DECOUPLED ELECTROMAGNETIC SIMULATION OF THE INDUCTION WELDING PROCESS OF CFRTP COMPOSITES

T. Hoffmann, M. Duhovic, P. Mang and P. Mitschang

Leibniz-Institut für Verbundwerkstoffe GmbH, RPTU Kaiserslautern-Landau, Erwin-Schrödinger-Straße 58, 67663 Kaiserslautern, Germany
thomas.hoffmann@ivw.uni-kl.de, www.ivw.uni-kl.de

Keywords: Induction welding, Numerical modeling, Process simulation, Multiphysics simulation

ABSTRACT

Induction welding is an efficient joining technology, particularly for carbon fiber reinforced thermoplastic polymer (CFRTP) composite structures. The combination of high heating rates, localized, intrinsic, and contactless heating allows for the efficient joining of large, complex formed parts. However, the temperature distribution in the joining zone, which largely influences the weld quality, depends on multiple factors and is not easy to predict without the use of advanced numerical modeling techniques. Within this study, a fast simulation approach for the induction welding of CFRTP was developed and validated. In this approach, the electromagnetic (EM) field and the resulting ohm-heating in the composite laminates is only calculated once in a sub-model of the weld geometry. The ohm-heating values are then mapped onto a thermal induction welding simulation of the full weld geometry. Initially, a static single plate induction heating simulation is set-up and validated experimentally both qualitatively and quantitatively. Afterwards, the continuous induction welding simulation is also validated by means of welding trials.

The simulation approach presented in this work was able to accurately predict the temperature development in the joining zone. Compared with state-of-the-art fully coupled electromagnetic-thermal-mechanical continuous induction welding models, calculation times for similar weld geometries could be significantly reduced, which further increases the viability of continuous induction welding or similar electromagnetic process simulations in industrial-scale applications.

1 INTRODUCTION

1.1 BASICS OF THE INDUCTION HEATING PROCESS OF CFRTP

During induction heating of CFRTPs, an induction coil is supplied with an alternating electric current. This leads to the generation of an alternating EM-field around the coil (Figure 1).

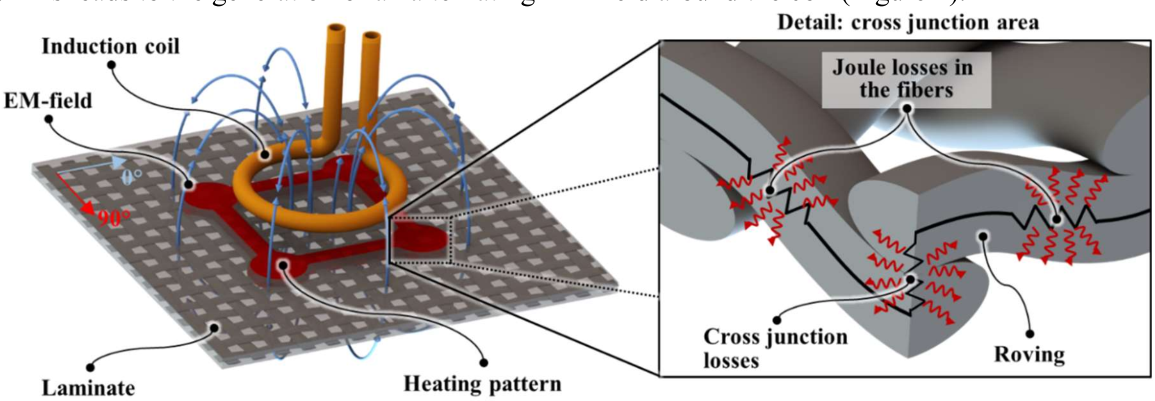

Figure 1: Schematic of the induction heating process in woven fabric reinforced CFRTP.

According to Faraday's law, this alternating EM-field leads to the induction of a voltage across the electrically conductive carbon fibers (CF) within the CFRTP laminate. If the carbon fibers allow the formation of electrical conductive loops (e.g. through cross-junctions), this voltage leads to the formation of electric currents. According to Joules law, these so called eddy currents lead to a rapid intrinsic heat generation within the laminate.

During the induction heating of CFRTP laminates, the heating occurs in two distinct regions of the reinforcing fiber structure (Figure 1, right). When passing through the electrically conductive CF, joule losses due to the electrical resistance of the CF occur. Since the electric current only runs along the electrically conductive CF, it has to alternate between the differently oriented rovings in order to form a closed loop. In these so called cross-junction areas, heat generation occurs due to the contact resistance between the physically contacting fibers of the crossing rovings. If there is no physical contact and only a thin polymer layer separates the crossing fibers, losses due to the dielectric properties of the polymer matrix can occur. However, recent research conducted by Becker et al. shows, that for CFRTP laminates considered in this work (PA66 matrix and Twill 2/2 reinforcement), dielectric losses can be assumed to be negligible [1].

1.2 BASICS OF THE CONTINUOUS INDUCTION WELDING PROCESS OF CFRTPs

Continuous induction welding of CFRTPs takes advantage of their rapid energy-efficient intrinsic induction heating capabilities. In continuous induction welding, the induction coil moves relative to the CFRTP welding partners leading to a continuous melting and resolidification of the polymer matrix at the weld interface (Figure 2). In order to allow for the load introduction of the following consolidation roller, the surface of the upper welding partners needs to remain solid and therefore must not be heated above the melting temperature. For this reason, the surface of the upper welding partner is usually actively cooled (e.g. by means of air cooling).

