

# DETERMINATION OF INPUT PARAMETERS FOR A MIX-MODE COHESIVE ZONE FORMULATION OF A NOVEL METAL-CFRP-JOINING INTERFACE

J. Kronsteiner<sup>a\*</sup>, S. Ucsnik<sup>a</sup>, S. Stelzer<sup>b</sup>

<sup>a</sup> LKR Leichtmetallkompetenzzentrum Ranshofen GmbH, Austrian Institute of Technology, 5282 Ranshofen, AUSTRIA

<sup>b</sup> Montanuniversität Leoben, 8700 Leoben, Otto Glöckel-Straße 2/II, AUSTRIA

\* johannes.kronsteiner@ait.ac.at

**Keywords:** COHESIVE ZONE, MIX-MODE, DOE, METAL-CFRP JOINT

## Abstract

*The simulation of joints between Carbon Fiber Reinforced Polymers (CFRP) and metals in modern lightweight structures needs input parameters which can only be partly determined by experiments. This paper presents a methodology for an automated determination of input parameters by reverse engineering. With the help of Design of Experiments (DOE) and optimization algorithms, the parameters are investigated by comparison of experimental results with results from numerical simulations of the same tests. The presented methodology is an effective way to speed up the parameter determination.*

## 1. Introduction and background

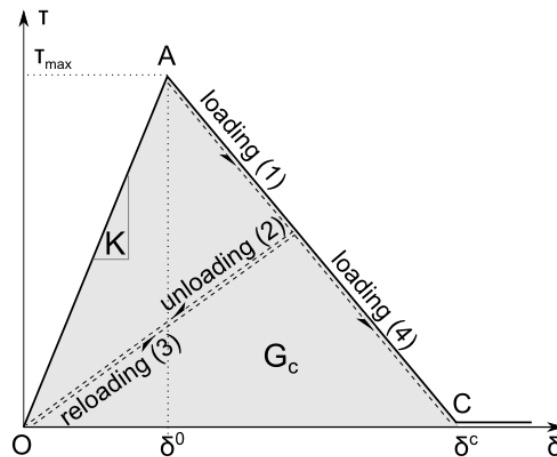
Lightweight design is a necessity for any modern means of transport. The design of lightweight structures very often requires a combination of miscellaneous materials. Although composite materials have many advantageous attributes, metals still prevail in many structures. Joining composite materials like CFRP with metal structures in an efficient way offers an enormous potential but also great challenge.

In adhesive bonds, the determination of the interface normal stress ( $\sigma$ ) and tangential stress ( $\tau$ ) can be easily performed by experiments for pure CFRP or metal bonds. Difficulties arise when co-cured metal-CFRP interfaces are considered. In the presented work, a titanium sheet is joined with CFRP beams in order to produce a specimen for the assessment of a metal-CFRP connection. Tests for tensile and shear loading allow the evaluation of the joint behavior under different loading conditions. The resulting debonding behavior is used for the characterization of the bonding properties using standardized tests [3][4]. One possibility of modeling the bonding-debonding-opening behavior of adhesive joints in numerical simulations is to use Cohesive Zone (CZ) contact formulations [1]. However, difficulties exist in the determination of all required input parameters solely through experiments.

The aim of this paper is to present a methodology for the fast and automated semi-empirical determination of all input parameters. These will be used in a CZ formulation for adhesive bonding of co-cured metal to CFRP interfaces in later simulations of pin-reinforced metal to CFRP joints [2].

## 2. The CZ Material Model

Interface delamination can be modeled by introducing fracture mechanism with softening relationships between tractions and separations. These relationships yield a critical interlaminar fracture energy which is required to break apart the initial bonding/contacting surfaces. The presented work uses a bilinear Cohesive Zone Material (CZM) model implemented in ANSYS<sup>®</sup> Mechanical<sup>™</sup>, which is based on the model proposed by Alfano and Crisfield [1]. A graphical representation of the model characteristics is given in Figure 1.



