Modeling of Flexible Adhesive Joints in Automotive Crash Simulations: Calibration and Application of Cohesive Elements

Olaf Hesebeck[1], Matthias Nossek[2], Heinrich Werner[3],
Markus Brede[1], Oliver Klapp[1] and Martin Sauer[2]

[1]Fraunhofer-Institut für Fertigungstechnik und Angewandte Materialforschung (IFAM),
Wiener Str. 12, 28359 Bremen, Germany, email: oh@ifam.fraunhofer.de

[2] Fraunhofer-Institut für Kurzzeitdynamik, Ernst-Mach-Institut (EMI),
Eckerstr. 4, 79104 Freiburg, Germany

[3]BMW AG, Knorrstr. 147, 80788 München, Germany

Abstract: The simulation of crashworthiness plays an important role in the design process of today's automotive industry. Recently, efforts have been made to consider the effects of adhesive joints in the simulations. Flexible adhesive bonds typically have a thickness of one to five millimeters. Therefore, a detailed three-dimensional representation of the adhesive layer is infeasible in industrial crash simulations. For quasi-static loading conditions the submodeling technique is one possible solution to this problem. But if the influence of the fracture process on the deformation behavior of the structure is of interest like in crash simulations, then submodeling is insufficient.

Cohesive zone models describe the adhesive layer in a much simpler way than detailed models. As they use fracture energy as failure criterion, they show less mesh sensitivity than standard stress or strain criteria and thus allow larger finite elements. Compared to classical fracture mechanical approaches they offer the advantage that they need no initial crack and no sophisticated evaluation of singularities. Although the cohesive zone models have been developed to describe very thin layers, they can be successfully applied to flexible adhesive joints with an adhesive thickness of a few millimeters. This will be shown in this paper, with the main focus on parameter identification and model validation.

First a brief introduction into cohesive zone models and cohesive elements in ABAQUS is given. Next we will discuss the parameter identification process, which has to consider the joint geometry, materials, strain rates and mesh sizes. Keeping in mind industrial application, the parameters have been used to predict the crash behavior of components of increasing complexity.

Keywords: Adhesive Joints, Cohesive Elements, Crashworthiness, Damage, Experimental Validation, Parameter Identification

1. Introduction

1.1 Cohesive zone models

Cohesive zone models (CZM) have been developed to describe the deformation and fracture behavior in a continuum mechanics framework. It is assumed that the damage process, e.g. the formation of microcracks to a macroscopic crack, is confined to a very thin interlayer, e.g. the resin rich interlayers in laminated composites or a thin adhesive bondline. Since the thickness is very small compared to other relevant dimensions of the structure, the state of the cohesive zone is described by the displacement jump between the two adherents without considering details of the deformation field in the interlayer. A CZM describes all micromechanical processes influencing the crack opening by an overall description of the stress and separation field around the crack tip. Plasticity and/or damage are modelled by internal variables which only depend on the position within the interface, the finite thickness of the interlayer is neglected. Furthermore, the in-plane components of the stress and strain tensors are considered of minor importance to the global behavior. For example, if the interface is in the xy-plane, only the following tensor components are taken into account:

$$\boldsymbol{\sigma} = \begin{pmatrix} 0 & 0 & \tau_{xz} \\ 0 & 0 & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_{zz} \end{pmatrix} \quad , \qquad \varepsilon = \begin{pmatrix} 0 & 0 & \frac{1}{2}\gamma_{xz} \\ 0 & 0 & \frac{1}{2}\gamma_{yz} \\ \frac{1}{2}\gamma_{xz} & \frac{1}{2}\gamma_{yz} & \varepsilon_{zz} \end{pmatrix}$$

This means that the continuum mechanics model of the three-dimensional, thin interlayer is simplified to a two-dimensional interface model. The constitutive response of the interface is defined by the relation between the interface tractions and the displacement jump across the interface.

Historically, the concept of CZM was first introduced by Barenblatt (1959, 1962). It has been used to simulate fracture processes in a number of different material systems including metals (Siegmund, 2000), ceramic materials (Camancho, 1996), bi-material systems in polymer matrix composites (Xu, 1994), metal matrix composites (Foulk, 2000) and fibre reinforced plastic composites (Espinosa, 2000; Rhagavan, 2005). The CZM approach is related to three-dimensional damage mechanics models (Klarbring, 1991; Edlund, 1994) as well as to fracture mechanics (Point 1996, 1998; Alfano, 2001; Feraren, 2004; Mazars, 1996).

1.2 Cohesive elements

Since version 6.5 (see manual version 6.6 chapter 26.5) ABAQUS provides cohesive elements for 2D and 3D simulations. For spatial analysis the element type COH3D8 can be used, which is an eight-node brick element with one stack direction corresponding to the thickness direction of the interface. The properties are defined in a special section definition:

```
*COHESIVE SECTION, RESPONSE=TRACTION SEPARATION
```

As the “response” parameter of the section definition indicates, the traction-separation definition of a CZM is not the only kind of constitutive response. The cohesive sections can also be assigned a gasket model or a continuum-based constitutive response. Similar to a CZM, in the continuum approach for the cohesive element the in-plane membrane strains and the in-plane shear strain are neglected. However, the stress tensor is not reduced but calculated based on some usual material model. This implicates that any lateral contraction in the adhesive layer is assumed to be zero, which is correct for an infinite layer between rigid adherents, but differs from more precise models at the borders of comparatively thick adhesive layers.

