depending on their application. Therefore extensive testing procedures are performed to get macroscopic information about the material behaviour for the calculation of S/N curves. The linear damage accumulation model of Palmgren-Miner works well for linear data, but is not able to take into account variable amplitude loading, or overloads. Also very low loads lead to local plastic deformation. From the global point of view there should be no plastic deformation in any one of the grains. The generation of data for fatigue lifetime prediction is performed regularly on servohydraulic or electromagnetic resonance test benches. The specimens tested in these machines have testing cross sections of many square millimeters and lengths of up to hundred millimeters. In-situ studying of damage evolution is very difficult unless the test is stopped and the specimen unmounted to perform micrographs, or a replica of the surface is picked. When performing replicas it is challenging to find the same area as it is with the whole specimen. Further it is not possible to study local deformation during a distinct test cycle. In this case a test bench for in-situ use should fulfill the needs to generate test data during the test even at the microscopic scale. Available test benches for in-situ use are operated via a spindle driven by a worm gearing, which limits the test frequencies so that investigation of material behaviour at higher load cycles is not possible. The task of this paper is to generate fatigue data and to compare this data with a simulation model to get an idea of fatigue from a microscopic point of view. To meet these demands a new test bench for SEM in-situ use was designed within this work by using finite element simulation in combination with topology optimization. The material studied was OFHC polycrystalline copper with nearly uniform grainsize distribution and equally distributed grain orientation. During the test local deformation analysis was performed with digital image correlation. For computation of a Voronoi tesselation for simulation purposes, the Multi-Parameter-Toolbox(MPT) for MATLAB 2009 by the Institut für Automatik, ETH - Swiss Federal Institute of Technology, CH-8092 Zürich, was used. A custom-programmed MATLAB 2009 routine calculates the vertices of each grain for further processing in the finite element software Abaqus 6.10-1. For automatic grain generation a custom-programmed Python code reads the calculated data points from the MATLAB routine and automatically generates the three dimensional representation of the Voronoi tesselation where each cell represents a grain with its own accessible element set. For the crystal plasticity simulations Abaqus 6.10-1 with Zmat Version 8.4.3 was used. Material data for OFHC copper were taken from literature.

Experimental test setup

SEM in-situ test rig. For generation of local deformation data depending on grain orientation and grain size, SEM in-situ testing is the key. Therefore a new SEM in-situ test bench depicted in Fig. 1 capable of achieving higher test frequencies than customarily available test benches driven by a spindle and worm gearing was developed.

