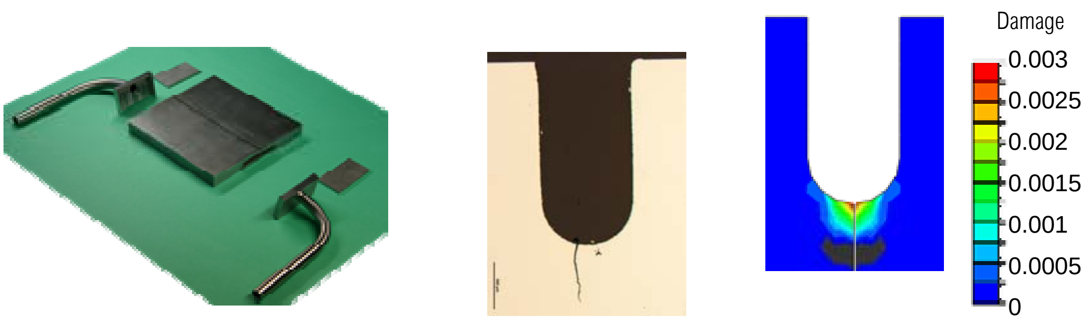
HIGH HEAT FLUX FATIGUE DAMAGE BEHAVIOR OF EUROFER97 STEEL

J.H. You[1*], T. Höschen[1], G. Pintsuk[2]

[1] *Max-Planck-Institut für Plasmaphysik, EURATOM Association, Boltzmannstr. 2, 85748 Garching, Germany*
[2] *Forschungszentrum Jülich, Institut für Energieforschung 2, EURATOM Association, 52425 Jülich, Germany*

Corresponding author: you@ipp.mpg.de

Currently Eurofer97 steel is considered as the main structural material for fusion reactors. For mechanical characterization extensive low cycle fatigue tests have been performed and the material data for an appropriate visco-plastic constitutive law were compiled for the expected operation temperature range [1]. The material parameters are identified by means of numerical regression using cyclic test curves and stress relaxation curves which are produced normally under uniaxial and isothermal testing conditions. On the other hand, the structural components of a fusion reactor will experience complex stress states with a transient thermal history under fusion-relevant thermal loads. When a finite element analysis of structural behavior is made, the input data of the material parameters are supposed to be interpolated and adapted to the temperature of an instantaneous load increment. However, it is still an open question if and how the realistic loading condition will affect the damage initiation and evolution laws. This may be a critical issue of material-design interface. Aim of this study is to test the performance of the identified constitutive law of the Eurofer97 steel under actual fusion loading conditions. A comparative investigation was carried out based on FEM simulation and experiment. Focus was put on the damage and fracture behavior. To this end, ad-hoc designed, dedicated water-cooled test modules with a surface notch were fabricated. Thermal fatigue experiment was conducted with high-heat-flux load cycles in an electron beam irradiation facility (JUDITH). To trace the damage evolution and crack growth 10 different fatigue loading conditions were tested and microstructure analysis was made. Finally the observed features of micro-crack initiation and propagation were compared with the prediction of finite element analysis using the material parameters and the constitutive model reported in [1].



(a) (b) (c)

Figure 1. (a) Test module before welding, (b) Metallographic section after test under heat flux of 3.5 MW/m² for 10s and 800 load cycles, (c) FEM simulation result showing damage parameter distribution

[1] J. Aktaa et al., Fusion Eng. Des., 81, 2006, pp 2221–2231