Prediction of the behavior of copper alloy components under complex loadings by electro-thermomechanical coupled simulations

M. Weber, D. Helm
Matthias.Weber@iwm.fraunhofer.de
Dirk.Helm@iwm.fraunhofer.de

Fraunhofer Institute for Mechanics of Materials IWM
Wöhlerstrasse 11
79108 Freiburg (Germany)

Abstract

Electronic devices must be served with electric power for functional or communication reasons. The necessary robust and reliable electric connectivity for electronic devices is realized by numerous electric connectors made of materials with an adequately high conductivity. For its leading chemical and electric conductivity properties, precipitation hardened copper alloys such as the investigated C19010 are widely used for designing connectors with high level mechanical or conductance properties. However, copper generally shows a characteristic stress relaxation phenomenon under mechanical and/or thermal loads.

In many industry branches finite element analysis (FEA) is the standard method to design and optimize components with respect to reliability and performance. For this reason the model parameter of an elasto-viscoplastic material model, developed at the Fraunhofer IWM, are determined by use of experimental data taken from tensile and relaxation tests of a C19010 copper alloy. The material model considers the characteristic of the mechanical properties, observed by the experiments, and allows for the simulation of time and temperature dependent elasto-viscoplastic material behavior. Finally, the model is validated against the experimental data and shows a good agreement.

The material model with the identified parameters is applied in electro-thermomechanical coupled finite element simulations of a clamp connector and a spring connector lamella with different load histories. The advantage of models as presented here is their capability to simulate and analyze complex loading histories in one simulation model, e.g. a forming process followed by a mounting procedure of the sample and an electro-thermomechanical loading straight afterwards. The goal of the simulations in the following is the analysis of the impact of the stress relaxation on the mechanical properties of systems and their behavior over time.

From the numerical results with the new model it is not only quantitatively shown, how stress relaxation influences the connector clamping forces or pressure, respectively, in dependence with time and/or temperature. Moreover, the simulation results documents that and why stress relaxation has to be taken into account in finite element simulations during the designing process of electrical devices.

Introduction

The development and improvement of modern devices need a downsizing of their components while keeping electric loads mostly on a constant level. Typical examples are mobile phones, components in electric cars or their controlling units and in particular highly functional systems such as minisatellites. Due to the materials conductance resistance downsizing consequently leads to an increase of the component temperatures. To combine the demand on high mechanical strength and high electrical conductivity, high performance copper alloys like CuNiSi-, CuNiSiMg- or CuNiSiAl-alloys are of a central importance for industrial branches, from innovative technology companies for electric mobility or private entertainment up to the aircraft and airspace industries [5-7].

Since nonlinear simulations of the mechanical behavior of devices or components in the designing process, e.g. for optimization reasons, are common these days, there still exists a deficit on material models and reliable data, for example to quantify the impact of stress relaxation with high accuracy. Hence, this aspect is still essentially and therefore a research project was initiated in 2013 with its goal to measure the time dependent mechanical properties of the precipitation hardened high performance copper alloy C19010. In addition, the mechanical properties of this alloy had to be modeled using an elasto-viscoplastic material model for its application in finite element simulations. The material model was developed at the Fraunhofer Institute for Mechanics of Materials IWM and implemented as a user defined material subroutine (UMAT) for Abaqus/Standard.

The core terms of the material model are presented below. By use of this model the device properties such as mounting forces with respect to the energization induced temperature and time are investigated by numerical simulations to understand how different systems and their clamp forces are influenced by the temperature dependent stress relaxation.

Mechanical characterization

Since the material properties of rolled copper alloys do vary with temperature, strain controlled tensile and relaxation tests are carried out under isothermal conditions on a ZWICK 1445 universal testing machine at the Research Institute for Precious Metals and Metal Chemistry [9]. As shown in the strain vs. time plot in figure 1 a), in all tensile and relaxation tests nominal strain rates are kept constant, while the nominal stress is continuously recorded. In the relaxation tests, the nominal strain is constant through the dwell times Δt at 1 or 3 %. The experiment parameter matrix is given in table 1.

Experiment Type	Temperature in °C	Nom. Strain Rate in s^{-1}	Dwell Time in s	Orientation in °
tensile test	20 / 150	10^{-4}	----	0/45/90 (20°C)
relaxation test	20 / 150	10^{-4}; 10^{-3}	1800	----

Table 1: Test parameters applied in the relaxation tests, used for the identification of the material model parameter for the C19010 copper alloy.

