Transactions of the SMiRT 27
Yokohama, Japan, March 3-8, 2024
Division III

THERMO-MECHANICAL FE SIMULATIONS ON A FULL-SIZE THERMAL FATIGUE PIPE TEST IN VIEW OF LIFETIME ASSESSMENT

Jens Arndt[1], Klaus Heckmann[1], Jürgen Sievers[1], Tim Schopf[2], Fabian E. Silber[2], Georg Veile[2], Stefan Weihe[2], Kai Donnerbauer[3], Marina Macias Barrientos[3], Frank Walther[3], Tobias Bill[4], Peter Starke[4], Ruth Acosta[5], Bharath Yerrapa[5], Christian Boller[5]

[1]GRS gGmbH, Cologne, Germany
[2]University of Stuttgart, MPA, Stuttgart, Germany
[3]TU Dortmund University, Chair of Materials Test Engineering (WPT), Dortmund, Germany
[4]University of Applied Sciences, WWHK, Kaiserslautern, Germany
[5]Saarland University, LZfPQ, Saarbrücken, Germany

INTRODUCTION

The Fukushima Daiichi nuclear power plant accident in 2011 led to a review of the design concepts and operating conditions of operating nuclear power plants worldwide, taking the entire plant infrastructure into account. In these so-called stress tests, the ability to control accident scenarios was assessed. The analysis of the Fukushima event and the safety standards of domestic and international nuclear facilities has led to a variety of changes in the operation of nuclear power plants (ONR, ENSREG).

As a result of load following operation, structures and components are exposed to thermal and mechanical load variation, which causes thermo-mechanical fatigue. This triggers an increased need for testing and evaluating the safety and integrity of structures and components, which is primarily associated with methods of material characterization and non-destructive testing (NDT) as well as structural mechanical analysis methods. While characterization, NDT and also monitoring can be developed on laboratory/specimen level, it is finally to be demonstrated at a component level. A thermo-mechanical fatigue test at component level poses experimental challenges for the test facility, as the evaluation does for the computational analysis.

In this paper, such a component test is analyzed. A pipe section was to be subjected to thermal loading under realistic conditions, whereby significant fatigue damage was to be caused by alternating loads. The pipe segment was instrumented with thermocouples, strain gauges. The FE (Finite-Element) code LS-DYNA was used to simulate the occurring stress state.

DESCRIPTION OF TESTING SETUP

The Fluid-Structure-Interaction (FSI) test rig at MPA Stuttgart is a multi-purpose test facility for fluid structure interaction tests, including investigations on thermal mixing phenomena, flow pattern, thermal fatigue and leak rates (Weihe et al., 2022). The test rig can be operated up to a pressure of 8.0 MPa at up to 280 °C temperature. The FSI test rig is depicted in Figure *1*.

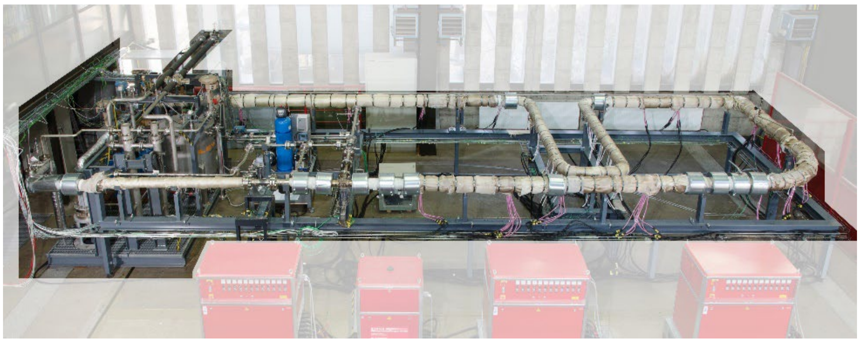

Figure 1. FSI test rig of the MPA University of Stuttgart (Swacek et al., 2022).

The installed pipe section (test planning phase: outer diameter 88.9 mm, wall thickness 10 mm) is a straight pipe made of austenitic steel (AISI 347/1.4450), made from nuclear grade stock material of a plant. The drawing of the pipe section is shown in Figure *2*.

