Methodology to simulate veneer based structural components for static and crash load cases

G. Piazza[1], D.B. Heyner[1], E. Beeh[1], Horst E. Friedrich[1]

[1]German Aerospace Center (DLR e.V.), Institute of Vehicle Concepts,
Pfaffenwaldring 38-40, Stuttgart, Germany

0. Abstract

The increased public interest in green technology combined with new environmental policies results in the need for light-weight vehicles with a reduced global environmental impact. A method that is gaining importance is the reduction of the cradle-to-grave carbon footprint of utilized materials. For structural components, one promising approach is the utilization of biomaterials such as veneer-based hybrid materials since wood is a natural carbon storage. The specific properties of wood are comparable to aluminum and magnesium, and thus have the potential to replace some structural and semi-structural components of a vehicle. When required, the hybridization of veneer-based materials with traditional materials, such as metal sheets, can further increase its structural performance.

While it is technologically possible to implement such a material concept, a key challenge is the application-oriented simulation of non-hybridized and hybridized wooden structures. A suitable simulation method and material model must be found and validated. At the Institute of Vehicle Concepts of the German Aerospace Center, the methodology to simulate beech-veneer based structural components for static and crash load cases has been developed over the last three years. The characteristics of the veneer were determined in order to fit a material model in LS-Dyna which was then implemented in various simulation approaches for wooden structures. The findings were also transferred to simulate hybridized structures.

This paper presents the chosen approach to simulate a hybridized generic structure that represents a door impact beam. It compares simulation results of different three-point flexural bending tests with testing results. The findings were generated in the project “For$^{(s)}$tschritt”.

The fully qualified simulation approach and material model will contribute to the structural application of non-hybridized and hybridized veneer-based composites in modern vehicle structures.

Keywords

Crash, Simulation, Veneer, Structural

1. Introduction

Lightweight construction has once again come to the fore of the automotive industry due to strict emission guidelines, rising energy costs and legal limits for the vehicle weight of electric vehicles. Options include the substitution by materials of lower density and the use of multi-material systems. A pure material substitution with lighter materials is only suitable if individual properties as well as the integration of the component into the assembly or the overall structure are considered and optimized. The economy must not suffer compared to existing construction methods. The same applies to the use of multi-material systems. Furthermore, in addition to lightweight construction, the ecological consideration of vehicles, both during utilization phase and production phase, is becoming a focus of politics, society and thus also the automotive industry. While greenhouse gas emissions can be reduced during utilization phase through lightweight construction, the greenhouse gas emissions during production phase highly depend on the materials used. Depending on the process route, steels as well as classic lightweight construction materials, such as aluminum or magnesium alloys, but also new lightweight construction materials such as carbon fiber reinforced plastics (CFRP) or glass fiber reinforced plastics (GFRP) generally tend to have a poorer balance in comparison to wood derivates [2] [3].

Given the political, economic and ecological boundary conditions mentioned above, there is a need for efficient, holistic and sustainable lightweight construction solutions. One approach is the use of renewable raw materials, such as wood and wood derivatives, in non-load-bearing but also in load-bearing structures in vehicles. With weight-specific material characteristics in fiber direction that are comparable to or better than classic lightweight construction materials, wood offers a sustainable and economical alternative. The last two columns in Tab. 1 show the weight-specific characteristics for bending, where applicable along fiber direction (l), according to Ashby [1].

Tab. 1: Technical comparison of materials [5] [8]

Material	Density [g/cm^3]	Young's Modulus E [MPa	Ultimate strength (UTS) [MPa]	$(E)^{1/2}$/Density* $[MPa]^{1/2}/[g/cm^3]$	$(UTS)^{2/3}$/Density* $[MPa]^{2/3}/[g/cm^3]$
Aluminum	2,30 - 2,80	70.000	45 - 500	95 - 115	5 - 27
Beech	0,54 - 0,91	~14.350 (l)	100 - 140 (l)	132 - 222	24 - 50
CFRP	~1,50	~140.000 (l)	~1.700 (l)	~250	~95
GFRP	~2,00	~44.500 (l)	~1.100 (l)	~105	~53
Magnesium	~1,74	45.000	100 - 300	122	12 - 26
Steel	7,85 - 7,87	210.000	340 - 1.800	58	6 – 19

*Higher values are better.

2. Properties of veneer and veneer-based components

2.1. Properties of veneer

Wood consists mainly of the three structural substances cellulose, hemicellulose and lignin. Analogous to CFRP materials, cellulose are the long, longitudinal fibers, while lignin forms the matrix. The hemicellulose consists of significantly shorter fibers and connects individual cellulose fibers crosswise with one another. This creates support in the cross direction. When manufacturing veneer a stem is typically peeled around its longitudinal direction. Therefore veneer-layers are orthotropic materials where the properties are defined in longitudinal (L), tangential (T) and radial (R) directions, see Fig. 1. Depending on the used material model, the directions in the simulation are also described with “A” or “1” for the longitudinal and “B” or “2” for tangential respectively “C” or “3” for radial direction.

In the studies carried out in project “For$^{(s)}$tschritt”, veneer made of beech wood was used. The properties in-plane were determined through simple tensile tests mainly in L and T directions (see Fig. 3 left) as well as compression tests (see Fig. 3 right). Testing showed the expected orthotropic material behavior.