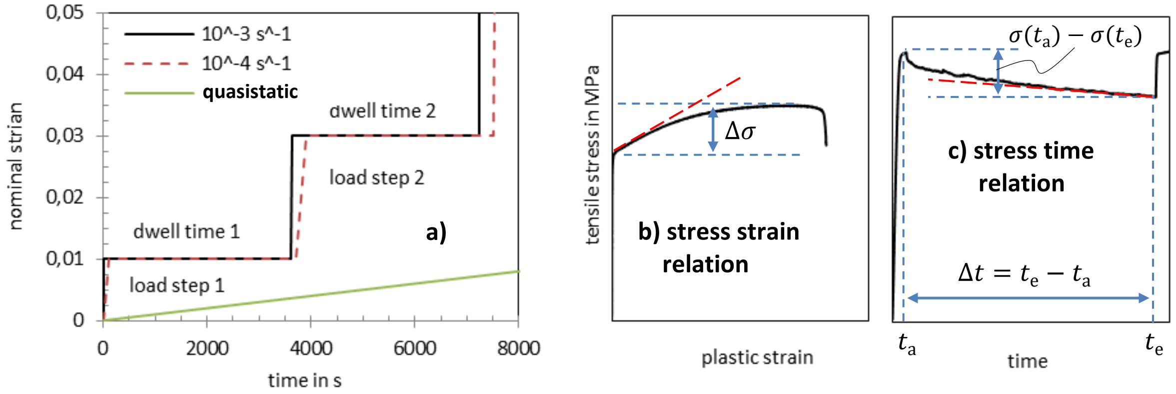


Figure 1: a) In the isothermal tensile tests the nominal strain rate is held constant at $\dot{\boldsymbol{\varepsilon}} = \mathbf{10^{-6}}\mathrm{s}^{\mathbf{-1}}$, while in the relaxation tests the nominal strain is increased by 1 and 2 % with a nominal strain rate $\dot{\boldsymbol{\varepsilon}} = \mathbf{10^{-4}}\mathrm{s}^{\mathbf{-1}}$. The dwell times are 1800 s each. b) By the measured stress-strain-curves of the tensile tests, the characteristics and time independent parameters of the material model are determined. c) The viscoplastic parameters of the applied material model are identified additionally using the stress-time-curves of the isothermal relaxation tests.

From the relaxation and tensile test results, or more precise by the stress strain relation and the stress time relation as shown schematically in figure 1 b) and c), the characteristic of the quasistatic and time dependent behavior of the C19010 copper alloy is determined.

Using the quantities of the slope of the stress strain relation at zero plastic strain or the stress difference $\Delta\sigma$ in the hardening curve as shown in figure 1 b), time independent model parameters are determined with respect to the temperature. From the identified quasistatic parameters, the visco-plastic model parameters are determined so that the relaxation characteristic in figure 1 b), namely the stress drop during or the slope at the end of the dwell time, are predicted by the model with a sufficient accuracy.

In figure 2 the stress strain relation observed for the rolled C19010 alloy is shown for the temperature of 20 and 150 °C. Therein, the dependence of the Young's modulus E or the value of the yield strength $R_{\mathrm{p0.2}}$ on the temperature can be seen as furthermore a characteristic linear hardening behavior, which is typically known from cold rolled materials with a face-centered cubic structure (see e.g. for reference [8]).

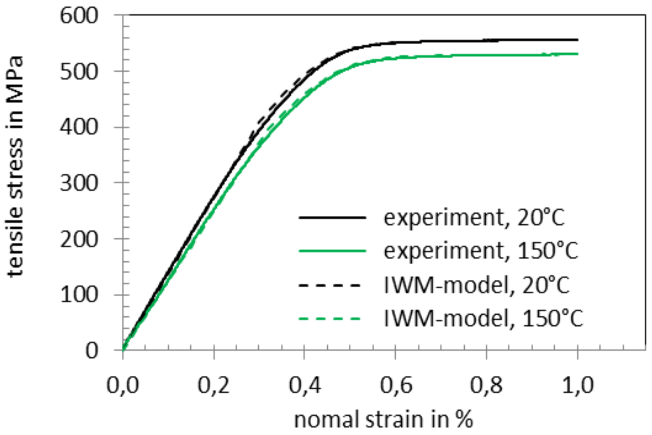


Figure 2: a) Experimental data of tensile tests in rolling direction for different temperatures of 20 and 150 °C in comparison with the material model. In these data the dependency of the elastic parameter of the Young's modulus E on the temperature is quantitatively shown. In the material model the Young's modulus is therefore considered with respect to the local temperature $\boldsymbol{\vartheta}$.