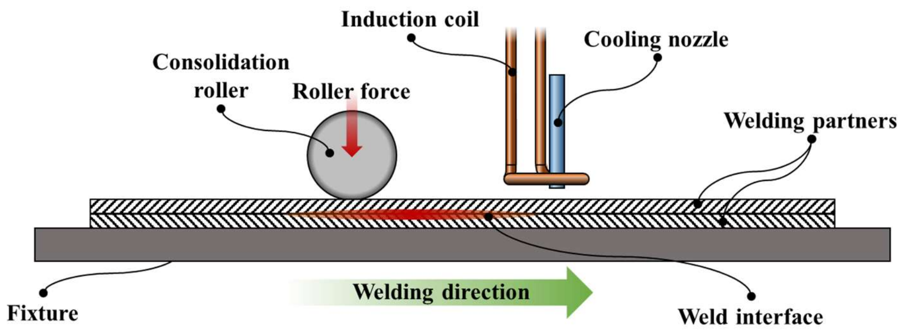

Figure 2: Schematic of the continuous induction welding process.

2 FE-MODEL SETUP

Numerical models of the continuous induction welding process of CFRTP usually come with one of two shortcomings:

- Continuous induction welding modeling approaches generally make use of simplified isotropic material models, where differences in the laminate layup leading to complex heating patterns cannot be modeled. [2]
- Detailed mesoscopic models, where different laminate layups are taken into account by discretely modeling unidirectional layers with anisotropic material properties lead to increased calculation times. Therefore, these models are usually only used to model static induction heating behaviour. [3]

In this work, a new approach combining the possibility of efficiently using any kind of electromagnetic material model to determine the heating behavior of a CFRTP laminate in combination with large weld geometries is developed and validated by means of static induction heating tests and continuous induction heating experiments. The modeling approach presented in this work was set up in LS-Dyna and consists of two sequential simulation steps:

Step 1) In the first simulation, a submodel of the full weld geometry is used to calculate the static ohm-heating (Figure 3, left). Since only the ohm-heating values are of interest, this simulation only makes use of the electromagnetic induced heating solver of LS-Dyna.
Step 2) Once the static electromagnetic simulation is carried out, the ohm-heating values are extracted and mapped onto a continuous thermal-mechanical simulation of the full weld geometry (Figure 3, right).

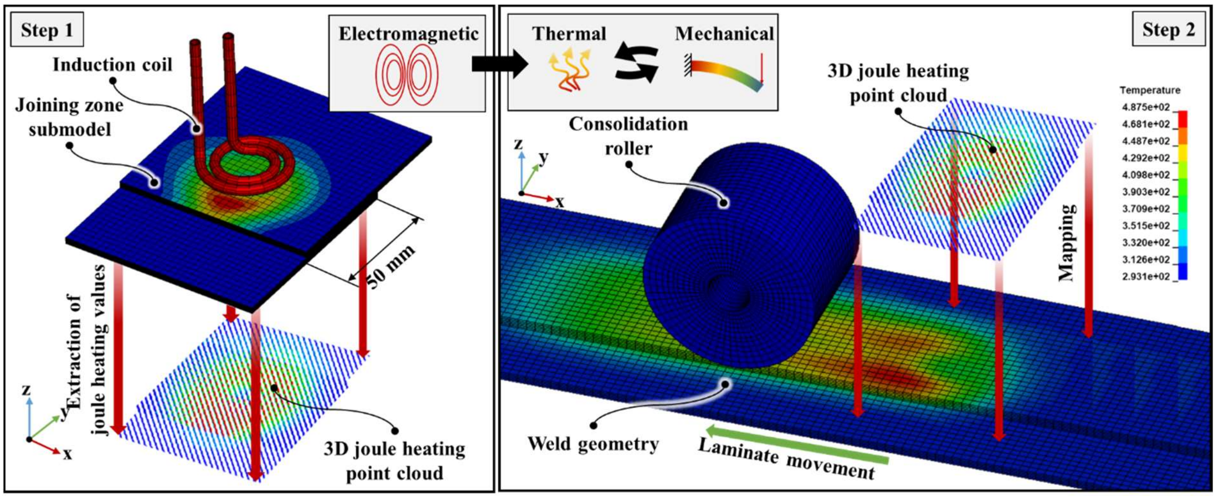

Figure 3: Visualization of the decoupled simulation approach for the continuous induction welding of CFRTPs using the example of a Single-Lap-Shear weld geometry.

Due to the large number of physical effects affecting the temperature distribution in the laminate, an additional single plate static induction heating model already presented in [4] is used to validate the mesoscopic electromagnetic material model prior to the continuous induction welding simulation.

2.1 SINGLE PLATE STATIC INDUCTION HEATING MODEL

To validate the electromagnetic material model, a simple static heating simulation for a single plate laminate-inductor configuration is set up. This model consists of a single laminate and the induction coil, closely replicating the static induction heating setup as shown in 3.1 (Figure 6).