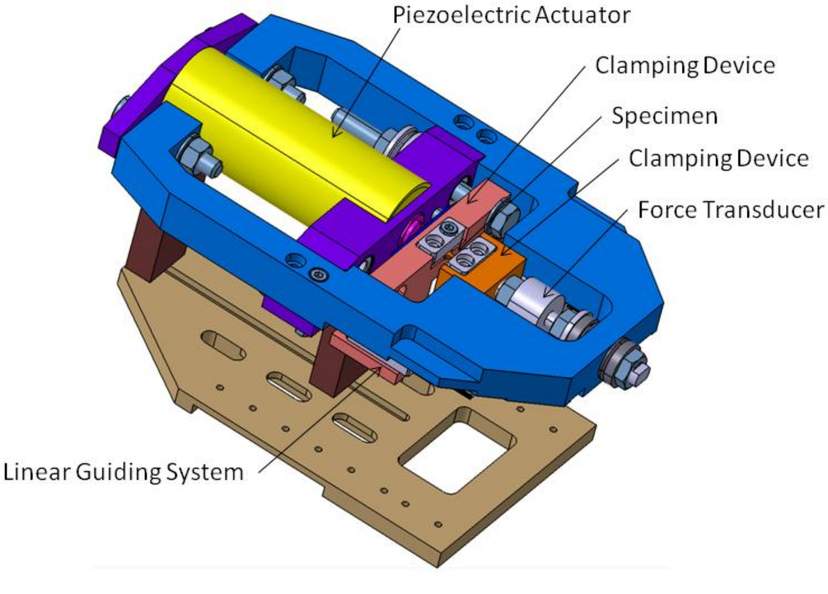

Fig. 1. SEM in-situ test rig

For operation inside high vacuum in the SEM chamber, a piezoelectric actuator was chosen, as it needs only an electric supply line and works without any pollution of the chamber. Global strain and force applied to the specimen are measured via an extensometer and force transducer. The load of the piezoelectric actuator is very high compared to the available range of displacement of the actuator. To make sure that there is no loss of elongation due to elastic deformation of the main frame, a finite element analysis of the whole test rig and especially of the main frame was performed. This frame holds the piezoelectric actuator which applies the load to the clamping devices that transfer it to the specimen. As the table of the SEM is not able to carry high loads, a lightweight and stiff design for the main frame is important. To achieve this design goal, the commercial topology optimization software tool TOSCA was used. As explained above, the load is applied by a piezoelectric actuator. To ensure, that the specimens are always clamped parallel and without any deviation at the bottom of the frame a linear guide system is used. This guide system consists of four linear guides two for each clamping device. The very low coefficient of friction of these guides secures very low losses. For controlling of the test rig a routine was written using National Instruments LabVIEW 2009. The program allows to control either the applied force or the actuator travel distance, and to select the shape of the amplitude curve, for example, sinus, triangle, or block signals. The routine is also able to perform step tests, with e.g., low loads at the start and higher loads after a distinct number of cycles followed by lower loads. This is necessary for studying the influence of overloads on the fatigue life of the specimen. To study local deformation digital image correlation was performed when the test was stopped at distinct points of the load cycle. Therefore high definition SEM images were taken. The displacement field is then obtained by using meX 5.1. In this paper 20 load cycles with constant strain amplitude of 0,045 in y-direction as depicted in Fig. 2 (a) were applied on the specimen.

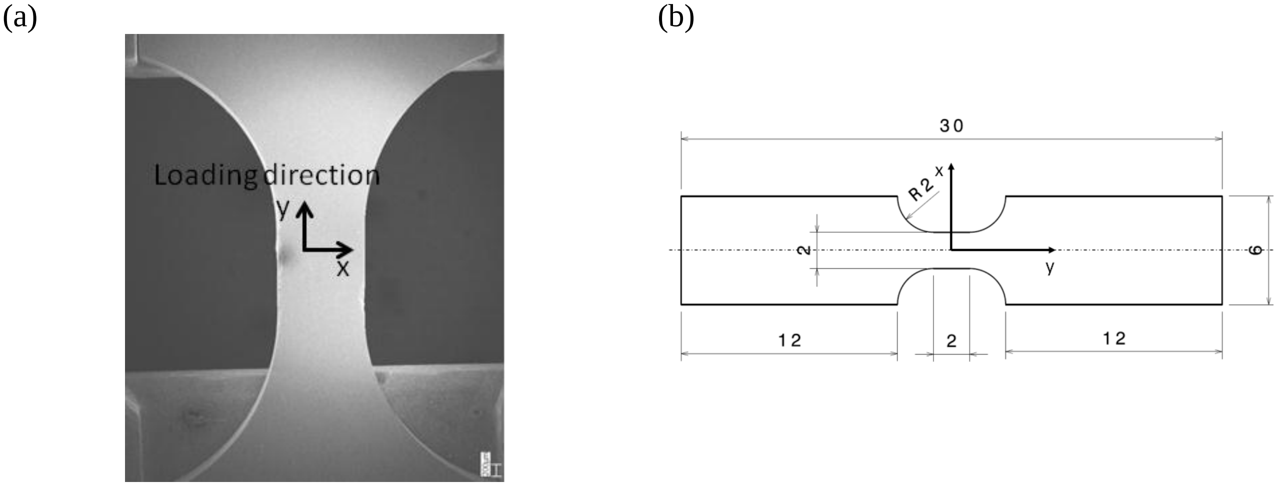

Fig. 2. (a) Coordinate system of the specimen and loading direction. (b) Geometry of the specimens used.