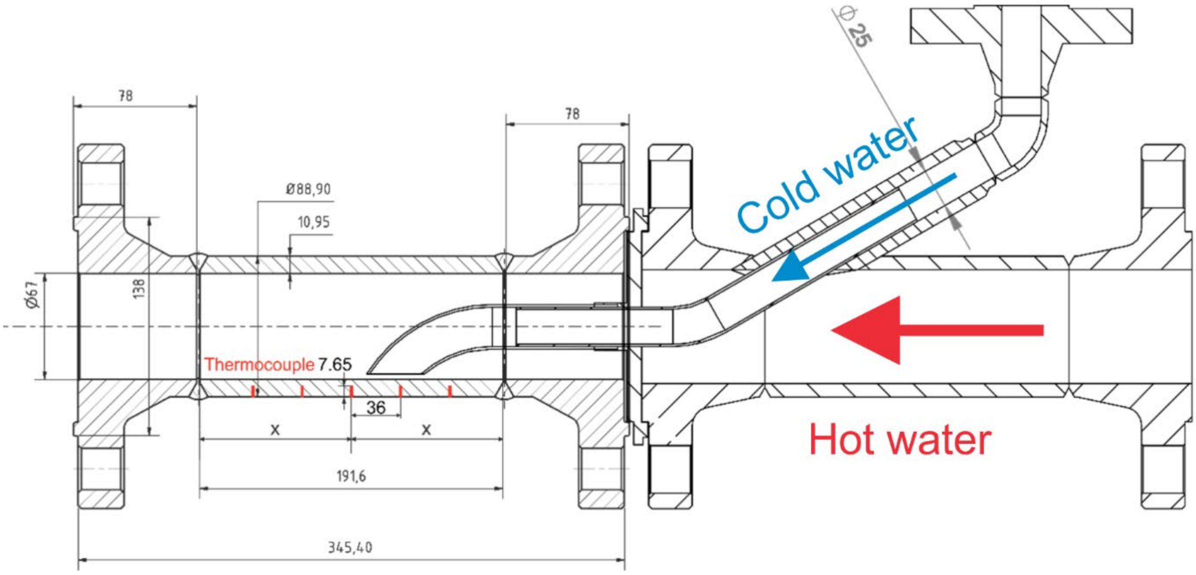

Figure 2. Technical drawing of the pipe section (MPA).

The initial state of the inserted pipe segment is pressurized water at 7.5 MPa and fluid temperature of ca. 265 °C (Weihe et al., 2022). Cold water of ca. 25 °C is injected by a Y-piece into the hot main loop for a selected duration of 30 s. By use of an inner elbow pipe, (see Figure 2) laminar flow in the main loop can be better penetrated by the cold water. Repeated cold-water-injections, interrupted by reheating phases, cause thermal fatigue effects on the piping at 6 o'clock position. The diameter of the inner piping is approx. 25 mm.

In this configuration, a pipe section can be tested for several thousand thermal cycles, each of a duration of one minute.

Thermal sensors were attached to the test component at several positions in the longitudinal direction and distributed around the circumference, some of which measure the fluid temperature (F) and some the solid temperature (S) in the area close to the wall. The solid thermocouples measure the temperature approximately 1 - 1.5 mm from the inner wall of the test component. The thermal sensors of the fluid measuring points protrude about 1 mm into the flow and record the temperature in the boundary layer (Figure 3).

