Transient Analysis of the Triggering Behaviour of Safety Fuses

Florian Loos*[1], H.-D. Ließ[1], Benoit Philippe[2]
[1] University of the German federal armed forces, Munich, Germany
[2] ENSEA, Paris, France
*Corresponding author: Werner-Heisenberg-Weg 39, 85577 Neubiberg, florian.loos@unibw.de

Abstract: In modern cars, the number of electrical devices increases more and more whereas the available space decreases. This effect contributes mainly to very high temperatures generated in today's automobile components. Safety fuses have to fulfil the important function to protect these devices from overheating and irreparable damages due to high currents.

In this paper, we present a 3D-model for the simulation of safety fuses coupling the current flow caused by an electrical potential with the heat distribution – a typical Joule heating application. The main purpose of this work is to investigate the triggering behaviour of safety fuses for mobile on board supply systems and to set up a solid simulation model that reduces the need of testing. Thus, the influence of different materials and shapes of the fuses as well as different surrounding temperatures and currents on the triggering behaviour are analysed and the simulation results obtained with COMSOL Multiphysics® are compared to experimentally achieved data.

Keywords: Safety fuses, Joule heating, Transient analysis, Triggering behaviour, FEM simulation.

1. Introduction

The permanent increase of the amount of electrical systems in today's automobiles and in parallel a lack of available space for the multitude of components implicate high temperatures in cables, connecting structures and other important electrical car elements. To protect these assemblies from overheating and irreparable damages, they are collateralised by fuses, especially by safety fuses. Safety fuses are classical current dependent protection devices that present a bottle neck in the electric circuit. They heat more intensely than the rest of the electric circuit because of their partially higher resistance. If the electrical current is too high, temperature rises until a galvanic isolation takes place, i.e. the fuse element melts and interrupts the electric circuit [1].

To be able to protect all the differently thermally loaded components, there exists a great number of safety fuses characterised by a certain shape and by their composition of various materials. A nominal current assigned to each fuse allows the classification of different fuse types (see figure 1). This nominal current determines the minimal current a fuse still resists, for higher currents it blows. Our practical experiences show that for currents fewer than 120 % of the nominal current, the fuses trigger after more than ten minutes whereas for 200 % and more, it takes less than three second.

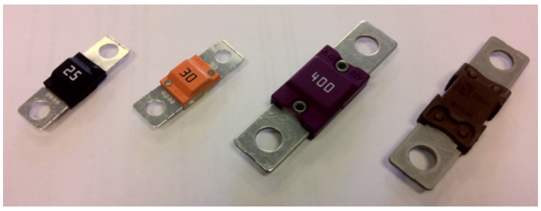

Figure 1. Different types of safety fuses with nominal current 25 A, 30 A, 400 A and 500 A.

In order to analyse the triggering behaviour of different types of fuses, we set up a model in COMSOL Multiphysics® simulating the transient heat distribution coupled to the current flow and an electrical potential.

In [2]-[9], several approaches for the simulation of fuses have been presented. The author of [2] uses the analogy of an electrical and thermal flow field to derive a thermal network that is solved iteratively. This approach enables a one dimensional calculation of the different parts of the fuses. In [3] and [4], the conservative averaging method is applied to transform the governing partial differential equations in three ordinary differential equations that are partly solved with finite differences. Other authors directly use finite differences to solve the thermal problem ([5] and [6]). In this work, we use the finite element method with COMSOL to simulate the heat distribution in current carrying fuses. Other commercial solvers that use the finite element method have been applied in [7] and [8]. In [9], equations to couple analysis of the current distribution in the fuse with the heat distribution are presented similar to our approach. A main difference of our approach to all these works is, apart from the use of COMSOL, to respect the temperature dependence of the heat transfer coefficients at the interface of solid material and air.

All mentioned approaches with different modelling and solving techniques have in common that finally the simulations are compared to experimental results to ensure accordance with the real world processes. If accordance of simulation and experimental setup is confirmed, producers of fuses have a cost and time saving possibility to test the effects of different shapes or materials and external influences to the fuses without having to manufacture and to perform experimental tests on.

The paper is organized as follows: in section 2 and 3, the geometry of our test fuses and the experimental setup are described. The numerical model with governing equations, boundary and initial conditions as well as the use of COMSOL Multiphysics are presented in section 4 and in section 5, the simulation and experimental results are compared. In section 6, we state some research results and an outlook to our future work is given in section 7.

