Automated process chain for calibrating material cards of punctiform and planar joint connections for explicit finite element simulations

Tim Wirtz[1]

[1]German Aerospace Center (Deutsches Zentrum für Luft- und Raumfahrt e.V.)

1 Introduction

Punctiform and planar joint connections are central joining technologies in automotive engineering. These include clinch joints, Semi-tubular self-pierce rivets and flow drilling screws [1]. They play a decisive role in the structural integrity of vehicle bodies [2]. Especially in dynamic load cases such as crash simulations, their mechanical properties are critical for the overall simulation. The accurate representation of these joints in FE simulations requires precise calibrated parameter values of the material cards [3]. However, the characterization of material parameters is time-consuming, especially for explicit FE simulations. The process of material card calibration is often based on manual, iterative steps that require repeated experimental campaigns and simulations. This leads to long development cycles and high resource requirements. The effort required increases considerably for dynamic applications. Therefore, there is an urgent need to optimize this process through automation [4] to significantly reduce the effort required for material card calibration. The approach targets punctiform and planar connections, emphasizing the interaction between the joining partners, the joining technology and joining materials rather than the specific joining technology or the materials used.

The material card calibrated with this process applies only to the selected configuration. As soon as a parameter of the connection is changed - for example the sheet thickness - a new calibration is necessary. The presented methodology is independent of the selected joining technique and can therefore also be applied to other joining technologies such as spot welds. The flexibility enables broad applicability in development, while automation reduces manual effort and sources of error, thereby increasing efficiency in vehicle development. An automated process chain is being developed for this purpose. The quasi-static properties of the joints are determined by performing shear and head tensile tests on a universal testing machine. The dynamic properties are determined using the IMPETUS® pendulum from 4a Engineering GmbH [5]. New clamping concepts and simulation models are developed in order to record the static and dynamic behavior of the joints. In addition, algorithms are implemented that automatically determine the material card parameters of the joint from the test results, taking into account specific boundary conditions. This technical paper presents the current state of development.

2 Materials and Methodology

The Materials and Methodology chapter describes the steps for calibrating material cards for punctiform and planar joints. Based on a systematic approach, the relevant joining methods, load types, test setups and simulation models are explained. For this purpose, specific clamping concepts for the test machines were developed, designed and manufactured to enable the experimental investigations. The modeling of the joints and the test setups are consistent with the real test specimen configurations and clamping concepts. All steps are designed in such a way that they are independent of the specific joining technology and can be used for different types of joints. The entire process chain forms the basis for an automated material card calibration process.

Fig.1: shows the abstracted process chain. Test specimens and methods were produced to represent common loading types of the joints whilst ensuring compatibility with the testing machines. These include quasi-static and dynamic impact tests so that strain rate-dependent material behavior can be measured. The test responses are used to derive initial material parameter values for the joint's material card. These are used as input for first test simulations, which are parameterized simulations of the test set-up. Determination of the initial parameter values is a critical point in the process chain and should be conducted carefully to enable the optimizer to identify the optimum joint parameters within an

acceptable time. An analytical metamodel was trained to predict the discrepancy between simulation results and real measurements. Optimal material parameters can be derived from this metamodel. Lastly, a verification simulation of the head and shear tension tests is conducted with the optimised joint parameters to ensure the response is within acceptable limits/bounds to the physical test data. After successful verification, the validated parameters are exported in material card format for use in explicit solvers.

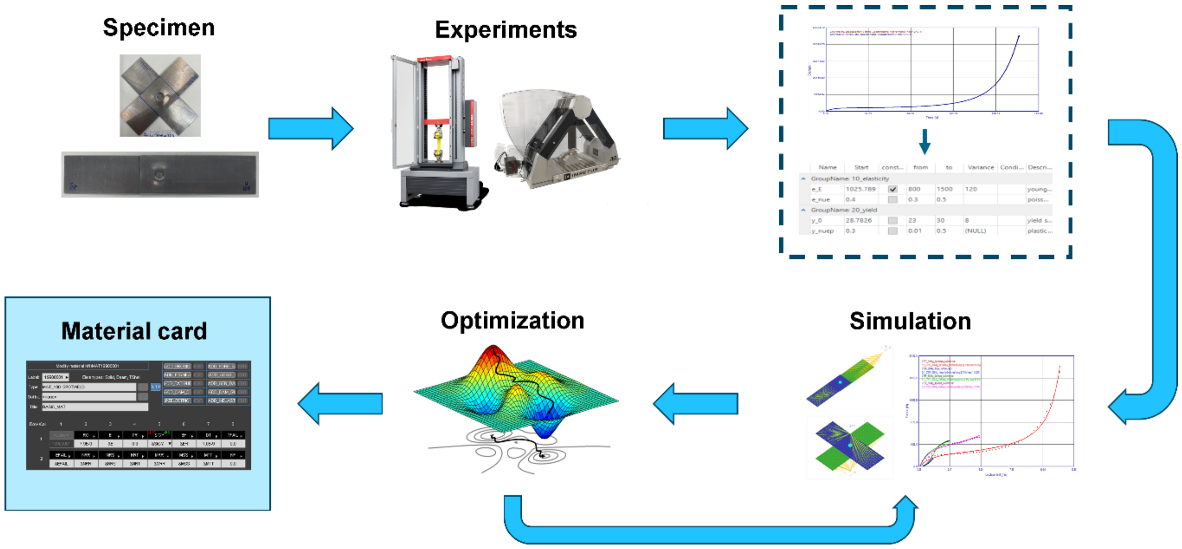

Fig.1: Process chain for the automated material card calibration

2.1 Selected joints and load types

The selection of the joints to be investigated was based on the common joints in automotive engineering, which is why semi-tubular self-pierce riveting, clinching and flow drilling screws were selected to represent punctiform joints. Adhesive joints are considered in the design and approach of the process but are not presented in this paper.

