Phenomenological driven Modeling of Joints

Matthias Bier[1], Herbert Klamser[1], Andre Haufe[2]

[1]Dr. Ing. h.c. F. Porsche AG, Weissach, Germany

[2] DYNAmore GmbH, Stuttgart, Germany

Abstract:

In the construction of automobiles different technologies are used to join sheets. Today the most common method for connections is resistant spot welding. There are thousands of spot welds in body-in-whites, which are determining the behavior of the structure under crash conditions. New research is striving after replacing these thermal joints by adhesives or rivets for optimization of the production process. It is essential to ensure a cost saving and time optimized car development to reduce the required number of experiments by using precise simulations. For the quality of such simulation models accurate reproduction of the mechanical joint behavior is necessary. Because of the fact that the element size is bound by the time step in explicit finite elements schemes, a detailed model for the joints is not applicable in typical body-in-white simulations under crash conditions. For this reason simplified models using beam or solid elements have to be used to represent the connection.

In the majority of load cases current modeling strategies do not show the required accuracy needed for design decisions. Therefore new ways of spot weld modeling and approaches for rivets modeling should be investigated. For spot welds several strategies of modeling exist. They all are based on the effort to reproduce the spot weld behavior by using specially adopted constitutive or structural models. For these models parameters have to be determined by comparing the maximum load under tensile, shear and peel conditions with corresponding numerical investigations. In the present paper it will be shown that a model with a relative small number of elements driven by a rigorous phenomenological approach can achieve better results. The quality of the proposed model will be evaluated by comparison of tests with KS2-specimen.

In the field of self-piercing rivets an established modeling technique doesn't exist. In this paper firstly capabilities of several modeling techniques will be investigated and secondly they will be compared with a model based on the aforementioned approach.

Keywords:

spot weld, constrained contact, spr, self-piercing rivet

1. Introduction

The quality demands of automotive crash test simulations are increasing continuously. The intention of this numerical simulation is an accurate prediction of the structural behavior. Especially the possibility to study different car configurations in early stages of development is an advantage of simulations. The failure behavior of the joints is decisive for the behavior of the entire structure under crash conditions. As it is not possible to use a detailed model of the joints in complete vehicle simulations, they must be replaced by simplified models. Actual modeling techniques are based on theoretical description of the transferred forces and moments. Such models reach their limits of capabilities. Therefor a new approach will be pursued, to represent the coarse structure and hence the occurring effects of point-shaped joints through a matching model. The intention is to achieve an improved reproduction of the joint's force level and failure modes. In the following paper two different joints will be investigated, first resistance spot welds and second joints with a semi tubular self-piercing rivet.

2. Failure of Joints

For a correct simulation of point-shaped joints under crash-conditions it is necessary to know their failure behavior. Therefore we have to distinguish between spot welds and semi tubular self-piercing rivets. For spot welds generally two different failure modes, namely interfacial and pull-out failure, exist which can regularly be observed in experiments. In case of failure by pull-out the fracture appears not in the region of the weld nugget, which represents the joint, but in the base material around the spot weld. The crack opens stepwise around the spot weld, which leads to finally to failure of the joint. After fracture at least in one of the joining partners damage remains in form of a void.

Contrary to the aforementioned failure mode an interfacial failure mode exists. In this mode the basic material around the spot weld is not impaired. The failure of the spot weld occurs in the region between the joined sheets and leads to disconnection.

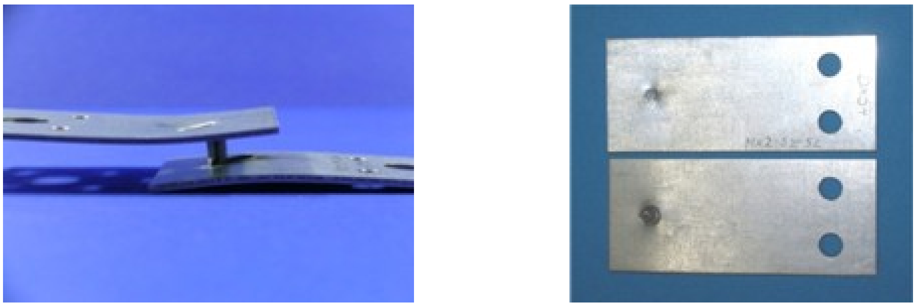

Figure 1: pull-out fracture (left) and interfacial fracture (right) of spot welds [1]

Which of the aforementioned failure mode eventually occurs depends on various parameters. They include, beside the loading scenario, the material type, the thickness of both metal sheets and the diameter of the spot weld. A pull-out fracture occurs typically on thick sheets with small spot weld diameters and an interfacial fracture on thinner sheets with large spot weld diameters.

Figure 2: punch side failure (left and matrix side failure (right) of semi tubular self-piercing rivet joints [2]

Regarding the semi tubular self-piercing rivet joints also two fundamental different failure modes can be observed. On the one hand failure may occur on the punch side, respectively on the side of the rivet head, and on the other hand failure may occur on the matrix side. Both failure modes are usually of pull out type.