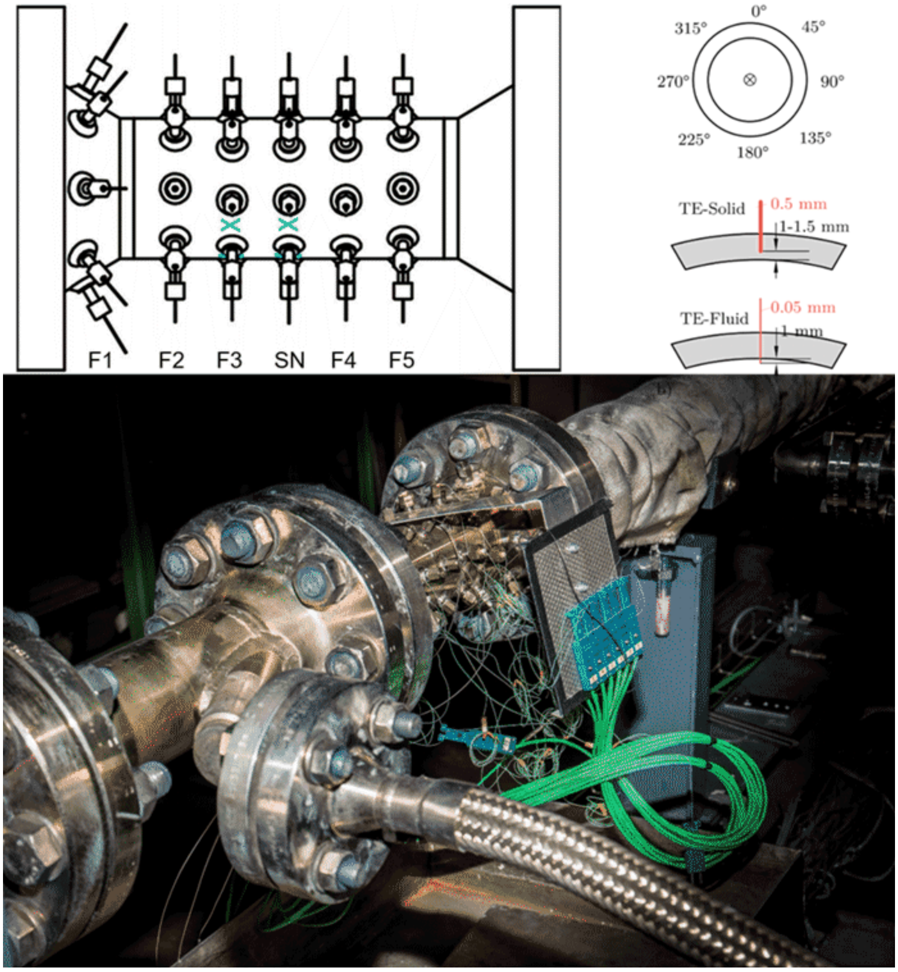

Figure 3. Test component with positions of thermal sensors (MPA).

POST-TEST EXAMINATION

Non-Destructive Testing

After the component test, the pipe segment is extracted and examined.

Destructive Testing

Specimens for fatigue tests are extracted from the pipe segment after the test campaign in the FSI test rig. With a pipe wall thickness of 10.0 mm, centered specimens with a diameter of 5.3 mm result in a distance of 2.35 mm from the inner surface.

The destructive post-test examination had not yet been carried out at the time of paper submission.

DESCRIPTION OF THE ANALYSIS MODEL (IDEALIZED APPROACH)

The objective of the simulation was to determine the local stress-time history at specific locations in the pipe segment. The simulation of the piping module by coupling CFD and thermo-mechanical FE codes would require a high effort of modelling and time-consuming computations times. Hence a simplified simulation method which is based on the thermo-mechanical FE method was developed, which is verified by temperature measurements. This approach simplifies the pipe section geometry and omits the fluid-mechanical part by considering the measured fluid conditions. The simplified FE model of the tube segment with cooling strip is shown as a half-model in Figure 4, assuming appropriate axial symmetry.

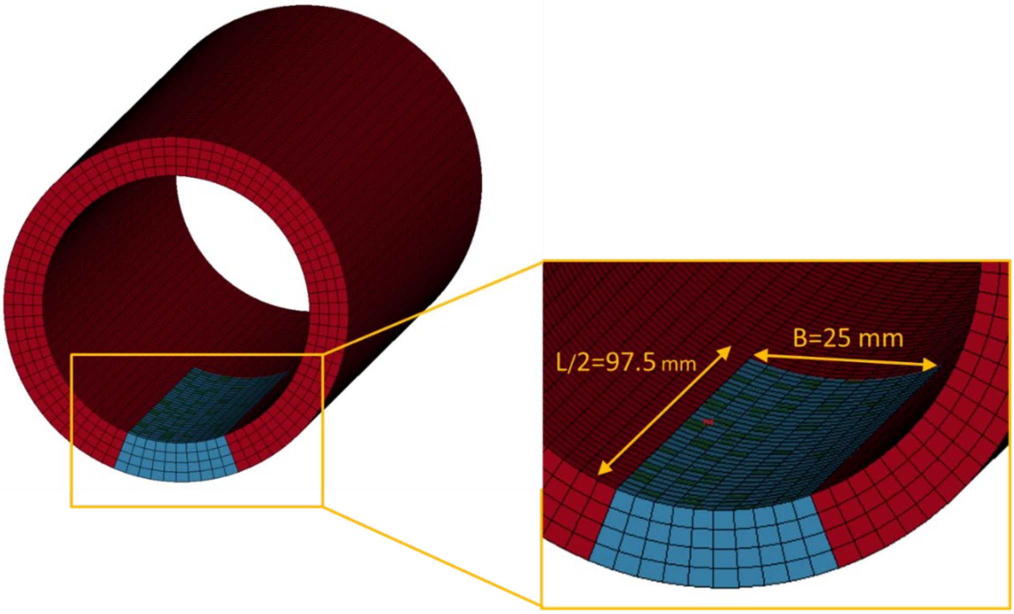

Figure 4. FE model of the pipe segment with cooling strip (half geometry).

THERMAL SIMULATION OF THE COMPONENT TEST

The model consists of 22176 solid elements and 330 shell elements, which transfer the cyclic thermal load to the structure of the tube segment in the area of the cooling strip via a contact definition. The total number of nodes is 28514. In the first approach, the cooling stripe is assumed at 6 o'clock position. The size of the stripe approximates a flow entrainment of the injected cold water. Further the constraints described below were assigned to the model. The model is supported on the bottom (x-y-plane) on a "sliding plane" in the z direction. This restricts any possible bending of the pipe segment. Likewise, the cross-sectional area facing away from the cooling strip is fixed in the axial (x) direction, which is intended to approximate the large stiffness of the flanges. The nodes on the symmetry surface are coupled in the axial direction, i.e., all nodes have the same x-displacement. As a thermal boundary condition on the outer surface of the pipe segment, it is assumed in a simplified way, that there is no heat exchange with the environment (thermally isolated). On the inside, an alternating load in the form of a step function results as a temperature boundary condition (Figure 5). A sequence of thermal cycles, following an initial uniform temperature (265 °C = 538 K), is simulated.