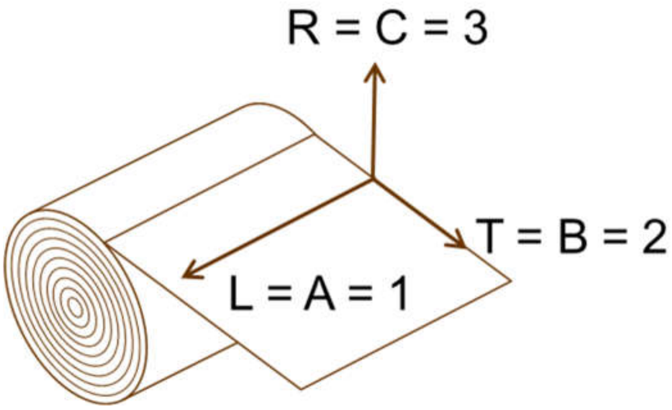


Fig. 1: Illustration of the orthotropic directions of veneer

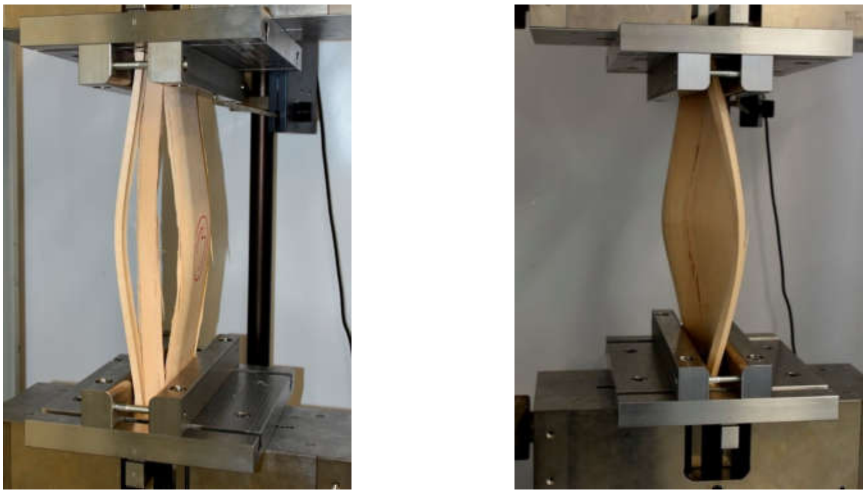
2.2. Properties of veneer-based layered composites

As a natural material wood in general has scattering characteristics. For example, scatter in Young's moduli are depending on the amount of fibers in a cross-section which changes depending on the position in a tree. Scatter in failure strength is depending on the amount and size of imperfections in a given specimen. By using veneer-based layered composite the scatter can be reduced since on the one hand the used veneers are preselected such as the number of imperfections within them are reduced and on the other hand the usage of multiple veneers leads to a statistically averaging of the characteristics.

The behavior can mostly be described by the classical laminate theory [9] with following assumptions:

- Each layer is ideally linear elastic
- The thickness of each layer is small compared to its width and length
- The thickness of each layer is constant
- Plane-stress due to small thickness of a layer
- Deformations are small in order to use first order deformation theory
- Euler–Bernoulli beam theory is valid
- Layers are glued together ideally

Failure occurs in tension regardless of orientation when failure strain is exceeded. In compression global buckling or local buckling of individual fibers with subsequent crack formation in loading direction occurs in grain respectively in a uni-directional (UD) laminate. A combination of local buckling with subsequent crack formation and shear failure occurs when individual layers are oriented differently towards each other (see Fig. 2). For bending, failure occurs mainly due to exceeding of failure strain on the side under tension. Delamination due to exceeding of normal stress (out-of-plane) in compression and of shear stress in bending can occur.



UD laminate $[0]_n$ with 5x1,5 mm **Plywood $[0\text{-}90]_n$ with 5x1,5 mm**

Fig. 2: Failure of a hat profile made of UD laminate (left) and plywood (right) in compression

3. Modelling of veneer-based layered composites

3.1. Finite-Element-Method and Solver LS-Dyna

In the present load case with high plastic deformations and crack formations, the non-linear material behavior of the veneer-based layered composite cannot be calculated using classical methods, e.g. the calculation of an analytical solution. An approximation method must therefore be used to calculate the complex behavior during impact simulations. Therefore, the finite element method (FEM) is used [4]. FEM is a numerical approximation method used in the present work to calculate strengths and deformations occurring due to the intrusion of a pole into the component.

With the help of computer-aided engineering (CAE) a network from elements and nodes that map the geometry as well as the definition of the properties of the component is created in a pre-processor. Boundary conditions, loads and contact definitions are then applied at individual nodes. The data is then transmitted to an equation solver. In this paper the solver

LS-Dyna was used [6]. Finally, the results of the solver are read in a post-processor and displayed graphically. For example, deformations or stresses occurring during the side impact can be made visible at defined time steps.

In this paper, keywords and input parameters from LS-Dyna will be shown in italics.

