FEM-Calculation of Different Creep-Tests with French and German RPV-Steels

H.-G. Willschuetz1), E. Altstadt1), B.R. Sehgal2), F.-P. Weiss1)

1)Forschungszentrum Rossendorf, Institute of Safety Research, Dresden, Germany
2)Royal Institute of Technology, Division of Nuclear Power Safety, Stockholm, Sweden

ABSTRACT

For calculations of Lower Head Failure experiments like FOREVER it is necessary to model creep and plasticity processes. Therefore a Finite Element Model is developed using a numerical approach which avoids the use of a single creep law employing constants derived from the data for a limited stress and temperature range. Instead of this a numerical creep data base (CDB) is developed where the creep strain rate is evaluated in dependence on the current total strain, temperature and equivalent stress. A main task for this approach is the generation and validation of the CDB. For an evaluation of the failure times a damage model according to an approach of Lemaitre is applied.

The validation of the numerical model is performed by the simulation of and comparison with experiments. This is done in 3 levels: starting with the simulation of single uniaxial creep tests, which is considered as a 1D-problem. In the next level so called “tube-failure-experiments” are modeled: the RUPTHER-14 and the “MPA-Meppen”-experiment. These experiments are considered as 2D-problems. Finally the numerical model is applied to scaled 3D-experiments, where the lower head of a PWR is represented in its hemispherical shape, like in the FOREVER-experiments.

An interesting question to be solved in this frame is the comparability of the French 16MND5 and the German 20MnMoNi55 RPV-steels, which are chemically nearly identical. If these steels show a similar behavior, it should be allowed to transfer experimental and numerical data from one to the other.

KEY WORDS: Creep Tests with French and German RPV-Steel, Finite Element Simulation, Core Melt Down Scenario

INTRODUCTION

The hypothetical scenario of a severe accident with core meltdown and formation of a melt pool in the lower plenum of a Light Water Reactor (LWR) Pressure Vessel (RPV) can result in the failure of the RPV and the discharging of the melt to the containment. One accident management strategy could be to stabilize the in-vessel debris or melt pool configuration in the RPV as one major barrier against uncontrolled release of heat and radionuclides [1].

To obtain an improved understanding and knowledge of the melt pool convection, the vessel creep, possible failure processes and modes occurring during the late phase of a core melt down accident the FOREVER-experiments (Failure Of REActor VEssel Retention) are currently being performed at the Division of Nuclear Power Safety of the Royal Institute of Technology, Stockholm. These experiments are simulating the behavior of the lower head of the RPV under the thermal loads of a convecting melt pool with decay heating, and under the pressure loads that the vessel experiences in a depressurized vessel scenario (cf. Fig. 1). The geometrical scale of the experiments is 1:10 compared to a prototypic LWR.

Due to the multi axial creep deformation of the vessel with a non-uniform temperature field these experiments are on the one hand an excellent source of data to validate numerical creep models which are developed on the basis of uniaxial creep tests. On the other hand the results of pre-test calculations can be used to optimize the experimental procedure and can help to make on-site decisions during the experiment.

Therefore an axisymmetric Finite Element (FE) model is developed based on the multi-purpose code ANSYS/Multiphysics®. Using the Computational Fluid Dynamics (CFD) module the melt pool convection is simulated and the temperature field within the melt pool and within the vessel wall is calculated. The transient structural mechanical calculations are then performed applying a creep model which takes into account the large temperature, stress and strain variations.

A main task for the numerical creep model approach is the development and validation of the creep data base (CDB). The source for the CDB are uniaxial creep tests, like the REVISA-experiments. The CDB includes the primary, secondary and tertiary creep stages. In the calculation the creep strain rate is then evaluated in dependence on the current total strain, temperature and equivalent stress.