The joints selected undergo two main types of loading, namely, shear tension and head tension [6]. The standards, DIN EN ISO 14273 and DIN EN ISO 14272 represent the loading types for shear tension and head tension respectively.

Due to the limited space in the IMPETUS®, it was necessary to deviate from the standard geometry of the shear tensile and head tensile test specimens. However, the width-to-length ratio of each joining partner was maintained in accordance with the standard geometry.

2.2 Test setups

New clamping concepts had to be developed for the dynamic tests with the IMPETUS®. The specimen clamping for the shear tensile test (Fig.2: and Fig.3) is carried out outside the IMPETUS®. The specimen (a) is clamped between the upper and lower parts of the front clamping (b) and the rear clamping (c). The clamps are adjustable to allow different thicknesses of the joining partners. The entire clamped specimen is then placed into the support with a force sensor (d) attached at the rear guide (e). The force response is used later in the optimization process. The front clamping is supported on the front guide (f). To conduct the test, the pendulum head is raised to a defined angle to achieve the required impact velocity. The pendulum head is then released so that the pendulum head swings in a downward arc due to gravity. The pendulum head strikes the impact surface (g) of the front lower clamping and transfers the energy to the front clamping. Due to the rear clamp being fixed, the movement of the front clamp along the impact plane exposes the joint to shear stress. An accelerometer (h) is attached to the front clamping to measure the acceleration pulse throughout the test phase. To prevent undesirable vibrations in the overall system as a result of metal-to-metal contact, a damping material is applied on the impact surface. The shear test set-up is adaptable to conduct peel tests with minor modifications to the clamping device. The data extraction method remains the same. For this reason, the peel test will not be discussed further in this paper as the automation and optimisation methodology remain the same.

The quasi-static shear tension tests are carried out using a standard test fixture.

Fig.2: Test setup for dynamic shear tension tests

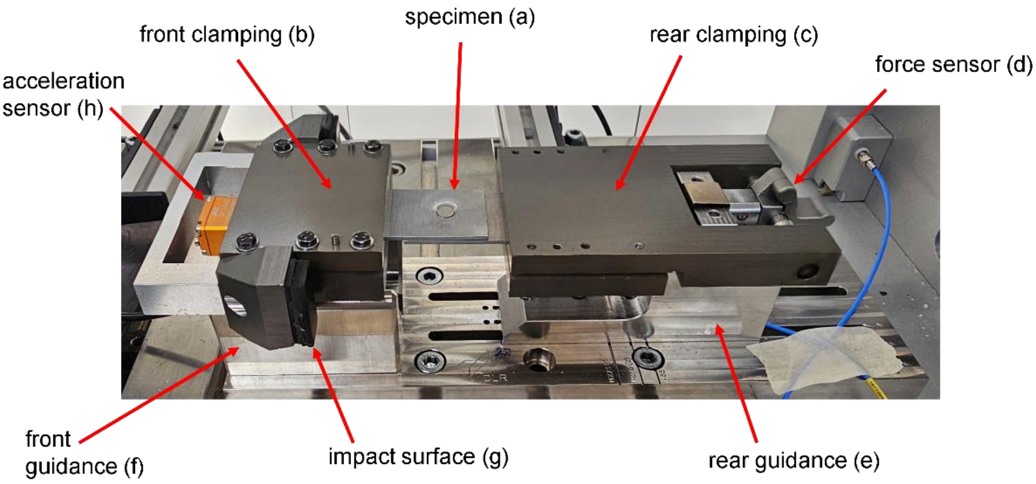

Fig.3: Test setup for dynamic shear tension tests in detail

The concept in principle of the head tensile device (Fig.4: and Fig.5:) for the dynamic tests is similar to that of the shear tensile tests. The specimen is clamped outside the IMPETUS®. For this purpose, the prepared head tension specimen (a) is placed in the vertical clamping (b) and is fixed with the corresponding clamping plates. The horizontal clamping (c) is then placed on the opposite side of the test specimen and clamped to it using the corresponding clamping plates. The topology of the horizontal clamping was optimized in terms of weight so that the accelerated mass could be kept as low as possible. This assembly is then placed into the support with a force sensor (d) attached to the ground plate using an adapter (e). As in the shear tensile test, a damping material is attached to the impact surface (f) to reduce noise in the response signal caused by strong metal-to-metal impact. Furthermore, an acceleration sensor (g) is attached to the horizontal clamping to measure its movement during the test. Specific test specimen configurations lead to higher accelerations than the sensor can record. Therefore, a pattern (h) was applied to the horizontal fixture to track the movement of the clamp. Using a high-speed camera and image analysis algorithms, the velocity profile of the horizontal clamping can be calculated after the test, which is required as input for the subsequent simulation.