3.2. Material Model *MAT_054

In order to model the single veneer layers for the FEM-Solver LS-Dyna the material model **MAT_ENHANCED_COMPOSITE_DAMAGE* (*MAT_054) is used. Fig. 3 shows the comparison between simulation and testing for beech veneer. While the behavior parallel to fiber in the simulation is in line with testing, for the behavior in compression transversal to fiber a trade-off has to be done between in ultimate strength and failure strain respectively absorbed energy.

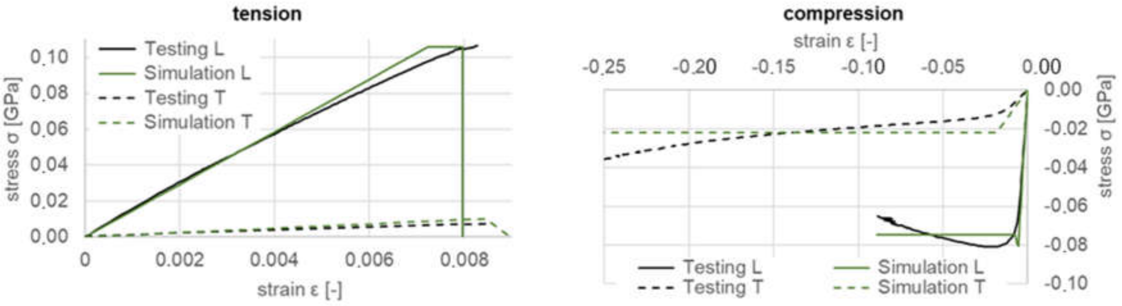


Fig. 3: Properties in tensile tests (left) and compression test (right)

3.3. In-plane properties

The properties of the single layers of a (sub-)laminate are represented through the layered shell approach. A shell layer is modeled at the center of the physical height of the laminate it represents. A numerical integration point is defined within the element for each veneer layer. The thickness and orientation of the single layers as well as their material models are defined locally in the integration points of the shell (see Fig. 4). Hence, different ply stacks can be defined in a single shell layer. Multi-material systems, which can for example also consist of different woods, can also be defined through the layered shell modelling approach.

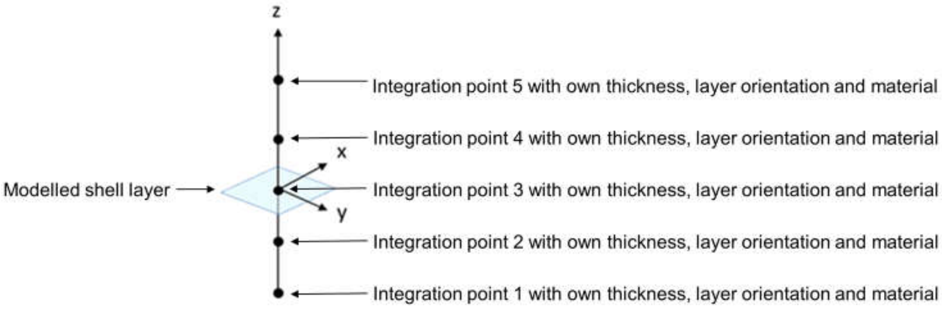


Fig. 4: Example of a layered shell representing a component with five layers

3.4. Out-of-plane properties

Testing showed that bending a laminated structure often leads to delamination in the middle of the component (see Fig. 5) due to exceeding of shear stress [10]. To enable delamination in the simulation, the components were modeled with at least two layers of shell elements, where each of the layer represented a sub-laminate as described above in chapter 3.3. The contact interface between the shell layers was modeled through the keyword **CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE* with *Option 11*. This activates a contact algorithm that is equivalent to using cohesive zone elements with a fracture model based on a bilinear traction-separation law, a mixed mode delamination criterion as well as a damage formulation [7].

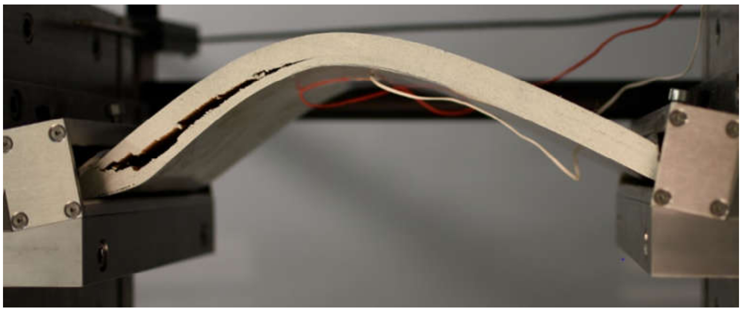


Fig. 5: Example of a component with delamination in the middle due to exceeding of sheer stress

Main parameters of the fracture model are the normal failure stress in tension (*NFLS*) and the shear failure stress (*SFLS*) between the two shell layers as well as the mode I and mode II critical strain energy release rates (G_{Ic} respectively G_{IIc}) when delaminating. The parameter *CN* determines the normal stiffness of the material in the interlaminar region. For the tangential stiffness *CT*, a scale factor (*CT2CN*) with a value between 0 and 1 is used on the normal stiffness – hence *CT* = *CT2CN x CN*. The description of the mixed-mode loading treatment will not be described in this paper but can be found in [7].

