Modeling of Punctual Joints for Carbon Fiber Reinforced Plastics (CFRP) with *MAT_054

Christian Liebold[1], David Moncayo[2]

[1]DYNAmore GmbH, Stuttgart, Germany
[2]Daimler AG, Sindelfingen, Germany

Abstract

The increasing amount of carbon fiber reinforced plastic (CFRP) components used in the automotive industry opens new questions concerning the numerical modeling of different joining techniques for such materials. Design engineers already know that the beneficial properties of carbon fibers can be best used when considering the necessary joining techniques from the very first step in the preliminary design of a CFRP part. The anisotropical behavior of composites has to be fully considered in numerical simulations as well, especially when failure and partial damage occurs. Using experimental data from bearing tests loaded in different directions, different modeling techniques for structural joints are tested. The achieved correlation between simulation and experimental results will be discussed for a discretization relevant for industrial applications in a vehicle crash environment. Besides a qualitative and quantitative evaluation, the proposed modeling techniques are evaluated in terms of an appropriate representation of failure compared to the failure patterns observed during the experiments.

Keywords: composite failure modeling, damage tolerance, fracture, joining techniques, bearing test, structural joints

1 Overview on composites in automotive applications

1.1 Material characterization process

The prognosis capability of full car simulations in digital prototypes relies on a systematic approach for the material characterization of the used materials and a correspondent validation of finite element (FE)- models at different levels of a building block approach as discussed by [1]. In order to reach a good prognosis capability through the development of damage tolerant composite parts for automotive applications, it is crucial to discuss an efficient balance between accuracy and efficiency in the full car simulations.

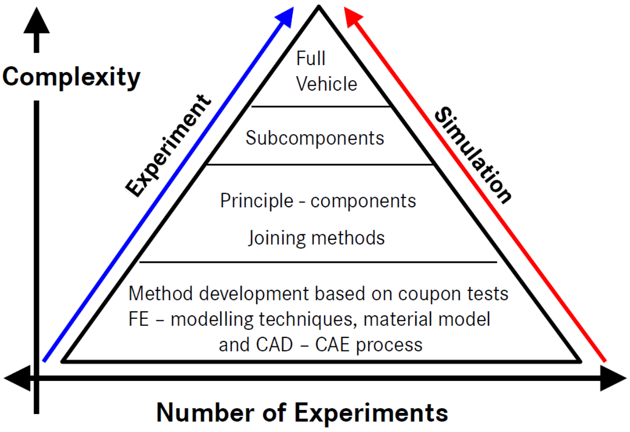


Fig. 1: Building block approach for fiber reinforced plastic (FRP) material characterization in the automotive sector

Figure 1 depicts a building block approach for the material characterization of composite material configurations. The possible wide range of material configurations and laminate definitions requires the validation of numerical models not only on a coupon test level, but also on principle components and subcomponents where the geometry and manufacturing issues can also be captured in the validation process.

1.2 Physical behavior of carbon composites laminates (non-crimp fabrics)

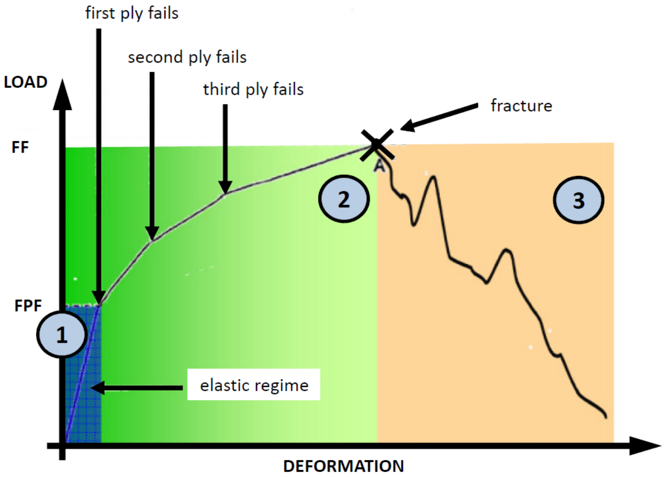


Fig. 2: General behavior of multidirectional laminates
1) elastic regime;
2) progressive damage;
3) catastrophic laminate failure

Figure 2 depicts the general behavior of moderate thick multidirectional laminates. The technical challenge for the crash analysis relies in the necessity to capture the material behavior after the first ply failure (FPF) in an appropriate manner. Especially, dealing with structural joints in composite parts, is a challenge for the simulating engineer and different approaches such as they are shown in this work have to be considered and analyzed. The main problem thereby is the loss of energy in the simulation after element failure which leads to a sudden drop of the already reached force level which will not be reached again during further analysis. This problem is also known for regular crash tube structures.