Both traction-separation and continuum-based response of the cohesive elements are capable of producing similar behavior of the simulated joint, if the constitutive responses are chosen accordingly. For the work reported in this paper a cohesive zone model, i.e. the traction-separation response was favored because of the following reasons:

The definition of the cohesive zone model by a traction-separation relation is relatively simple and clear. The predefined CZM in ABAQUS is based on a separate description of the crack opening modes and their interaction and thus can easily be related to experiments dominated by peel and shear loads for parameter identification. It allows a flexible choice of different failure criterions and a straightforward extension by user subroutines (VUMAT).

Any model discretized by cohesive elements, especially with element sizes suitable for application in automotive simulations, is unable to describe the stress concentrations and localized damage processes at the edges of the adhesive layer. These stress concentrations occur on a scale smaller than the adhesive thickness, while the entire adhesive thickness is discretized by only one cohesive element. Therefore, a validated material model of an adhesive used with cohesive elements and the continuum-based approach does not necessarily lead to good results. The constitutive response of the cohesive elements has to describe the failure behavior of the adhesive joint despite the coarse mesh, so that in general it is different from an adhesive material model for a detailed simulation using a very fine mesh. If cohesive elements with the traction-separation response are used, it is quite obvious that this is a simplified model using a coarse mesh, but it should be noted that this is also the case for the continuum-based approach.

It should be noted here that the advantages and drawbacks of the continuum approach compared to a traction separation formulation for the description of adhesive joints are currently being investigated (FOSTA, 2007).

1.3 Modeling flexible adhesive joints

Flexible adhesive joints in cars typically have thicknesses between one and five millimeters and use hyperelastic adhesives with an elastic modulus in the order of magnitude of a few MPa. They provide significantly less stiffness but can sustain higher strains than so-called structural adhesive joints which use adhesives with an elastic modulus of about a GPa and a layer thickness less than half a millimeter. Flexible adhesive joints are often used to compensate for a mismatch in thermal expansion between dissimilar materials or to provide damping for noise reduction.

If an adhesive layer of a few millimeters thickness and ten to thirty millimeters width connects two metal sheets with a thickness of about one millimeter, the adhesive layer is not very thin compared to all other relevant dimensions in the structure. This means that the basic assumption of cohesive zone models and cohesive elements is not fulfilled, so that it is questionable if a cohesive element simulation of a flexible joint can yield accurate results. On the other hand, the use of cohesive elements gives the chance to include adhesive joints in a crash simulation at a reasonable computational cost. Therefore, BMW decided to investigate methods to simulate flexible adhesive joints by cohesive elements.

In the next section the cohesive zone model used to describe the flexible adhesive joints will be described briefly. Section 3 will explain the parameter identification process. Some difficulties encountered in the modeling with cohesive elements will be discussed in section 4. The experimental verification will be presented in section 5. Finally, the results will be summarized and an outlook on the application of cohesive elements in crash simulations at BMW will be given.

2. Cohesive zone model

The theoretical background for the CZM used here can be found in the literature (eg. Camanho, 2002) and is also explained in the ABAQUS documentation. Therefore, no formula will be given in the following short description of the CZM we used. A bi-linear traction-separation relation was chosen to describe the adhesive layer behavior in pure mode I and mode II loadings, for mode I, it is illustrated in Figure 1. Here, τ_I denotes the interface traction, τ_{Iu} the ultimate traction where damage starts, G_{Ic} is the fracture energy, δ_m^0 and δ_m^f are the effective displacements at damage initiation and at complete failure, respectively.

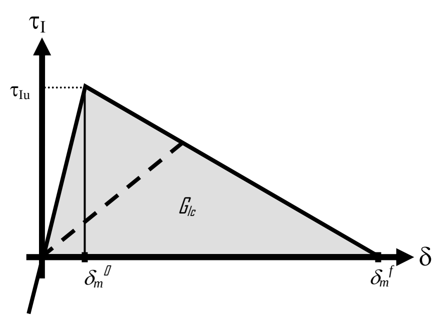


Figure 1. Bi-linear traction-separation relation in mode I.

The mode interaction in damage initiation is defined by a quadratic stress criterion, while the interaction in damage evolution follows the model of Benzeggagh-Kenane (Benzeggagh 1996).

The model is realized in ABAQUS by the following input lines:

```
*COHESIVE SECTION, ELSET=name, MATERIAL=adhesive, RESPONSE=TRACTION
SEPARATION, THICKNESS=GEOMETRY
*MATERIAL, NAME=adhesive
*ELASTIC, TYPE=TRACTION
modeI_stiffness,  modeII_stiffness,_modeII_stiffness
*DAMAGE INITIATION, CRITERION=QUADS
```

```
modeI_ult_tract,  modeII_ult__tract,  modeII_ult_tract
*DAMAGE EVOLUTION, TYPE=ENERGY, SOFTENING=LINEAR, MODE=ENERGY, MIXED MODE
BEHAVIOR=BK, POWER=eta
modeI_fract_energ, modeII_fract_energ, modeII_fract_energ
*DENSITY
density
```

A typical traction limit surface on the plane of mode I and mode II displacements is displayed in figure 2 to illustrate the mode interaction criteria.