For illustration a mode I crack opening model is shown in Fig. 6. The bilinear behavior shows first the rise of stress with the slope *CN* from zero stress up to a value of *NFLS* and then the drop to zero. The area under the curves corresponds to the mode I critical strain energy release rate G_{Ic}. After reaching the displacement at which failure occurs ($\delta_{failure}$), the interface between master segment and slave nodes turns to a surface to surface contact and can only be loaded on compression [7]. Mode II crack opening is described analogously with the corresponding parameters.

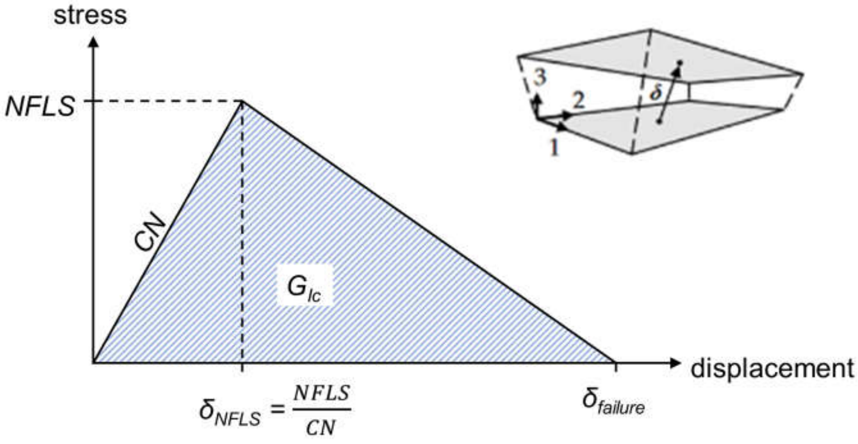


Fig. 6: Bilinear law used for mode I crack opening to describe delamination

4. Simulation of generic veneer-based components

To represent a realistic load case a generic structure comparable to a door impact beam was locked at. Door impact beams are a safety feature in todays vehicles that protect the driver respective the passengers during a side impact. As the name suggests, it is mounted in the door and absorbs energy from a side impact while maintaining structural integrity.

4.1. Calibration through quasi-static three-point bending flexural test (3PB)

To calibrate the contact interface described in chapter 3.4, three-point bending flexural tests (3PB) on a universal testing machine (UTM) were carried out and simulated. A 10-inch diameter pole (254 mm) impacted the generic beam made of 16 veneer layers with 2,0 mm thickness and a steel strip with a thickness of 1,2 mm. The test setup and the trapezoidal cross section can be seen in Fig. 7.

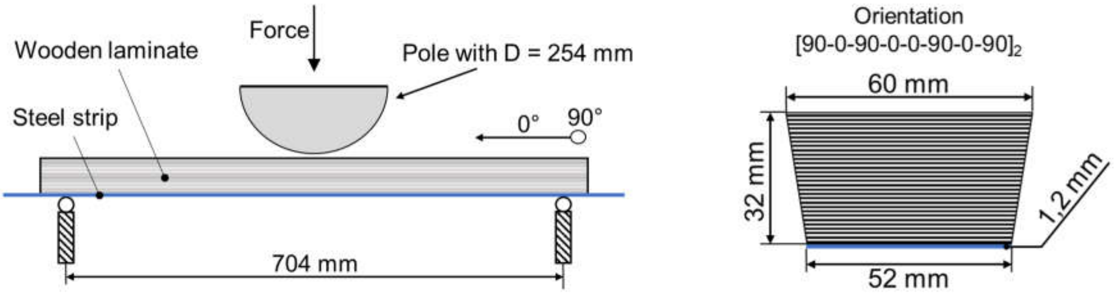
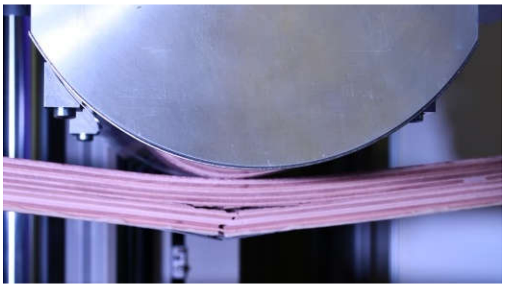
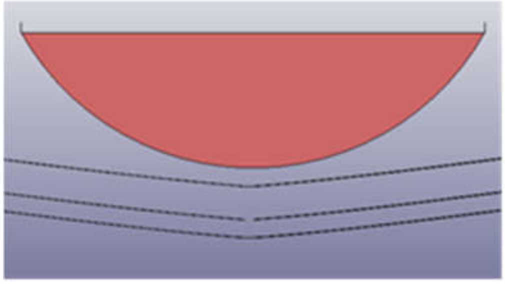


Fig. 7: UPM - Sketched test setup (left) and cross-section (right)

In simulation, the beam was modeled by two shell layers for the wooden part each representing a sub-laminate with the orientation [90-0-90-0-0-90-0-90] (see chapter 3.3) in order to allow delamination in the wooden part in simulation (see chapter 3.4). The steel strip was modeled through a third shell layer and was on purpose not included into the lower sub-laminate in order to allow a possible delamination between steel strip and wooden part. The three layers of shell elements can be seen in Fig. 8 top right.