1.3 Numerical modeling of structural joints

Capturing the real behavior of structural joints in automotive applications is crucial for realistic load paths through the structure under crash loads. Currently, mesh sizes in FE- models consider an element length between 1 mm until 10 mm depending on the application and the correspondent modeling of fragmentation. The computing time has an influence on the efficiency through the product development. Therefore, FE- models for full car simulations prioritize shell elements for modeling shell like structures. Furthermore, figure 3 depicts common used modeling techniques for structural joints.

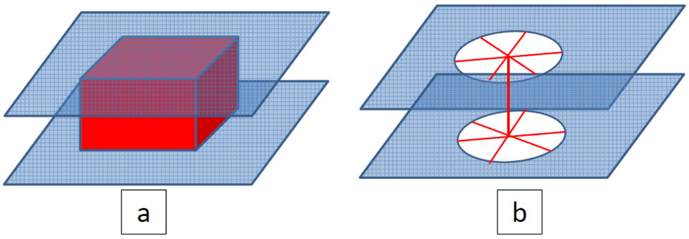


Fig. 3: Modeling of structural joints:
a) solid definition;
b) beam definition

The validity of the FE-Models depends on the mesh size and the material definition for the structural joint and joined partners. Furthermore, the load transmission, especially under bearing loads is represented with these kinds of models. The connection through a TIED_CONTACT definition (figure 3 a) or a beam definition (figure 3 b) distributes the load in the joined partners different than in reality. Nevertheless, several projects have shown that this modeling can be acceptable for full vehicle analysis. In terms of efficiency, it has to be discussed how these simplifications do influence the accuracy of the simulation results compared to the experimental data. An explicit consideration of the transmitted bearing loads can be neglected with coarse meshes such as they are used in full vehicle crash structures. When considering crushing loads, conventional metal parts (joined partners) would depict a plastic deformation in highly loaded regions. Composite parts instead show a high degree of local fragmentation and delamination which has to be captured during the simulations and therefore, it is required to take a closer look at the behavior of structural joints since these locally damaged regions can trigger further local fragmentation and delamination. In terms of damage tolerance, an appropriate modeling technique for structural joints is required and has to be validated to improve the interpretation capabilities of the crash simulations with complex assemblies.

2 Experiments

For this work, different experiments are performed in order to find the force-displacement behavior of bolted joints under a tensile load. In preliminary experiments the ideal position of a bolted joint has to be found in order to get a representative scatter of experimental results. Moderate thick multi-directional (MD) laminates as are used for a various number of structural parts are tested, having a thickness between 2 mm and 6 mm. In a second series of experiments, two different stacking sequences for the laminate and different loading directions (cut-out direction) relative to the 0° ply orientation of the laminates are considered in order to capture the influence of fiber orientation on the specimens. The experimental set-up for both series of experiments is shown in figure 4.

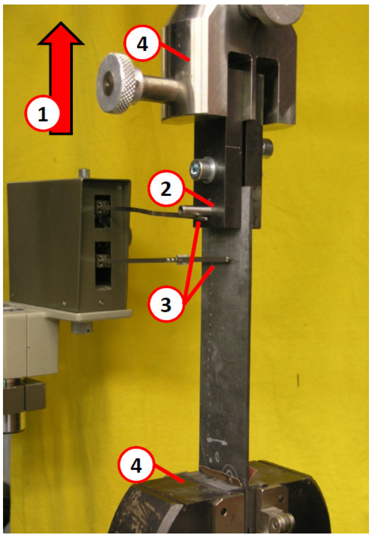
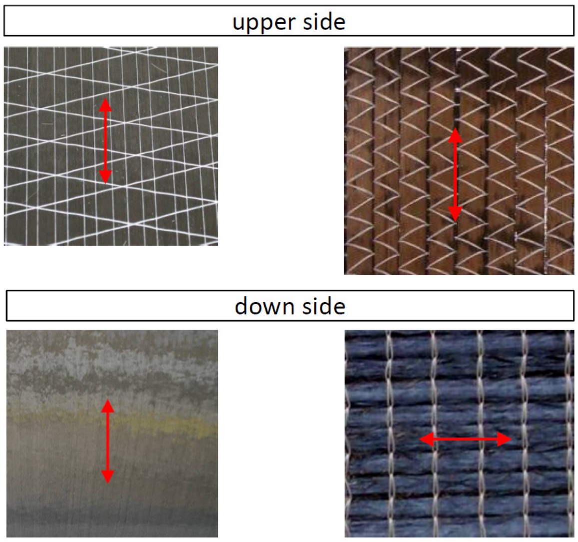