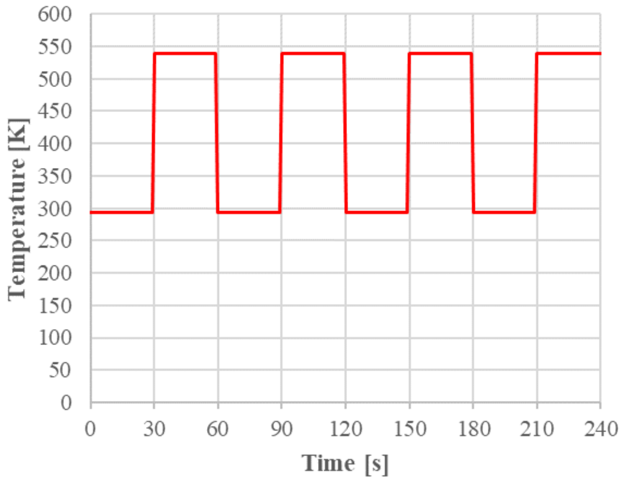

Figure 5. Applied alternating thermal loading.

The simulation performed with the FE program LS-DYNA was thermo-mechanically coupled. For this purpose, both the mechanical and the thermal FE solver require suitable material models. The material model *MAT_ELASTIC_PLASTIC_THERMAL enables the consideration of temperature-dependent, mechanical characteristic values for Young's modulus, Poisson's ratio, thermal expansion coefficient, yield stress and tangent modulus. The used thermal material model includes the heat capacity and the thermal conductivity of the employed material (*MAT_THERMAL_ISOTROPIC). The existing, temperature-dependent expansion coefficients, which refer to a reference temperature of 288 °C (Sievers et al., 1999), were adapted to a lower reference temperature of 265 °C. In Table 1 the values of the temperature support points calculated in this context are listed.

Table 1: Adapted coefficients of thermal expansion.

T [°C]	α_T $T_{reff} = 288$ °C	α_T $T_{reff} = 265$ °C
0	1.50E-05	1.470E-05
20	1.50E-05	1.470E-05
100	1.60E-05	1.569E-05
200	1.70E-05	1.655E-05
300	1.90E-05	1.855E-05
400	2.10E-05	2.055E-05

Table 2 summarizes the relevant material properties employed in the material model *MAT_ELASTIC_PLASTIC_THERMAL for the mechanical calculation.

Table 2: Material parameters used in the mechanical calculation from (Sievers et al., 1999).

Temperature [° C]	0	20	100	200	300	400
Young’s modulus [Pa]	1,95e11	1,95e11	1,72e11	1,55e11	1,49e11	1,42e11
Poisson's ratio	0,276	0,277	0,282	0,288	0,294	0,3
Coefficient of thermal expansion α_T [1/K]	1,47e-05	1,47e-05	1,569e-05	1,655e-05	1,855e-05	2,055e-05
Yield stress [Pa]	3,2e+08	3,2e+08	3,2e+08	3,2e+08	3,2e+08	3,2e+08
Tangent module [Pa]	2,0e+09	2,0e+09	2,0e+09	2,0e+09	2,0e+09	2,0e+09

The material properties which are used for the thermal calculation in the material model *MAT_THERMAL_ISOTROPIC are a specific heat capacity of 500 J/(kg · K) and a thermal conductivity of 16 W/(m · K).

The contact definition used in the LS-DYNA calculation for the thermal cyclic load contains on the one hand the thermal conductivity of the fluid and on the other hand a thermal conductivity with closed contact gap (*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL_ID). As simplified approach, the value of approx. 0.6 W/mK was used for the thermal conductivity of the fluid, i.e. the temperature and pressure dependence of the thermal conductivity of the water was not taken into account. Due to the lack of possibility to define within the contact definition a temperature dependence of the heat transfer coefficient with closed contact gap either directly via a temperature dependence or indirectly via a time-dependent curve, time-controlled contacts were defined as an alternative. A heat transfer coefficient of 3800 W/m²K was introduced for the low temperature level and a heat transfer coefficient of 300 W/m²K for the high temperature level. A time control was used to activate or deactivate the respective contacts in 30 s cycles analogous to the thermal alternating loads. To avoid a mechanical influence, a material law (MAT_Null) was used for this model group, which has no mechanically relevant properties. The radially acting internal pressure was applied as compressive load to the inner surface of the pipe segment model. The axial effect of the internal pressure was applied as axial stress to the annular surface of the pipe segment model. The value is obtained from the internal pressure and the ratio of the areas of fictitious cross-sectional internal surface and the annular surface of the pipe. The temperature loads are applied to the structural nodes as boundary conditions. Figure 6 shows evaluation points within the area of influence of the cold water feed. These are approximately located in the center of the cooling strip in the half-model of the pipe segment and allow an evaluation from the inside to the outside.