The comparison between testing and simulation showed that a similar course of the force-displacement curve (see Fig. 8 bottom) could be achieved. Furthermore, the fracture pattern could be mapped to good approximation (see Fig. 8 top).





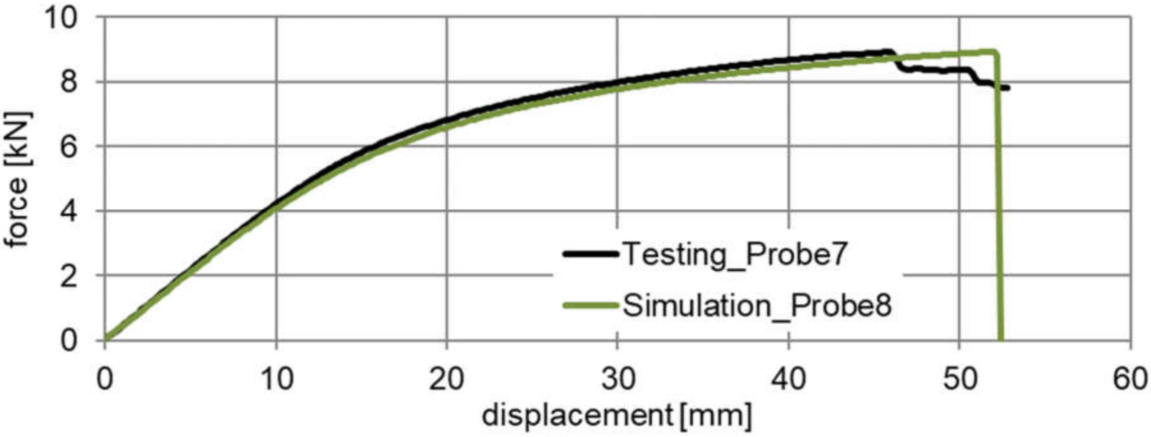
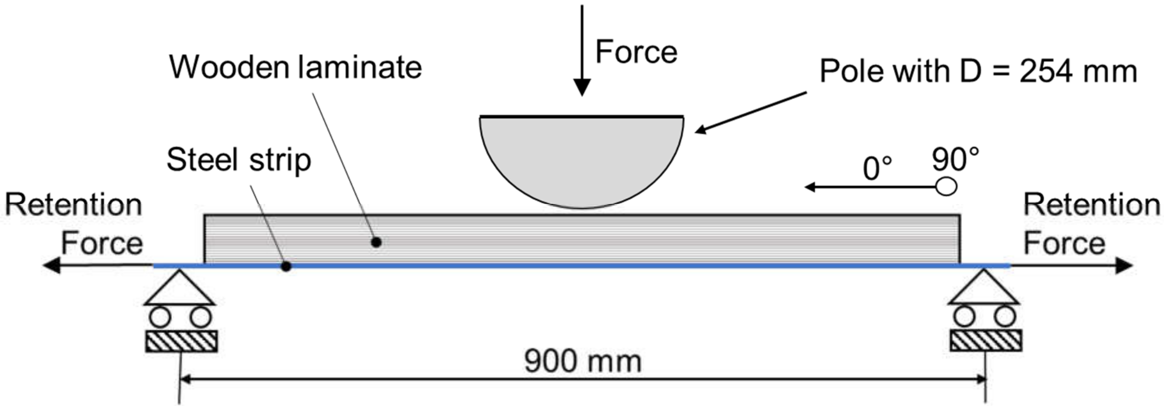


Fig. 8: UTM - Comparison between testing and simulation model with calibrated material card of bending line (top) and of force-displacement curves (bottom)

4.2. Validation through quasi-static 3PB with boundary conditions

On the static load test facility (SLTF) a hydraulic cylinder pushed the pole into the hybridized wooden beam, in approximation to the quasi-static door impact test according to FMVSS 214S with a pole of 10-inch diameter (254 mm). In contrast to the tests at the UPM, self-constructed absorber units (AU) were mounted on the steel strip at the beam endings (see Fig. 9 top and middle for test setup and Fig. 9 bottom for absorber units). The AU provided a defined motion while maintaining a constant retention force of about 10 kN by pulling a blade through an aluminum tube.