**Figure 1:** Normal contact stress versus contact opening curve for bilinear CZ formulation.

The critical interlaminar fracture energy ( $G_c$ ), determined by experimental tests, corresponds with the integral over the area under the OAC curve (see Figure 1). In order to define the triangle OAC, the contact stiffness  $K$ , the maximum contact stress  $\tau_{max}$  and the critical contact opening  $\delta^c$  need to be determined. When  $\tau_{max}$  is assumed,  $\delta^c$  can be calculated and used to identify the current value of the debonding parameter  $d$ .

The same procedure is applied for the normal and tangential directions of Mode I as well as for Mode II. Therefore, by using the same values for the parameters of all the contact elements, the load solution for the whole specimen can be found.

The implemented CZM model of ANSYS<sup>®</sup> deals with the following parameters: the fracture energies  $G_{Ic}$  and  $G_{IIc}$  the critical stresses  $\sigma_{max}$ ,  $\tau_{max}$  and the stiffness parameters  $K_n$ ,  $K_t$ . The stiffness parameters  $K_n$ ,  $K_t$ , contact stress parameters  $\sigma_{max}$ ,  $\tau_{max}$  as well as parameters for artificial damping  $\eta$  and tangential slip under compressive normal contact opening  $\beta$  are unknown. In the present work, values for artificial damping are used from literature and tangential slip has been activated. The remaining parameters are unknown and need to be determined using the presented methodology herein.

## 3. Procedure for parameter determination

Since only the critical interlaminar fracture energies ( $G_{Ic}$  and  $G_{IIc}$ ) can be efficiently obtained through experimental tests, the stiffness and maximum strength values ( $K_n$ ,  $K_t$ ,  $\sigma_{max}$  and  $\tau_{max}$ ) need to be found by a “Reverse Engineering” approach. This is done through a comparison of experimental force-displacement curves with simulation results of the same tests which allow for an estimation of the unknown material parameters for the CZM contact formulation.

DOE is an efficient way to perform a parameter study by using statistical methods for the definition of parameter sets. By defining the boundary values for each investigated parameter, the software can create “design points” in a multidimensional “design space”. The main task

for the DOE software is to obtain the required information with a minimum of design points. There are several methods used in DOE such as full factorial and fractional factorial designs with different advantages and disadvantages. For the presented work a “Sequential Response Surface” method [5] in combination with an “Adaptive Simulated Annealing” algorithm [6] is used to find appropriate parameter combinations.

The developed procedure combines the mechanical simulation capabilities of ANSYS<sup>®</sup> Mechanical<sup>™</sup> with the DOE and Optimization algorithms used in LS-Opt<sup>™</sup> by LSTC<sup>®</sup>. A special routine has been devised to couple both software packages in order to perform a combined determination of missing parameters  $\sigma_{max}$ ,  $K_n$ ,  $\tau_{max}$  and  $K_t$ . Two Finite Element (FE) simulations are representing the Double Cantilever Beam (DCB) and End Notched Flexure (ENF) tests (see section 2)

The aim of the automated determination routine is to find one set of input parameters for the contacting interfaces with a CZ model which gives good results for the numerical simulation of both the DCB and ENF tests.

Therefore, both numerical models are performed with the same parameters and the result of each test is compared with the corresponding data from the experimental tests via a curve mapping method. The curve mismatch error between simulation and experimental data for both tests is calculated by the DOE software and used to find more suitable design points for the next iteration. In order to automate the procedure, the FEM software ANSYS<sup>®</sup> Mechanical<sup>™</sup> is coupled with the optimization software LS-Opt<sup>™</sup> of LSTC<sup>®</sup> via a proprietary developed routine. A sequence consists of the determination of parameter combinations, calculation of the corresponding results, comparison of the results with experiments and identification of improved parameter combinations. Figure 2 illustrates the optimization procedure used to determine the missing parameters for the Cohesive Zone Material (CZM) contact formulation.