Specimen preparation routine. For the in-situ tensile tests, specimens with a quadratic cross section of 2 x 2 mm and a gauge length of 2 mm were used. Specimen geometry is shown in Fig. 2 (b). The material was grade 99,99 % OFHC copper delivered in form of rods with a diameter of 25,4 mm. For equal grain size distribution and no favoured grain orientation the rods were cold rolled before machining the specimens from these sheets, after milling, the specimens were heat treated for 1h at 500 °C followed by air quenching. This procedure leads to a relatively uniform grain size distribution and no favoured grain orientation, although the grain size is not completely homogeneous. Surface preparation was done by a sequence of grinding with SiC paper, mechanical polishing with diamond paste of grain size 1μm, 20 seconds electrolytical polishing and etching in Fe(III)Cl+HCl solution. In Fig. 3 (a) a microsection of the copper specimen is shown. A well-known problem with copper is the presence of twins, especially when thinking of modelling the grain structure with an automatic Voronoi tesselation. Due to the cold rolling process no favoured grain orientation is visible in the EBSD map in Fig. 3 (b).

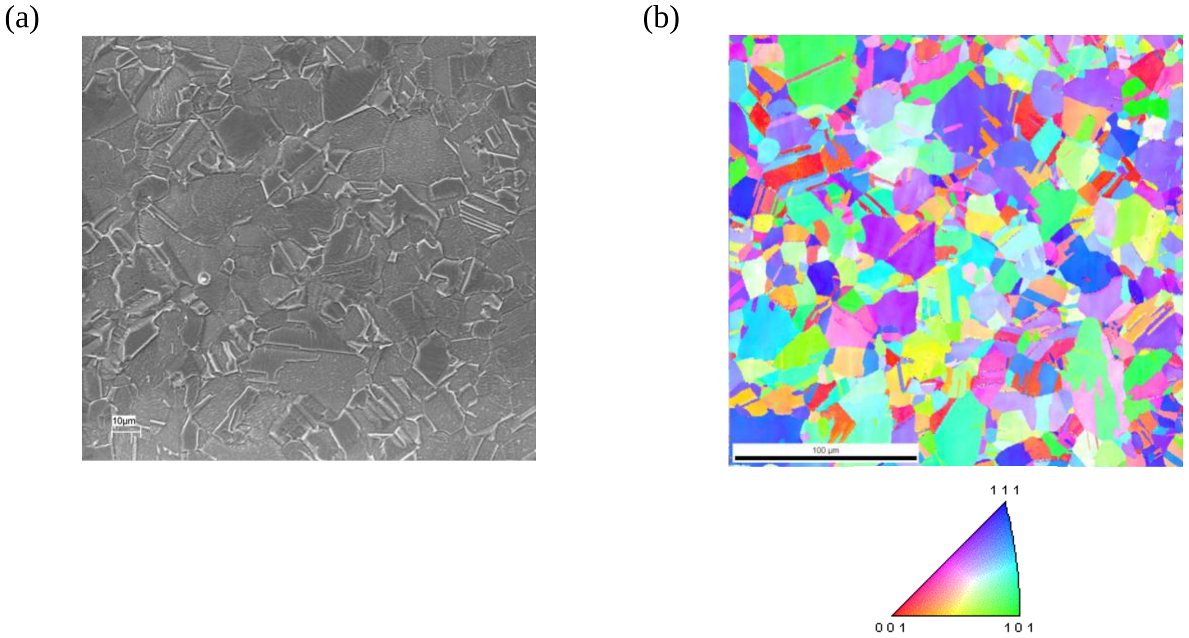

Fig. 3. (a) SEM image of copper specimen. (b) EBSD map of copper specimen.