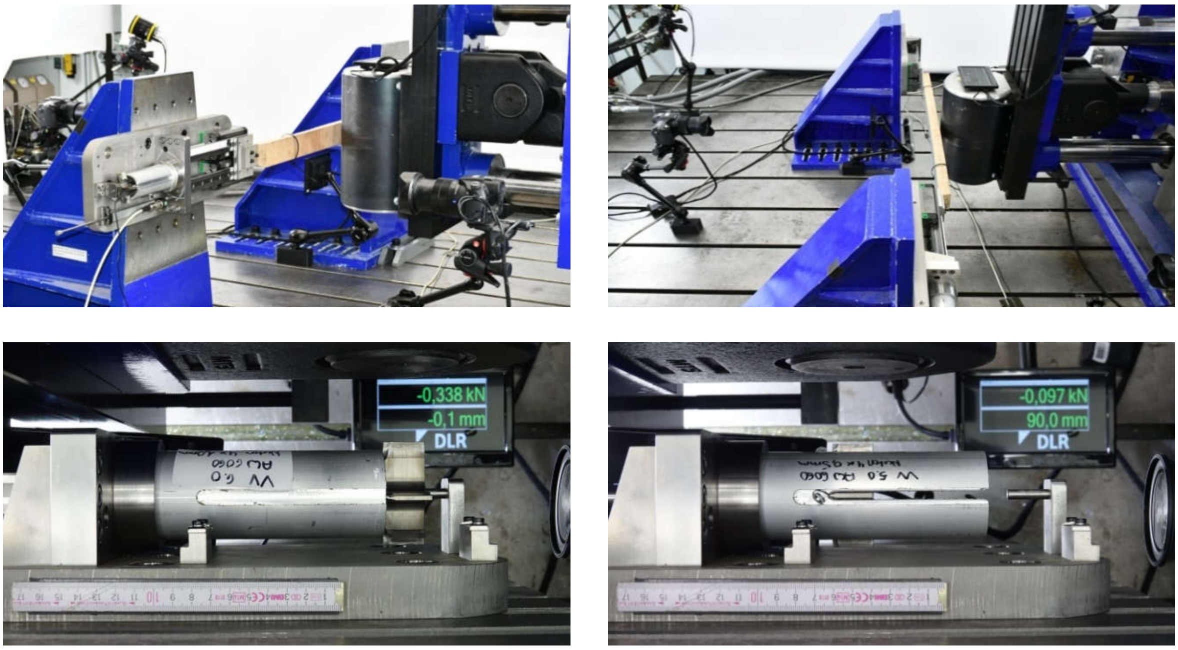


Fig. 9: SLTF - Sketched test setup (top), pictures from testing (middle) and absorber units (bottom)

The AU were modeled using solids in combination with **MAT_HONEYCOMB* in the simulation. A parameterized force-displacement curve is used such that the force level was adapted to the same retention force planned for the experiment.

In testing, the two blades cut the tubes very asymmetrically (99.8 mm with 59.6 mm) due to friction between pole and beam, manufacturing tolerances of blades and aluminum tubes of the absorber units as well as imperfections in the testing setup. In simulation, the blades cut 80.7 mm into one AU and 86.8 mm into the other. Since the simulation is idealized, this more symmetrical distribution was expected. The simulation showed that the location and type of failure in the laminate led to small asymmetry. The combined penetration depth of both blades added up to 167.5 mm in simulation and 159.4 mm in testing. Hence the overall kinematics in simulation is in good approximation to testing.

In Fig. 10 testing and simulation are compared. The qualitative course of the two force-displacement curves at the pole are similar although absolute values in simulation tend to be smaller. The height of the first peak in simulation is 8.67 kN in comparison to 9.93 kN in testing (-12,7 %). Furthermore, a second failure (red circle) occurs in simulation. The energy required during testing was 2765 J, while it was 2573 J (-6,9 %) in simulation.