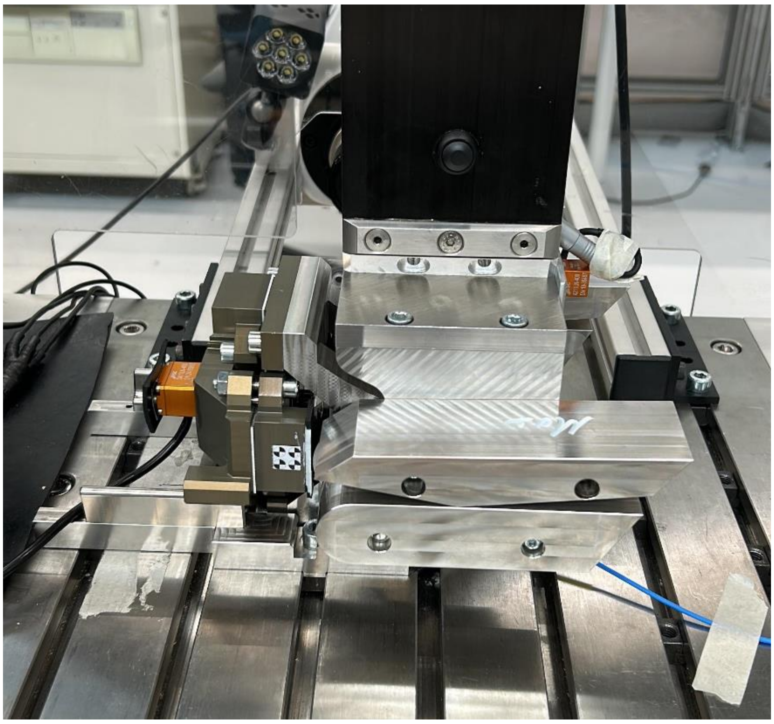

Fig.4: Test setup for dynamic head tension tests

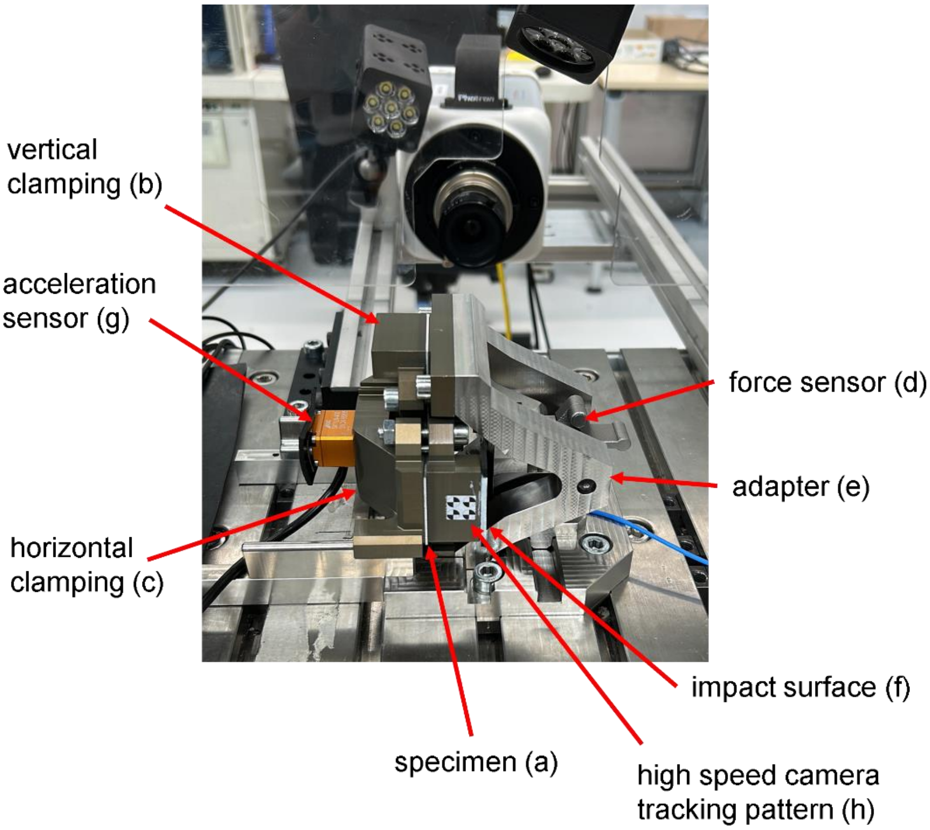

Fig.5: Test setup for dynamic head tension tests in detail

A new clamping device (Fig.6:) was also developed for the quasi-static head tensile tests, which are carried out on a universal testing machine. Each joining partner is clamped in a fixture. This fixture is attached to a 3-axis joint adapter, which is clamped in manual vise grips. The 3-axis joint is necessary to compensate tolerances in specimen production, clamping of the specimen and alignment of the standard tensile test clamping.