Fig. 4: Experimental set up:
1) loading direction;
2) bolt;
3) displacement measurement device
4) fixation

The used preform is made out of MD plies [0°/ 90°]. The so called “non–crimp” fabrics are manufactured by stitching two uni-directional (UD) fiber layers in a certain pattern in order to achieve a good draping behavior for complex geometries. The waviness of the fibers in cured laminates depends on the manufacturing process, geometry and stitching pattern. Furthermore, it has been observed that these effects are not always neglectable. In this study, fiber waviness will be no further discussed for simplicity reasons (figure 5).



a) b)

Fig. 5: Non-crimp fabric preforms
a) uniaxial;
b) biaxial

3 Modeling techniques

In order to find an adequate modeling method which represents both, the measured force-displacement curves as well as the physical behavior of the CFR laminate, different modeling techniques for punctual joints will be introduced and discussed. For this work, the main focus is on the modeling itself. Additional constraints or elements shall improve the simulation results. The goal is to find a simplified approach which can be used in automotive industries for full vehicle crash analysis. Prior the explanation of the different modeling techniques, the material model used for this work, *MAT_ENHANCED_COMPOSITE_DAMAGE (*MAT_054) shall be introduced shortly.

3.1 Material Model *MAT_ENHANCED_COMPOSITE_DAMAGE (*MAT_054)

This material model is a standard orthotropic material model which is often used for the modeling of composite materials. Figure 6 depicts the stress–strain relationship in this material routine. While *MAT_054 [7] in the current version of LS-DYNA® considers a strain rate dependency of the strength values, this implementation has no damage or degradation of the elastic properties beyond the maximum strength. Nevertheless, this material model allows the analyst an ad-hoc fit of material cards based on experimental results. It has to be mentioned that in order to reach a good compromise between experimentally measured material values for strain and strength and good predictive simulation on a component level, some parameter fitting is usually unavoidable and might be a tedious work.

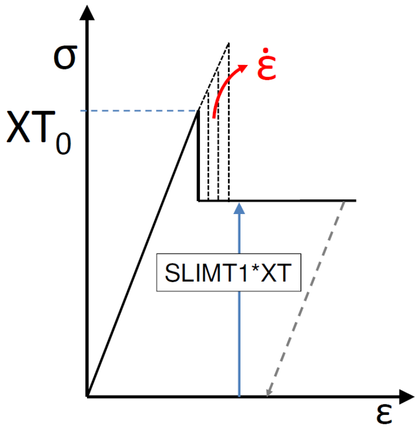


*Fig. 6: *MAT_054*

The Chang-Chang failure criterion [2] is used in this routine. In addition, this material uses a special criterion for compressive failure [3] and a damage model for transverse shear strain is also available to model interlaminar shear failure. Minimum stress limits can be defined to keep the element from deletion prior reaching a user-defined effective failure strain. For simplicity reasons, strain rate dependency will not be considered in the following load cases.

3.2 Laminate lay-up

The experiments presented here are performed on two different kinds of laminates with different stacking sequences and a different number of layers, respectively different thicknesses. Two layers are always stitched together to form a [0°,90°] preform such as it is shown in Figure 6.

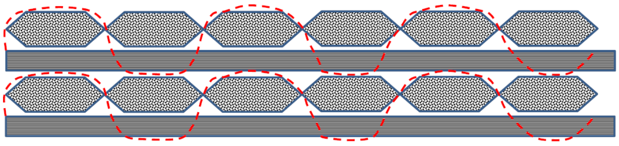


Fig. 7: Laminate lay-up with stitching lines

Furthermore, there are different modeling techniques to model laminates as presented in [4]. Figure 8 a) depicts a conventional abstraction, modeling one integration point through the thickness in each FE-element for each physical composite ply in the stacking sequence. A sublaminate definition (Figure 8 b)) considers a group of physical plies per integration point. The second option is especially relevant for industrial applications, since a better compromise between efficiency and accuracy can be reached by reducing the number of mathematical operations, model output size and smearing the heterogeneous ply properties through an appropriate material card; representative for a certain sublaminate definition. In this case, it is also necessary to adapt and validate the failure parameters in the used material model. The definition of the stacking sequence can be considered through a "ply based" modeling technique or a "zone based" modeling technique as described in [1] using the *ELEMENT_SHELL_COMPOSITE keyword syntax in [5] or the *PART_COMPOSITE keyword syntax in [5]. Therefore the use of biaxial non-crimp fabrics in the stacking sequence defines a sublaminate definition. For the [0°, 90°] ply group. It is favourable, then, considering carefully these two directions for a sublaminate (Figure 8 b)) in the material characterization scheme as explained in figure 1.

