Christian Sager

Student	Christian Sager
Examiner	Prof. Dr. Pierre Jousset
Subject Area	Innovation in Products, Processes and Materials - Industrial Technologies

Fracture Mechanic Applied to Compact Tension Specimen Using FEM

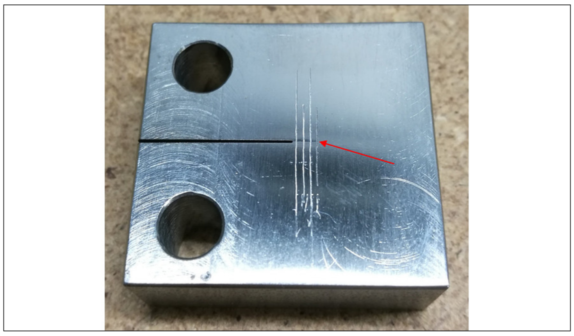

Fig. 1: CT specimen with fatigue crack
Own presentment

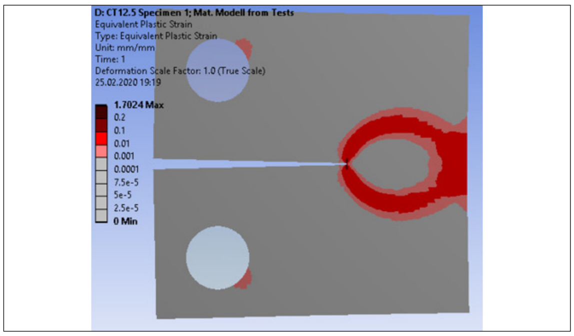

Fig. 2: Equivalent plastic strain at last step
Own presentment

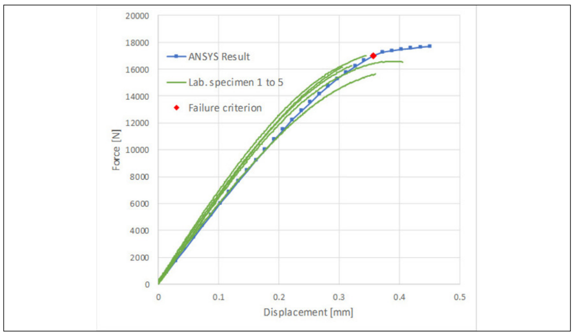

Fig. 3: Force displacement diagram
Own presentment

Introduction: Fracture mechanics deals with fracture and failure concepts in engineering materials and structures. Contrary to the theory of strength of materials in the field of fracture mechanics, it is assumed that every structure contains cracks due to manufacturing or service loads. In assessment concepts, fracture mechanics plays a role in evaluating the results of non-destructive testing (NDT), avoiding brittle fracture and in determining inspection intervals. In the case of significant plasticity at the crack tip, which is the case in ductile materials, the concept of EPFM (elastic plastic fracture mechanics) is necessary and the method of finite elements (FEM) must be used to determine the load at the crack tip since this non-linear problem is not solvable with analytical methods. The load at the crack tip is usually expressed with the stress intensity factor K for small plasticity and the j-integral J when the effects of plasticity are no longer negligible.

Problem: In order to gain an understanding of the design of pressurized piping systems in nuclear power plants the present work uses the commercial program ANSYS Workbench R2 to set up a FEM model. Furthermore, laboratory tests are carried out to derive the material data required for the FEM and to compare the test results with the FEM results. The structure modeled in ANSYS is a CT12.5 specimen (Fig. 1, Compact tension with 12.5 mm thickness) which is a standard specimen used in laboratories to derive fracture mechanic material data. The specimen is modeled using a non-linear 2D-Analysis (Fig. 2) assuming plane strain condition while the crack is modelled directly on the geometry using a pre-meshed crack. Two types of laboratory tests were performed using the material S235J2 (1.0117), which was chosen because of its high toughness values with the aim to get a significant amount of ductile crack growth at room temperature. First, uniaxial tensile tests using flat tensile specimen were performed to derive the hardening curve and the strength of the material. Additionally, fracture mechanic tests were conducted using side grooved CT12.5 specimen with a fatigue induced crack and evaluated according to ASTM E1820. The side grooves increase the stress triaxiality at the crack tip and therefore justify the assumed plane strain condition in the FEM analysis. This test provided fracture mechanic properties such as the critical fracture toughness J_{Ic} and the force-displacement curve.

Result: Consequently, a comparison between the force-displacement curve from FEM and the laboratory test was done (Fig. 3). As a failure criterion in ANSYS the critical fracture toughness was used. The test curves and the calculated curve show a very good agreement. Plastic zones are developing around the crack tip (Fig. 2) which leads to the formation of a plastic hinge and subsequently to the flattening of the force-displacement curve. The verification of the FEM model was performed using formulas from literature to calculate the stress intensity factor in the range of low load with only a small amount of plasticity present. The deviation between numerical and analytical stress intensity factor is only 1.2%.