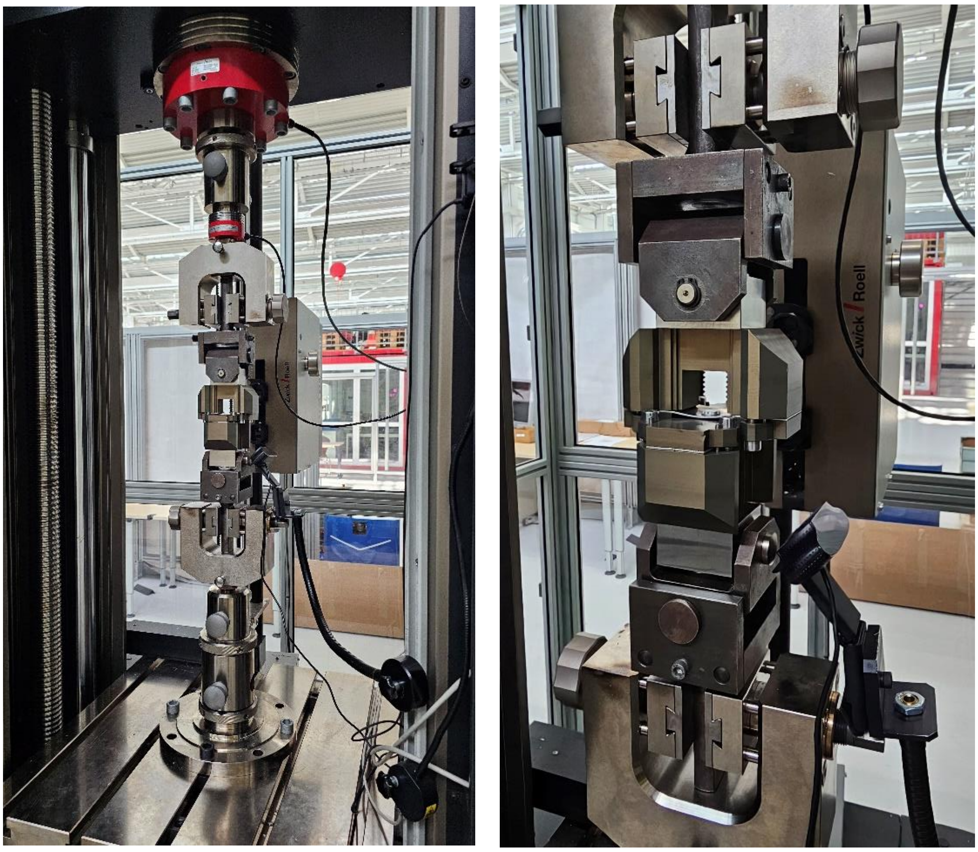

Fig.6: Testing of quasi-static head tensile tests shortly before failure of the joint

2.3 Simulation models

To efficiently optimize the material card parameter values, the simulation models must reproduce the real test setups with sufficient accuracy without being too computationally expensive. Therefore, various levels of simplifications of the experimental setup were made and analyzed to ensure a good balance between simulation quality and computation time [7]. The different simplification levels aren't described in this paper. As with the test setups for the shear and head tension tests, the simulation models for both types of loading are similar in principle (Fig.7:). The modeling in this paper is carried out for the FE explicit-solver LS-DYNA R12.1 MPP Double Precision.

The joining partners (metal sheets) are modeled using shell elements. The material cards of the joining partners are of type ***MAT_024** (***MAT_PIECEWISE_LINEAR_PLASTICITY**) and were calibrated and validated in advance. The joint can be modeled element-based or constraint-based [8,9], depending on the user's preference. However, this paper will be focused on element-based connection modeling. A beam element or a solid element is available for selection. Element formulation 9 is used for the beam element and element formulation 1 for the solid element. Additionally, material type ***MAT_100** (***MAT_SPOTWELD**) or ***MAT_100_DA** (***MAT_SPOTWELD_DAIMLER_CHRYSLER**) can be selected as the material description, especially as they can represent strain rate-dependent behavior. If ***MAT_100_DA** is used, the card ***DEFINE_CONNECTION_PROPERTIES** must also be used. The contact modeling between the fastener and the two joining partners is carried out for ***MAT_100** using ***CONTACT_SPOTWELD** and for ***MAT_100_DA** using ***CONTACT_TIED_SURFACE_TO_SURFACE**.

The components of the front clamping as separate ***PARTs** are not necessary and are modeled as a ***CONSTRAINT_NODAL_RIGID_BODY** (NRB). All masses, mass inertia effects and centers of mass are already taken into account in the movement data recorded by the acceleration sensor on the front clamping. Using ***CONSTRAINED_EXTRA_NODE**, a node is connected to the NRB of the front clamping at point of impact of the pendulum head on the impact surface of the front clamping. A ***BOUNDARY_PRESCIRBED_MOTION** is applied to this node, utilizing the velocity-curve response extracted from the physical test. The data of the velocity curves are calculated from the data of the acceleration sensor. A simulation model is set up for each test speed. For this purpose, an average velocity curve is generated from each valid test series.

The two components of the rear clamping are also replaced by a ***CONSTRAINED_NODAL_RIGID_BODY**. All translational and rotational degrees of freedom, with the exception of the translational degree of freedom in the direction of movement of the pendulum arm during impact, are constrained. A beam element is connected at the end of the NRB. The other node of the beam element is located at the height of the force sensor of the test setup. A ***BOUNDARY_SPC_NODE** is set on this node, in which all translational and rotational degrees of freedom are constrained. This allows the force to be recorded in the simulation and compared to the real test for purposes of verification.

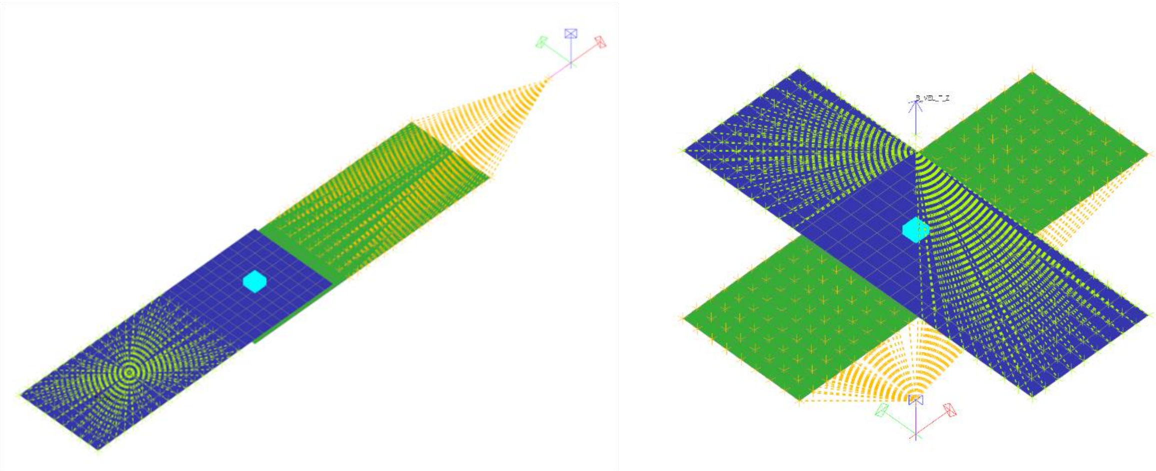

Fig.7: Simulation model shear tensile test (left) and head tensile test (right)

2.4 Simulation and optimization process

The software VALIMAT [4] must be used to carry out the dynamic tests on the IMPETUS®. This is also used to record the test responses. There is a database system in the background into which the data from the quasi-static tests can be integrated. At the beginning of the simulation and optimization process chain (Fig.8:), the test specimen and recorded data is automatically extracted from the database and the test folders and is prepared for the process chain. Three input files are automatically generated. The master input file holds key information of the model set-up and optimisation variables, while the other two input files contain information of the joining partners and joint separately. An initial simulation is carried out with these files to test whether the simulation is working correctly. One model is generated for the head tensile tests and one for the shear tensile tests.