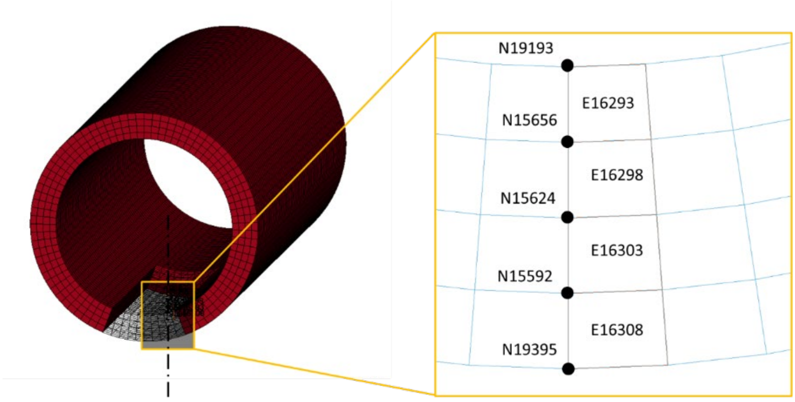

Figure 6. Element and node numbers on the evaluation line from inside to outside (enlarged section), position: 6 o'clock position, axial x=48.75 mm.

The simplified methodology enables calculation of strains, stresses and temperature distributions. In the following, results of a calculation with the described analysis model are presented. The alternating thermal load (see Fig. 6) is applied to the structure. The following assumptions are made:

- 4 cycles with cooling and heating phases of 30 s each.
- 5 evaluation points from inside to outside

Figure 7 shows the temperature distribution in the cut pipe segment after 30 s. The temperature gradient due to the cold water feed is clearly visible.

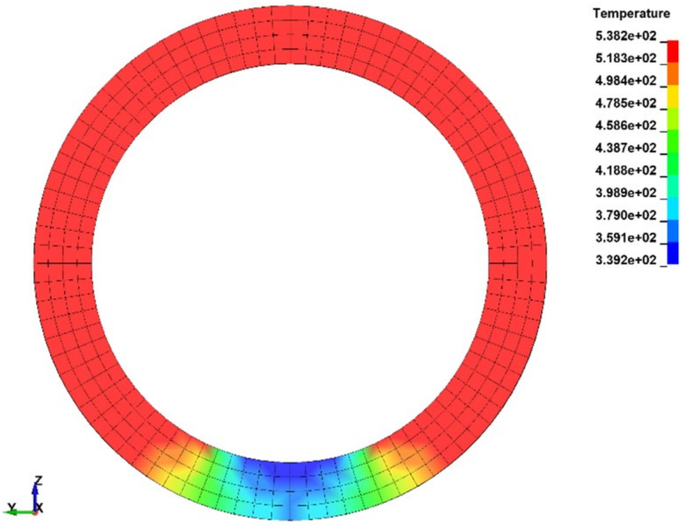

Figure 7. Temperature distribution [K] of cross section after 30 s.

Figure 8 shows the time histories of 5 nodes on the evaluation line and measured temperatures at the inner surface of the pretest XT2. The agreement is satisfactory, especially concerning the temperature range. The temperature profile follows the time cycle of cooling and heating. It is obvious that the inner nodes reach a lower temperature level during the cooling phase.