2. Geometry of the simulated safety fuses

The simulated fuses consist of four components: a metallic part that connects the fuse to other devices like cables, a fuse element that influences the triggering behaviour of the fuse essentially, a plastic housing that surrounds the fuse element and an air cavity between the fuse element and the plastic housing.

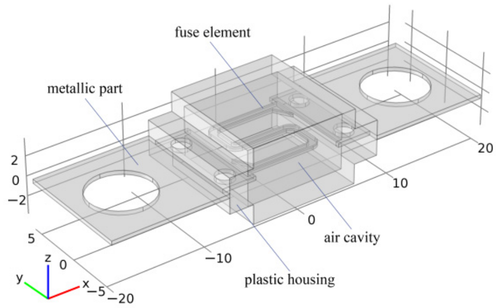

Figure 2. Geometry of the fuse Midi 25 A.

The material, the thickness and the length of the fuse element have an important influence to the triggering behaviour. In our simulations, the metallic part and the fuse element are made of E-Cu, but on the fuse element, an additional layer of tin is applied. As soon as the temperature of the fuse element reaches the melting point, the galvanic isolation takes place and the electric circuit is interrupted. It is the challenge for the designers to create fuses that ideally would carry their rated current indefinitely, and melt quickly on a small excess. Furthermore, the element must not be damaged by minor harmless surges of current, and must not oxidize or change its behavior after possibly years of service.

3. Experimental setup

To measure the triggering times of different fuses, the following test setup is used:

Figure 3. Test setup to take measurements concerning the triggering behaviour of fuses.

The principle of the experimental setup is that a virtual oscilloscope, connected to a computer via a USB cable, measures the voltage at a shunt. Since the resistance of the shunt is known, the electrical current in the fuse can be calculated on the computer by Ohm’s law. The current source is also connected to the computer, such that the magnitude of voltage and current can be controlled by the user via the computer.

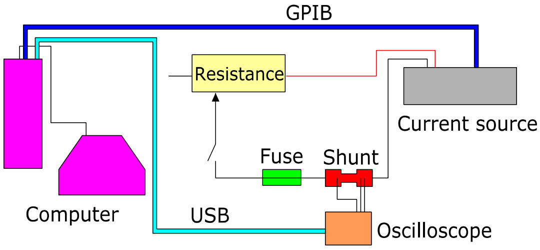

Figure 4. Basic circuit diagram of the experimental setup.

To be able to measure the correct triggering times, we have to know the moment at which the current edge flank rises mostly. This is the start of the time measuring and it ends as soon as the fuse blows. Both, beginning and stopping of the current flow in the fuse element, are determined automatically by a computer programme implemented in LabVIEW®.

The described procedure is performed several times for each type of fuse with 120 % of the nominal current up to 300 % in several steps. For each current value, the measurement is taken three times and fitting of the average values for each current delivers the characterising triggering curve of a fuse type.

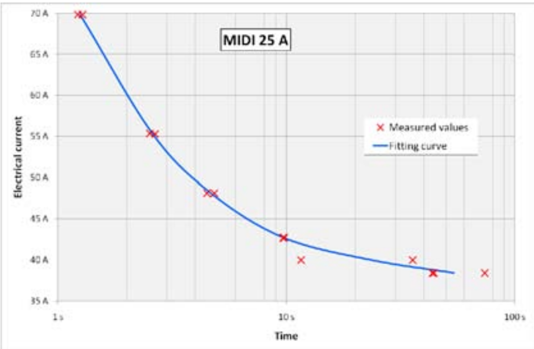

Figure 5. Triggering graph of Midi 25 A with measured values and fitting curve.

4. Numerical Model

4.1 Governing equations, boundary and initial conditions

As it was mentioned already, the heat in the described fuse generated by electrical current is a classical Joule heating problem. A given electrical current I [A] induces an electric potential U_0 [V] whose magnitude depends on the material's characteristics, especially the resistance R [Ω]. The passage of electrical current through a conductor releases heat due to resistive losses in the material.