The material model of the CFRTP laminate consists of discretely modeled unidirectional layers. For the material used in this work (Material: Tepex-Dynalite 201-C200(9)/50; Manufacturer: Bond-Laminates GmbH), each of the 20 fabric plies is separated into two cross-ply unidirectional (UD) layers (see Figure 4, detail). Since the reinforcing twill 2/2 fabric used in the laminates is balanced, all UD-layers are assumed to be identical with a fiber volume content of $\varphi = 50$ %.

To model the electromagnetic properties of the UD-layers, an orthotropic material model is used:

- σ_{0° describes the electric conductivity of the UD-layers in fiber direction, and is therefore used to model the joule losses in the fibers. This value is obtained by the rule of mixtures (volume ratio) using literature values for the electrical conductivity of the fibers. The electrical conductivity of the PA 66 matrix is assumed to be negligible.
- σ_{90° describes the in-plane electrical conductivity transverse to the fiber direction. This value was obtained using micromechanical approaches. [5]
- σ_z defines the electrical conductivity in thickness direction of the laminate and corresponds to the cross-junction losses. This value was obtained empirically.

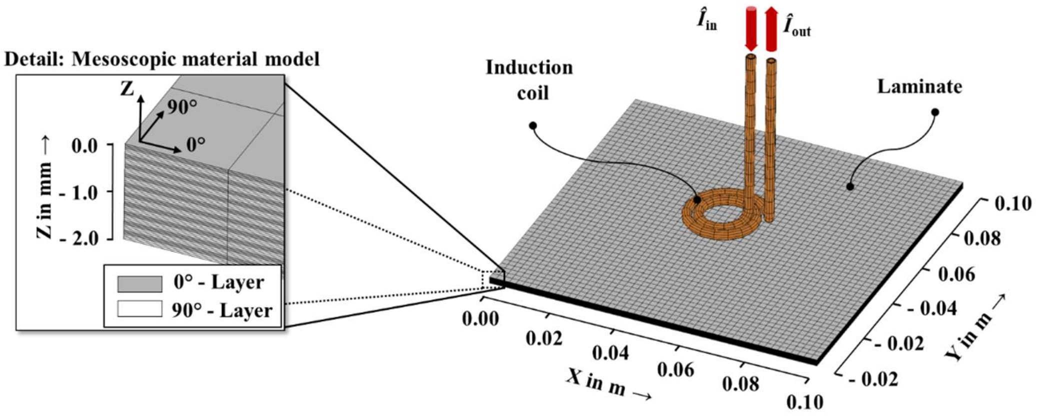

Figure 4: Image of the electromagnetic-thermal FE-Model of the static induction heating process of CFRTP. The electromagnetic boundary conditions $\hat{I}_{in}$ and $\hat{I}_{out}$ are defined at the end faces of the inductor.

The voltage, amplitude and frequency are defined as electromagnetic boundary conditions at the end faces of the inductor. The thermal properties of the CFRTP are modeled macroscopically. The values presented in Table 1 are calculated using micromechanical approaches and the laminate theory according to [5] based on literature values.

	PROPERTY	UNIT	VALUE CFRTP UD-Layer	Inductor (Copper)
ELECTRO-MAGNETIC	**Material Properties**			
	$\sigma_{0°}$	$S \cdot m^{-1}$	16,000	$5.998 \cdot 10^{7}$
	$\sigma_{90°}$	$S \cdot m^{-1}$	1.92	
	σ_{z}	$S \cdot m^{-1}$	1.92	
	Boundary Conditions (corresponds to 40 % generator power)			
	Voltage (Amplitude)	V	ND	112.76
	Frequency	Hz	ND	520,000
THERMAL	**Material Properties**			
	$\lambda_{x,y}$ (in plane)	W/(m·K)	2.764	390
	λ_{z} (thickness direction)	W/(m·K)	0.413	
	Heat Capacity	J/kg·K	TD (DSC Measurements)	383
	Density	kg/m^3	1,568	8,960
	Melting temperature	°C	260	ND
	Melting enthalpy	J/kg	35,890	ND
	Boundary Conditions			
	Free Convection	$W/(m^2 \cdot K)$	TD	ND
	Radiation	$W/(m^2 \cdot K)$	$\varepsilon = 0.98$	ND

*ND = Not defined **TD = Temperature dependent

Table 1: Overview of the electromagnetic and thermal material properties of the CFRTP UD-layers and the copper inductor used in the static induction heating simulation model.

To closely resemble the experimental static induction heating setup used to validate the material model, thermal boundary conditions are applied to the inductor facing and opposing surfaces of the laminate. The vertical free convection occurring in the experiments is modeled using an analytical approach according to [6]. Thermal radiation is also taken into account and an emission coefficient, ε of 0.94 is assumed [7].

2.2 CONTINUOUS INDUCTION WELDING SIMULATION

Electromagnetic joining-zone submodel

To calculate the ohm-heating values of the continuous induction welding laminate configuration, the previously introduced single plate static induction heating model is adapted. The initial single laminate is replaced with two laminates placed in a 50 mm overlapping configuration (Figure 3, left). The length (Figure 3, x-direction) and width (Figure 3, y-direction) of the model is dimensioned liberally to avoid any unwanted edge-effect at the outer edges of the submodel geometry. The inductor is positioned identically to the experimental continuous induction welding setup. The electromagnetic properties of the model are identical to the values presented in Table 1. Since only the electromagnetic ohm-heating is of interest, no thermal material properties or boundary conditions are defined.