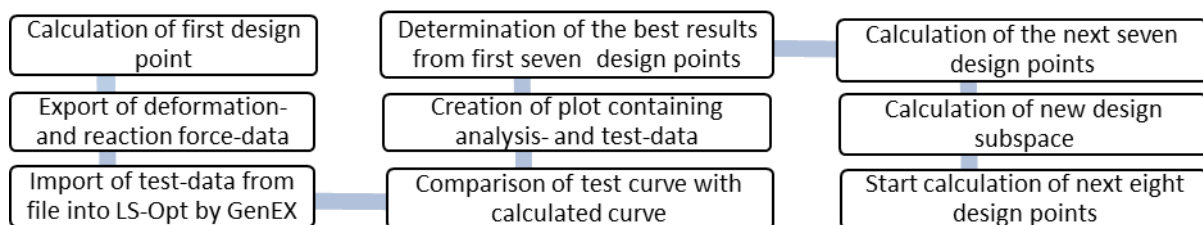


Figure 2: Optimization procedure

#### 4. Experiments

Standardized tests are used for both deliver input data and to permit a comparison of simulation with test results. There are two load forms which need to be considered. The first form (Mode I) with the corresponding DCB test standardized by DIN EN 6033 [3]. The second form (Mode II) induces in plane shear tractions. The commonly used test is ENF test, standardized by DIN EN 6034 [4]. The main purpose of these tests is to obtain the critical interlaminar fracture energy  $G_c$  for Mode I and II which are the only parameters of the CZ formulation to be obtained efficiently by experiments.

#### 4.1. Test specimen

The investigated test specimen for both tests consists of two CFRP sheets (material designation textile: Tenax HTS biaxial NCF with resin: Hexcel RTM 6) with a titanium sheet (Ti-6-Al-4V with 0.4 mm thickness) in between. Each composite beam consists of a double symmetric, quasi-isotropic stacking sequence [0/90/±45] with 10 layers. The metallic surfaces of the titanium sheet are treated by sandblasting. A poly-tetra-fluor-ethylene (PTFE) film at the laminate mid-plane provided the 60mm long starter crack (see Figure 3).

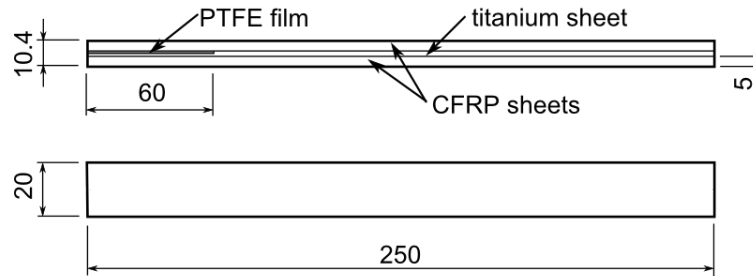


Figure 3: DCB/ENF test specimen dimensions.

#### 4.2. Double Cantilever Beam Test

The DCB test method for Mode I loading introduces peel forces in the area of the crack tip perpendicular to the crack plane (see Figure 4 left). The peel forces drive the crack through the pre-notched interface of the test specimen [1]. The critical interlaminar fracture energy for Mode I ( $G_{Ic}$ ) is calculated through Equation ( 1) according to standard [3];

$$G_{Ic} = \frac{A}{a * w} 10^6 \quad (1)$$

where A is the total energy required to propagate the crack through the specimen, a is the propagated crack length, and w is the width of the specimen. A corresponds to the integral over the area under the load-displacement curve.