3.3 Usage of the material properties

In a first approach, it is more than conceivable to fully use the capabilities that *MAT_054 material model offers – and, as will be shown later on, a good agreement between simulation and experimental results can be reached when only considering the tensile, compressive and shear strengths of the UD layers. Therefore, high strains and large element deformations have to be allowed which cause a decrease of the time step size and therefore, longer simulation times. Of course, this is not acceptable for full vehicle analysis and therefore, criteria which lead to an elimination of the highly deformed elements have to be defined. These criteria can be effective failure strains such as they can be defined for *MAT_054. As known from the simulation of standard crash tubes, the deletion of elements leads to a loss of contact forces, respectively a loss of energy between the crash tube and the impactor. When looking at the simulation results of such experiments, one can see a steady rise of the force-displacement curve, followed by a sudden drop of the maximum force level. Since this cannot be obtained during experiments, these phenomena have to be considered by the simulation as well and therefore, different approaches as they are presented below are necessary.

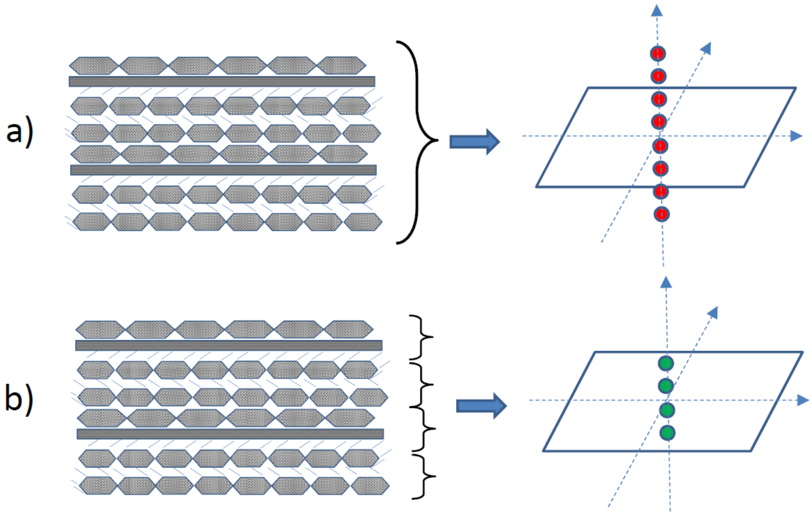


Fig. 8: Lay-up abstraction in FE-Models

3.4 Beam modeling technique

There are two ways to use beam elements in order to reinforce the material even though shell elements have already been deleted. Both of them are shown in figure 9. The idea behind these models is to keep the contact force between the bolt and the remaining elements at the level where the first shell elements are deleted [6]. Additionally, this method allows a coarse mesh with an acceptable quality which is crucial for the crash analysis in industrial applications. For the model shown on the left (figure 9 b)), the additional beam elements are attached to the shell nodes, perpendicular to the motion of the bolt. The beam elements therefore provide an additional support once the underlying shell elements are removed. Connecting the moving bolt directly with the specimen such as shown on the right (figure 9 c)) basically leads to the same results, even though this modeling technique already transforms stresses into the upcoming elements and depends on a pre-defined loading direction. Both methods require an additional effort during the model creation which might be acceptable for this kind of load cases.

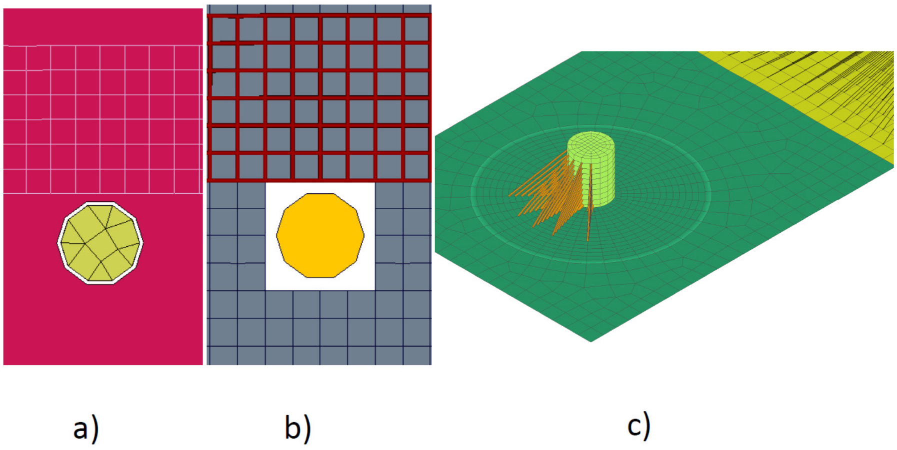


Fig. 9: a) Beam elements attached to the shell mesh
b) Mesh simplification + attached Beams
c) Beam connection between the bolt and the shell mesh.