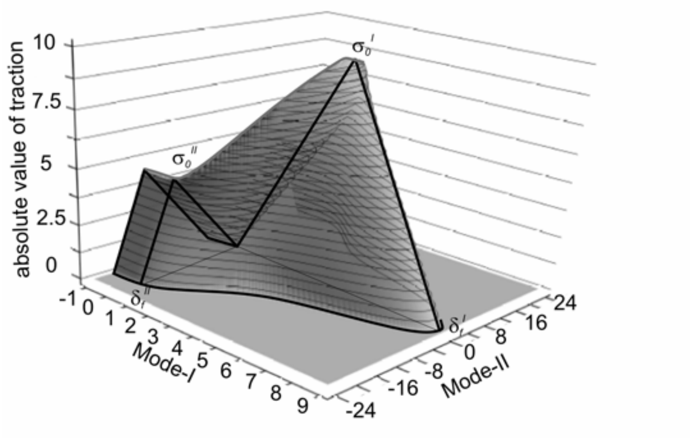


Figure 2. Traction limit surface for combined quadratic/B-K-interaction criteria.

3. Parameter identification

The set of model parameters consists of values indicating the stiffness, strength and energy dissipation of the adhesive layer in both mode I and mode II, and a parameter which describes the mode interaction in the damage evolution law (eta in the input file fragment shown above). Only the latter has been obtained from literature, the other parameters were directly measured in experiments (mode II stiffness and fracture energy) or obtained by fitting simulations to measured force-displacement curves (mode I stiffness, ultimate tractions).

3.1 Stiffness parameters

To characterize the selected adhesive, uniaxial tensile tests have been performed on bulk specimens at -30 °C, 23 °C and 80 °C. The strain rate of the quasi-static tests was in the order of magnitude of 0.01 1/s. Additional tests at room temperature with strain rates up to 100 1/s have been used to investigate the rate dependence. The stress-strain curves showed a typical shape for hyperelastic materials. A first order Ogden model (see e.g. ABAQUS documentation) can nicely be fitted to match these curves. For the investigated adhesive, the different test conditions result in different stiffness parameters μ but can be described by a single exponent parameter α of the Ogden model. Of course, the chosen linear CZM can not describe the nonlinear elastic behavior of the adhesive. As it was considered most important for the overall structural response to get a good approximation of the initial stiffness of the adhesive joint, the stiffness parameter of the cohesive zone model for mode II was chosen equal to the shear modulus μ of the Ogden model. Therefore, for large shear strains, some deviation of the model compared to experiments must be expected.

The mode I stiffness parameter can not be transferred as easily from the adhesive material parameter μ, because it has to consider the restriction of lateral contraction by the adherents which depends on the geometry of the adhesive joint. While the lateral contraction is nearly completely restricted for structural (thin) adhesive joints, this is not the case for thick adhesive layers. We have calibrated the parameter to calculate the correct secant stiffness at a moderate deformation (0.5 mm) for a T-peel specimen. The question whether this choice is optimal for the simulation of automotive structures will be a subject of future investigation.

The necessary reference stiffness for the calibration can either be obtained by experiment or by detailed simulations using the hyperelastic material law identified in the bulk specimen investigations. Figure 3 shows the finite element meshes used in the calibration by simulation. On the left side there is the model for the detailed analysis using the hyperelastic adhesive model. The adherents are represented by shell elements, a simplification which has been checked by prior simulations with continuum elements. The right side of the figure displays the model using cohesive elements with an element size of 2 mm and an adhesive layer thickness of 5 mm (as the CZM does not consider lateral strains, it is sufficient to reduce the model to one row of elements). The mode I stiffness parameter of the CZM has been adjusted so that both models have equal stiffness. Since no simulation of damage processes is required for this calibration, it can be performed within a few minutes.

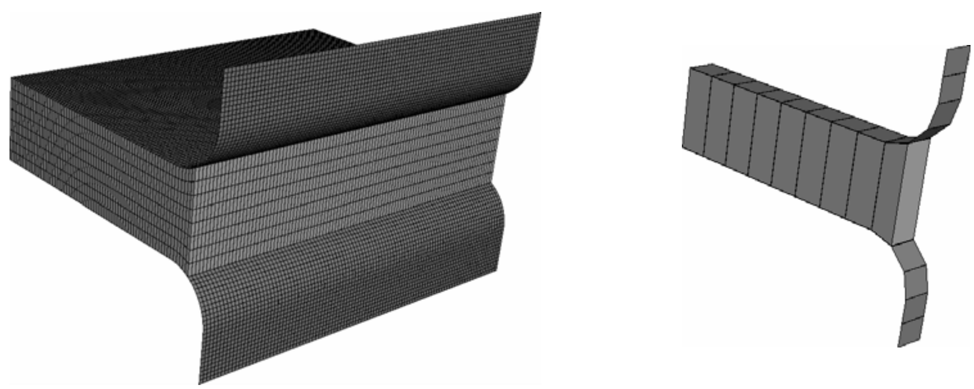


Figure 3. Detailed and cohesive element model for mode I stiffness calibration.