#### 4.3. End Notched Flexure Test

For the ENF test, the pre-cracked specimen from the DCB test is loaded in a three point bending fixture until the pre-induced crack propagates further (see Figure 4 right). There are several requirements for the test. These are defined in the standard [4]. The critical interlaminar fracture energy for Mode II ( $G_{IIc}$ ) is calculated from the initial crack length, a, the load at crack propagation onset, P, and the displacement, d, (see Equation ( 2)). The width of the specimen, w, and the span length between the bearings, L, are also involved.

$$G_{IIc} = \frac{9 * P * a^2 * d * 1000}{2 * w * \left(\frac{1}{4}L^3 + 3a^3\right)} 10^6 \quad (2)$$

### 5. Numerical Model setup

For the numerical analysis, ANSYS<sup>®</sup> Mechanical<sup>™</sup> is used. For each experimental test (ENF and DCB), a corresponding numerical model has been prepared.

### 5.1. Geometry

The same beam structure described in section 4.1 is reproduced for the simulation model in ANSYS<sup>®</sup> Mechanical<sup>™</sup> although due to symmetric condition only half of the test setup is modeled. The CFRP sheets are connected to the intermediate titanium sheet by a full bonding contact with an additional CZ formulation for the bonding-debonding-opening behavior. The same parameter set is used for both contact areas (top and bottom surface of the titanium sheet). The initial 60 mm long crack is captured by a friction-sliding contact. The same method is used for the model of the ENF specimen but with a larger pre-cracked area of separable contact (see Figure 4).

### 5.2. Loads and boundary conditions

The loading of both specimens is realized by a displacement in z-direction of +12 mm for the DCB and -3 mm for the ENF test (see Figure 4). The connection of the cylindrical support with the beam specimen of the DCB-test is realized by using aluminum blocks. The blocks are bonded to the beam and are used to transmit the displacement as well as to realize the cylindrical support. The displacement for the ENF-test is simply assigned to an edge of the model, while cylindrical rigid bodies are used for the vertical support (see Figure 4). The distance between the supports is 100 mm according to the standard [4].

### 5.3. Mesh

As many calculation cycles are necessary for the determination of the CZ-parameters, a hexahedral mesh is used for the whole FE model. For a stable calculation with several cycles for different parameters, it is of utmost importance to have a mesh which is fine enough in areas where material or geometric changes happen, and to reduce the number of elements in areas where only minor deformations, strains and stresses occur. The mesh size is thus reduced in areas where the crack is located and is gradually expanded with increasing distance from the crack (see Figure 4).

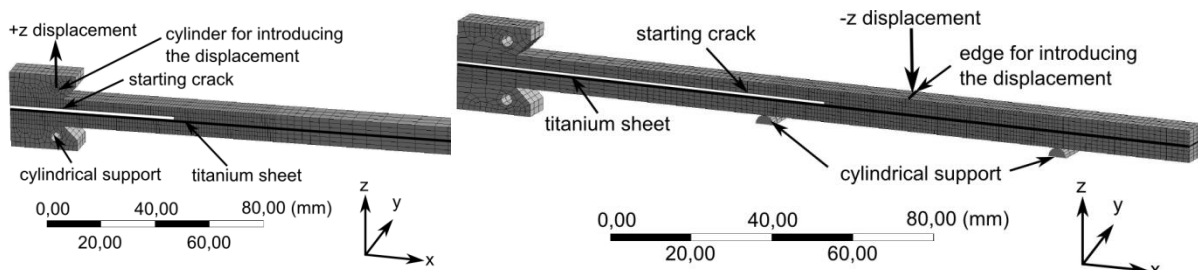


Figure 4: (a) Meshed DCB and (b) ENF-specimen.

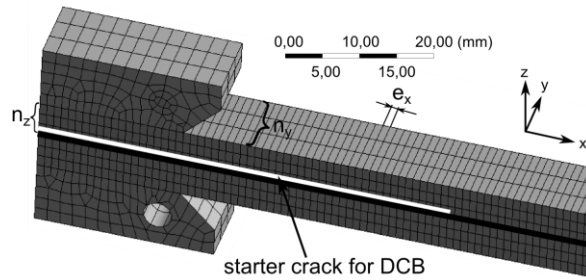
To find an appropriate coarse mesh size with a sufficient accuracy and a reasonable number of elements, several combinations of three different mesh size parameters (see Figure 5) have been investigated in a mesh sensitivity analysis.