Numerical Simulation

Generation of Voronoi tessellation. In this work, the focus is on studying the material behaviour when applying fatigue loads. Therefore, for reasons of computational speed, not a special EBSD scan will be rebuilt for use in a finite element simulation. Instead it is planned to use a Monte Carlo approach with randomly generated Voronoi tesselations in order to study the material behaviour in a statistical way. For generation of three dimensional Voronoi tesselations the MPT for MATLAB was used. The MPT computes a Voronoi tesselation of a uniform distributed point set. For further use of the computed Voronoi tesselation data a MATLAB 2009 routine was written, to get the dataset of each polyhedron representing a grain. As each grain consists of faces represented by polygons, the vertices and their dedicated connection direction is needed for automatic implementation into a finite element software. The faces of each polyhedron are splitted into triangles for further computing as shown in Fig. 4 (a). The length of the edges of the cube is 50µm.

Implementation of Voronoi tesselation into Abaqus 6.10-1. For implementation of the computed Voronoi tesselation into the finite element software Abaqus 6.10-1, a Python code was written which allows automatic grain generation by using the dataset computed by the MATLAB routine. The code automatically generates element and material sets for each grain for easy assignment of material parameters. Due to extensive computational capacity needed for computing polycrystalline plasticity, the finite element model in this work consists of 20 grains only. Each face represents a geometry set which allows application of (symmetry) boundary conditions.

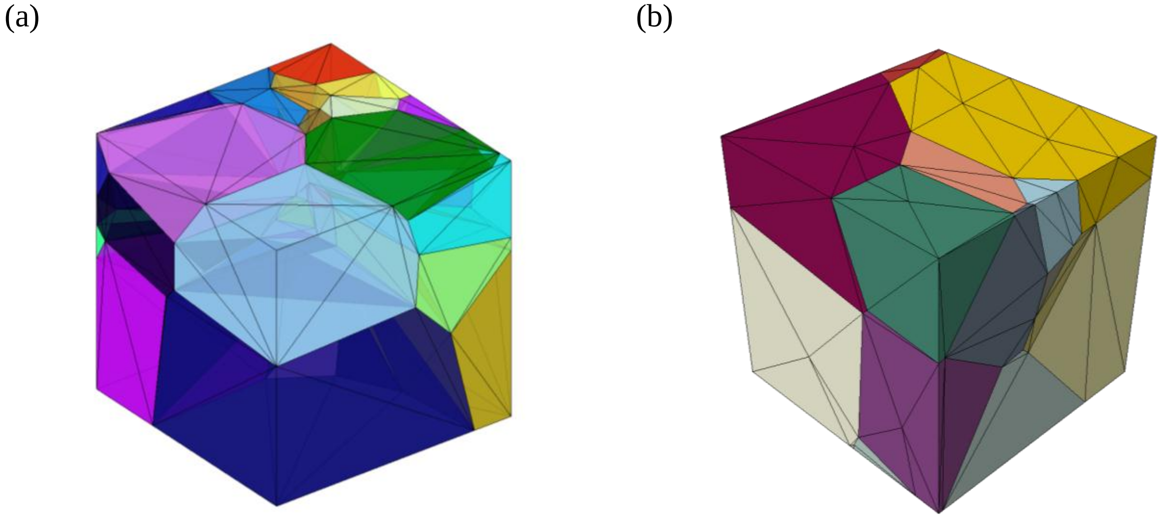

Fig. 4. (a) MPT Voronoi tessellation. (b) Finite Element representation of the generated Voronoi tessellation.

Crystal Plasticity The meshed aggregate of the finite element software is the input for simulation of crystal plasticity. The software used for simulation was Zmat Version 8.4.3. For a detailed description of the crystal plasticity model see [1, 2, 3]. The parameters used are shown in Table 1 and are the ones used in [3]; only the strain rate exponent was changed to a somewhat lower value of n =10.