The parameters that can be controlled by the optimizer are integrated in the master input file. These contain the individual parameters of the material card of the joint. For the material card type ***MAT_100**, these would include the modulus of elasticity, yield stress, plastic hardening modulus and various parameters that define the failure. An optimizer called CS-OPT, developed at DLR e.V., is used to perform a DOE, create metamodels, optimize and perform a validation simulation. To ensure that the following optimizer can find a result in an acceptable time, the limits of the parameter values, the starting values and the step size of these must be chosen carefully. A DOE is performed with this information. For each simulation carried out in the DOE, the distance between the real force measurement curve of the rear clamping and the force curve from the simulation is calculated. A metamodel is generated using the input parameter values and the distances between the curves. The optimization process attempts to minimise the difference between the simulated and real results by using the learned responses of the metamodel to converge to a solution of optimum values for the design variables. Once the optimizer has found a suitable combination of parameter values, a validation simulation is carried out with these values and the metamodel is validated. If they match, the parameter values of the validation simulation are the output in the form of a material card.

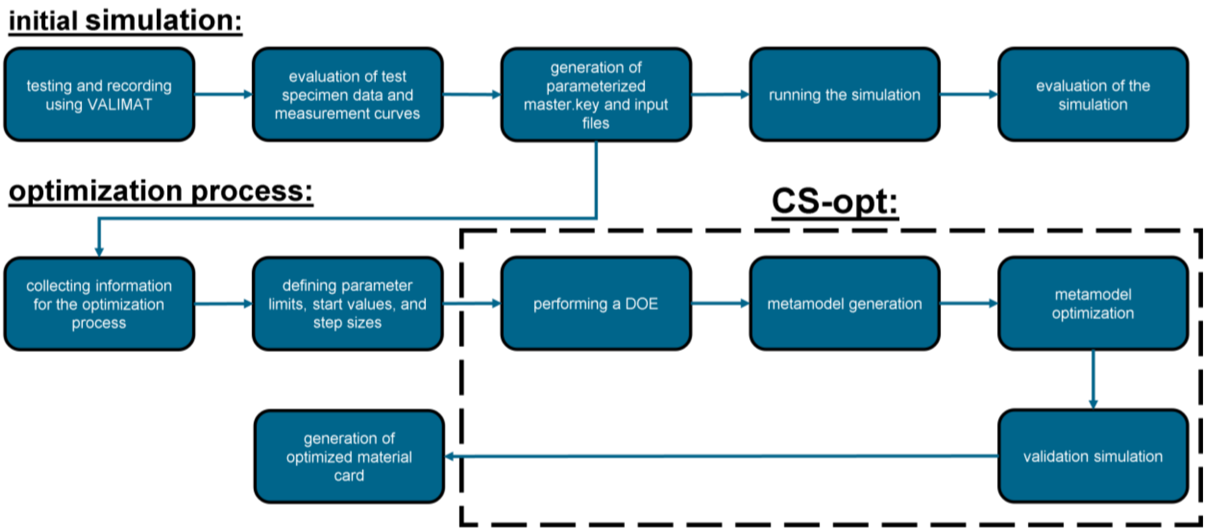

Fig.8: Schematic representation of the simulation and optimization process chain

3 Results and Discussion

Notable findings were first obtained from real tests, particularly with regard to the behavior of joints under dynamic loads. In addition, key knowledge regarding the robustness, mesh (in)dependency of the models and element formulation effects were obtained. The learned lessons support the development of the automated process chain that has not yet been finalized. For example, the combination and joint optimization of shear tension and head tension simulations - with the aim of creating a material card that accurately represents both load cases simultaneously - has not yet been implemented. Therefore, a complete, calibrated material card cannot be presented in this paper. Instead, selected results of the real tests and individual results from simulation comparisons are presented, which form the basis for the further development of the process chain. These findings underline the need for further optimization and integration of the individual steps in the calibration process.

3.1 Lessons learned from the tests

Various joining partner configurations with different material thicknesses and materials were used to investigate the dynamic behavior of joints: 1-mm-thick EN-AW-5083-O aluminum sheets, 2-mm-thick EN-AW-5083-O aluminum sheets and 0.8-mm-thick DX56 steel sheets. The joining technologies included clinch joints, semi-tubular self-pierce rivets joints and flow drilling screw joints. The load types were defined as shear tension and head tension to cover a wide range of mechanical requirements.

Only selected results from shear tensile tests (Fig.9:) are presented in this paper, whereby six tests were carried out per impact velocity. Mean value curves were determined from the individual results and displayed graphically. A detailed interpretation of the results is purposely omitted, as the focus is on demonstrating the significance of the dynamic behavior of materials and joints. The results underline that the consideration of velocity effects and non-linear material behavior is crucial, especially for the accurate simulation of crash scenarios.