The initial set of CZ input parameters for the DCB model which has been used for the FE analysis in Mix-Mode are shown in Table 1.

$\sigma$ [N/mm <sup>2</sup> ]	$\tau$ [N/mm <sup>2</sup> ]	$G_{Ic}$ [N/mm <sup>2</sup> ]	$G_{IIc}$ [N/mm <sup>2</sup> ]	$K_n$ [N/mm <sup>3</sup> ]	$K_t$ [N/mm <sup>3</sup> ]	crack length [mm]
4	30	0.52	1.42	1e4	1e6	35

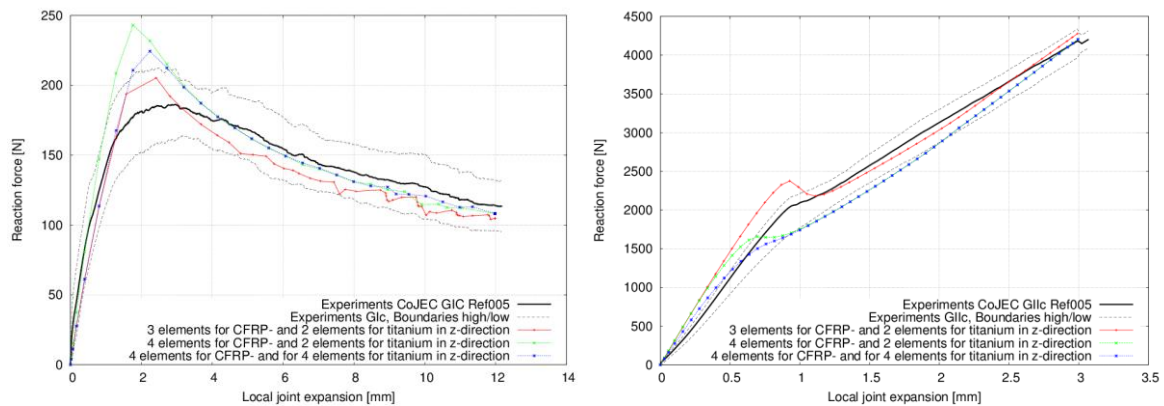
Table 1: Starting CZ input parameters for mesh investigations under Mode I loading.

In Table 1, the critical interlaminar fracture energy for Mode I and Mode II ( $G_{Ic}$  and  $G_{IIc}$  respectively) are determined by DCB and ENF tests. For the stiffness and maximum contact stresses ( $K_n$ ,  $K_t$ ,  $\sigma_{max}$  and  $\tau_{max}$  respectively) the data from the previous investigations are used.



**Figure 5:** Detail of the mesh for the DCB-specimen.

The outcomes of the mesh sensitivity studies were shown that the element length in x-direction  $e_x$  and the number of elements in y-direction  $n_y$  have only minor influence on the numerical results. The number of elements in z-direction  $n_z$  however, has a major influence on the resulting force-displacement curve after the first linear expansion. Figure 6 shows the results of the mesh investigations together with the mean curve and boundary (upper and lower bound) curves from the experiments. The mean and boundary curves are calculated from 11 independently-run experiments. Although the first (red) curve in Figure 6a seems to be already a good match with the mean experimental curve, it shows a poor result (see Figure 6b) compared to ENF experimental result. A major problem is that an optimized parameter set for Mode I would not give an equally good result for Mode II.



**Figure 6:** Results of the mesh sensitivity analysis for (a) DCB and (b) ENF tests in Mix-Mode.

Mesh sensitivity studies led to the conclusion that a mesh with a size of  $e_x = 1$  mm,  $n_y = 3$ ,  $n_z = 4$  and  $n_{z, titanium} = 4$  needs to be used in order to get good results in a reasonable time.