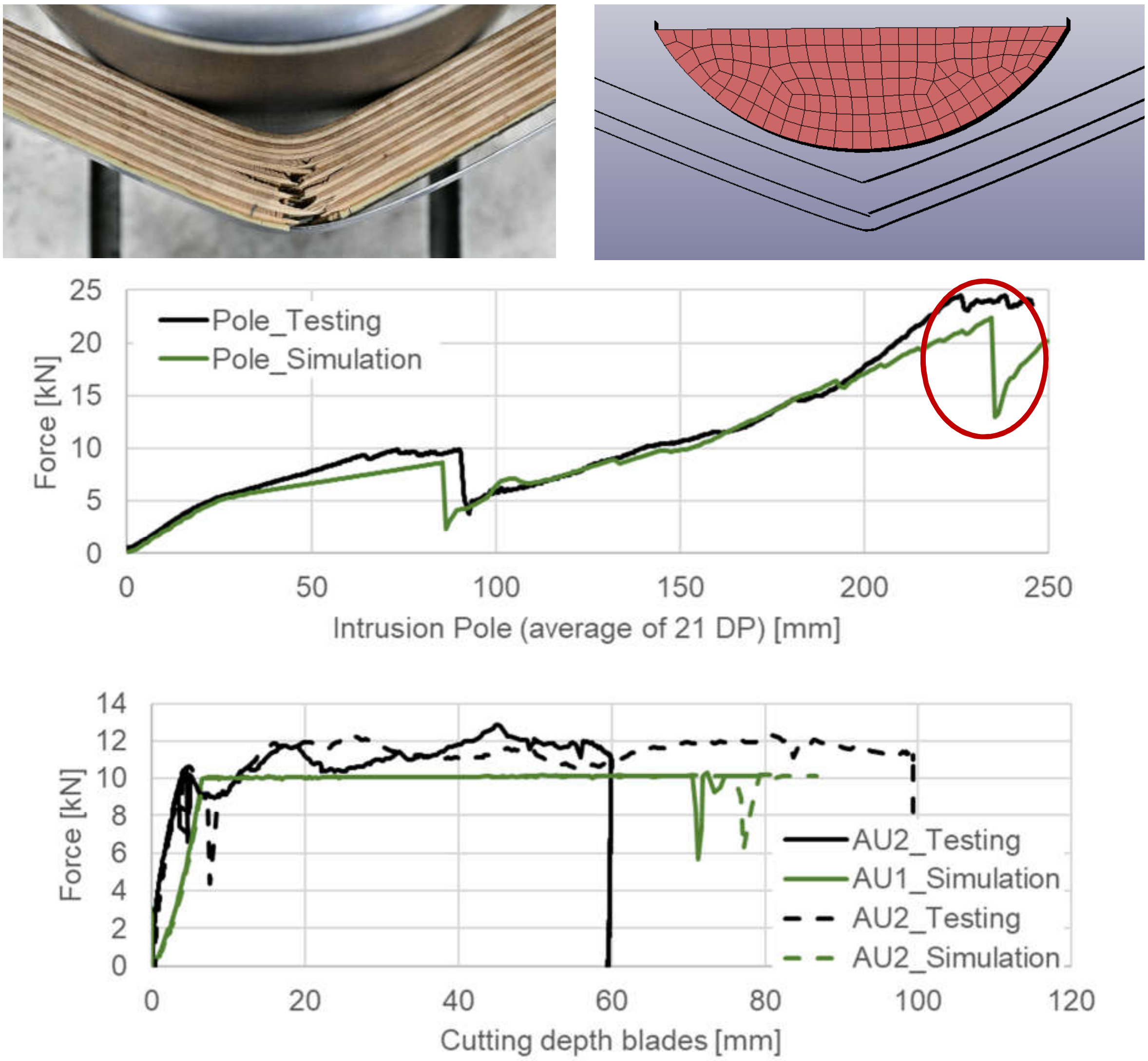


Fig. 10: SLTF - Comparison between testing and simulation of bending line (top), force-displacement curves of poles (center) and force-displacement curves of the absorber units (bottom)

The discrepancy in peak-force could be partly attributed to higher forces in the AUs in testing (around 11 kN instead of the expected 10 kN). The difference at an intrusion greater 200 mm was attributed to failure of single integration points and numerical deletion of those integration points in the layered shell on the side under compression.

With these findings, the simulation approach and the material model are considered validated.

4.3. Expansion of dynamic behavior through high-speed 3PB with boundary conditions

To simulate high-speed events like crash or impact, dynamic material properties have to be transferred into simulation.

For this purpose, a drop tower was used in combination with the absorber units. The pole is used as an impactor with a mass of 84,3 kg and an impact velocity of close to 5 m/s. The test setup can be seen in Fig. 11. Boundary conditions are analogous to the tests carried out on the static load test facility (see Fig. 9 top).

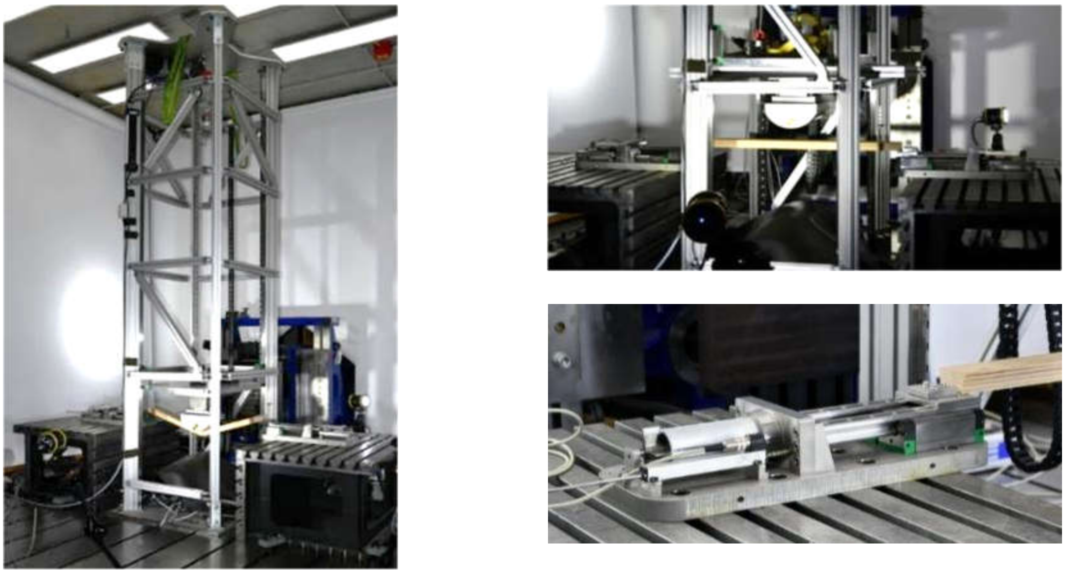


Fig. 11: Drop tower – pictures of test setup

A reduced test setup was transferred to the simulation. With knowledge generated within the project “For$^{(s)}$tschritt” by high-speed testing of small specimen, the static material model of **MAT_054* was roughly expanded by introducing strain-rate-dependent curves for the individual strengths. The material model was afterwards refined with the knowledge gained from the high-speed three point bending flexural tests.

Fig. 12 shows the result of the comparison between testing and simulation for the drop tower test. Test results can be reproduced to a good approximation.