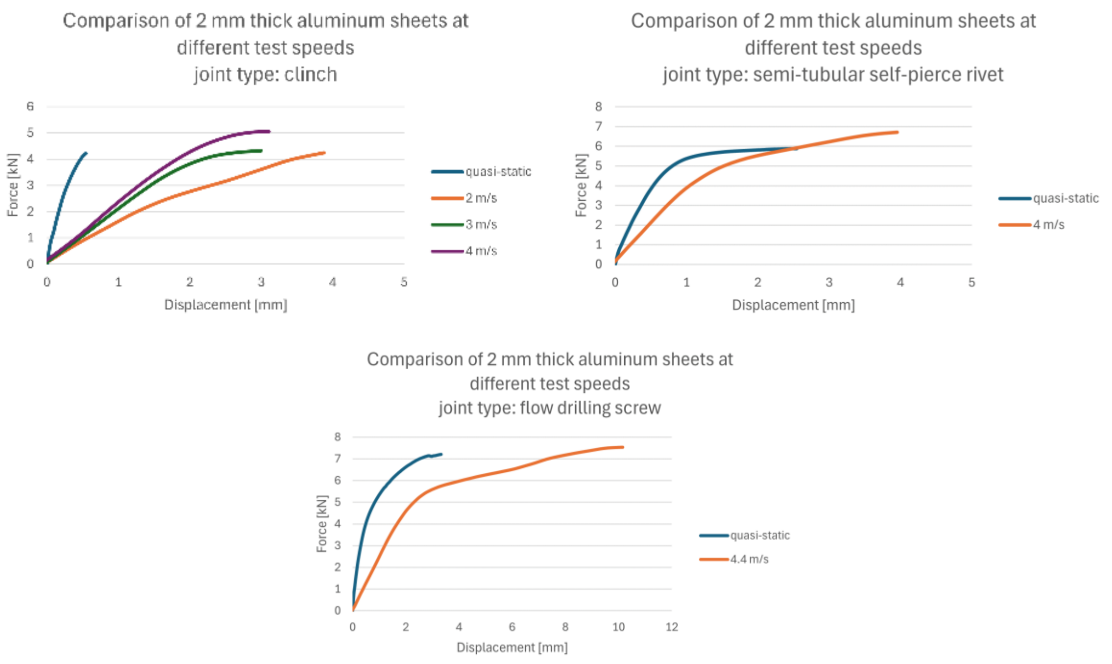

Fig.9: Selected shear tensile test results of joined 2mm thick aluminum sheet clinch joint semi-tubular self-pierce rivet joint; flow drilling screw joint

The low bending stiffness of the test specimen in the head tension test results in strong vibrational response of the specimen [10]. Therefore, the impact of the pendulum head on the impact surface of the clamping must be damped in order to ensure a stable and measurable load situation. Specific tests are required for each joint in order to optimally adjust the damping. Without sufficient damping, excessive vibrations (Fig.10:) occur, which distort the measurement signal and make it difficult to precisely adjust the simulation. The vibration amplitude can be reduced by using rubber as a damping element. However, it must be noted that the remaining energy used to destroy the joint decreases with increasing thickness. Furthermore, the vibrational effects arise below a specific threshold of the damping element thickness, highlighting the importance of current damping strategies for each test configuration. Identifying the correct damping and filter of the measurement signal (Fig.11:) are necessary to enable reliable coupling between experimental data and numerical simulations to ensure a valid calibration of the material cards.

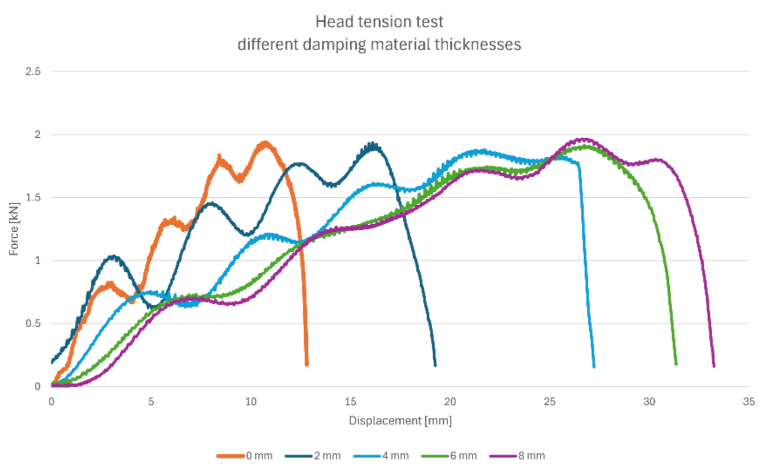

Fig.10: Head tensile test results of 1 mm thick aluminum sheet with different damping material thicknesses joined with semi-tubular self-pierce rivets

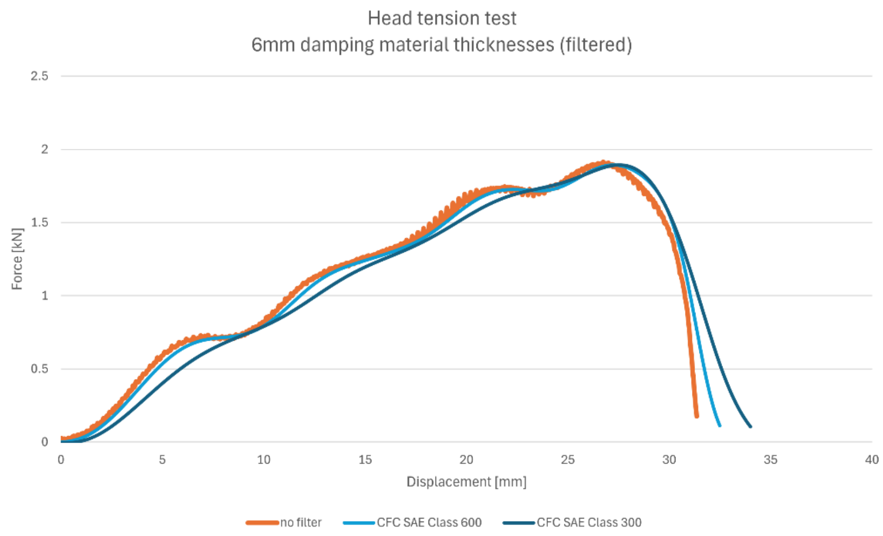

Fig.11: Head tensile test results of 1mm thick aluminum sheet with 6mm damping material thickness joined with semi-tubular self-pierce rivets; unfiltered and filtered measurement curves

3.2 Lessons learned from the simulations

This section presents the key findings from the simulations carried out with different element formulations and meshing, which provide information on the robustness, mesh independence and efficiency of the modeling of joints.