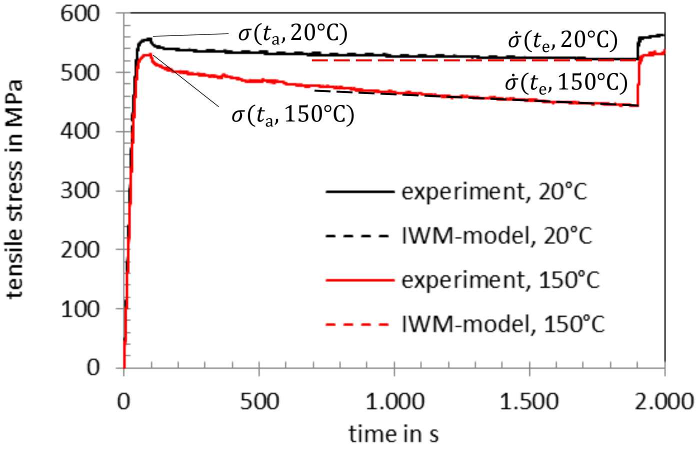


Figure 3: a) Experimental data of the relaxation test in rolling direction for different temperatures of 20 and 150 °C, compared with relaxation curves predicted by the material model. The influence of the temperature on the viscosity, shown by these data, has to be considered by the material model.

In addition in figure 3 the measured stress time relation is shown for the temperature of 20 and 150 °C. From the figure it is quantitatively shown, how temperature influences the strain rate dependence or the viscosity, as found in the maximum yield stress $\sigma(t_{\mathrm{a}})$ at the start of a dwell time and also by the relaxation rates at the end of a dwell time $\dot{\sigma}(t_{\mathrm{e}})$.

Material Model

The material model developed at the Fraunhofer IWM includes different approaches to describe a wide range of phenomena in material behavior characteristics (see for reference also in [10]), from which only the activated parts will be introduced in the following. The model features described as follows has been developed and implemented as a user defined material model at the Fraunhofer IWM for its use with Abaqus/Standard.

With the set of parameters, derived from tensile and relaxation test and validated against experimental data, the material behavior of the investigated copper alloy can be described with a high order correlation and the model is therefore usable for and applied in electro-thermomechanical simulations of the mechanical behavior of copper alloy connectors.

Set	Model parameter	Value at 20 °C	Value at 150 °C
	Young's modulus E	$1{,}367 \cdot 10^5$ MPa	$1{,}252 \cdot 10^5$ MPa
	Poisson's ratio μ	0,3	0,3
	R_0	$3{,}250 \cdot 10^2$ MPa	$2{,}750 \cdot 10^2$ MPa
Set No. 1	γ_1	$3{,}417 \cdot 10^5$ MPa	$3{,}084 \cdot 10^5$ MPa
	β_1	$2{,}440 \cdot 10^3$	$2{,}400 \cdot 10^3$
	α_1	$7{,}500 \cdot 10^{-5}$ s^{-1}	$6{,}300 \cdot 10^{-4}$ s^{-1}
Set No. 2	γ_2	$8{,}000 \cdot 10^2$ MPa	$6{,}700 \cdot 10^2$ MPa
	β_2	0,0	0,0
	α_2	0,0 s^{-1}	0,0 s^{-1}
	η	$6{,}500 \cdot 10^{60}$ s	$2{,}800 \cdot 10^{60}$ s
	r	$1{,}000 \cdot 10^0$ MPa	$1{,}000 \cdot 10^0$ MPa
	p	$3{,}075 \cdot 10^1$	$2{,}823 \cdot 10^1$

Table 2: Material parameter set of the model applied in the simulations.