Thermal-mechanical continuous induction welding model

Once the heating values are processed and scaled according to the desired power level of the induction heating generator, they are used as a user defined heat source in a continuous thermal-mechanical model of the CFRTP induction welding process. This model only consists of the laminate setup and the tooling (Figure 5). The inductor and consolidation roller are only modeled for visualization purposes and do not take part in the thermal calculation.

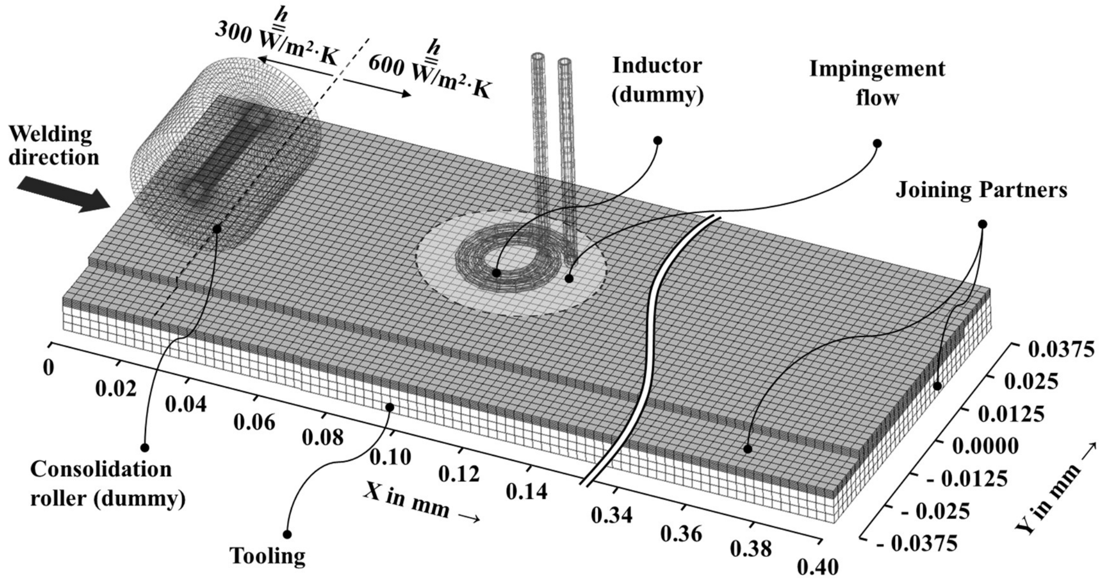

Figure 5: Thermal-mechanical continuous induction welding simulation model.

The impingement flow is modeled analytically based on an empirical model as used by Hofmann et al. [8].Since all necessary electromagnetic calculations are performed in the EM-submodel, only the thermal material model (using the parameters specified in Table 1) is used. There is no mesoscopic material model present in this model. Therefore, no UD-layers need to be spatially resolved by the mesh. For this reason, a reduced mesh density in the thickness direction (5 elements) of the laminate can be used.

Surface cooling plays a major role in the continuous induction welding process. Therefore, realistic thermal boundary conditions are of major importance and highly influence the resulting temperature distribution in the welding partners. The thermal boundary conditions included in this model can be distinguished by surfaces on the inductor facing side or opposing side of the weld geometry.

Thermal boundary conditions on the inductor facing side of the welding partners need to take the effect of the air cooling into account. This also includes modeling the influence of the positioning of the consolidation roller on the airflow. To model the complex air flow and thereby resulting convection coefficients, three different areas are defined:

- In the direct circular vicinity of the impingement airflow, the convection coefficients are modeled using an approach as developed by Hofmann et al. [8]. For the air-nozzle inner diameter used in the experiments, the impingement airflow model is valid within a 20 mm distance from the nozzle center.
- In the area in front of the consolidation roller, where the approach from Hofmann et al. is not valid, the heat transfer is modeled via a constant convection coefficient $h = 600$ W/(m^2·K). This value was obtained by means of an iterative calibration for a process speed of $v = 200$ mm/min.
- The consolidation roller obstructs the air flow originating from the cooling nozzle. For this reason, a lower but still moderately high convention coefficient of h = 300 W/(m2·K) is used. This value is also obtained by the iterative calibration process for $v = 200$ mm/min.

Previous experiments at the IVW have shown, that the heat absorption of the consolidation roller has no significant cooling effect on the temperature distribution in the joining zone and only a brief direct influence on the surface temperature distribution. For this reason, the heat transfer to the consolidation roller is not considered in the model.

Inductor opposing side thermal BC's

The inductor opposing side of the laminates is placed on a thermal insulating material K-Therm® AS 600 M from AGK Hochleistungswerkstoffe GmbH. The thermal properties of the tooling are taken from the manufactures specification sheet. The heat transfer coefficient from the laminate surface to the tooling surface is defined as a constant $h = 1{,}500$ W/(m^2·K).