To approximate the simulation as good as possible to the experimental setup, we perform it as follows:

On one side of the fuse element, the electrical potential is set to the value U_0 that is calculated via the current and the material resistance. On the other end of the fuse element, we set another boundary to ground, i.e. the electrical potential on this border is equal to zero. Consequently, the entire resistive loss essentially occurs in the fuse element:

$$U_0 = I \cdot R = I \cdot \frac{l}{A} \cdot \rho_R. \qquad (1)$$

The resistance of the fuse element depends on its length l [m], its cross sectional area A [m^2] and the electrical resistivity ρ_R [Ω·m]. For the fuses Midi 25 A and Midi 30 A, the fuse element consists of E-Cu and a tinny layer:

$$U_0 = I \cdot l \cdot \frac{\rho_{Cu}\,\rho_{Sn}}{A_{Cu}\rho_{Sn} + A_{Sn}\rho_{Cu}}, \qquad (2)$$

where ρ_{Cu} and ρ_{Sn} represent the resistivity of E-Cu and tin and A_{Cu} and A_{Sn} the corresponding cross sectional areas of the fuse element.

The rise of resistance for higher temperatures T [K] than the reference temperature T_{ref} [K] is approximated linearly by the temperature coefficient α_ρ [1/K]:

$$\rho_R = \rho_0 \cdot \left[1 + \alpha_\rho\left(T - T_{ref}\right)\right]. \qquad (3)$$

The electrical power P [W] generated in the fuse element is calculated as follows:

$$P = U_0 \cdot I = R \cdot I^2 = \frac{l}{A} \cdot \rho_R \cdot I^2. \qquad (4)$$

The heat equation is the main equation governing the time dependent heat distribution in the solid parts of the fuse Ω_{sol} , i.e. the fuse element, the metallic part and the plastic housing:

$$\rho C_p \frac{\partial T}{\partial t} = k\,\Delta T + Q \quad \text{in } \Omega_{sol}. \qquad (5)$$

The quantities ρ [kg/m^3], C_P [J/(kg·K)] and k [W/(m·K)] represent respectively the material density, heat capacity at constant pressure and thermal conductivity. The change of temperature in time t [s] depends on the total power dissipation density Q [W/m^3]:

$$Q = \frac{P}{V}, \qquad (6)$$

where V [m^3] is the volume of the fuse element.

In contrast to the energy generation in the fuse, heat dissipates via the surface that is described by a Robin boundary condition:

$$k\frac{\partial T}{\partial n} = \alpha\left(T\right)\left(T_{env} - T\right) \quad \text{on } \partial\Omega_{ex} \qquad (7)$$

with $\partial T/\partial n$ [W/m] the normal derivative of T, $\partial\Omega_{ex}$ the exterior boundary of the fuse and T_{env} [K] the ambient temperature. The heat transfer coefficients α_o, α_l and α_v (summarized by $\alpha(T)$ [W/(K·m^2)]) for the upper, lower and vertical surfaces are temperature dependent and have important influence to the solution of the system. They consist mainly of two parts, radiation α_r and convection α_c:

$$\alpha = \alpha_r + \alpha_c. \qquad (8)$$

The radiation part of the heat transfer coefficients is calculated via the Stefan-Boltzmann law. The rate of heat exchange between the surface and its surrounding by radiation, q_r [W], is dependent on the Stefan-Boltzmann constant σ = 5.67e-8 W/(m^2·K^4) and the emissivity ε:

$$q_r = \varepsilon\sigma\left(T_{env}^4 - T^4\right) = \alpha_r\left(T_{env} - T\right). \qquad (9)$$

The second presentation of q_r in (9) is stated to make the expression compatible to the heat convection formula and the required input format in COMSOL. For the determination of the convective part, formulas of Ilgevicius' dissertation [10] which approximate the calculation system of [11] by fitting of empirical data with polynomials of forth degree are used.

The dominating physical effect in the air cavity is radiation from surface to surface. For the governing equations, we refer to COMSOL's help 'Theory for the Surface-to-Surface Radiation Interface' [12].

As initial condition, the temperature of the solid materials is set to the ambient temperature T_{env} and the electrical potential to zero.

4.2 Use of COMSOL Multiphysics

For the geometry and composition of the fuses, we got precise information from the fuse producers that make it possible to build the geometry exactly. The construction of the fuse geometry in COMSOL is described in [13].

Concerning the mesh, COMSOL automatically generates meshes that satisfy our needs, thus, no post-editing is necessary. Exemplarily, for the fuse Midi 25 A, a finer mesh with 54101 tetrahedral elements corresponding to 77235 degree of freedom is created.