3.2 Energy and strength parameters mode I

The fracture energy parameter G_{Ic} and G_{IIc} of the CZM describes the energy per unit area dissipated in the adhesive layer by plasticity and fracture processes. For mode I it was identified by a special kind of double cantilever beam test which will be described elsewhere. Alternatively, it is possible to obtain the parameter by fitting the simulations to T-peel tests as explained below for the strength parameter τ_{Iu}.

The strength parameter was calibrated using simulations of a T-peel specimen (Figure 3). The parameter was tuned so that the calculated maximum load agrees with the average experimental value, see for example Figure 4 (left). As expected, initial stiffness and maximum force of the calibrated model match the experiment, the shape of the curve differs. This is due to the nonlinear behavior of the adhesive at large strains and to the effects of restricted lateral contraction, which are not covered by the linear model. As mentioned above, one has to choose either to match the initial stiffness or the displacement at failure initiation.. Nevertheless, the model can be sufficient to describe the main features of adhesive joint failure for the simulation of bonded structures.

As a first test of the model and the identified mode I parameters, another kind of T-peel test was performed and simulated. The specimen was only half as wide as the specimen in Figure 3 but five times as long, so that a steady state peeling at a constant load could evolve. Figure 4 (right) shows that the simulation gave an excellent prediction of the peel force and good values for maximum load, deformation at failure and energy dissipation. This experiment could alternatively be used to calibrate both energy and strength parameters.

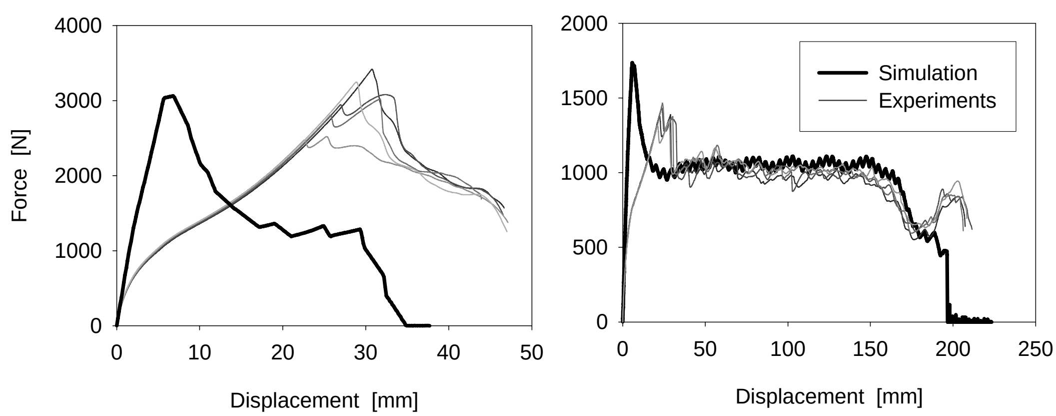


Figure 4. Simulation and measurements of short (left) and long (right) T-peel test.

3.3 Energy and strength parameters mode II

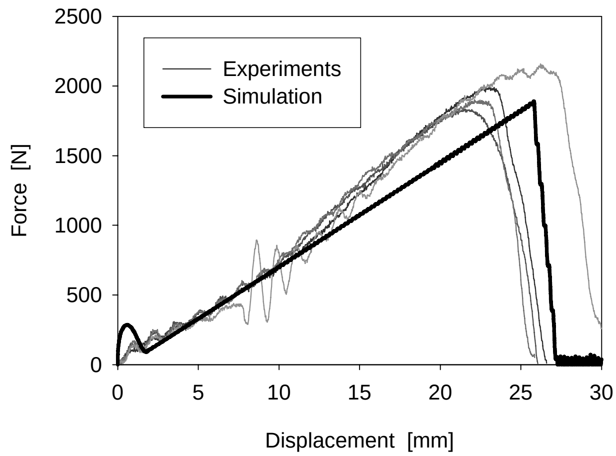


Figure 5. Simulation and measurements of lap shear tests.

While much research has been done on parameter identification for mode I loadings, mode II, up to now, has received less attention and poses more experimental difficulties. For example, the T-peel specimen is a simple joint of metal sheets which enables us to observe a stable, mode I dominated crack growth, but for shear dominated loads no comparable test exists yet. Currently the strength and energy parameters under mode II are identified by lap shear tests. Figure 5 shows force-displacement curves of lap-shear tests at a loading velocity of 5 m/s and the corresponding simulation. Alternative methods of mode II parameter identification are currently being investigated.

4. Challenges and limitations

Three of the difficulties encountered during parameter identification und simulation of flexible adhesive joints shall be reported in this section.

In automotive production, the process of bonding often takes place after varnishing of the adherents. This implies that not only the adhesive but also the varnish can be the weak link in the joint. While a failure of the varnish did not occur at room temperature, it was observed in some tests at other conditions. The varnish failure usually happened very sudden compared to the more stable crack growth of cohesive failure, and it could not be described well by the model.

Care must be taken that the model is not supplied with contradicting parameters. If we consider pure mode I, the condition that the damage initiation occurs before complete failure is

$$G_{Ic} > \frac{t_a\, \tau_{Iu}^2}{2\, E_I},$$

where t_a denotes the adhesive layer thickness, E_I the mode I stiffness parameter, τ_{Iu} the strength parameter and G_{Ic} the energy parameter. A similar condition exists for mode II. Since the parameters are identified using more than one experiment, it is possible that the inequality is violated and the parameters have to be adjusted.