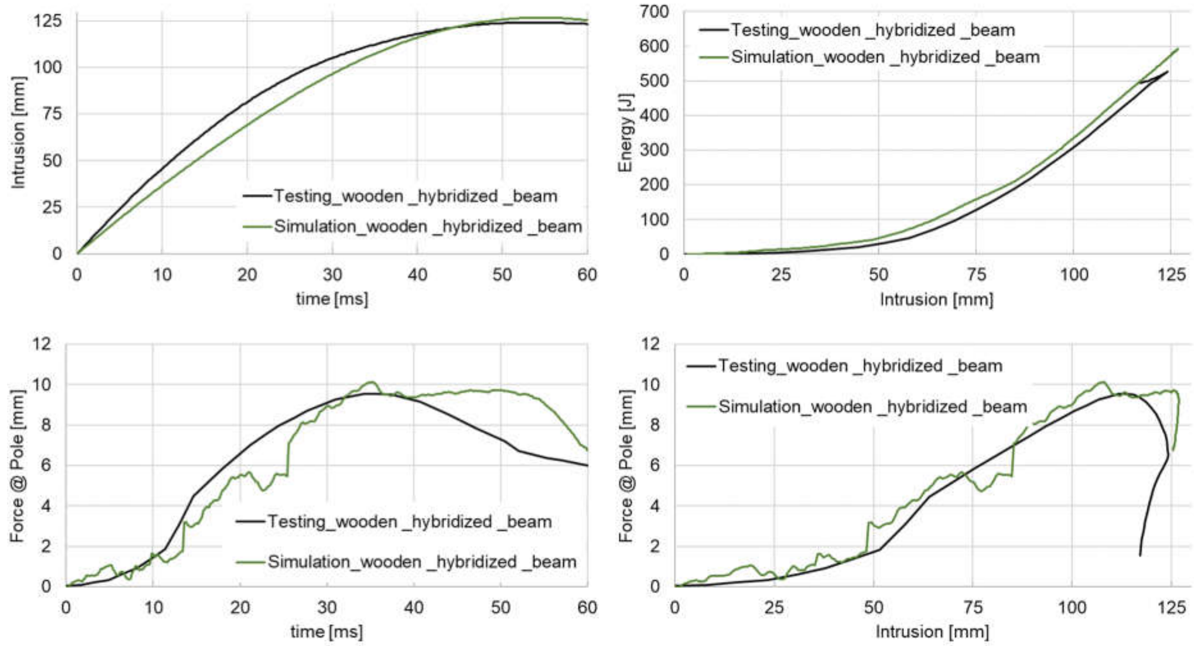


Fig. 12: Drop tower – displacement-time curve of poles (top left), energy-displacement curve (top right), force-time curve (bottom left) and force-displacement curve (bottom right)

5. Summary

The presented paper puts forward a validated simulation approach for veneer-based multi-material components. Comparison between test and simulation with subsequent evaluation has taken place and shows good agreement. The global behavior in bending but also the local behavior in the event of failure could be mapped to a good approximation.

Hence, a transversal isotropic material model is available to simulate quasi-static, e.g. equivalent loads, and high-speed events, such as impact or crash, in LS-Dyna.

Work is still needed on refining the dynamic parameters of the material model and hence on reducing the discrepancy between simulation and testing.

Afterwards approval-relevant crash load cases as well as those required by consumer protection agencies can be simulated in order to determine the potential of a specific assembly in comparison to the reference assembly.

6. Acknowledgments

The authors acknowledge the financial support by the German Federal Ministry for Economic Affairs and Energy (BMWi) through the TÜV Rheinland. We thank our colleagues from the project “For$^{(s)}$tschritt” who provided insight and expertise that greatly assisted the research.

7. References

[1] Ashby, M.F.: “Materials Selection in Mechanical Design”, 3rd edition, Elsevier, 2004

[2] Bergman, R., Alanya Rosenbaum, S.: “Cradle-to-gate life cycle assessment of laminated veneer lumber production in the United States”, Forest Products Journal, vol.67, November 2016

[3] Kelly J.C., Dai Q.: “Life-Cycle Analysis of Vehicle Lightweighting: A Review”, Meyers R. (eds) Encyclopedia of Sustainability Science and Technology, Springer, New York, 2019

[4] Klein, B.: “FEM - Grundlagen und Anwendungen der Finite-Element-Methode im Maschinen- und Fahrzeugbau“, 9th edition, Vieweg+Teubner Verlag, 2012

[5] Klein, B.: Lightweight-Construction. Wiesbaden, 2011

[6] Lee, R.: “LS-DYNA for Engineers – A Practical Tutorial Book”, 1st edition, BW Publications, 2019

[7] LS-DYNA, Keyword user's manual, Volume II: material models, Livermore Software Technology Corporation (LSTC), 2013.

[8] Niemz, P.: “Holzphysik”, Zürich: Institut für Baustoffe, Eidgenössische technische Hochschule, Zürich, 2006

[9] Reddy J.N.: “A Generalization of Two-Dimensional Theories of Laminates Composite Plates”, Communications in Applied Numerical Methods, vol.3, pages 173-180, 1987

[10] Roubins D.H., Reddy J.N.: “Modelling of Thick Composites using a Layerwise Laminate Theory”, International Journal for Numerical Methods in Engineering, vol. 36, pages 655-677, 1993