#### 5.4. Material properties

The carbon fiber laminate is modeled with a linear elastic orthotropic model. The 9 independent input parameters are derived from basic material constants of fibers, resin, fiber volume fraction and laminate lay-up by means of a rule of mixture approach and the classical laminate theory. The 9 orthotropic constants are given in Table 2. A linear elastic isotropic material definition is used for the titanium sheet. Additionally, multi-linear isotropic hardening is used in order to incorporate hardening effects. For the aluminum blocks, a linear elastic isotropic material is used (see Table 2).

Material	CFRP laminate	Titanium	Aluminum
$E_x=E_y$ [MPa]	54690	117200	70000
$E_z$ [MPa]	10631	117200	70000
$G_{xy}$ [MPa]	20920	25926	44737
$G_{xz}=G_{yz}$ [MPa]	4159	25926	44737
$\nu_{xy}$	0.307	0.31	0.35
$\nu_{xz} = \nu_{yz}$	0.255	0.31	0.35
$\rho$ [kg/m <sup>3</sup> ]	1480	4510	2710
$\alpha$ [1/°K]	4.5E-7	8.9E-6	2.35E-5

Table 2: Material data

### 5.5. Simulation settings

Convergence poses a great challenge due to the debonding behavior of the bilinear CZM contact formulation and the large deformations. The incorporation of those nonlinearities calls for a robust sparse direct solver (according to [7]) with the possibility of large deflections. For the same reason, automatic time stepping is used to allow the solver to reduce the time step where necessary (down to  $1e^{-6}$  s) while minimizing the number of calculation cycles for the final solution by increasing the time step size again (up to 0.01 s).

## 6. Results of DOE Studies

The final mutual parameter set determined by the combined DOE-LS-Opt optimization of input parameters for DCB- and ENF-tests is given in Table 3.

In order to get good results in a reasonable time, a mesh size of  $e_x = 1\text{mm}$ ,  $n_y = 3$ ,  $n_z = 4$  and  $n_{z, \text{titanium}} = 4$  is used with quadratic elements for all investigations. For one optimization cycle, 56 simulations have been executed where the spectrum of results was very wide.

$\sigma$ [N/mm <sup>2</sup> ]	$\tau$ [N/mm <sup>2</sup> ]	$G_{lc}$ [N/mm <sup>2</sup> ]	$G_{llc}$ [N/mm <sup>2</sup> ]	$K_n$ [N/mm <sup>3</sup> ]	$K_t$ [N/mm <sup>3</sup> ]	crack length [mm]
6.74	29.35	0.56	1.5	10637	75275	35

Table 3: Final parameter set for CZM contact formulation.

Figure 7 shows the comparison between the DCB and ENF models with according experimental results.