The modeling approach and validation is done in 3 steps: starting with the simulation of single uniaxial creep tests, which is considered as a 1D-problem. In the next level so called “tube-failure-experiments” are modeled: the RUPTHER-14 and the “MPA-Meppen”-experiment. These experiments are considered as 2D-problems. Finally the numerical model is applied to scaled 3D-experiments, where the lower head of a PWR is represented in its hemispherical shape, like in the FOREVER-experiments.
In the frame of this work the comparability of the French 16MND5 and the German 20MnMoNi55 RPV-steel is investigated. If these 2 steels show a similar behavior, it should be allowed to transfer experimental and numerical data from one to the other.

**CREEP AND DAMAGE MODELING IN THE TRANSIENT MECHANICAL CALCULATIONS**

Because of the large spatial and transient temperature and stress changes within the vessel wall of a 3D-experiment like FOREVER, an advanced approach for the numerical creep modeling has been developed. Usually creep is described by analytical formulas (creep laws) with a number of free coefficients. The coefficients are used to adapt the creep laws to creep test results performed at constant load and temperature. However, it is difficult to achieve a satisfying adjustment for a wide range of temperatures and stresses with only one set of coefficients. Therefore a supplementary tool for the ANSYS® code has been developed, which allows to describe the creep behavior of a material for different stress and temperature levels independently by means of a creep data base (CDB). The Digital® Fortran Compiler (Rev. 6.1A) was used for programming and for generating the customized ANSYS-executable on a Windows/NT® platform [2, 3]. The creep data base has been generated based on an analysis of the measured data performed by Ikonen [4]. Due to the uncertainties of the creep fracture strains measured in the uniaxial tests the creep fracture strain $\varepsilon_{frac}$ was set conservatively for each temperature level. It is ranging from 35 % at 600 °C to 65 % at 1000 °C. The plasticity of the material is modeled by using the multilinear isotropic hardening option of ANSYS® [5]. The plastic fracture strain $\varepsilon_{frac}^{pl}$ is evaluated from the last point of the stress-strain curve.

For the prediction of a failure time it is necessary to calculate a damage criterion. The material damage due to significant creep and plastic strains is modeled by a damage measure $D$ which is incrementally accumulated at the end of a time step or substep. $D=0$ means "no damage", which is the initial value for all elements. If the element damage reaches the value of $D=1$, the element is killed by setting its death flag to 1, i.e., this element does no longer contribute to the wall strength. The implementation of this model is described in [2].

Describing the viscoplastic behaviour, the most difficult part is the tertiary creep stage. The creep laws, or alternatively the creep data base, in FE algorithms must be based on true stress and true strain. However, usual creep tests are load controlled (i.e., the force applied to the tension bar is constant). Consequently, the true stress is not constant during the test because of the uniform reduction of the cross section in the early test stage and because of the necking later on. The increasing creep strain rate observed in the late test phase is a consequence of two effects:

- geometrical creep acceleration due to reduction of cross section and necking
- decreasing material creep resistance due to micro structural changes (e.g. micro cracks, creep cavities)

In FE-models, the geometrical creep acceleration is automatically considered if the large strain option is activated. To describe the material tertiary creep there are two basic options: i) to use a creep law or a creep data base where the derivative of the creep strain rate is positive ($\dot{\varepsilon} > 0$), ii) to use a creep strain rate that is not only coupled to the creep strain $\varepsilon_{cr}$, the stress and the temperature $T$, but also to the material damage [6]:

$$\dot{\varepsilon}_{cr} = \tilde{f}(D;\varepsilon_{cr};\sigma;T) = \frac{1}{1-D} \cdot f(\varepsilon^{cr};\sigma;T).$$

Eq. 1

In this case the CDB itself does not consider tertiary creep; the material creep acceleration is realized by the damage coupling, i.e. by the factor $(1-D)^{-1}$.

Sometimes the creep acceleration is considered as a consequence of the geometrical effect only [7, 8]; i.e. if the effective true stress and the temperature are constant, the creep strain rate would be constant, too. In other words, there would be no tertiary creep from the material point of view, but only from geometrical point of view. This approach seems to be too simple. At least, it can not be generalized for all materials. This is demonstrated by the following example. Fig. 2 shows the 2D-model of a tensile bar with circular cross section. To initiate the necking process observed in creep tests a small region with a slightly reduced diameter is introduced (reduction by 1% at $y = 0$).