3 EXPERIMENTAL SETUP

In this work, two experimental setups where used. A static single plate induction heating setup already presented in [9] was used to validate the electromagnetic material model introduced in 2.1. The thermal-mechanical simulation model of the continuous induction welding process is validated using the continuous induction welding setup available at the IVW.

3.1 STATIC SINGLE PLATE INDUCTION WELDING SETUP

In order to conduct the static induction heating experiments, each specimen was clamped vertically in a fixture as shown in Figure 6.

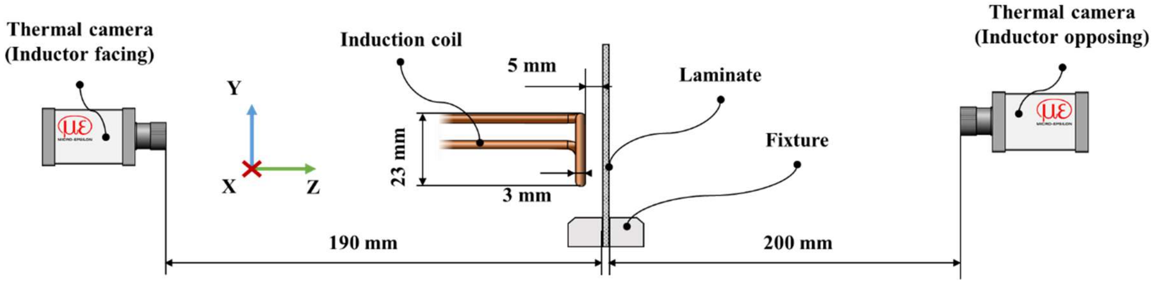

Figure 6: Schematic of the static induction heating setup used at the IVW.

Each specimen is inductively heated with a pancake inductor (23 mm in diameter and a coupling distance of 5 mm). The inductor is powered by a Powercube PW3-32/400 high-frequency generator from C.E.I.A S.p.A (Italy). The temperature was measured with two TIM 160 infrared cameras (MICRO-EPSILON MESSTECHNIK GmbH & Co. KG, Germany). To provide an even emissivity of the measured laminate surface, emissivity tape from TESTO SE & Co. KgaA (Germany) with an emissivity coefficient of $\varepsilon = 0.95$ was applied to the laminate surface.

3.2 CONTINUOUS INDUCTION WELDING SETUP

The induction welding unit used for the validation of the continuous induction welding simulation consists of a consolidation roller, an induction coil and an air cooling nozzle (Figure 7). The unit mounts to an industrial robot KUKA KR 125 (Kuka AG, Germany). The measurements of the induction coil are identical to the one used in the static induction heating trials. The induction coil is connected to the same generator used in the static induction heating setup.

To validate the temperature distribution in the joining zone, six Type E thermocouples (Wire diameter 0.08 mm) are placed in the joining zone according to the positions specified in Figure 10.

The validation of the temperature on the laminate surface is carried out with a TIM 160 infrared camera.

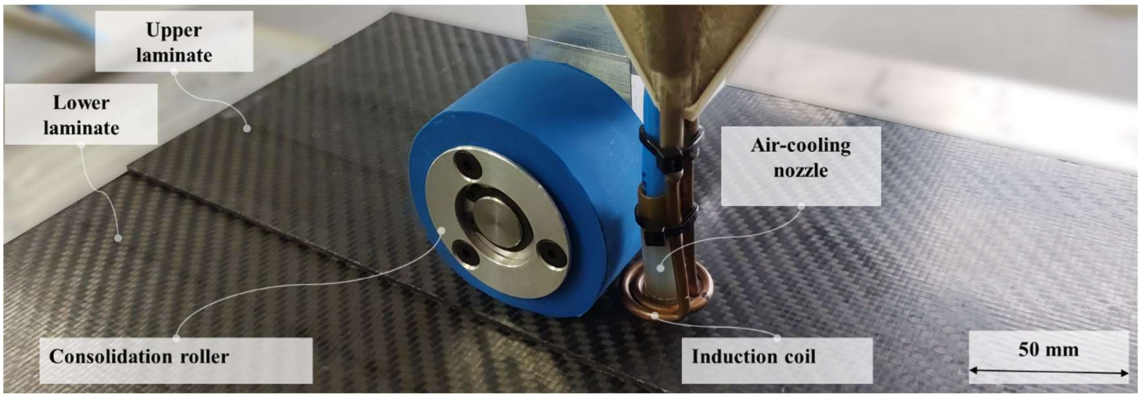

Figure 7: Continuous induction welding setup used for the validation experiments.

4 EXPERIMENTAL VALIDATION

As established, the experimental validation of the continuous induction welding simulation is split into the static validation of the electromagnetic material model by means of static induction heating experiments and the final validation of the continuous induction welding simulation by means of induction welding tests.

4.1 STATIC VALIDATION OF THE ELECTROMAGNETIC MATERIAL MODEL

For validation of the electromagnetic material model, the temperature development of the single plate static induction heating model (2.1) is compared to static induction heating tests performed with the setup introduced in 3.1. The parameters used in the simulation of the associated heating tests are provided in Table 1. The validation of the electromagnetic material model is carried out both qualitatively and quantitatively.