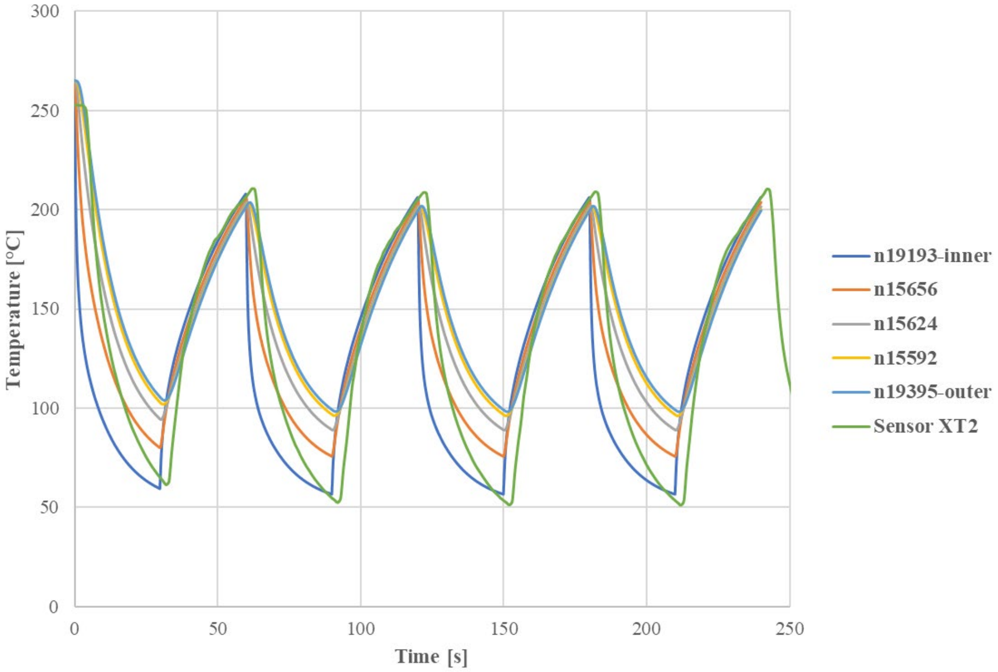

Figure 8. Temporal temperature curves at the evaluation points:
Comparison simulation (node 19193-inner surface) with experiment (XT2).

FATIGUE ASSESSMENT

The ASME methodology (ASM 2010) is employed to determine the fatigue life. This is briefly explained below. Both elastic and plastic components are taken into account, with the respective components being considered separately. Since the elastic-plastic material model used in the LS-DYNA calculation does not allow a separate evaluation with regard to elastic and plastic portions of the strain (see LS-DYNA Keyword Manual, 2021), a purely elastic calculation was also performed. Based on these elastic calculation results, the plastic portion of the strains was determined according to the superposition principle from the elastic-plastic calculation. The effective equivalent strain ε_{eff} is the sum of the plastic equivalent strain $\Delta\varepsilon_{peq}$ and the elastic equivalent strain calculated from elastic equivalent stress $\Delta\sigma_v$ and Young's modulus E_{ya}.

$$\varepsilon_{eff} = \frac{\Delta\sigma_v}{E_{ya}} + \Delta\varepsilon_{peq} \tag{1}$$

The elastic equivalent stress $\Delta\sigma_v$ is calculated here using the components of the stress tensor. The components for the calculation are available as output of the elastic LS-DYNA calculation.

$$\Delta\sigma_v = \left[\begin{matrix} (\Delta\sigma_{11} - \Delta\sigma_{22})^2 + (\Delta\sigma_{11} - \Delta\sigma_{33})^2 + (\Delta\sigma_{22} - \Delta\sigma_{33})^2 \\ +6(\Delta\sigma_{12}{}^2 + \Delta\sigma_{13}{}^2 + \Delta\sigma_{23}{}^2) \end{matrix}\right]^{0.5} \tag{2}$$

The comparative plastic strain $\Delta\varepsilon_{peq}$ is calculated analogously using the outputs of the strain tensor components, subtracting the result of the elastic calculation from that of the elastic-plastic calculation.

$$\Delta\varepsilon_{peq} = \frac{\sqrt{2}}{3}\left[\begin{matrix} (\Delta\varepsilon_{11} - \Delta\varepsilon_{22})^2 + (\Delta\varepsilon_{22} - \Delta\varepsilon_{33})^2 + (\Delta\varepsilon_{11} - \Delta\varepsilon_{33})^2 \\ +1.5(\Delta\varepsilon_{12}{}^2 + \Delta\sigma_{23}{}^2 + \Delta\sigma_{31}{}^2) \end{matrix}\right]^{0.5} \tag{3}$$

The relevant half stress range σ_{alt} is calculated as the product of Young's modulus on the fatigue curve being utilized E_{yf} and effective comparative strain ε_{eff}. The temperature dependence of the Young's modulus is taken into account.

$$\sigma_{alt} = \frac{E_{yf} \cdot \varepsilon_{eff}}{2} \tag{4}$$

Based on an evaluation with Excel, which additionally interpolates stresses and strains, a half stress range of 203 MPa can be determined, whereby the elastic portion amounts to approx. 82 %. This stress range is about 35 % larger than the equivalent stress range previously determined in the elastic-plastic calculation. The number of cycles and consequently the estimated service life can be determined from the S-N curve in KTA 3201.2 (2017) (see Figure 9). This results in a fatigue life of approx. $5 \cdot 10^4$ cycles, i.e. with a cycle duration of 1 min about 35 days of test time. Neglection of the bending possibility of the pipe and using the KTA-curve may result in an overestimation of the stresses and consequently an underestimation of the service time in form of load cycles.