In figure 2 and figure 3 a temperature dependent elasto-viscoplastic material behavior is observed for the investigated C19010 copper alloy. Taking these data into account, the model as presented here includes an exponential isotropic hardening law in combination with a recovery term. Further, the dependency of the yield surface on the plastic strain rates, mainly responsible for the stress relaxation effect in the simulations, is described using Perzyna's model [2].

The yield function f is given by

$$f\left(\varepsilon_p, \dot{\varepsilon}_p, T\right) = \bar{\sigma}_{\mathrm{V}} - \sigma_{\mathrm{y}} \leq 0\,. \tag{1}$$

Therein $\bar{\sigma}_{\mathrm{V}}$ is the von-Mises-stress and σ_{y} the value of the actual yield stress at a given material point. As shown in equation (2) the yield surface evolves following a strain dependent isotropic hardening law $R(\varepsilon_p)$ and a strain rate dependent part $R_{\mathrm{ov}}(\dot{\varepsilon}_{\mathrm{p}})$.

$$\sigma_{\mathrm{y}} = \sqrt{2/3}\,\left[R_0 + R(\varepsilon_p) + R_{\mathrm{ov}}(\dot{\varepsilon}_{\mathrm{p}})\right] \tag{2}$$

The model parameter R_0 denotes the initial value of the yield surface and also depends on the temperature ϑ. As written above, the isotropic hardening is modelled with a Voce-type yield law including a recovery term, given by its rate formulation as

$$R(\varepsilon_p) = \textstyle\sum_i R_i(\varepsilon_p), \;\; \text{with: } \dot{R}_i = \gamma_i \dot{\varepsilon}_{\mathrm{p}} - \left(\beta_i \dot{\varepsilon}_{\mathrm{p}} + \alpha_i\right) R_i\,. \tag{3}$$

In equation (3) γ_i and β_i are the hardening parameters of the parameter set i and α_i is the corresponding recovery parameter, all taken into account as function of the local temperature. Finally, the strain rate influence on the yield stress is included in R_{OV} using the Perzyna-approach with

$$R_{\mathrm{ov}}\left(\dot{\varepsilon}_{\mathrm{p}}\right) = r\,\left(\eta \dot{\varepsilon}_{\mathrm{p}}\right)^{1/p}. \tag{4}$$

The parameter r has the dimension of stress and is necessary for normalizing reasons, while the parameters η and p are the temperature dependent viscosity and strain rate hardening exponent.

Simulation models and simulation results

The dimension of a connector is related to many different parameters, for example the connector type, the voltage or thermal load due to conductive resistance, the minimum clamping force which has to be reached under operation conditions, the long-term conductance or reliability and many more. However, the benefit of using fully coupled thermal-electrical-structural finite element analysis with material models as given in equations (1-4) is demonstrated exemplarily for two different types of connectors in the following, which are shown in figure 4:

1. the electrical clamp contact in a), using about 42.200 elements (type Q3D8) and
2. a single spring contact lamella in b), using about 9.100 elements (type Q3D8).

The model of the single lamella represents the characteristic unit of a connector which is built up by multiple instances of identical lamellae. By a downsizing to the characteristic unit computation time is reduced while the knowledge, obtained by the finite element simulation results, can be transferred directly to larger systems made of many lamellae of the same type. The geometries of both connectors are adapted to real systems. The thickness of the clamp connector and its height are 1.0 mm or 6.0 mm, respectively. The spring connector thickness is 0.5 mm and the undeformed lamella length and width 26.7 mm or 1.75 mm, respectively.

Model parameter	Temperature in °C
friction coefficient (dry, Cu on Cu)	0,05
thermal conductance in the contact area	1,0
electrical conductance in the contact area	1,0

Table 3: Contact and conductance parameters applied in the simulation.