The connection of the beam element to the FE mesh shows a strong dependency on the position of the nodes of the joined plates. There are clear differences depending on whether the beam element is connected in a node of the joining partner mesh or in the middle of an element (Fig.12:). The position of the attachment point in particular has a significant influence on the resulting behavior, as the load transfer and the local stress distributions change. In comparison, the difference between a connection at the upper joining partner in the node and at the lower joining partner in the center of the element - or vice versa - is negligible.

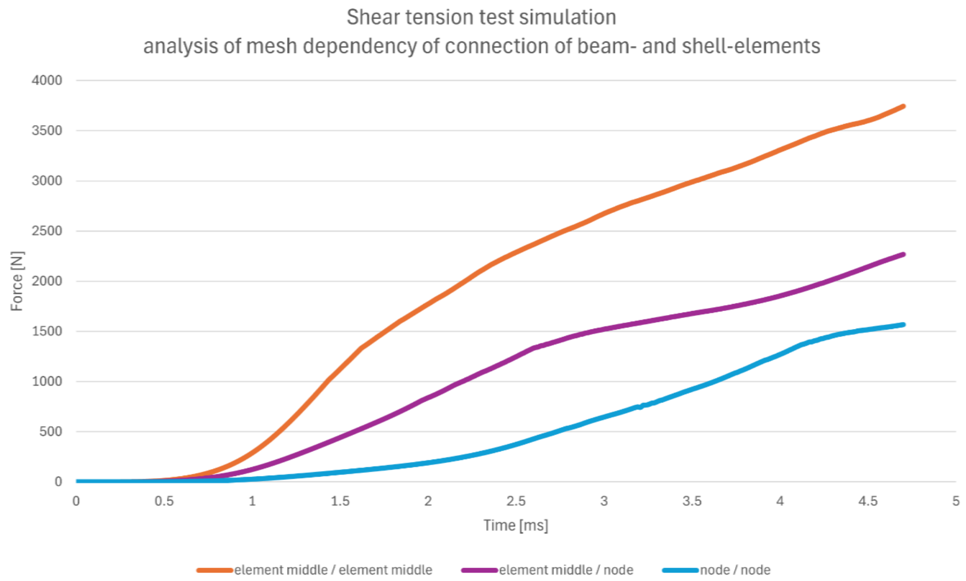

Fig.12: Comparison of the mesh dependency of a spotweld beam element

The dependency of the connection for punctiform joints modeled as a solid element is significantly lower compared to beam element-based approaches (Fig.13:). The volumetric representation of the weld spot in the FE mesh makes the load transfer more uniform and less sensitive to the exact position of the point of application in the mesh. The position within the joining partner element therefore only has a minor influence on the overall behavior. This leads to more stable and reproducible results, regardless of whether the solid element is located at a node or in the center of an element. Modeling as a solid element thus reduces the mesh dependency and improves the robustness of the simulation, especially for complex joint configurations.

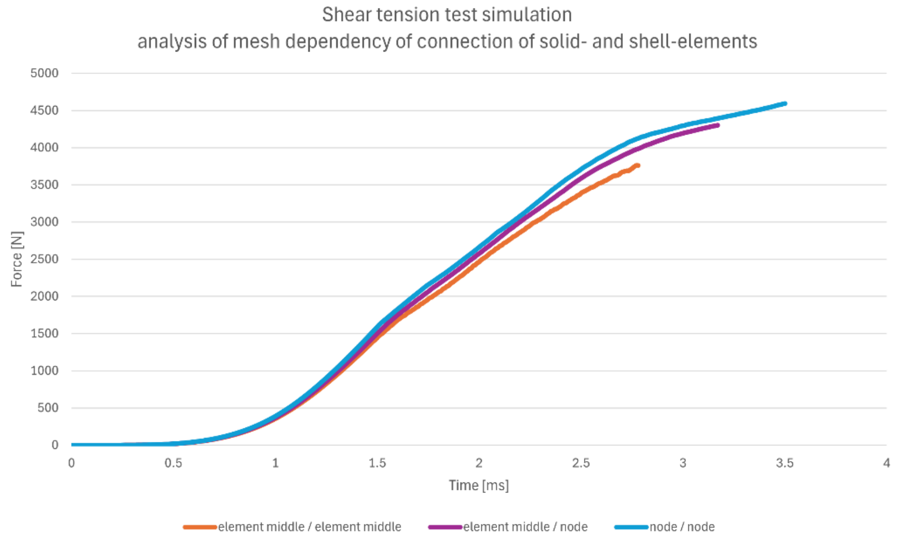

Fig.13: Comparison of the mesh dependency of a spotweld solid element

The difference between the element formulations ELFORM 2 and ELFORM 16 (or -16) in the modeling of punctiform joints as solid elements is comparatively small (Fig.14:) with regard to the mechanical behavior and the accuracy of the results. The load reactions, stresses and failure curves show only minor deviations between the two formulations. Nevertheless, there is a clear difference in the simulation time: ELFORM 2 leads to significantly shorter calculation times compared to ELFORM 16. This is due to the different numerical stability and the computational load of the respective formulation [11], with ELFORM 16 requiring a higher computational effort due to its higher accuracy and stability in the case of severe deformations. The choice of formulation should therefore be based on a compromise between accuracy and computation time.