Figure 3 shows several post-test calculations of a REVISA creep test (nominal strain versus time). The CDB-PS curve shows the calculation with constant strain rate at constant true stress for the whole range where the the creep strain is greater than the primary creep strain: $\varepsilon \geq \varepsilon_{p}$, i.e. the CDB considers only primary and secondary creep.
Though the pure geometrical effect leads to some acceleration, this effect alone is much too small in comparison with the experiment. Moreover, metallographic post-test investigations at the FOREVER vessel wall show clearly the existence of creep cavities (cf. Fig. 14), which highly suggests an influence onto the material behaviour. The CDB-PST curve shows the simulation with consideration of tertiary creep in the CDB. The CDB-PS-DMG curve shows the ANSYS simulation with a CDB that only considers primary and secondary creep, i.e. the same as for the CDB-PS-curve. The material creep acceleration in this case is achieved by the damage coupling of the creep strain rate according to Eq. 1. These curves agree well with the test. So it can be concluded that material creep acceleration has to be considered in the FEM simulation either by a CDB that accounts for tertiary creep or by coupling the creep strain rate to the damage.

THE CONSIDERED FRENCH 16MND5 AND GERMAN 20MNMONI55 STEELS

In this work we consider 2 nominal types of RPV-steel in different tests: the French 16MND5 and the German 20MnMoNi55. At first the Creep Data Base (CDB) is developed from uniaxial creep tests [9, 4] of the French 16MND5 RPV-steel and then the comparability of the 2 steels is investigated.

Figure 4 shows the region that is covered by the CDB. The arrays 1 to 4 show the points where uniaxial creep test were performed. There are 8 temperature levels in the CDB starting from 873 K up to 1573 K in steps of 100 K. At each temperature level there are 5 equidistant stress levels ranging from 20% of the yield stress of the next higher temperature level to the ultimate stress level of the next lower temperature level. I.e. the numerical CDB provides also temperature and stress combinations where the stress is higher than the ultimate stress which is physically unrealistic. However, these areas of the CDB are never used because the plasticity model of ANSYS [5] causes a failure after reaching the
ultimate stress.

As an example Fig. 5 shows the comparison of the REVISA tests at 700°C with the calculated results from ANSYS. There are some deviations between the calculated and experimental results, but they are considered as acceptable in view of: i) only small deviations of less than 20% for the short time runs (like the 90 and 70 MPa-runs) and ii) a conservative behavior for long time runs (like the 40 MPa-run). The reason is that the main application of this CDB is related to experiments and prototypic scenarios where a short to medium failure time range is investigated or expected, i.e. typically between 1 and 20 hours. On the other hand it is not known whether each experimental creep curve is really representative, because there is a large scatter even for different specimen of the same heat when tested at the same temperature and stress level.

This can be seen in Fig. 6, which shows uniaxial creep tests for the French and the German steel at 800°C and an engineering stress of 65MPa. There was only 1 test of 16MND5 (CEA), whereas there were 5 tests of 20MnMoNi55 (MPA) with the fastest and the slowest creep curve shown. The failure occurred after 4,700s in the slowest test and after 3,800s in the fastest test. This corresponds to a difference of some 20%, and gives an idea about the scatter that can be expected for the 16MND5 tests, too. Finally, the red curve shows the calculated ANSYS curve corresponding to the developed 16MND5-based CDB.

Table 1 lists the chemical composition, the thermal treatment, and the mechanical properties at room temperature. Comparing the chemical composition it seems that the differences between the two 16MND5 heats are