3.5 Constrained modeling

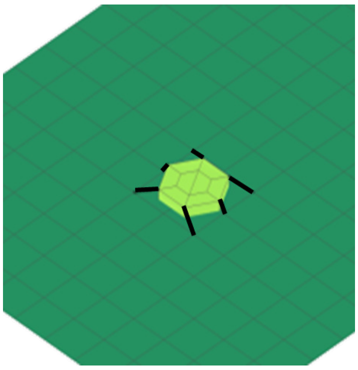


*Fig. 10: *Constrained-interpolation model*

Another option is to use constrained interpolations instead of beam elements to transfer reaction forces onto the elements right behind the front of failed elements. This method works quite well when using one *CONSTRAINED_INTERPOLATION – card per beam element from the method shown before. This of course does not simplify the modeling approach but additional parameter fitting which might be necessary to model the beam elements is avoided. The usage of only one fixed point on the bolt which connects to several points on the specimen is not recommended with the *CONSTRAINED_INTERPOLATION – card. The problem is, that once an element and its nodes are deleted, the full *CONSTRAINED_INTERPOLATION fails and therefore, the amount of time for which the force-level can be kept at the maximum of the first element failure is relatively short. This modeling method for an already simplified mesh is shown in figure 10.

3.6 Tied modeling

As well as the constrained modeling technique, this approach tries to simplify the prior proposed beam model, avoiding the direct connection between nodes of the bolt and the specimen's node. Instead, a simplified model of the bolt which could also be used for the modeling of spot welded joints is tied with a *CONTACT_TIED_SHELL_EDGE_TO_SURFACE – formulation onto the specimen. The advantage of this modeling approach is its simplicity since a regular mesh with element sizes between 3 mm to 5 mm can be used without applying any additional holes into the geometry. Nevertheless, this model is a strong simplification of the real situation so it might be necessary to combine this modeling technique with one of the above presented.

3.7 Meshing methods

As already mentioned, different meshing methods are considered for this work. In order to do a material parameter fitting, a fine mesh is used in the area of the bolted joint. For the simplified models using a constrained or a tied modeling method, a regular mesh is used with two different element sizes. The different meshes are already shown in figure 9 a) & b) and 9 c). For real crash simulation applications, a simplified mesh such as shown in figure 10 is preferred due to calculation time.

4 Simulation Results

4.1 Usage of the *MAT_054 – material model

As it is shown in figure 11 (left), using the possibilities of the *MAT_054 material model leads to good results for all three kinds of cut out directions. It has to be mentioned that the experimental results of all three directions are presented in one curve due to resemblance. Figure 11 (right) shows the cut-out directions of the different specimen. Nevertheless, using this modeling technique leads to higher calculation times due to strong element deformations after reaching the first level of the force plateau. These results can be gained with a modeling technique using one integration point for a [0°/90°]-layer such as it is shown in figure 8 b). For a second series of experiments, similar results to the once shown below can be observed when using two integration points per [0°/90°]-layer.

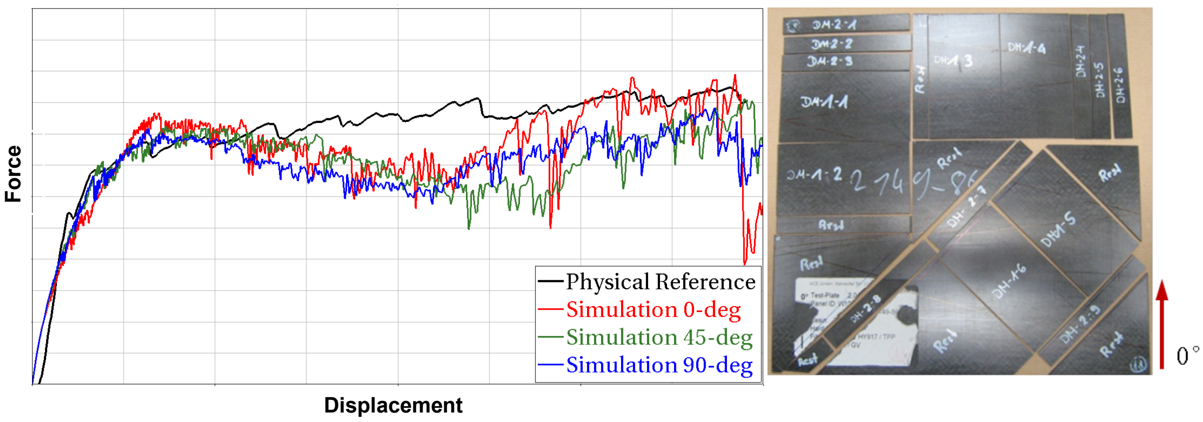


Fig. 11: Simulation results for three different cut-out directions (left) and the cutting plan (right).