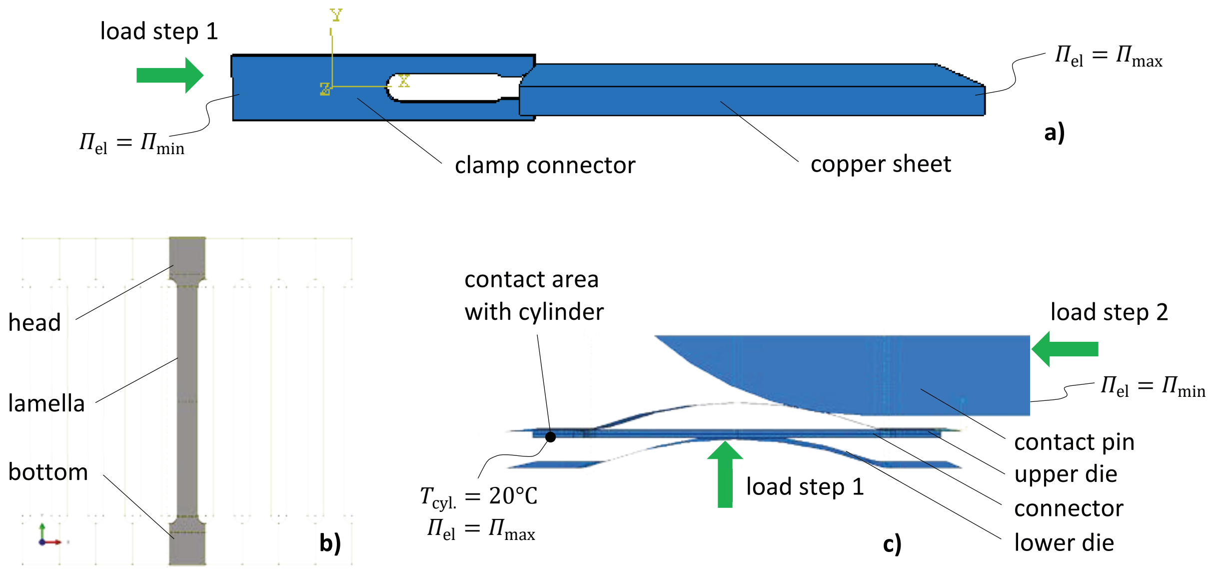


Figure 4: Simulation models of the clamp connector a) and the single lamella taken out from a multi lamellae spring connector b) and the boundary conditions for displacements, temperature and electrical potentials. The forming process of the lamella is applied by a stamp load realized by the two dies (rigid bodies) shown above or underneath the undeformed connector in c).

The goal of the simulations is the study on how mounting conditions and the influence of energization impacts the clamping forces over time, when self-heating due to the electrical resistance of the components are taken into account. The coupled thermal-electrical-structural simulations are carried out with Abaqus/Standard, which means that the mass inertia is neglected. The electrical and thermal transition resistance applied in the simulations are pressure and temperature independent and both set to zero, so that the conductance in this study is overestimated. These parameters can be defined accordingly in case that appropriate data are available as functions of contact pressure or the temperature in the contact area.

Additionally, the load histories of the systems are shown in figure 4. The clamp connector in a) is firstly mounted on a metal sheet in a load step 1, and afterwards loaded with an electrical potential gradient $\Delta \Pi_{\mathrm{el}} = \Pi_{\max} - \Pi_{\min}$ corresponding to the voltage of 12 V. Since heat radiation and convection is not taken into account, the system is isolated from the environment. In simulations for real systems parameters for radiation and convection have to be adapted to measurement results. However, in this numerical study of connector clamping forces with respect to a given electrical load their knowledge is not absolutely necessary.

In opposite to the clamp connector the lamella of the spring contact undergoes first a plastic forming process by stamping, from which it gets its arched shape shown in figure 8. In load step 2 a contact pin is pushed into its contact position, before an electrical potential of the same level as in the first model is applied. In real systems the spring connector is positioned between two bulk parts, with which it gets into contact: on the upper side there is the copper pin shown in the simulation model in figure 4 c) and on the lower side it is a cylinder, multiple times larger than the thin lamella and also made of copper, from which it gets the electrical potential load. The heat, converted due to the electrical resistance, is transferred to the pin and the bulk cylinder. In the simulation these is taken care of by the heat transfer simulation to the pin and a temperature boundary condition of 20 °C, applied to the lower surface which is in contact with the cylindrical part.