<table>
<thead>
<tr>
<th>Chemical composition</th>
<th>16MND5 RUPHER 14</th>
<th>16MND5 FOREVER EC</th>
<th>20MnMoNi55 MPA-Meppen</th>
</tr>
</thead>
<tbody>
<tr>
<td>Source: [10] [11] [8]</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>0.17</td>
<td>0.105</td>
<td>0.21</td>
</tr>
<tr>
<td>Si</td>
<td>0.25</td>
<td>0.241</td>
<td>0.24</td>
</tr>
<tr>
<td>Mn</td>
<td>1.44</td>
<td>1.26</td>
<td>1.48</td>
</tr>
<tr>
<td>P</td>
<td>0.004</td>
<td>0.0017</td>
<td>0.008</td>
</tr>
<tr>
<td>S</td>
<td>0.002</td>
<td>0.0006</td>
<td>0.005</td>
</tr>
<tr>
<td>Cr</td>
<td>0.2</td>
<td>0.249</td>
<td>0.2</td>
</tr>
<tr>
<td>Ni</td>
<td>0.75</td>
<td>0.581</td>
<td>0.8</td>
</tr>
<tr>
<td>Mo</td>
<td>0.51</td>
<td>0.568</td>
<td>0.52</td>
</tr>
<tr>
<td>V</td>
<td>0.004</td>
<td>n/a</td>
<td>0.02</td>
</tr>
<tr>
<td>Cu</td>
<td>0.01</td>
<td>0.115</td>
<td>0.07</td>
</tr>
<tr>
<td>Al</td>
<td>0.016</td>
<td>0.0379</td>
<td>0.015</td>
</tr>
<tr>
<td>Sn</td>
<td>0.001</td>
<td>n/a</td>
<td>0.005</td>
</tr>
<tr>
<td>As</td>
<td>0.001</td>
<td>n/a</td>
<td>0.02</td>
</tr>
<tr>
<td>Quenching</td>
<td>877–891°C - 8.7h/W.C.</td>
<td>920°C - 6.5h/W.C.</td>
<td></td>
</tr>
<tr>
<td>Tempering</td>
<td>635–652°C - 9.0h/A.C.</td>
<td>655–660°C - 8.0h/A.C.</td>
<td></td>
</tr>
<tr>
<td>Simulated Stress Relieving</td>
<td>618–625°C - 6.3h/F.C.</td>
<td>n/a</td>
<td></td>
</tr>
<tr>
<td>Resulting microstructure:</td>
<td>bainitic</td>
<td>bainitic</td>
<td>bainitic</td>
</tr>
<tr>
<td>Mechanical Properties at room temperature:</td>
<td>Yield strength</td>
<td>473-488MPa</td>
<td>567-624MPa required: &gt;430MPa</td>
</tr>
<tr>
<td>Tensile strength</td>
<td>620-724MPa</td>
<td>635-726MPa req.: 570-710MPa</td>
<td></td>
</tr>
<tr>
<td>Elongation</td>
<td>25%</td>
<td>22%</td>
<td></td>
</tr>
<tr>
<td>Reduction of area</td>
<td>73%</td>
<td>64-69%</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: Comparison of properties and manufacturing data of the investigated steels.
in the same range as the differences of the 16MND5 steel to the 20MnMoNi55 steel. Also the thermal treatment seems to be rather similar, which is supported by the resulting bainitic microstructure of all specimen. There are some differences in the mechanical properties, but even the weaker 16MND5 values for the yield and tensile strength would fulfill the regulation values according to the German KTA 3201.1.

After analyzing these figures we assume that an application of the 16MND5-based CDB to creep tests with 20MnMoNi55-steel is reasonable.

RUPTHER POST TEST CALCULATION

The considered RUPTHER-14-experiment was performed at CEA, France [10]. Fig. 7 shows the principal configuration of this tube failure experiment. The test pipe of 16MND5 was 270mm long and 88.9mm in diameter. The wall thickness was 2mm. Due to the centered external heating coil the resulting vertical temperature profile had its maximum in the vertical center, too. Therefore the maximum displacement and the failure can be expected at the vertical center as shown in Fig. 9.

Fig. 8 contains the loading history and the central diameter increase of the RUPTHER-14 experiment. After increasing the pressure to 8bar the temperature was increased to 1000°C at the hot spot. This regime was kept until 18,000s, when the pressure was slightly reduced to reach a level of 6bar after 25,200s. But the tube failed earlier: at 22,180s.