The simulated heating curves of the local temperature hot-spots at the inductor-facing and opposing surfaces of the laminates correlate very well with the experimental values (Figure 8). Temperature differences mostly trace back to the initial 0.25 s of the static heating process, where the experimental heating rates are briefly below the simulated heating rates. This is most likely due to start-up effects of the induction generator, which are not considered in the simulation. In continuous induction welding processes, these differences are not of interest, since the induction generator is running at a constant power output.

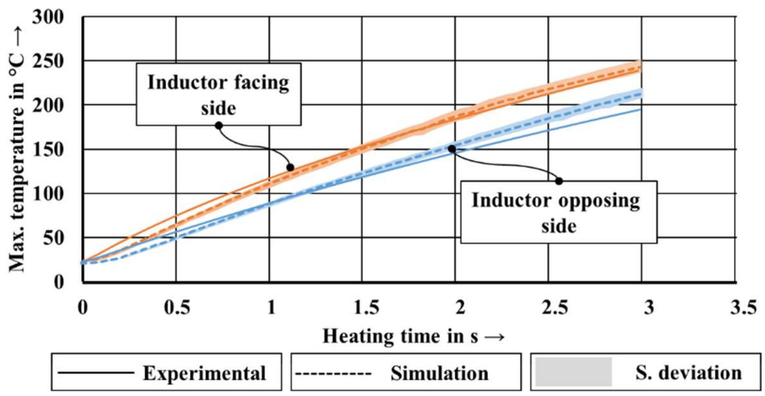

Figure 8: Simulated and experimental temperature curves at the local temperature hot spots of the inductor facing and opposing sides of the specimens.

Furthermore, a qualitative evaluation of the simulated heating pattern is performed. For this, an average heating pattern of the experimental thermography recordings is calculated (Figure 9). Since the inductor facing surface of the laminate is partially covered by the induction coil, this is only performed for the inductor opposing side of the laminates. The average heating pattern as well as the simulated heating pattern are evaluated for a maximum temperature of 150 °C. The simulated heating pattern corresponds well with the averaged experimental heating pattern.

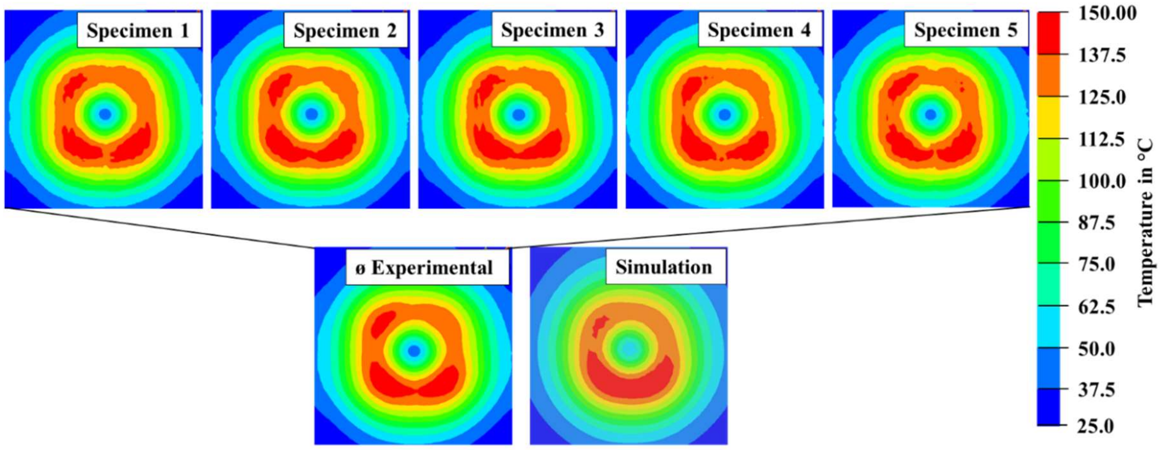

Figure 9: Comparison of the simulated heating pattern to an average of the experimental heating patterns of the inductor-opposing side of the laminates. The single heating patterns are evaluated at the time of a hot-spot temperature of 150 °C.

4.2 VALIDATION OF THE CONTINUOUS INDUCTION WELDING SIMULATION

Step 1: Electromagnetic joining-zone simulation

The electromagnetic simulation of the ohm-heating values is performed with the model introduced in 2.2. The simulation is carried out with a voltage amplitude and frequency matching a generator power setting of P = 40 % which corresponds to the values as specified in Table 1. After the calculation of the ohm-heating, the heating values are read out for every element of the laminate configuration. These values are then transferred into a volumetric heat source. Different induction generator power settings are modeled by linearly scaling the heating values.

Even though the same voltage as well as frequency values (compared to the static single plate validation trials) were applied to the inductor according to the *Ceia Power Cube Assistant* software, the resulting heating rate of the simulated 50 mm overlapping configuration was highly overpredicted. For this reason, a one-time calibration of the heating values is necessary, in order to achieve realistic temperatures for the thermal simulation. Initial calibration experiments have shown, that a reduction of the ohm-heating values to 75 % of the initial simulated values corresponds well to the induction welding experiments. The calibration was carried out via a welding test at 100 mm/min without cooling and reduced generator power (10 %). Thermocouples TC 1, TC 2, TC 3 and TC 6 along the weld centerline where used for the calibration.