Table 1. Material parameters of the single crystal model [3]

K [MPa.($s^{\frac{1}{n}}$)]	n	r_0	Q [MPa]	b	c [MPa]	d
8	10	15	4	12	32000	900
h_0	h_1	h_2	h_3	h_4	h_5	
1	1	0,2	90	3	2,5	

In a randomized routine each grain was assigned a crystal orientation. For simulation of the tension-compression test mirror symmetry boundary conditions were applied on the faces of the cubic representative volume element (RVE). A normal displacement was applied in z direction on the face –z. The simulation was performed for 20 displacement-controlled load cycles.

Results and Discussion

The evolution of global measured stress vs. globally measured strain is depicted in Fig. 5. As expected the stress shows a rise from the first to the last cycle at constant strain amplitude. Label A in Fig. 5 shows the strong hardening when comparing the peak stresses after first loading and after the first full cycle. Label B in Fig. 5 shows that hardening is still obvious from the first to the last cycle. But of course is much lower than in the first one.

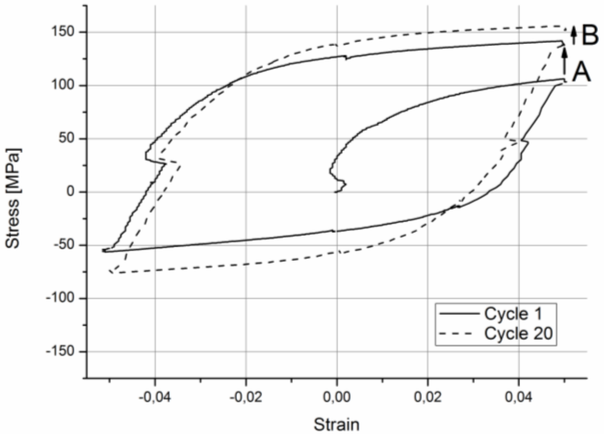

Fig. 5. Evolution of measured global stress vs. global strain from the first to the last cycle.

For comparing global parameters with local ones, local deformation analysis with meX 5.1 and a software code [5] was performed. During the whole test cycle, peak strain was kept constant, but local strains ε_{yy} in tension direction do show non-zero values as depicted in Fig. 6 (a). Also a tension strain field in ε_{yy} direction is noticeable in Fig. 6 (b) which corresponds to the peak of the first cycle to the peak of the last cycle.