Despite the application of different boundary conditions and slight temperature changes it was not possible to get numerical results for the time dependent diameter increase showing exactly the same behavior as measured. The calculations rupt02 to 05 differ in the slightly changed temperature (+5K, -5K respectively) and the assumed rupture strain in rupt02 was reduced to 50%, while it was normally 60%. Especially the strong radius increase just after reaching the high temperature level at 1,800s and the accelerated creep at the pressure reduction stage can not be represented by the code.

If the reason for this discrepancy is not the numerical model, one experimental uncertainty might be the temperature. The temperatures might have been higher in the wall - especially at the beginning of the high temperature level - than the thermocouples show, because they are mounted on the wall. Additionally a distance change between the tube and the induction coil can cause a temperature change. This could be considered for the last stage, when the pressure was dropping, but the creep process accelerated.

Another reason can be the scale of the experiment. A wall thickness of 2 mm is relatively thin and small deviations from the design state – either geometrical or material – have a large influence. The suspension effect of
relatively thick components does not apply to this thin tube.

MPA-Meppen post test calculation

The considered MPA-Meppen test was performed at a test site of the German army in Meppen, Germany [8]. Fig. 10 shows the principal configuration of this tube failure experiment. The vertically positioned test pipe of 20MnMoNi55 was 2,700 mm long and had an internal diameter of 700 mm. The wall thickness was 47 mm. Several external heating coils were placed vertically around the pipe and the resulting vertical temperature profile had its maximum in the vertical center with a measured maximum at the end of the test of 735 °C.

Therefore the maximum displacement and the failure site can be expected at the vertical centre as shown in Fig. 11, which shows the upper half of the deformed FE-model. The loading history of the MPA-Meppen test is given in Fig. 12. Starting with a pressure of 120 bar, the pressure was increased to 165 bar and the temperature was increased in three stages to 735 °C at the hot spot. This regime was kept until failure. Because temperatures around 700 °C were only achieved during the last 1,200 s significant deformation is also only recorded for this period. Fig. 12 shows the comparison of the radius development between the test and different calculations for the last stage. The calculations UR1 to UR3 differ only in their temperature field, which has been shifted by 5K up or down.

A relatively good agreement between calculation and measurement was achieved. Again a main uncertainty might be the temperature. It is difficult to read and analyse the temperatures from the found literature [8] and additionally there might have been some measurement error. Another reason could be the higher yield stress of the material applied in Meppen compared to the yield stress in the numerical model, which is based on 16MND5.

Calculation of the EC-FOREVER-2-Experiment

The hemispherical bottom head of the considered experiment EC-FOREVER-2 was made of the French RPV steel 16MND5 with an internal radius of 188 mm and a wall thickness of 15 mm. The oxidic melt employed was a CaO-B₂O₃ mixture (30-70 wt.-%) which has a solidus temperature Tₛ=1250K. The heater construction was modified to preclude the uncovering of the heater welding joint, where the heater had failed in an earlier test.

After performing successful post-test calculations of the experiment FOREVER-C2 [3], the numerical model was applied to develop a promising heat generation and pressure load schedule for the experiment EC-FOREVER-2.

For the evaluation of the temperature field within the vessel wall the CFD-
module FLOTRAN® of the FE-code ANSYS® is used. A 2D-axisymmetric model with appropriate boundary conditions and material properties is developed for the vessel wall and the melt pool region. Due to the melt convection inside, the hottest region of the vessel wall is the upper part of the hemisphere just below the surface of the melt. The local temperature differences over the vessel wall correspond to the local heat flux across the wall. The maximum heat flux is 140kW/m² and the calculated maximum temperature difference is around 80K. The measured temperatures on the outside are lower than calculated because the thermocouples are mounted on the wall and can not measure exactly at the surface. But due to the integral character of the FOREVER-experiment the calculated temperature field is assessed, again indirectly, by the mechanical response of the vessel to the pressure load, which will be discussed later.

