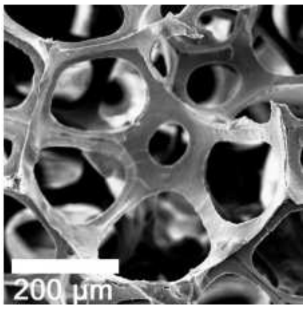
Diplomarbeit / Masterarbeit



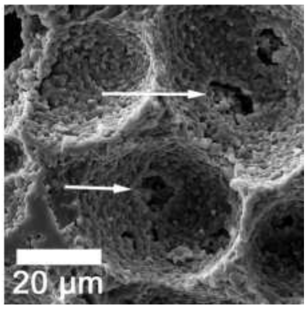


Fig.: Microstructure of the ceramic preform before and after infiltration

Development of a multi-surface smeared crack model with consideration of crack closure effects and implementation into ABAQUS

The development of high-tech composites has attracted gradual importance during the past decade. So-called "Preform Metal-Matrix-Composites" (Preform-MMCs) represent one of these materials. They consist of a porous ceramic matrix (the *preform*) that is in very close geometric accordance to the final part already. This preform is then interpenetrated by a ductile metal such as aluminium (Al). The resulting composite material has adjustable physical properties: thermal expansion coefficient, electric conductivity and increased creep resistance under thermo-mechanical loading to mention only a few. Field of application for such materials are e.g. parts in high performance engines, gearboxes and in bearing industry.

The numerical simulation of the thermo-mechanical behaviour of such materials is a challenging field of research. First, the microstructure of such materials is not periodic. This disqualifies a large class of homogeneisation techniques for application to the considered materials. Second, the two constituents show large differences in their physical properties and particularly in their inelastic behaviour. The plastic behaviour of the metal phase has extensively been studied in the past while the damage behaviour of the brittle ceramic preform is still subject of current research.

To model the inelastic behaviour of an e.g. Al_2O_3 preform, we intend to use a *multi-surface smeared crack model*. It depicts the initiation and growth of micro-cracks by treating them as a single *smeared* crack with changing orientation and size. The cracked structure then shows *anisotropic* elastic response. Additionally unilateral effects can be observed: if the material is subjected to compression, the micro-cracks can be closed and the initial stiffness of the material is (in good approximation) recovered. The Finite Element Method (FEM) is intended to be used for the numerical part of the work. Therefore a subroutine for ABAQUS/Standard should be developed. Several numerical simulations and a discussion of the results will report on the suitability of the model for the application of the damage model in the context of interpenetrated preform MMCs.

The thesis is scheduled for a total of four to five months. Basic knowledge of tensor calculus is needed. A brief tutorial will be given at the beginning if required. In order to enhance language abilities, we encourage the elaboration of the thesis in English.

Literatur

[1] Govindjee, S.; Gregory, J. K.; Simo, J. C. Anisotropic modelling and numerical simulation of brittle damage in concrete. *International Journal for Numerical Methods in Engineering*. Vol. 38, p. 3611–3633, 1995.

Betreuer

Dipl.-Ing. Felix Fritzen fritzen@itm.uni-karlsruhe.de
Prof. Dr.-Ing. Thomas Böhlke boehlke@itm.uni-karlsruhe.de