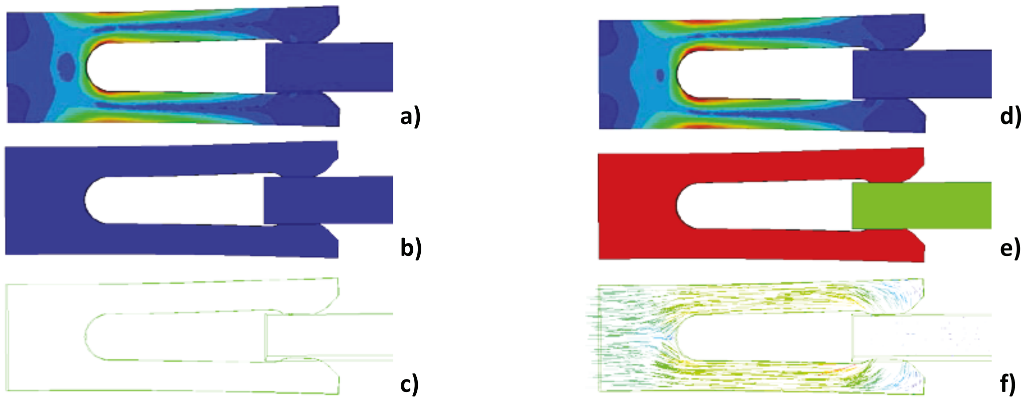


Figure 5: Simulation results of the clamp connector under isolated conditions, obtained by using the Fraunhofer IWM material model. In the left column the results a) to c) show the von-Mises-stress, the temperature and the electrical current per unit area being zero after the mounting process has been completed and before the electrical potential is applied. The results of the right column d) to f) show the identical result quantities in the same order, 10^3 s after closing the circuit.

As it is seen in the simulation results in figure 5 a), the mounting process itself introduces a stress maximum of about 545 MPa in the inner radius of the clamp connector while the temperature stays at 20 °C and the electrical current per unit area is zero. After closing the circuit, the current in f) immediately reaches its stationary value and, due to the conductance resistance of the clamp connector and the negligence of heat conduction and convection or radiation in this model, in e) the maximum temperature reaches the level of about 200 °C within the simulation time of 10^3 s. In d) is shown, that in the same time the maximum stress relaxes to about 475 MPa.

The calculated temperature of more than 150 °C, although it is only of theoretical significance here due to the negligence of heat conduction and convection, is of special interest: copper alloys such as the investigated C19010 are certified with few exceptions for their application within the temperature range from room temperature to 150 °C. However, by the calculated temperature increase in a quite short time range the known sensibility of connectors with small form factors against their environment conditions is confirmed and, even though this temperature level will not be reached as fast as in the simulations for real systems, it is also shown that in simulations of small electrical devices ambient temperatures or heat transfers must be taken into considerations adequately.

For a quantitative approach how large the relaxation influence on the clamping force would be with respect to the temperature of the connector, isothermal simulations are done for 20 and 150 °C. For the higher temperature, the load history includes two load steps: first the system is heated up to the target temperature linearly over 100 s, followed by 24 h with a constant temperature of the system. For room temperature the time range is identical but the temperature is kept constant at 20 °C throughout the simulation. The comparison of clamping forces evolving over time for 24 h for these temperatures is given in figure 6.

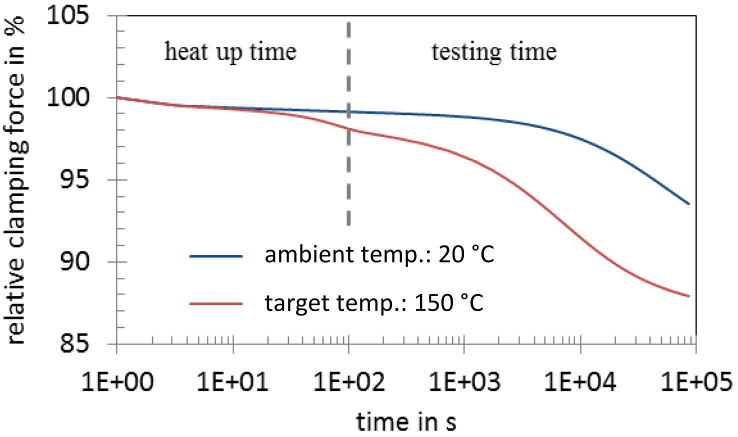
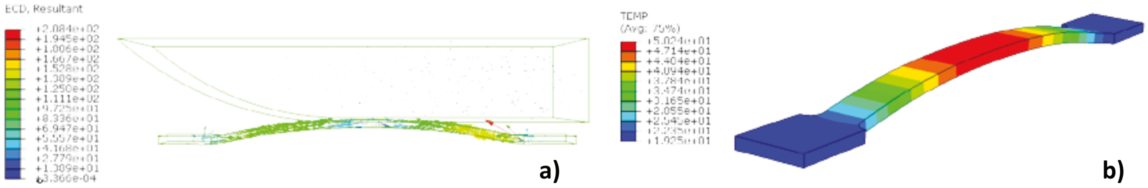


Figure 6: Numerical analysis of the calculated clamping force in dependence of the system temperature. While the force at the lower temperature decreases moderately by about 6.5 %, the force drop at the higher temperature of 150 °C is at the same time more than 12 %.