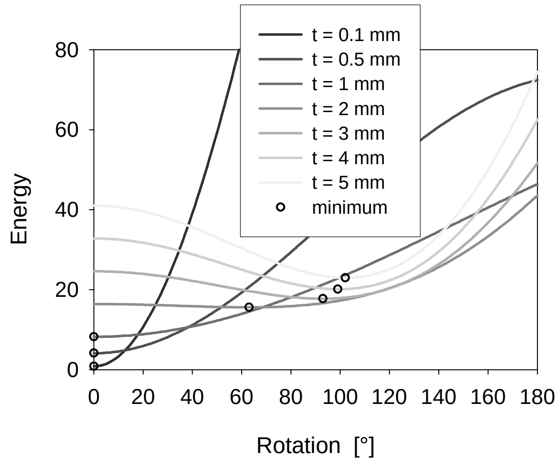
While the conditions for pure mode I and mode II can easily be checked, the condition for mixed mode is less simple and depends on the interaction criterion and interaction parameter. Using a quadratic stress criterion, we obtain an elliptical damage initiation curve in the plane of normal and tangential separations. The failure curve defined by the B-K-model or by an exponential criterion has a different shape, however. Thus it is possible to trigger the failure criterion before the damage initiation criterion even in some cases when there is no conflict in pure mode I and mode II. Currently, there is no consistency check implemented in ABAQUS.

A third problem encountered during the simulations originated from the strain evaluation in the cohesive element. This evaluation is designed for small strains, while the simulated flexible joints reach large strains before failure. The resulting unusual element behavior has been investigated to learn more about the limitations of the element. For illustration we shall consider a two-dimensional deformation of a single cohesive element without damage. First a strain of 100 % in thickness direction (mode I loading) is applied, then one of the adherents rotates by a certain angle about its center. Figure 6 (left) displays the resulting strain energy depending on the angle of rotation. Obviously, the energy should be minimal if no rotation is added to the strain, i.e. at an angle of 0°. But this requirement is violated for certain element geometries. At an element width of 2 mm, different element thicknesses between 0.1 and 5 mm have been checked. While thin elements like in structural adhesive joints showed a proper behavior, thick elements may cause trouble.

In addition to one-element-tests, this phenomenon has been studied analytically, starting from the basic equations of the element. It turns out that in order to avoid the inappropriate element behavior, the aspect ratio has to fulfill the inequality

$$\frac{w}{t_a} > \sqrt{(1+\varepsilon)\left(\varepsilon - (1+\varepsilon)\frac{E_{II}}{E_I}\right)},$$

where w/t_a is the ratio of element width to thickness, ε denotes the strain in thickness direction and E_{II}/E_I the ratio of mode II and I stiffness parameters. Figure 6 (right) displays this condition for two stiffness parameter ratios. It is violated for typical adhesive layer thicknesses and failure strains of flexible adhesive joints. Nevertheless, in most simulations this caused no problems, because either the much stiffer adherents restricted the cohesive element deformation or the mode II failure criterion was triggered before unreasonable deformation states could occur. However, the restriction has to be kept in mind when analyzing results with large mode I displacements.



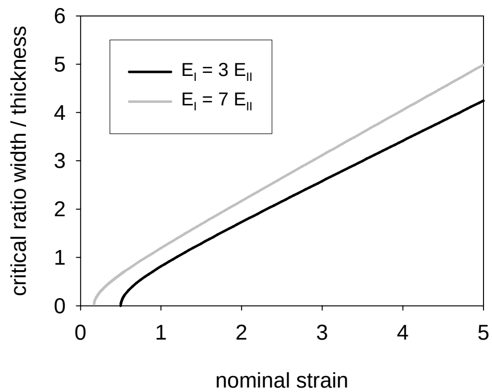


Figure 6. Test of cohesive element for large strains.

5. Experimental Validation

On the path from model formulation and parameter identification by tests of simple specimens to the industrial application in the crash simulation of an entire car, intermediate validation steps of increasing complexity have to be taken. Therefore, the specimen displayed in figure 7 has been subjected to impact tests, and has been simulated using cohesive elements. It consists of a steel sheet which is bonded on a u-shaped steel frame using a flexible adhesive joint. The steel sheet was loaded by a hemispheric impactor at three different impact velocities (quasistatic, 500mm/s, 3m/s) and three different temperatures (30°C, RT, 80°C).

The movement of the impactor causes large deformations and local buckling of the steel sheet. The adhesive layer sustains large deformations before a crack starts to grow from the location of the buckling (figure 8 left) along the sides of the specimen. The crack opening is observed by two high speed cameras. Furthermore, the force and impactor displacement are measured. The force measurement is performed by a piezoelectric load cell at the upper end of the impactor.

For the numerical simulation a model with an element size of about 4 mm has been created (figure 7 right). The u-shaped steel frame is modeled by solid elements (C3D8R) with linear elastic behavior, while its support is represented by shells (S4R). The behavior of the steel sheet, discretized by S4R elements as well, is described by an elastoplastic, strain rate and temperature dependent model. The dependence of the adhesive properties on temperature and strain rate is considered by using sets of parameters identified at different conditions. For model validation, deformation, crack growth and force-displacement curves were compared to the experiments. In addition, the mixed mode ratio and the sensitivity of the results to mesh refinement were analysed.