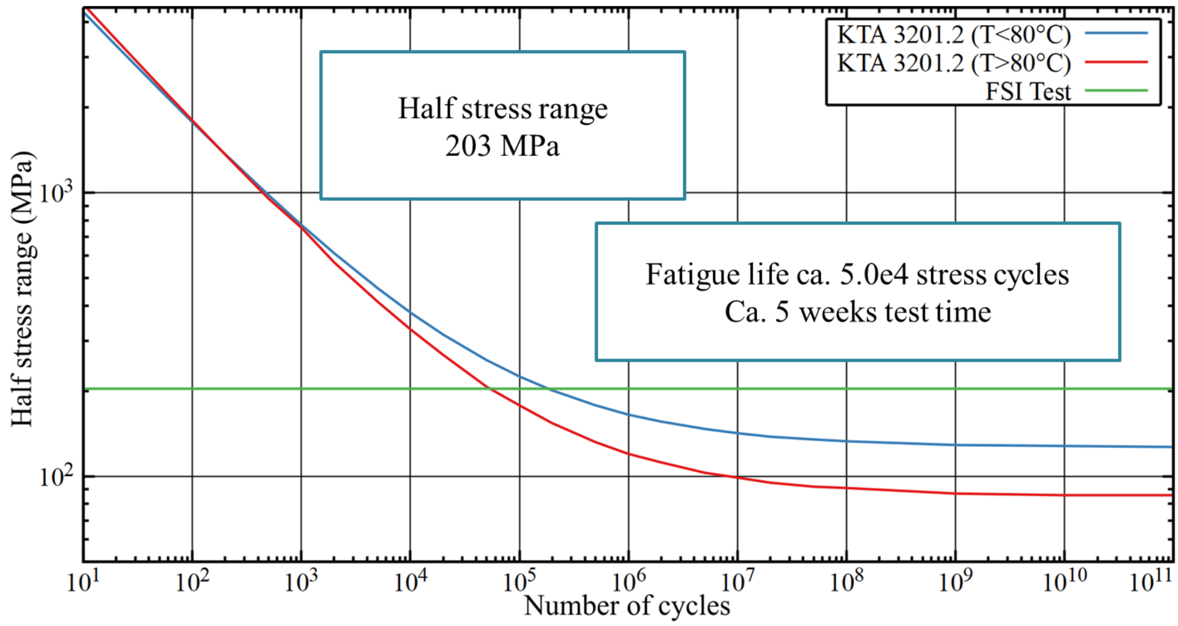

Figure 9. Assessment of the results in the S-N curve of KTA 3201.2 (2017).

COMPARISON OF TESTING, CHARACTERIZATION AND SIMULATION

The presented thermo-mechanical simulation and fatigue analysis allows a quantification of the fatigue usage in a component test. This numerical result can be compared with the results from the non-destructive evaluation (NDE) and with the destructive evaluation based on fatigue life test of extracted samples. A flowchart for the detailed evaluation of the component test is displayed in Figure 10.