The input of the equations is done by the help of the Joule heating interface and additionally, surface-to-surface radiation is activated. The interaction of temperature and electric potential is realized by the activation of the option 'Total power dissipation density' as general heat source for the thermal calculation (see equation (4), (5) and (6)) whereas the temperature dependence of the electric conductivity is respected in the electrical calculation via the option 'Linearized resistivity' (see equation (3)). For condition (7), the boundary condition 'Convective cooling' was activated at the corresponding boundaries and for the heat transfer coefficient, the implemented functions for formula (8) are called.

To determine the triggering time, we first perform stationary simulations with lower currents to see if the melting point of the fuse element is attained at all. Afterwards, the simulation is repeated several times with rising currents where for each simulation the time to heat up the fuse element until the melting point is noted. In this way, we obtain the characteristic simulation curve for the fuse as it is depicted in figure 5.

For the time dependent solver, the generalized α-method with time step adapted to the estimated triggering time, an amplification for high frequency of 0.75 and a linear predictor is used. To solve the linear system, we use the direct solver PARDISO with the nested dissection multithreaded preordering algorithm and a pivoting perturbation of 1e-8.

5. Comparison of experimental measurements and simulation results

The result of our simulation is the entire heat distribution of the fuse at any point in time as it is shown for the geometries of Midi 25 A after 31 seconds with low current in the figures 6 and Midi 30 A after 0.2 seconds with high current in figure 7:

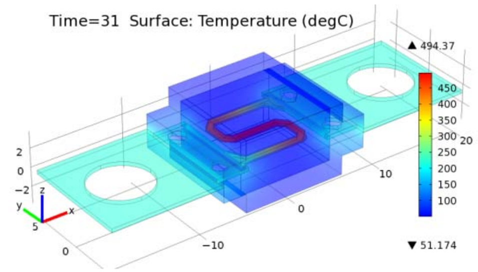

Figure 6. Calculated heat distribution in Midi 25 A after 31 seconds with current 32.5 A and ambient temperature 50 °C.

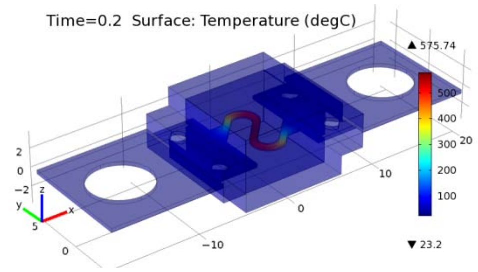

Figure 7. Calculated heat distribution in Midi 30 A after 0.2 seconds with current 90 A and ambient temperature 20 °C.

Accordance of the measurement results and the simulations is the main objective of our investigations. One problem in this context is that we cannot be sure that heating the fuse element up until the melting point is a unique indicator for the triggering of the fuse. The composition of the fusible plug with 60 percent of E-Cu (melting point at 1083 °C) and 40 percent of tin (melting point at 231 °C) makes it even more difficult to find a reliable criterion when the electrical circuit is interrupted. This is the reason why in the figures 8 and 9 we compare the simulation curves for different maximal temperatures in the fuse element with the measurement results for Midi 25 A and Midi 30 A.

For example, the green curve in figure 8 illustrates when, according to our simulations, the fuse element reaches 500 °C for different currents relative to the nominal current 25 A. For 200 % of the nominal current, 500 °C are attained after about 1.1 seconds.

In [14], it is stated that the melting point of a copper-tin-alloy is about 730 °C what could approximate our material. Figures 8 and 9 show that our calculated triggering curves accord best with the measurements for the maximal temperature values between 700 and 750 °C.

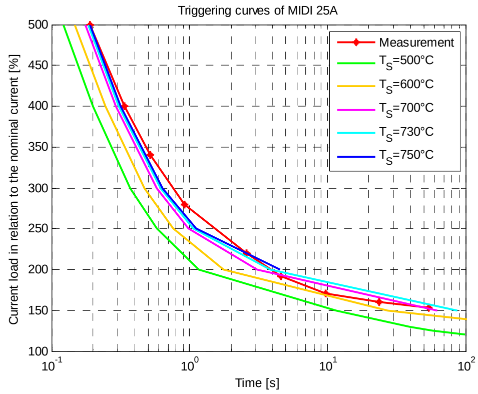

Figure 8. Measured and calculated triggering curves for Midi 25 A.