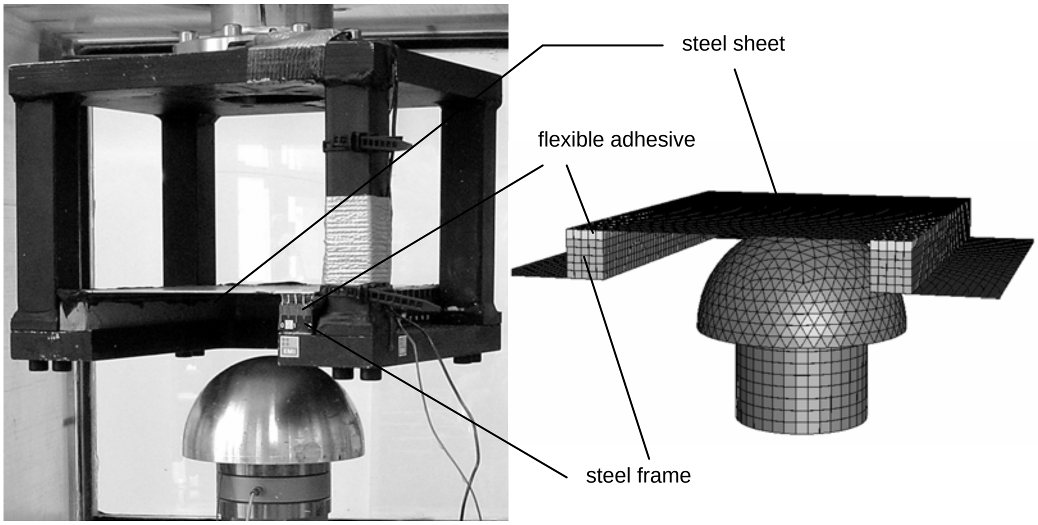


Figure 7. U-shaped Specimen: experimen (left) and model (right).

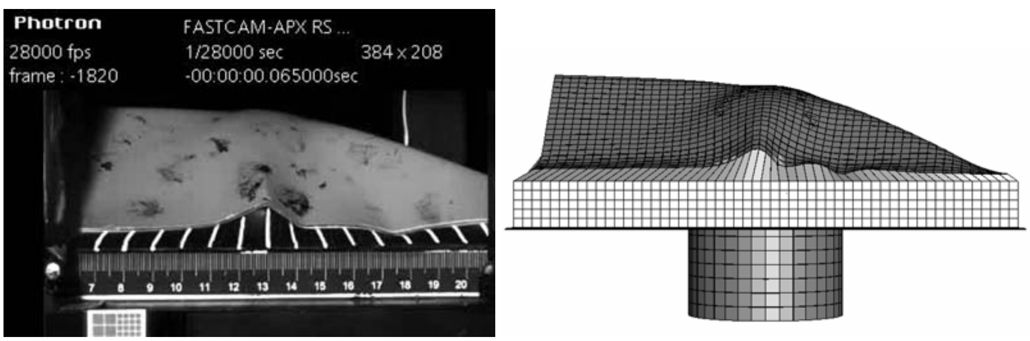


Figure 8. Deformation of the u-shaped specimen at crack initiation

Figure 8 illustrates that the location of damage initiation in the adhesive is predicted well by the simulation. An analyses of the crack growth velocities showed that they decrease as the crack advances in both measurement and simulation. A precise quantitative evaluation of the crack propagation is difficult, but, within the accuracy possible in the measurement, the simulation results could be confirmed.

The force-displacement curves for an impact velocity of 3 m/s at all three test temperatures are displayed in figure 9. At -30 °C an excellent agreement between experiment and simulation concerning stiffness, maximum force and plateau force at larger displacements has been achieved. Maximum force and force decrease have been predicted well for the room temperature tests. At 80 °C, however, the simulation differs significantly from the experiments. An obvious reason for this deviation is the failure of the varnish, which occurred in both the experiments for parameter identification and in the tests of the u-shaped specimen, but in a different proportion to cohesive failure.