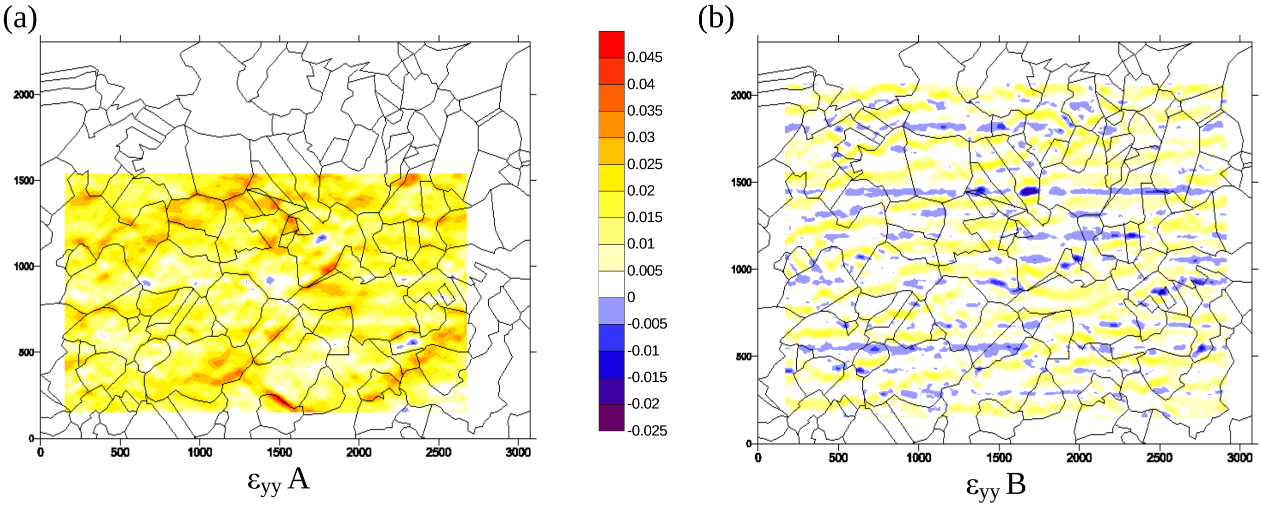

Fig. 6. (a) Local strain field εyy corresponding to A in Fig. 5. (b) Local strain field εyy corresponding to B in Fig. 5.

In Fig. 7 (a) the von Mises stress at the peak of the first cycle is shown, whereas Fig. 7 (b) depicts the von Mises stress distribution at the peak of the last cycle. As expected the stress states in the grains change from the first to the last cycle, and deformation localizes in distinct regions due to the differing grain orientations.

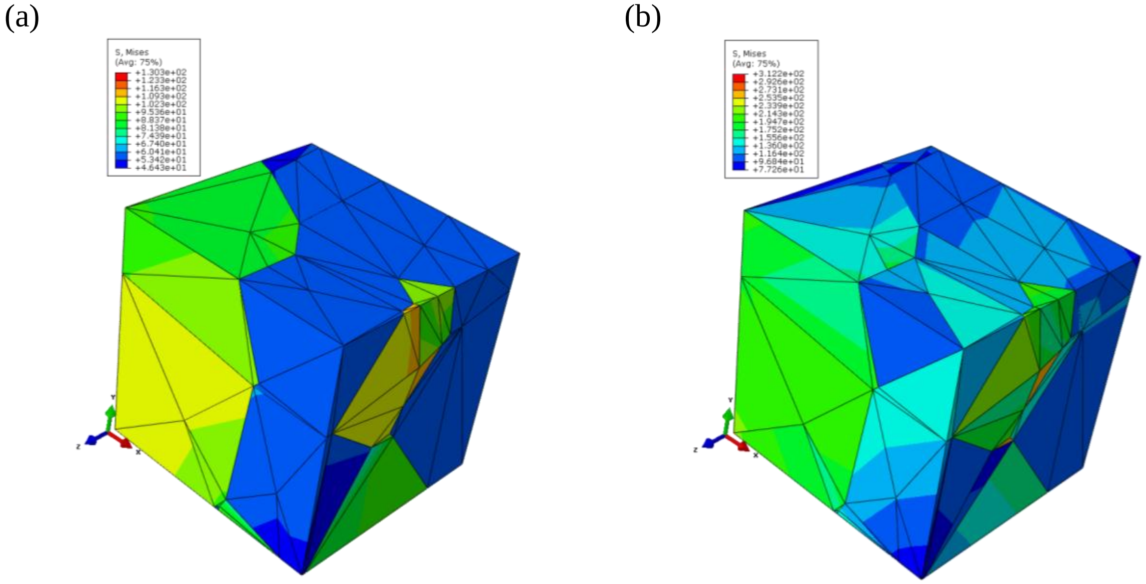

Fig. 7. (a) Von Mises stress distribution at the peak of the first cycle corresponding to A. (b) Von Mises stress distribution at the peak of the last cycle corresponding to B.

The evolution of the homogenized stress of the representative volume element over all 20 load cycles is depicted in Fig. 8. The displacement amplitude was kept constant, the stress increases steadily due to hardening. The simulated data show good accordance with the measured data in the tension phase shown in Fig. 5. In the compression phase the calculated data show a rather big difference. Therefore future simulation efforts will concentrate on the evolution of strain localization (cf. [3]) and on fatigue damage; of course, much more detailed RVEs will be used to this purpose.