Step 2: Thermal simulation of the continuous induction welding process

In order to validate the continuous induction welding simulation, welding tests at two different process speeds (100 mm/min and 200 mm/min) were conducted (Figure 10). The corresponding power settings to reach the target temperature of 310 °C where determined in pretrials. To track the temperatures in the welding zone, six thermocouples where positioned according to Figure 10. Thermocouples TC 1, TC 2, TC 3 and TC 6 were placed along the center line at the joining interface of the welding stack. Two additional thermocouples, TC 4 and TC 5, where placed 8.3 mm from TC 3 across the weld width, where the maximum process temperatures are expected due to the induction coil geometry.

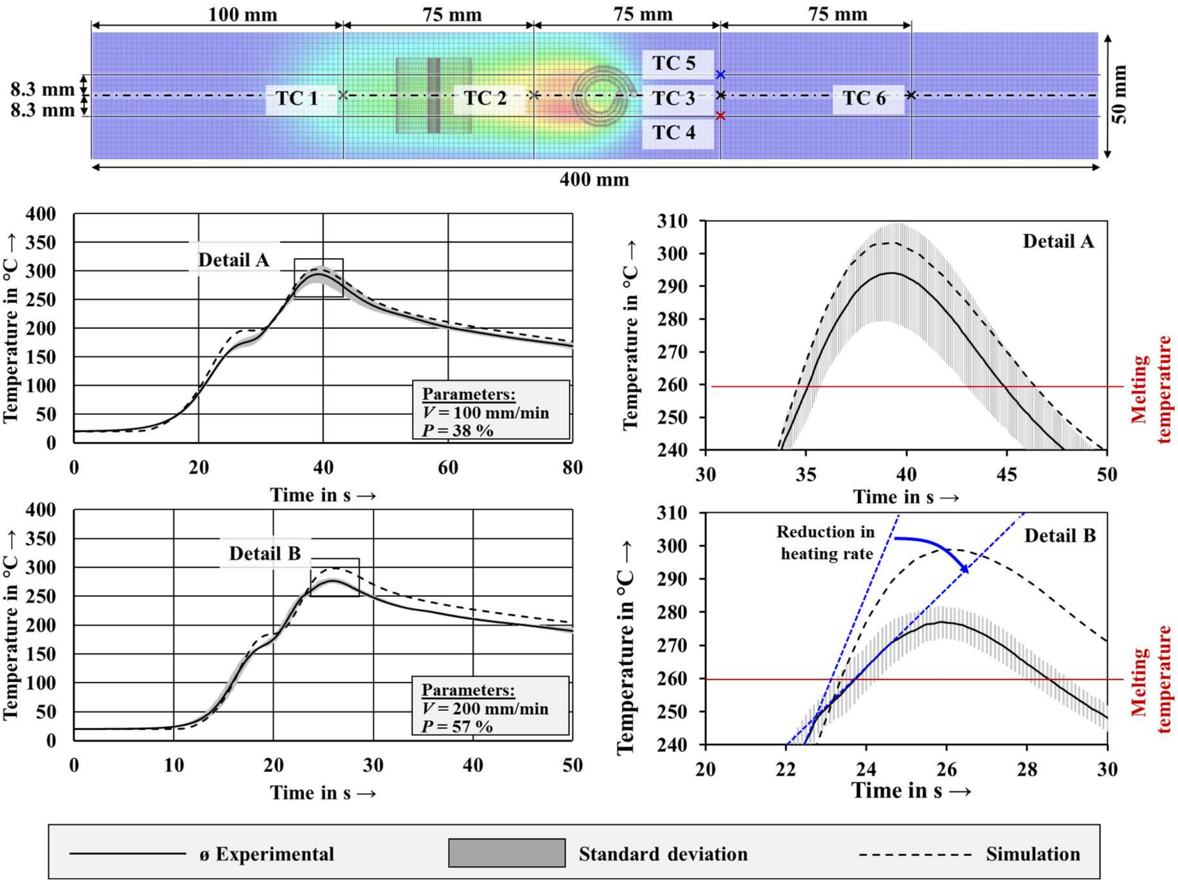

Figure 10: Comparison of the experimental temperature curves and the simulation results. The thermocouples are positioned analogously to the marked positions TC 1 – TC 6. For the sake of clarity, the experimental temperature curves TC 1, TC 2, TC 3 and TC 6 were combined into a single curve with a standard deviation.

The temperature prediction of the simulation along the weld centerline shows a very good fit for a process speed of 100 mm/min (Figure 10, top left graph). Even above the PA 66 melting temperature (260 °C) the simulated values only result in a slight overestimation of the experimental temperature measurements.

The 100 mm/min process speed simulation also gives a very good prediction up to temperatures of approximately 250 °C (Figure 10, bottom left graph). From this temperature onwards, the gradient of the experimental curve drops significantly resulting in lower temperature than predicted by the simulation. This difference may be caused by a number of effects influencing the real heating behavior above melting temperature, which are not included in the presented model (e.g. different contact conditions at the cross-junctions due to delamination). Due to the drop in EM-field intensity, layers closer to the inductor may already be above melting temperature and already have an effect on the heating behavior directly at the weld interface. While this effect is not as obvious at lower process speeds, the significant increase of the standard deviation above 250 °C supports this conclusion.