Another conclusion of figure 11 (left) is that the cut-out angle seems to have little influence on the measured and simulated force-displacement curves of these laminates. Therefore, the following diagrams show the force-displacement curves of specimen with a cut-out angle of 0° in order to reduce the number of curves and to highlight other conclusions from the proposed modeling techniques.

4.2 Contact modeling technique

Using a *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET – contact formulation to be able to use a regular mesh such as it is shown in figure 10 and a simplified representation of the bolt leads to the results which are shown in figure 12 (left). In order to decrease the influence of the calculation time cause by the high deformations of the different elements, the parameter EFS is varied between 0.2 and 0.8 such as it is shown in the diagram. Allowing a shell element deformation of 80% leads to acceptable results when comparing the simulation with the experiments but for standard crash applications, usually no more than 25% of element deformation is allowed in order to avoid highly distorted elements. Compared to figure 11 (left), the curve of the physical reference looks different since this time, simulations are preformed for the second series of experiments which do have a different laminate lay-up.

Another criterion for the quality of the simulation results is the deformation along the movement of the bolt. This is shown in figure 12 (right) for the presented modeling technique and will be compared with the experimental results later on. For this modeling technique it can be concluded that still high numbers for EFS are necessary to get a good match between the simulation and the experiments. Therefore, further investigations have to be made.

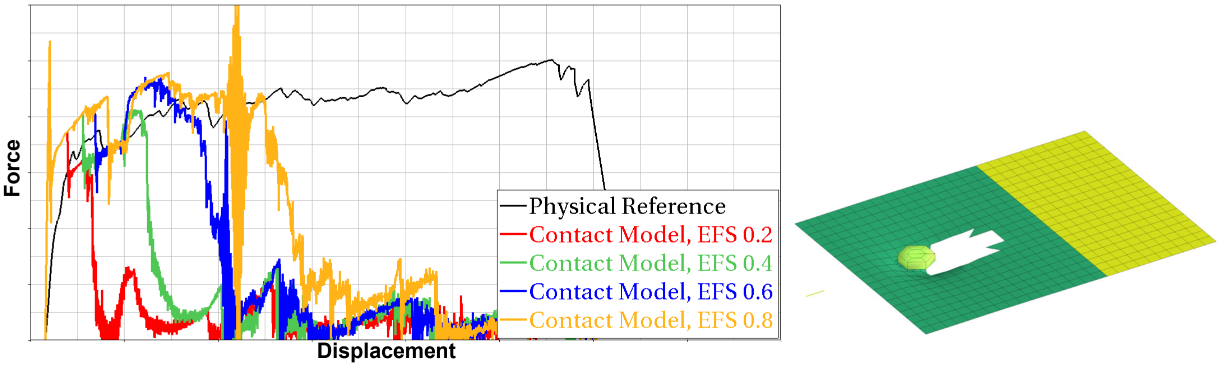


Fig. 12: Simulation results for a contact modeling strategy with different values for EFS.

4.3 Beam modeling technique

The approach using additional beam elements to transfer specific loads onto the shell mesh is already explained in section 3.4. The results given in figure 13 (left) show the force-displacement curve of such a model, even though compared to figure 9 c), the bolt is modeled in a simplified way. From each node of its outer corners, several beam elements are spread towards the underlying shell mesh, generating a constant load distribution onto the composite part. The beam elements are modeled with a *MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM – material model. With the *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET – contact definition, the model of the bolt is again attached to the underlying part which generates more a combination of the model presented before with the additional beam elements. Nevertheless, better results compared to the ones shown in figure 12 (left) can be gained, also for the low EFS value of 20%. All the curves show a slight overestimation of the fist peak in the force-displacement curve. In case of the model using EFS = 0.4, this leads to a good match of the force–plateau measured during the experiments and the one obtained from the numerical simulation. When comparing the element failure in figure 13 (right) with the one shown in figure 12 (right), it is obvious, that even more elements do fail due to the load distribution onto the other elements. As will be shown later on, the failure pattern of the modeling with a simple contact definition is by far more realistic than the failure pattern which can be observed at the model using beam elements.