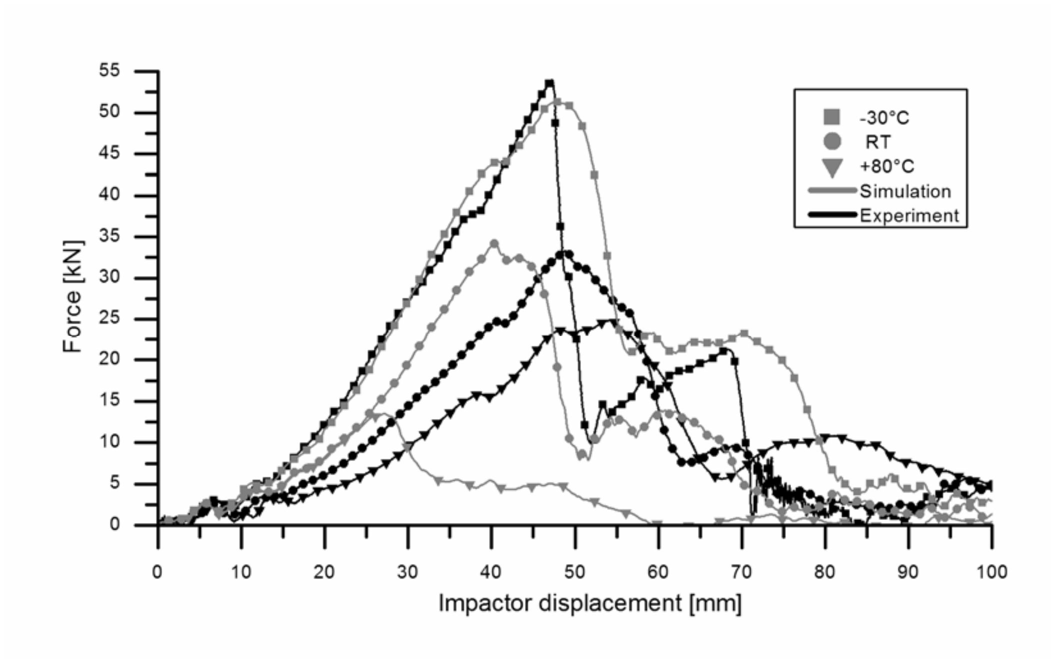


Figure 9. U-shaped specimen: Force displacement curves of experiments (black) and simulations (gray).

An evaluation of the mixed mode ratio in the cohesive elements showed that the loading of the major part of the adhesive layer is dominated by mode II. Only at the adhesive layer edges closest to the impactor the mode I contribution is predominant.

Different discretizations with an element size in a range between 1.5 and 4 mm have been tested. The global response of the specimen was quite independent of the mesh in the investigated range. In general, we observed little mesh dependence in simulations with cohesive elements as long as the elements are smaller than the size of the damage process zone.

6. Outlook on future development steps

Applying the cohesive element model and parameter identification strategies to industrially relevant test cases is the next logical step. Due to their importance for crash behaviour this will be done first for structural adhesives. The methodology described above for flexible adhesive joints has been used in a similar way to calibrate a model for a structural adhesive whose stress strain relationship is characterized by an elasto-plastic behavior ($E \approx 1000 \dots 3000$ MPa). The simulation model shown in figure 9 is based on a concept study which uses adhesive bonding extensively in the body in white. From the right part of figure 10 an impression of the bond lines (shown in dark gray) may be received.

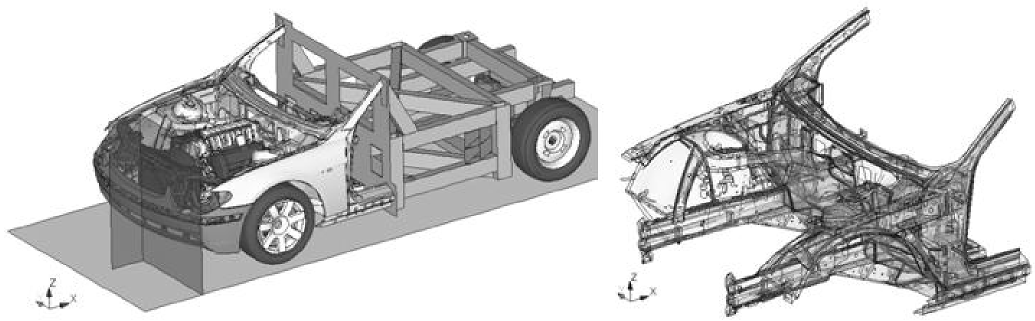


Figure 10. Car crash simulation model and details of the bond lines.

With respect to the cohesive element model the main objectives of the study are:

- identification of critically loaded areas within the bond lines including potential crack growth.
- sensitivity of the numerical result with respect to the descretization of the adhesive as well as the adhering metal flanges. This investigation will include overmeshing of the cohesive elements as well (edge length of the cohesive element is smaller than edge length of adhering shell elements). Since a fine discretization might impose too severe restrictions with respect to the stable time increment, the influence of mass scaling has to be checked.
- sensitivity of the numerical result with respect to systematic variations of the cohesive element parameters. The current implementation of the cohesive element offers no option to include strain rate effects in case RESPONSE=TRACTION SEPARATION is defined. Therefore, a compromise in the cohesive element parameters has to be found. Alternatively a VUMAT with rate dependent damage initiation may be used.
- determination of the best strategy to define the thickness of the adhesive layer in an industrially relevant model. The option to define the cohesive element thickness within the material data input is particularly attractive in large scale simulation models. The inevitable geometrical imperfections in the distances of bonded flanges in these models would otherwise require costly mesh generation techniques to ensure the desired bond layer thickness which is typically in the range of 0.2 to 0.5 mm for structural adhesives. The sensitivity of the results with respect to deliberately altered bond layer thickness and width will be investigated.

Based on the experience gained in this concept study for structural adhesive joints, similar investigations have to be performed for flexible adhesive joints to prepare the method for industrial application.

7. Conclusions

An approach to simulate flexible adhesive joints in automotive structures using a cohesive zone model with a bi-linear traction-separation relation has been developed. Flexible adhesive joints typically have an adhesive layer thickness of a few millimeters, which is not negligible compared to other relevant dimensions of the joint and the adherents. Therefore, a basic assumption of the cohesive zone models is not fulfilled. Nevertheless, the representation of adhesive layers by cohesive elements is of high practical interest for the automotive industry, because it promises the chance to consider adhesive joints in crash simulations at a reasonable numerical cost.