In this figure it is found, that stress relaxation occurs for both conditions and in the first 100 s very moderately only at room temperature. Nevertheless, the time and temperature dependent drop in the clamping forces over the first 24 h are about 6.5 % at 20 °C and more than 12 % at 150 °C. Since this effect is significant and observable for only 24 h, by the result it is shown, that and how finite element simulations with appropriate material models can support the understanding of working principles of electrical devices and components, undergoing electro-thermomechanical loads, at the beginning of the designing process.



a)

b)

Figure 7: Simulation result of the electrical current per unit area a), which is applied to the system for 1000 s, and the resulting temperature b). As with the clamp connector before, in this study of the system heat radiation and convection are neglected. Due to this, the lamella reaches a temperature of about 50 °C within 1000 s under the electrical current and with its material conductance resistance.

Finally for the lamella system in Figure 7 the system state is analysed with its energization. As it is seen in the simulation result in figure 8 a), the maximum von-Mises-stress of about 694 MPa appears in the region of the small bending radius which is generated during the forming process. The equivalent plastic strain is in figure 8 b) in the radius region about 15,9 %. After the energization for 1000 s, the stress decreased to 658 MPa while the plastic strain reached a maximum of more than 16 %.

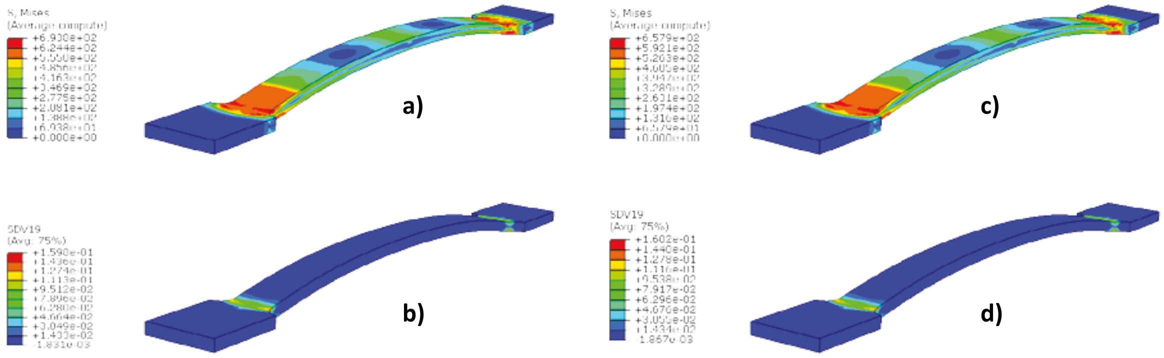


Figure 8: Simulation results of the von-Mises-stress a) and equivalent plastic strain b) of the spring contact lamella in a mounted state before the energization and after 1000 s of energization in c) and d).

In accordance with previous results, the influence of stress relaxation on the clamping force is found in a significant manner in figure 9 even without any energization. Therefore, the modelling of stress relaxation is expected to be relevant for the investigation of the system behaviour of the small connector system. However, with energization and a resulting temperature of more than 50 °C the relaxation effect increases, as also is seen in the simulation results in Figure 9.