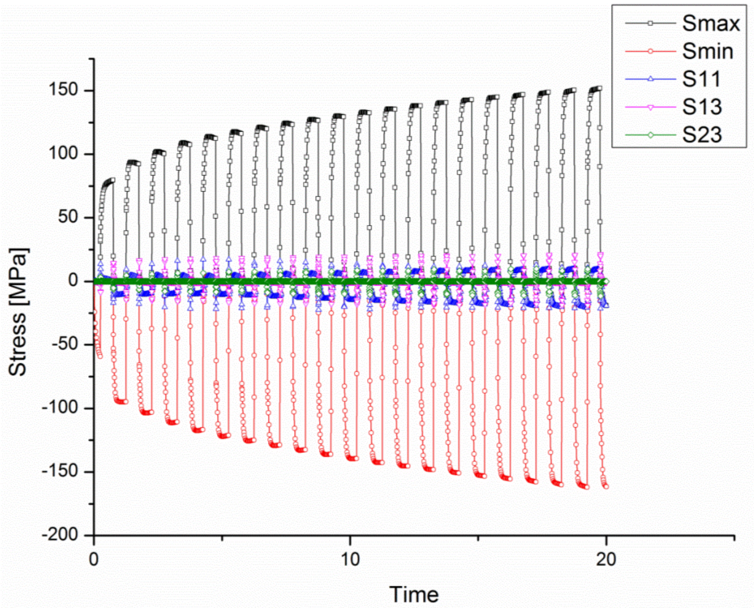

Fig. 8. Evolution of the calculated homogenized stress of the representative volume element.

Summary

A fatigue test rig for SEM in-situ use was developed for measuring strain localization at high load cycle numbers; the rig design was assisted by topology optimization. Tests were conducted on polycrystalline OFHC copper subjected to special heat treatment in order to obtain near-homogeneous grain sizes and orientation distributions. These tests were simulated numerically in a qualitative manner by using randomly generated representative volume elements (RVEs) consisting of 20 grains. For computational generation of a Voronoi tesselation representing a copper aggregate the MATLAB MPT was used. For automatic mesh generation from these Voronoi polyhedra, a MATLAB and Python routine was programmed. The simulations were performed using the finite element package Abaqus; for the numerical integration of the crystal plasticity constitutive equations the material library ZMAT was used. Future efforts will concentrate on the experimental and numerical assessment of the evolution of strain localization (cf. [3]) and on fatigue damage.

Acknowledgements

Financial support by the Austrian Federal Government and the Styrian Provincial Government, represented by Österreichische Forschungsförderungsgesellschaft mbH and Steirische Wirtschaftsförderungsgesellschaft mbH, within the research activities of the K2 Competence Centre on “Integrated Research in Materials, Processing and Product Engineering”, operated by the Materials Center Leoben Forschung GmbH under the frame of the Austrian COMET Competence Centre Programme, is gratefully acknowledged.

References

[1] F. Barbe, L. Decker, D. Jeulin, G. Cailletaud, “Intergranular and intragranular behavior of polycrystalline aggregates”. Part 1: F.E. model, International Journal of Plasticity 17, (2001), 513-536

[2] F. Barbe, S. Forest, G. Cailletaud, “Intergranular and intragranular behavior of polycrystalline aggregates”. Part 2: Results, International Journal of Plasticity 17, (2001), 537-563

[3] C. Gérard, F. N’Guyen, N. Osipov, G. Cailletaud, M. Bornert, D. Caldemaison, “Comparison of experimental results and finite element simulation of strain localization scheme under cyclic loading”. Computational Materials Science 46, (2009), 755-760

[4] A. Musienko, A. Tatschl, K. Schmidegg, O. Kolednik, R. Pippan, G. Cailletaud, “Three-dimensional finite element simulation of a polycrystalline copper specimen”. Acta Materialia 55, (2007), 4121-4136

[5] A. Tatschl, “Neue experimentelle Methoden zur Charakterisierung von Verformungsänderungen”. Dissertation, Montanuniversität Leoben, 2000