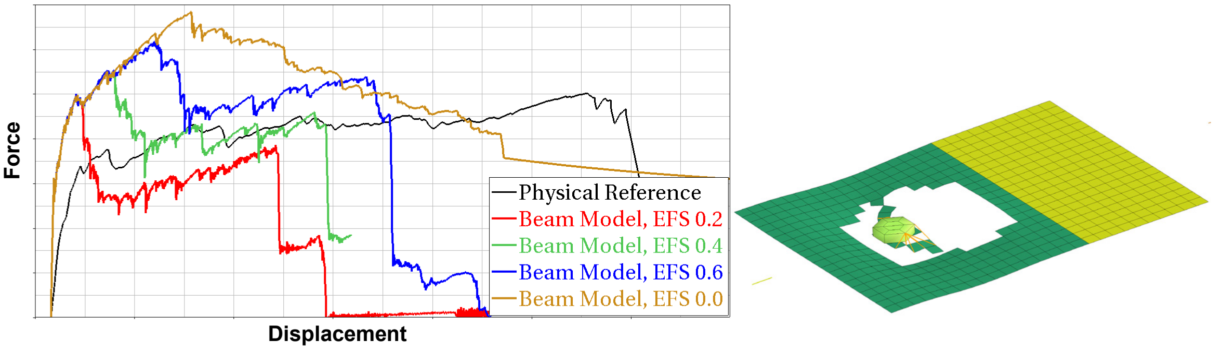


Fig. 13: Simulation results with a beam-model for different values for EFS.

4.4 Constrained-Interpolation modeling

In figure 14 (left), the simulation results of a modeling technique using *CONSTRAINED_INTERPOLATION such as it is described in section 3.5 is used to model the bearing behavior for the laminate which was analyzed in the first series of experiments. For an acceptable value of EFS = 0.2, good results are gained for the force-displacement curve for at least 40% of the total way of the bolts movement. This is the case for all the three cut out directions such as they are shown in figure 11 (right). When comparing the fracture patterns of the above shown specimen used for the simulation, the *CONSTRAINED_INTERPOLATION – model seams to be a compromise between the first two shown facture patterns since not as many elements do fail as for the modeling using beam elements.

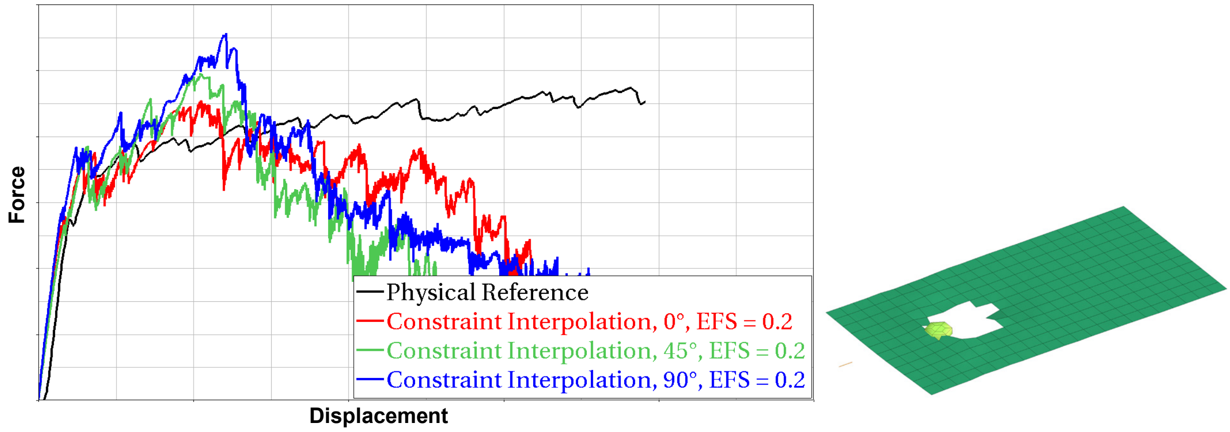


*Fig. 14: Simulation results using *CONSTRAINED_INTERPOLATION modeling technique.*

4.5 Fracture pattern

When comparing all the failure patterns from the simulations with the patterns which can be obtained during experiments (figure 15), one can observe that the fracture patterns of the experimentally tested specimen are almost similar. This is also an explanation for the similarity of the force-displacement curves. The composite material is basically pushed forward and the material thickens in front of the moving bolt. For specimen one and two (u.l. & u.m.), the 90° orientation on the upper surface is visible through the cracks running from left to right. Specimen three and four (u.r. & l.l) are the ones tested with a 45° cut-out angle and specimen five and six (l.m. & l.r.) show the failure pattern of a bolt being moved along the upper and lower 0° orientation of the fibers. These patterns are also specific, especially when taking a look at the travelling failure front. Compared to those failure patterns, the one of figure 12 seems to be the most promising, but as mentioned before, the main drawback of this modeling technique is the sudden drop of the force-displacement curve right after element failure.