Additionally, the maximum predicted weld-zone temperatures perpendicular to the process direction are validated by thermocouples TC 3, TC 4 and TC 6 (Figure 11). For both considered process speeds, the simulated maximum temperatures correspond very well with the experimental values.

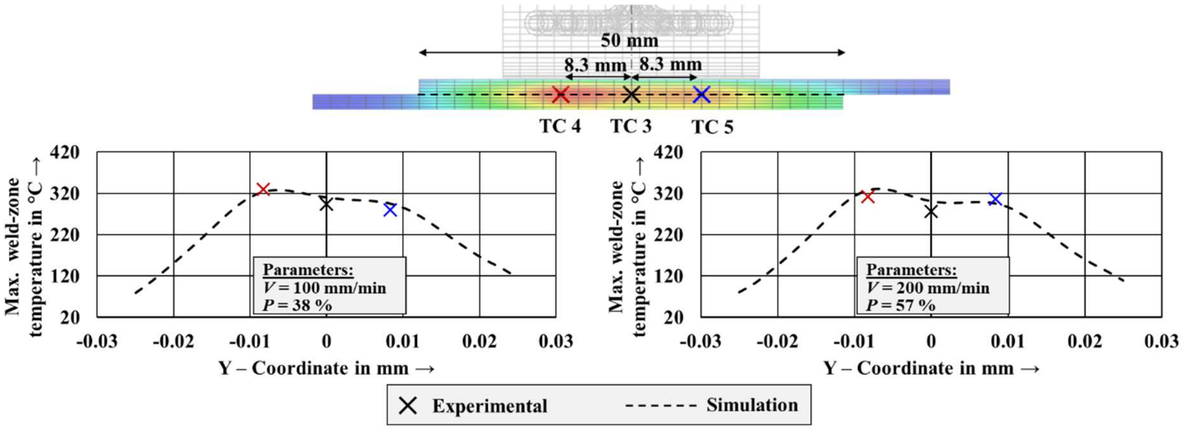

Figure 11: Simulated prediction of the maximum temperature along the width of the weld-zone compared with the experimentally determined maximum temperatures.

5 CONCLUSION

In this work, a decoupled approach for the simulation of continuous electromagnetic induction welding of CFRTPs was introduced. Compared to conventional, fully coupled EM-TH-ME simulations, calculation times for similar weld geometries could be reduced by a wide margin from more than 1 week to less than 24 hours for the initial simulation (EM + coupled TH-ME) and less than 12 hours for every subsequent parameter variation (coupled TH-ME). Additionally, this approach allows for the use of more complex electromagnetic material models while still keeping the mesh size of the full weld geometry manageable.

The combination of a mesoscopic electromagnetic material model with the newly developed decoupled approach was able to generate very accurate simulation results which were validated for process speeds up to 200 mm/min.

However, due to the large number of physical effects in the continuous induction welding process of CFRTPs, numerical approaches are not yet able to fully replace physical testing. A small number of initial induction welding tests are still necessary to calibrate the model and achieve accurate simulation results.

ACKNOWLEDGEMENTS

This work has been conducted within the framework of the project ''Process Optimization of Induction Welding of Continuous Carbon-Fiber Reinforced Thermoplastics by Process Simulation'' which is funded by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) – 272768988.

REFERENCES

[1] S. Becker, M. Michel, P. Mitschang und M. Duhovic, „Influence of polymer matrix on the induction heating behavior of CFRPC laminates," *Composites Part B 231,* 2022.

[2] F. Lionetto, S. Pappadà, G. Buccoliero und A. Maffezzoli, „Finite element modeling of continuous induction welding of thermoplastic matrix composites," *Materials and Design 120,* pp. 212-221, 2017.

[3] F. Lundström, K. Frogner und M. Andersson, „A numerical model to analyse the temperature distribution in cross-ply CFRP during induction heating," *Composites Part B 202,* 2020.

[4] T. Hoffmann, S. Becker, M. Duhovic und P. Mitschang, „Simulating the induction heating behavior of CFRTPC Laminates," in *13th European LS-DYNA Conference 2021*, Ulm, 2021.

[5] H. Schürmann, Konstruieren mit Faser-Kunststoff-Verbunden, Darmstadt: Springer-Verlag, 2007.

[6] P. Stephan, VDI Wärmeatlas, 11. Auflage, Darnstadt: Springer-Verlag, 2013.

[7] E. K. C. M. J. H. A. Adibekyan, „Review of PTB Measurements on Emissivity, Reflectivity and Transmissivity of Semitransparent Fiber-Reinforced Plastic Composites," *International Journal of Thermophysics,* 2019.

[8] H. M. Hofmann, M. Kind und H. Martin, „Measurements on steady state heat transfer and flow structure and new correlations for heat and mass transfer in submerged impinging jets," *International Journal of Heat and Mass Transfer 50,* pp. 3957-3965, 2007.

[9] S. Becker und P. Mitschang, „Influence of the Thread-count of Carbon Twill Textile Reinforced Polyamide 66 Laminates on the Inductive Heating Behavior," *Journal of Thermoplastic Composite Materials,* 2019.