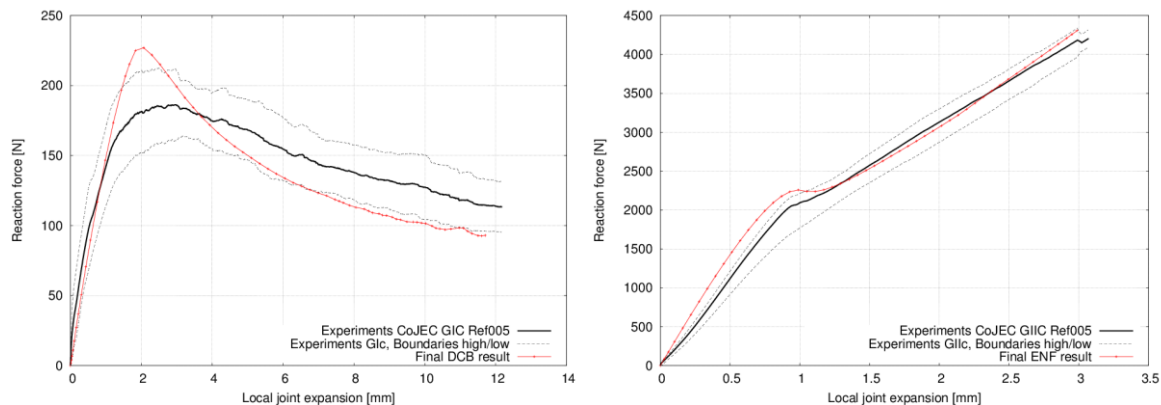


Figure 7: Optimized results for (a) DCB and (b) ENF test with final parameter set.

In Figure 7a, the linear elastic part (approximately 1 mm joint expansion) gives a good match to the mean curve of the experiments. In Figure 7b, results fit quite good after approximately 1 mm joint expansion but the linear elastic part shows higher stiffness.

For the DCB test, the peak force values are mostly higher and the gradient of decrease is often steeper than in the experiments for DCB-tests (see Figure 7a). The agreement of the results between simulation and experiment for the ENF test is much better (see Figure 7b). The simulations yield higher initial stiffness and a distinctive transition from linear elastic behavior to the crack initiation. Part of the problem is that the parameter set needs to be a compromise between DCB and ENF tests. High stiffness values for the ENF tests give rather low initial stiffness for DCB tests and vice versa (compare Figure 7a and Figure 7b).

## 7. Results and discussion

To answer the basic question of how to make a better compliance between experimental and simulated results, more details need to be elaborated. One reason for the very smooth peaks in the experimental curves could be an effect called “fiber-bridging” [8]. This effect describes the phenomenon of isolated fibers still connecting the crack fronts. Therefore, higher loads can be absorbed by the specimen and the transition from the linear elastic to the crack propagation regime is much smoother. This effect is not considered in the numerical model herein. However, the process of automatic determination of parameters with DOE and optimization methods has proved to be effective.

## References

- [1] G. Alfano and M.A. Crisfield, *Finite Element Interface Models for the Delamination Analysis of Laminated Composites: Mechanical and Computational Issues*, International Journal for Numerical Methods in Engineering, Vol. 50, 2001, pp. 1701-1736.
- [2] S. Ucsnik et al., *Experimental investigation of a novel hybrid metal-composite joining technology*, Composites: Part A, Elsevier, 2009, pp. 369-374.
- [3] *Bestimmung der interlaminaren Energiefreisetzungsrates Mode I, GIc*, Normenstelle Luftfahrt (NL) im DIN Deutsches Institut für Normung e. V., Köln, 1996.
- [4] *Bestimmung der interlaminaren Energiefreisetzungsrates Mode II, GIIC*, Normenstelle Luftfahrt (NL) im DIN Deutsches Institut für Normung e. V., Köln, 1996.
- [5] *LS-Opt User's Manual Version 4.2*, Livermore Software Technology Corporation, Livermore, CA, 2011.
- [6] L. Ingber, *Very fast simulated re-annealing*, *Mathematical Computer Modeling*, 12, 1989, pp. 967-983.
- [7] Ansys Help Release 14, *Mechanical APDL: Theory Reference*, SAS IP Inc., Canonsburg, 2011.
- [8] M. Shahverdi et al., *Modeling effects of asymmetry and fiber bridging on Mode I fracture behavior of bonded pultruded composite joints*, *Engineering Fracture Mechanics* 99, Elsevier, 2013, pp. 335-348.

## Acknowledgement

The authors would like to thank the Austrian Research Promotion Agency (FFG), the Federal Ministry for Transport, Innovation and Technology (BMVIT) and Austrian Institute of Technology (AIT) for the financial support. The authors would also like to thank the project partners FACC, Fronius, Rübiger, Fill, RECENDT, LWPK MUL and EADS for the financial and technological support for the project CoJEC in the framework of TAKE OFF.