The FOREVER-FE-model has 8 elements across the wall in the high temperature region of the vessel. A sufficient number of elements over the wall thickness is necessary to model the changing material properties and the body load due to the temperature field which is taken from the CFD analysis. To save computational time, regions of lower temperatures were meshed with 4 element layers over the wall thickness. A detailed description of the results is given in [1].

It is found that the most damaged element is on the vessel outside, some 60mm below the welding. This corresponds very well to the failure position observed in the experiment, where a horizontal crack of some 150mm length was created about 50mm below the welding. Figure 13 shows this region of the model.

According to the calculations the failure location is independent of a temperature shift (+10K), i.e. it is nearly independent of the failure time. This applies also for the pressure: different pressures mainly change the failure time and have little influence on the failure location.

The evaluation of the pre-test calculation was of substantial help during the experiment. Finally it can be stated that the developed Finite Element Model is quite well validated for arrangements and loading schemes like in this scaled experiment.

9. Metallographic Observations

Metallographic examinations of samples taken from different positions of the vessel wall were performed in order to detect experiment-induced microstructural changes. Additionally, annealing experiments with subsequent microstructural investigations, microhardness measurements, ion microprobe analysis and scanning electron microscopy with energy dispersive X-ray analysis of selected samples were also carried out (FOREVER C1 and C2, [12]). In this way the temperature regime and the temperature profile could be determined with an accuracy of +/- 20°C.

The result of the metallographic examination is a two-dimensional profile of the microstructure over the wall thickness and along the height position. Regions of microstructural appearances that are correlated to special transformation temperatures or environmental effects could be defined. In this way the region of highest thermal loads could be identified and the axial and radial thermal gradient could be proven. The investigations revealed that creep pores are formed at highly loaded positions. They indicate a remarkable creep damage.

Figure 14 shows a cross sectional view over the vessel wall just above the final crack position in EC-FOREVER-2. On the left side, i.e. vessel inside, nearly no creep pores are found. Towards the outside the number of pores increases significantly and finally the coalescence of the growing pores is clearly horizontally oriented and small cracks are formed. Comparing Figure 14 with 13, which shows a detailed damage contour plot of the most damaged region a qualitative agreement between the microscopically observed damage distribution and the calculated damage parameter “D” is found.

Figure 13: Contour plot of the damage parameter D [-] just before calculated failure.

Figure 14: Micrograph of a polished, non etched sample just above the final crack position (ca. 50mm below the welding). Dark areas indicate voids: creep pores and cracks.
12. Conclusions

After the development of a CDB from the uniaxial REVISA test of the French 16MND5 steel the numerical creep and plasticity model has been applied to 2 tube-failure experiments at different scales and of nominal different steels. The comparison shows that not all effects can be represented by the numerical model, but the reason might not be the different kinds of steels rather than temperature or different heats of the same steel. This assumption has to be investigated further.

Finally a successful application of the numerical model to the 3D-experiment EC-FOREVER-2 has been performed. The availability of the pre-test calculation was of substantial help during the FOREVER-experiments. It can be stated that the developed Finite Element Model is quite well validated for an arrangement and loading sequence like in this scaled experiment. The failure location and vessel geometry was predicted very well, while there are minor uncertainties concerning the time of failure. Additionally microscopic investigations were made to get insights into the complex structural behavior within the vessel wall and the distribution of the creep pores. The microscopic examinations have proven to be a valuable tool for the analysis of the conditions and the sequence of the FOREVER experiments. Qualitatively, the agreement between the metallographic and the numerical results is quite well.

References

9. Sainte Catherine, C., Tensile and creep tests material characterization of pressure vessel steel (16MND5) at high temperatures (20 up to 1300°C), Rapport SEMT/LISN/RT/98-009/A, CEA, France, 1998.

Acknowledgement

This work has been funded by the German Ministry of Economics and Labour BMWA under Contract No. 150 12 54. The FOREVER experiments are performed under the sponsorship of the ARVI Project of the 5th-Framework Programme of the EU and the APRI Project jointly supported by SKI, Swedish and Finish Power Companies, USNRC, and HSK. This work is supported by the ARTHUR UND AENNE FEINDT-STIFTUNG, Hamburg.