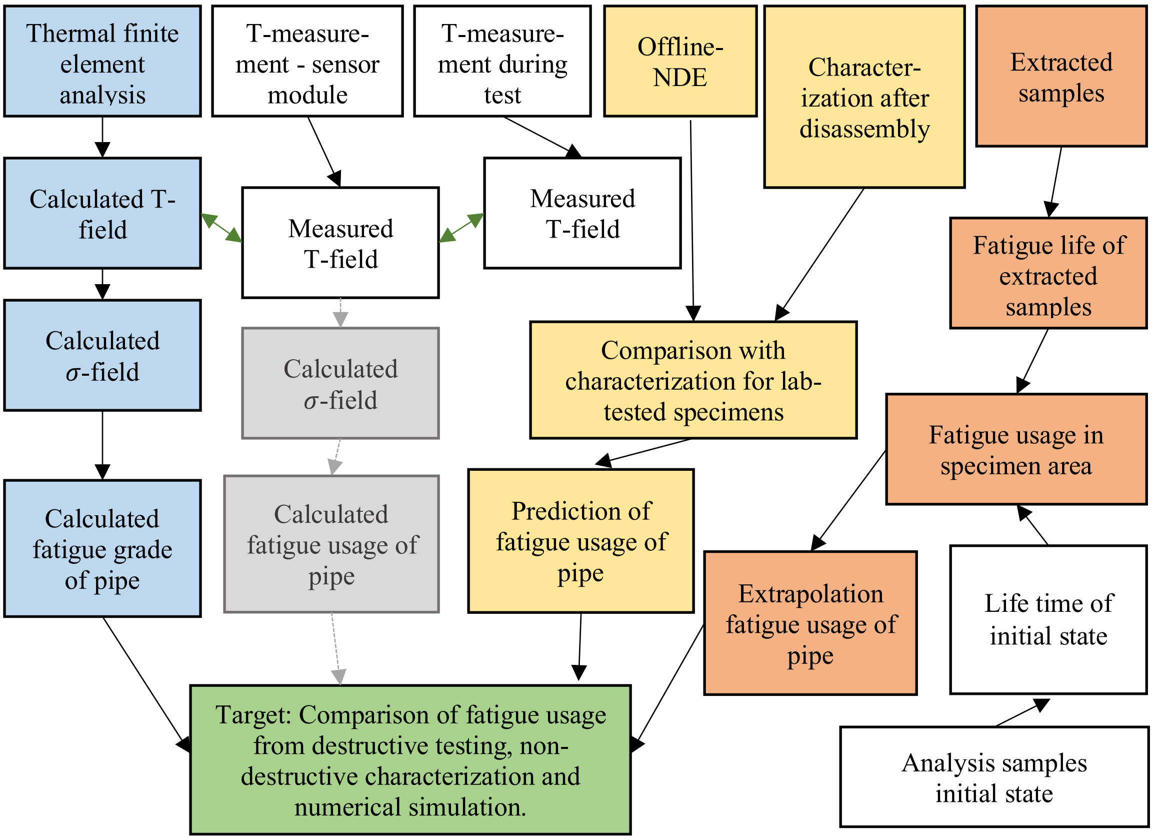

Figure 10. Flow chart for the evaluation of the component test.

This evaluation approach allows for the comparison of the numerical fatigue usage (blue) with the result obtained by the non-destructive evaluation (yellow) and the fatigue life tests on extracted samples (orange). Further verification paths are indicated in grey.

SUMMARY, CONCLUSION AND OUTLOOK

A method for the numerical evaluation of a thermo-mechanical fatigue pipe test is presented. A simplification of the complex fluid-structure interaction problem was proposed by calibrating the heat transfer (influenced significantly by the flow conditions) to measurements. The simplified methodology enables calculation of strains, stresses and temperature distributions. The service life can be estimated using S-N curves. This analysis allows to interpret destructive and non-destructive post-test examinations of the pipe section and to compare the observed fatigue signatures with the computed load collective.

It should be noted that the load approximation is an intentional choice for reducing complexity, motivated by the inevitable uncertainties in the fatigue life. A coupled fluid simulation would be able to determine the area cooled by the inner elbow pipe more precisely and consider the heating of the fluid, but finally this improved load characterization would not equally improve the fatigue life prediction.

ACKNOWLEDGEMENT

The work was carried out in the frame of a nuclear safety research project funded by the Federal Ministry for the Environment, Nature Conservation, Nuclear Safety and Consumer Protection (Bundesministerium für Umwelt, Naturschutz, nukleare Sicherheit und Verbraucherschutz, BMUV), within the framework of the joint project “Microstructure-based determination of the maximum service life for corrosion fatigue loaded materials and components of nuclear technology”, part II, funded grant numbers 1501610A, 1501610B, 1501610C, 1501610D and RS1594E.

REFERENCES

Acosta, R., Boller, C., Heckmann, K, Sievers, J., Schopf, T., Bill, T., Starke, P., Lücker, L., Donnerbauer, K., Walther, F. (2022). *Microstructure-based lifetime assessment of austenitic steel AISI 347 exposed to corrosion and fatigue*, Transactions, SMiRT-26

ASME Boiler and Pressure Vessel Code, Volume VIII, Division 2, *Alternative rules, rules for construction of pressure vessels*, 2010

ENSREG: National Action Plans Workshop, Summary Report, Brussels, 22-26 April 2013

Heckmann, K. Sievers, J. (2015) *Code Development for Piping Integrity Assessment with Respect to new German Safety Standard*, Transactions of Structural Mechanics in Reactor Technology SMiRT-23, 10-14. Aug 2015, Manchester, UK.

KTA 3201.2: *Components of the reactor coolant pressure boundary of light water reactors Part 2: design and analysis*, version 2017-11

LS-DYNA KEYWORD USER'S MANUAL, *VOLUME II Material Models*, 09/27/21 (r:14196) LS-DYNA R13, https://www.dynasupport.com/manuals/ls-dyna-manuals

ONR: National Final Report on European Council “*Stress tests*” for UK Nuclear Power Plants, December 2011, ONR Report ONR-ECST-REP-11-002 Revision 0

Sievers, J., H. Schulz, B. R. Bass, C. E. Pugh
Final report on the International Comparative Assessment Study of Pressurized Thermal-Shock in Reactor Pressure Vessels (RPV PTS ICAS), NEA/CSNI/R(99) 3, GRS-152, 1999

Swacek, C., Stumpfrock, L., Rudolph, J., Herbst, M.: *Joint project investigations into the influence of component-relevant stresses on the fatigue strength of austenitic and ferritic steels including welded joints, sub-project on load parameters and medium influence*, Materials Testing Institute (MPA), University of Stuttgart, MPA Report-Nr. 8480 000 000, 2022 (in German)

Weihe, S., Silber, S., Schopf, T, Swacek, C. (2022). *Investigations on the fatigue behavior of a full-scale pipe component with dissimilar weld under thermo-mechanical loading*, Presentation (D06 - Fatigue and Thermal Design II), Transactions, SMiRT-26