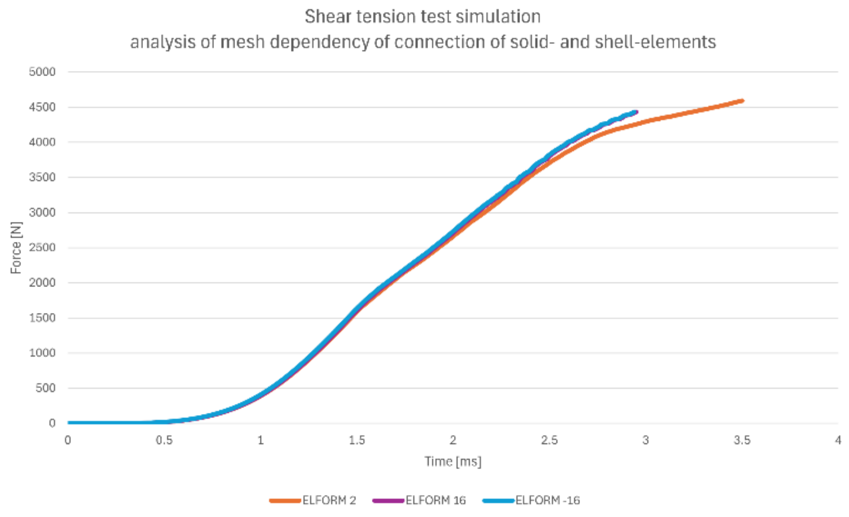

Fig.14: Comparison of the shell element formulation of the joining partners of a spotweld solid element

4 Conclusion and Outlook

In this paper, a process chain for the simplified and automated calibration of material cards for punctiform and planar joints was developed and described in detail. The test setups for dynamic and quasi-static tests, in particular shear tensile and head tensile load cases, were realized and documented. Specific clamping concepts were developed for the test machines to ensure reproducible and physically sensible load transfer. Corresponding simulation models were created for both test configurations which simulated the experimental conditions. Significant findings were obtained from the tests and simulations, firstly, a clear difference between the quasi-static and dynamic behavior of the joints was demonstrated experimentally. Secondly, a significant mesh dependency of the modeling was shown, both with regard to the meshing structure and the element formulation used. These findings underline the necessity of careful modeling and the consideration of dynamic effects in the material card calibration. The presented process chain represents an important step towards a more efficient and robust simulation of joints in the development of vehicle concepts.

Future work will concentrate on the optimization and completeness of the automated process chain. A central focus will be on developing a reliable approach for determining suitable initial values - either through a practicable method or by designing an algorithm that enables both a sensible initial value specification and a realistic estimation of the parameter limits. In addition, the definition of optimal simulation parameters, such as the appropriate element formulation (ELFORM), the mesh size the degree of detail in the area of the joint, is important. It is necessary to check whether fine meshing is required over the entire component or whether local fine meshing around the joining point is sufficient. The integration of peel tensile tests into the process chain will also make it possible to increase the robustness of the material cards under realistic dynamic conditions. Another key step is the combination of shear tensile, head tensile and peel tensile load cases in a joint optimization in order to generate a comprehensive material card that adequately represents several failure mechanisms. Finally, the entire process chain is to be validated on an additional component in order to demonstrate its transferability and applicability in the real development process.

5 Literature

[1] Pischinger, S.; Seiffert, U. (2016): Vieweg Handbuch Kraftfahrzeugtechnik. 8., aktualisierte und erweiterte Auflage, Springer Vieweg, pp. 598 ff. DOI: 10.1007/978-3-658-25557-2, Wiesbaden/Germany

[2] Altvater, S.; Sikora, S; Siefkes, T. (2022): Transition between flow-drill screwing systems considering joining process and joint characteristics. In: Advances in Industrial and Manufacturing Engineering, Vol. 5, DOI: 10.1016/j.aime.2022.100091; Stuttgart/Germany

[3] Sturm, R. (2023): AUTOMATED MATERIAL CARD DEVELOPMENT FOR THE CRASH DESIGN OF COMPOSITE VEHICLE STRUCTURES. At: 20th International Conference on Experimental Mechanics; Porto/Portugal

[4] Sturm, R. et. al. (2017): Optimization Strategies for Crash Relevant Vehicle Structures. At: 6th International Conference on Integrity-Reliability-Failure; Lisbon/Portugal

[5] 4a Engineering GmbH: Impetus. Online: https://www.4aengineering.at/4a-impetus, last check 30.09.2025

[6] AWS D8.9M:2012, 2012: Test Methods for Evaluating the Resistance Spot Welding Behavior of Automotive Sheet Steel Materials.

[7] Livermore Software Technology (ed.) (2020): LS-DYNA Keyword User’s Manual. https://www.dynasupport.com/manuals/ls-dyna-manuals/ls-dyna_manual_volume_i_r12.pdf, last check 30.09.2025

[8] Sønstabø, Johan Kolstø; Morin, David; Langseth, Magnus (2016): Macroscopic modelling of flow-drill screw connections in thin-walled aluminium structures. In: Thin-Walled Structures 105, pp. 185–206. DOI: 10.1016/j.tws.2016.04.013

[9] Hanssen, A. G.; Olovsson, L.; Porcaro, R.; Langseth, M. (2010): A largescale finite element point-connector model for self-piercing rivet connections. In: European Journal of Mechanics - A/Solids 29 (4), pp. 484–495. DOI: 10.1016/j.euromechsol.2010.02.010

[10] DYNAmore GmbH (ed.) in cooperation with Seeger, F; et. al. (2005): An Investigation on Spot Weld Modelling for Crash Simulation with LS-DYNA. 4. LS-DYNA Anwenderforum; Bamberg/Germany

[11] Wirtz, Tim (2024): Development of an automated process for material card calibration of Punctiform and Planar joints for explicit FE-simulations. 4a Technology Days 2024, 2024-03-11 - 2024-03-13; Schladming/Austria.