5 Conclusion and Outlook

In this work, different modeling techniques for bearing test loadings were tested in order to find a proper model being able to simulate the failure mechanism of anisotropic composite materials. For validation purposes, experiments were made with two different laminate lay-ups and the obtained force-displacement curves were taken as reference for the simulation results.

A first approach was to model the area of the bolted joint with a fine mesh compared to the surrounding area and to neglect element failure by strains. With this approach, it is possible to more or less predict the measured force-displacement curve with FE-analysis. Main drawback of this method is that the calculation time increases during the simulation. This is caused by the strong deformations which are taking place at the front of the moving bolt. Therefore, three other modeling techniques were tested, a tied modeling technique which places the bolt via a contact definition onto a simplified shell mesh, a beam modeling approach, which transfers the loads from the bolt onto the shell mesh in front of it, even

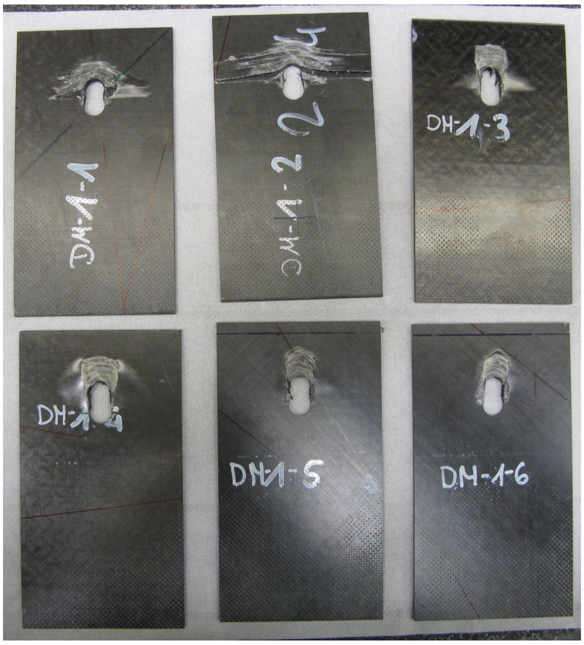


Fig. 15: Failure pattern of the bearing test specimen.

though elements are failing, and a *CONSTRAINED_INTERPOLATION model which is somehow similar to the beam modeling, but no additional elements and material cards have to be introduced.

From all these models, the beam and the constrained modeling technique seem to be the most promising when in comes to crash simulations in full vehicle analysis. The force-displacement curves can be represented quite good with these models, even though the failure pattern of the beam model is not as realistic as the one of the *CONSTRAINED_INTERPOLATION model when it comes to the comparison with the real specimen. Therefore, further work has to be done to modify and improve these modeling techniques in order to get a high quality and simply representation of these punctual joints.

Another conclusion of this work is that it is possible to either model the laminate layers as a classical orthotropic material with a high Young’s modulus in one direction which leads to a modeling with an integration point per layer. More realistic and even faster is the usage of Young’s moduli which are equal in a- and b- direction of the material. Therefore, it is necessary to smear the further material- and failure parameter in a way that a [0°/90°] non-crimp fabric layer can be represented by one integration point. This reduces the number of integration points through the thickness of each element and therefore leads to faster simulations.

6 Literature

[1] Moncayo D., Koegl M.: “Berechnung von FVK Bauteile”, EDM CAE Forum 2011, Stuttgart Germany

[2] Chang, F. K., Chang, K.Y.: "Post Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure", Journal o. Composite Materials, 1987, 809-823

[3] Matzenmiller, A., Schweizerhof, J. K.: “Crashworthiness Considerations of Composite Structures – a First Step with Explicit Time Integration in Nonlinear Computational Mechanics-State-of-the-Art“, Springer Verlag, 1991

[4] Moncayo D., “Numerical simulation of low impact delamination in composite materials”, CANCOM 2007, 14-17 August 2007, Winnipeg Canada

[5] Livermore Software Technology Corporation (LSTC): „LS-DYNA Keyword User’s Manual – Vol. I”, Livermore, CA, USA, 2012

[6] Devynck, J.: “Fragmentation and delamination in carbon composite parts”, DRT Thesis, DAIMLER AG 2012

[7] Livermore Software Technology Corporation (LSTC): “LS-DYNA Keyword User’s Manual – Vol. II – Material Models”, Livermore, CA, USA, 2012