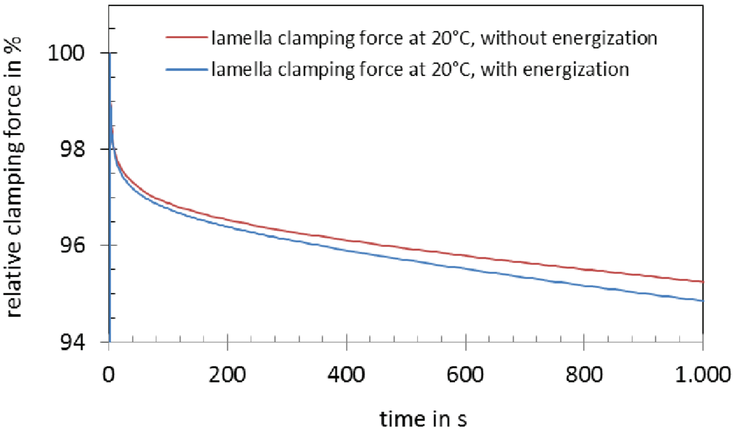


Figure 9: Simulation result of the spring contact lamella clamping force, with no (blue curve) or with an energization (red curve). Although no electrical potential is applied, the force shows a significant drop over time due to stress relaxation. With an increased temperature of about 50 °C after 1000 s (see for reference figure 7 b)), the higher relaxation influence can be seen on a measurable level in the clamping force.

Conclusion and future prospect

In a numerical study, using electro-thermomechanical finite element simulations, the impact of electrical loads on the clamping or contact force, respectively, is demonstrated for two different connector types. As seen by the exemplarily chosen calculation of their clamping forces, the necessity of adequate material models and reliable model data is shown and furthermore of central importance. However, with material models such as the applied Fraunhofer IWM model, quantitative measures for time and temperature influences on the mechanical properties, performance or reliability of devices can be determined before cost intensive prototypes are set up for testing purposes. Nevertheless, when small systems have to be analyzed in detail, the influence of the environment on the thermal balance of the whole system has to be taken into account accordingly.

The actual and upcoming work at the Fraunhofer IWM is concerned to prove the model parameters with longer dwell times, as they are for example common in industrial testing setups. In addition, an anisotropic yield behavior is observed in tensile tests with theirs parameters chosen on a random base. Since this yield behavior is successfully modelled with the same material model using Hill's yield criteria (see for reference [3]), for an appropriate extension of the copper material parameter database isothermal tensile tests with different tensile orientations become necessary.

Bibliography

[1] D. Helm, Stress computation in finite thermoviscoplasticity, International Journal of Plasticity, Vol. 22, Issue 9, pp. 1699–1727, 2006

[2] P. Perzyna, The constitutive equations for work-hardening and rate-sensitive plastic materials Proc. Vibrational Problems, Vol. 4, pp. 281–290, 1963

[3] R. Hill, A theory of the yielding and plastic flow of anisotropic metals, Proc. Roy. Soc. London, Vol. 193, pp. 281–297, 1948

[4] Final Report (AiF-Abschlussbericht) of the Project No. 17278N, Untersuchung und Modellierung des Relaxationsverhaltens von höherfesten und hochleitfähigen Cu-Legierungen, 2014

[5] Q. Lei et al., A new ultrahigh strength Cu-Ni-Si alloy, Intermetallics, Vol. 42, pp. 77–84, 2013

[6] Z. Sierpinski, J. Gryziecki, Phase transformations and strengthening during ageing of CuNi10Al3 alloy, Mater. Sci. Eng. A, Vol. 264, Issues 1–2, pp. 279–285, 1999

[7] R. Monzen, C. Waranabe, Microstructure and mechanical properties of Cu-Ni-Si alloys, Mat. Sc. and Eng. A, Vol. 483–484, pp. 117–119, 2008

[8] C. Watanabe et al., Microstructure and mechanical properties of Cu-Ni-Si alloys, J. Soc. Sci. Jpn., Vol. 54, pp. 717–723, 2005

[9] Weber, M. et al., Numerische Analysen zur Ausscheidungskinetik und zum viskoplastischen Verformungsverhalten von Cu-Ni-Si-Legierungen, METALL: Fachzeitschr. f. Metallurgie 69/3, pp. 465–470, 2015

[10] R. Uppaluri, D. Helm, Thermomechanical characterization of 22MnB5 steels with special emphasis on stress relaxation and creep behavior, Mat. Sc. and Eng. A, Vol. 658, pp. 301–308, 2016

Funding Notes

The project (No. 17278N) of the Research Community for the Research Institute for Precious Metals and Metal Chemistry (fem) has been funded by the AiF within the program for sponsorship by Industrial Joint Research (IGF) of the German Federal Ministry of Economic Affairs and Energy based on an enactment of the German Parliament.