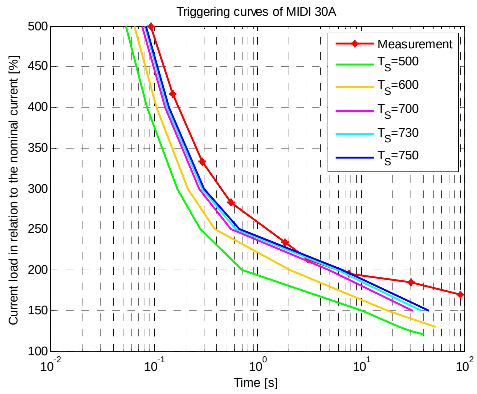

Figure 9. Measured and calculated triggering curves for Midi 30 A.

By trend, the triggering times of our simulations are shorter than the measured ones. There are three possible reasons to justify this tendency: firstly, as already mentioned, we do not really know the exact melting temperature of the fuse element, possibly it is higher than supposed. Secondly, some further energy is needed to melt the material what we call latent heat. Especially for lower loads (less than 150 %), this effect could strongly influence the triggering times. By now, the change of the (aggregate) state and especially the energy necessary for this process is not respected in our simulation. Thirdly, in reality heat dissipates via the attached cables which could also contribute to slower triggering times.

Nevertheless, the measured triggering curves and the maximal temperature curves have the same deviation. If we suppose the melting temperature of the fuse element to be at 730 °C, our simulation results excellently agree with the measured results, especially for higher loads. The metrological justification for this assumption is still outstanding.

6. Results

For the cooperation with some fuse producers, the COMSOL simulation will be applied to test external and design influences regarding the triggering behaviour of the fuses. Exemplarily, we show two examples how to employ the simulation for estimations regarding the design of fuses.

To investigate the influence of the plastic housing, tests with and without plastic housing were performed for the fuse Midi 25 A. For the simulation without plastic housing, the effect of surface to surface radiation is not present because of the absence of the air cavity. Instead, our temperature dependent formulas for convection and radiation at the interface of fuse element to air are used with appropriate parameter values.

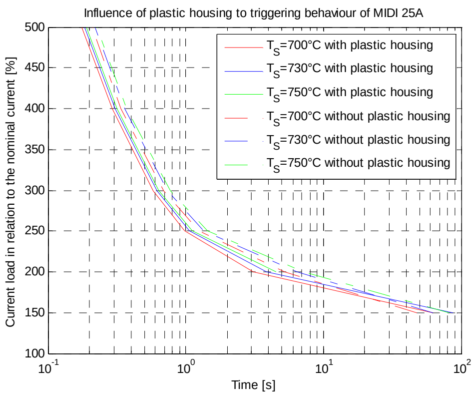

Figure 10. Influence of the plastic housing to the triggering behaviour of the fuse Midi 25 A.

Figure 10 illustrates that the simulation results with and without plastic housing hardly differ in the triggering times. Although the effect of the plastic housing regarding the triggering times is nearly negligible, it makes sense to build the fuses with plastic housing. Of practical experiences we know that for high currents an electric arc appears that could damage devices near the fuse. The plastic housing reduces the intensity of the generated electric arc and so protects other devices close to the fuse.

In section 5, the importance of the tin layer regarding the triggering behaviour of the fuse was already mentioned. To examine this aspect more intensely, simulations for the fuse Midi 25 A are performed where in one case the fuse element consists of pure E-Cu, in the other case of pure tin.

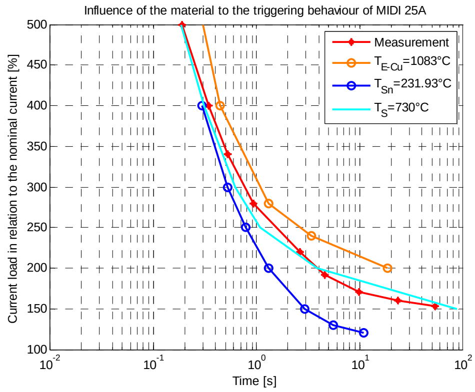

Figure 11. Triggering behaviour of Midi 25 A for different materials.

In Figure 11, we see that the triggering times for a pure tin fuse would be much shorter than of the real Midi 25 A. This is not surprising as the resistivity of tin is higher than the one of E-Cu and the melting point lower. For a pure copper fuse, the effect is inverted, i.e. the triggering times would be longer because of the lower electrical resistance and the higher melting point. Thus, the important influence of the tinny layer to the triggering behaviour is shown by the simulation.