The parameter identification was based on simple experiments like tensile, T-peel, lap shear and DCB tests. For the mode I stiffness parameter the partially restricted lateral contraction in the adhesive layer must be considered. The mode interaction parameter was not measured but taken from literature.

During the parameter identification and simulations of other joints some difficulties and limitations have been encountered. Care must be taken that the choice of parameters and mode interaction criteria does not imply a contradiction in mixed mode by predicting failure to occur before damage initiation. At large deformations the cohesive elements which have been developed to model thin layers may produce an unreasonable response. Generally, the prediction of joint failure is very difficult if the proportions of cohesive, adhesive and adherent failure change.

Despite of these limitations, the model was successfully applied to simulate impact tests of a steel sheet bonded to a u-shaped profile. At three different loading velocities and two different temperatures the simulations agreed qualitatively and quantitatively well with the experimental observations. Only at a high temperature a failure of the varnish instead of the adhesive occurred and could not be predicted by the model.

The next step in the verification of the approach will be an application to an industrially relevant test case.

8. References

1. Alfano, G., and Crisfield, M.A., “Finite Element Interface Models for the Delamination Analysis of Laminated Composites: Mechanical and Computational Issues,” Int. J. Numer. Meth. Engng., vol. 50, pp. 1701, 2001.
2. Barenblatt, G.I., “The Formulation of Equilibrium Cracks during Brittle Fracture. General Ideas and Hypothesis,” PMM, vol. 23, pp. 434-444, 1959.
3. Barenblatt, G.I., “Mathematical Theory of Equilibrium Cracks in Brittle Fracture,” Journal of Advances in Applied Mechanics, vol. VII, pp. 55-129, 1962.

4. Benzeggagh, M. L., and Kenane, M., ”Measurement of Mixed-Mode Delamination Fracture Toughness of Unidirectional Glass/Epoxy Composites With Mixed-Mode Bending Apparatus,” Composites Science and Technology vol. 56, pp. 439-49, 1996.
5. Camancho, G.T., and Ortiz, M., “Computational Modelling of Impact Damage in Brittle Materials,” Int. J. Solids Struct., vol. 33, pp. 2899-2938, 1996.
6. Camanho, P.P., and Dávila, C.G., “Mixed-Mode-Decohesion Elements for the Simulation of Delamination in Composite Materials”, NASA/TM-2002-211737, pp. 1-37, 2002.
7. Edlund, U., “Surface Adhesive Joint Description with Coupled Elastic Plastic Damage Behaviour and Numerical Applications,” Comput. Meth. Appl. Mech. Engng., vol. 115, pp. 253-76, 1994.
8. Espinosa, H.D., Dwivedi, S. , and Lu, H.C., “Modeling Impact Induced Delamination of Woven Fiber Reinforced Composites with Contact/Cohesive Laws,” Comput. Methods Appl. Mech. Eng., vol. 183, pp. 259-290, 2000.
9. Feraren, P., and Jensen, H.M., “Cohesive Zone Modelling of Interface Fracture near Flaws in Adhesive Joints,” Engng. Frac. Mech., vol. 71, pp. 2125-2142, 2004.
10. FOSTA project P676, “Methodenentwicklung zur Berechnung von höherfesten Stahlklebverbindungen des Fahrzeugbaus unter Crashbelastung,“ current research project sponsored by Forschungsvereinigung Stahlanwendung e. V., 2007.
11. Foulk, J.W., Allen, D.H., and Helms, K.L.E., “Formulation of a Three-Dimensional Cohesive Zone Model for Application to a Finite Element Algorithm,” Comput. Methods Appl. Mech. Eng., vol. 183, pp. 51-66, 2000.
12. Klarbring, A., “Derivation of a Model of Adhesively Bonded Joints by the Asymptotic Expansion Method,” Int. J. Engng. Sci., vol. 29, no. 4, pp. 493, 1991.
13. Mazars, J., and Pijaudier-Cabot, G., “From Damage to Fracture Mechanics and Conversely: a Combined Approach,” Int. J. Solids Structures, vol. 33, no. 20-22, pp. 3327, 1996.
14. Point, N., and Sacco, E., “A Delamination Model for Laminated Composites,” Int. J. Solids Structures, vol. 33, no. 4, pp. 483, 1996.
15. Point, N., and Sacco, E., “Mathematical Properties of a Delamination Model. Math. Comput. Modelling 28, 4-8, 359 (1998)
16. Rhagavan, P., and Ghosh, S., “A Continuum Damage Mechanics Model for Unidirectional Composites Undergoing Interfacial Debonding,” Mechanics of Materials, vol. 37, pp. 955-979, 2005.
17. Siegmund, T., and Brocks, W., “A Mumerical Study on the Correlation between the Work of Separation and the Dissipation Rate in Ductile Fracture,” Eng. Fract. Mech., vol. 67, pp. 139-154, 2000.
18. Xu, X.P., and Needleman, A., “Numerical Simulation of Fast Crack Growth in Brittle Solids,” J. Mech. Phys. Solids, vol. 42, pp. 1397-1434, 1994.