7. Conclusions and future works

Strong electrical stresses in modern cars and less available space reinforce the generation of high temperatures in electrical devices and so enhance the importance to create reliable safety fuses. In this work, a model for the simulation of the instationary heat distribution in safety fuses was presented that allows to investigate internal and external influences as well as the design of the fuse to the triggering behaviour. By our simulations, it was shown that the influence of the plastic housing concerning the heat distribution is nearly negligible, but the tinny layer influences the triggering behaviour essentially.

In our future work, we will perform more measurements where the temperature of the fuse element and the attached cables will be measured. Thus, we will be able to improve our simulation by integrating the influence of the attached cables, of the chemical reactions taking place for lower loads over longer periods and by the exact knowledge when the electric circuit is interrupted.

Concerning our cooperation with fuse producers, the simulation helps to probe different influences of material, shape and environment. The possibility of investigating these influences will contribute to save costs because new fuse types do not have to be produced and tested in a great number but these studies can be performed on the computer by the help of COMSOL.

8. References

1. Münzenmaier A., Einsatz von Hochvoltsicherungen bei Fahrzeugen mit elektrischem Hauptantrieb, *Bachelor Thesis*, University of Applied Sciences Esslingen, Faculty of automotive engineering (2010).

2. Hoffmann G., Kaltenborn U., Thermal Modeling of High Voltage H. R. C. Fuses and Simulation of Tripping Characteristic, *Proceedings of the International Conference on Electrical Fuses and Their Applications*, pp. 174-180 (2003).

3. Vilums R., Buikis A., Conservative Averaging and Finite Difference Methods for Transient Heat Conduction in 3D Fuse, *WSEAS Transactions on Heat and Mass Transfer*, Issue 1, **Vol. 3**, 2008, pp. 111-124 (2010).

4. Vilums R., Ließ H.-D., Buikis A., Rudevics A., Cylindrical Model of Transient Heat Conduction in Automotive Fuse Using Conservative Averaging Method, *Proceedings of the 13th WSEAS International Conference on applied mathematics*, pp. 355-360 (2008).

5. Agarwal M. S., Stokes A. D., Kovitya P., Pre-arcing behaviour of open fuse wire, *J. Phys. D: Appl. Phys.*, **Vol. 20**, pp. 1237-1242 (1987).

6. Giurgiu V., Oarga G., Purdel S., Analysis of thermal phenomena in high-voltage fuse-links, *Proceedings of the International Conference on Electrical Fuses and their Applications*, pp. 65-68 (1999).

7. Torres E., Mazon A. J., Fernandez E., Zamora I., New FEM Model for thermal analysis of medium voltage fuses, *19th International Conference on Electricity Distribution*, Paper No 0576, Vienna (2007).

8. Kawase Y., Miyatake T., Ito S., Heat Analysis of a Fuse for Semiconductor Devices Protection Using 3-D Finite Element Method, *IEEE Transactions on Magnetics*, **Vol. 36**, No. 4, pp. 1377-1380 (2000).

9. Feshki Farahani H., Asadi M., Kazemi A., Analysis of Thermal Behavior of Power System Fuse Using Finite Element Method, *Proceedings of the 4th International Power Engineering and Optimization Conference*, Shah Alam, Selangor, Malaysia, pp. 189-195 (2000).

10. Ilgevicius A., Analytical and numerical analysis and simulation of electrical conductors and fuses, *Ph.D. Thesis*, University of the German federal armed forces, Faculty of Electrical Engineering and Information Technology (2004).

11. VDI-Wärmeatlas, ed. 9, Springer-Verlag, Berlin, New York, pp. Fa 1-Fa 6 (2002).

12. COMSOL Multiphysics, *Comsol Multiphysics Reference Guide*, Theory for the Surface-to-Surface Radiation Interface (2011).

13. Nolet B., Thermische Simulation elektrischer Bauteile und Leitungen und messtechnische Überprüfung, *Internship report*, University of the German federal armed forces, Faculty of Electrical Engineering and Information Technology, p. 14-16 (2011).

14. Deutsches Kupferinstitut, Kupfer-Zinn-Knetlegierungen (Zinnbronzen), I*nformationsdruck i.*15, Auflage 07/2004, Düsseldorf, p. 4 (2004).

9. Acknowledgements

This work is part of the project no. AZ946-10 supported by the Bayerische Forschungsstiftung München.