3. Modeling techniques

In the following different models for the simulation of spot welds and joints with semi tubular self-piercing rivets will be presented and eventually the predictive quality of these models will be elaborated by comparing simulation results with corresponding experiments.

3.1. Spot weld models - Constrained-Modeling

On the basis of the knowledge of different failure modes a model for the simulation of spot weld joints shall be derived. The objective is different from the majority of conventional modeling methods since the failure model will take interfacial and pull-out fracture into account separately.

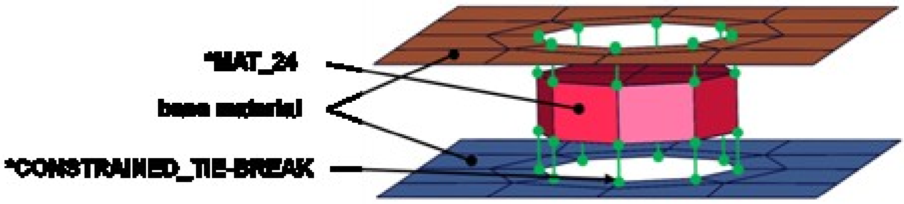

Figure 3 Constrained-Modeling

The model consists on the one hand of a mesh for the base material which is adapted to the spot weld and on the other hand of four hexahedron-elements that are modeled with *MAT_PIECEWISE_LINEAR_PLASTICITY (*MAT_24) [3] that represent the spot weld. The diameter of the four hexahedron-elements is identical to the diameter of the spot weld. The joint between the hexahedron-elements and the base material of the metal sheet is defined by a constrained contact (i.e. *CONSTRAINED_TIE_BREAK [3]). These contact definitions couple two nodes of identical coordinates, namely a node of the hexahedrons and the respective base material. However, the possibility exists to loosen the joint connection via a failure criteria in the contact definition. By means of this a more realistic modeling could be achieved such, that an interfacial fracture by the hexahedrons and a pull-out fracture by a stepwise loosening of the constraint conditions can be represented.

Calibration is possible by means of single spot weld samples (e.g. KSII samples) of the connection of interest. For the presented model three parameter definitions are critical:

- $\sigma\ \varepsilon_{pl}$ — *MAT_24 — interfacial fracture
- ε^F_{pl} — *MAT_24 — interfacial fracture
- ε^F_{eff} — *CONSTRAINED_TIE_BREAK — pull-out

Here $\sigma\ \varepsilon_{pl}$ defines the stress-strain behavior of the hexahedron-elements and it is defined such, that the force-displacement path of a single spot weld sample showing interfacial fracture is well matched with the corresponding simulation. ε^F_{pl} represents the maximum plastic strain that can be defined in *MAT_24 when the hexahedrons shall be deleted.

Calibrating of pull-out failure is done only with ε^F_{eff} defined in *CONSTRAINED_TIE_BREAK. ε^F_{eff} represents the maximum value of the effective plastic strain in the neighboring shell elements. In case the given value will be exceeded the connection between the corresponding tied nodes will be deleted. ε^F_{eff} is chosen in such a way that the starting point of the pull-out and therefore the maximum force coincide in both the simulation and the experiment.

In case no tests for the adaption of the parameters are available, an engineering approach to derive a formula can be used as a first guess:

$$\sigma\ \varepsilon_{pl} = \frac{t_n}{t_n}\ \sigma\ \varepsilon_{pl} \qquad n = 1{,}2 \qquad (1)$$

$$\varepsilon^F_{eff,n} = \frac{d}{d}\ \frac{t_n}{t_n}^{2}\ \varepsilon^F_{eff,n} \qquad n = 1{,}2 \qquad (2)$$

The corresponding yield curve will be scaled from an available curve $\sigma\ \varepsilon_{pl}$ of a calibrated connection with the help of the sheet thicknesses t_n and t_n (see eqn. (1)) The same holds for the failure strain $\varepsilon^F_{eff,n}$. Here again the corresponding sheet thicknesses t_n and t_n as well as the spot weld diameters d and d are used for scaling (see eqn. (2)).

Clearly, in order to match the respective experiments as good as possible the weld node diameter and correspondingly the other parameters have to be derived as exact as possible.

3.2. Semi tubular self-piercing rivet models

The adjustment of all model parameters for the rivet joints is done with the help of KS-2 specimen test data. In the following a short overview of the examined modeling techniques and the applied constitutive models is given.

3.2.1. *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE (*MAT_240)

The first model is created by means of *MAT_240 and cohesive solid elements. This constitutive model is based on the theory of cohesive zones and was developed by Marzi et al. [4]. The mesh of the base material consists of regularly meshed shell-elements and the actual rivet joint is build up by *MAT_240 and one single hexahedron connected to the base material by contact definition (see Figure 4).

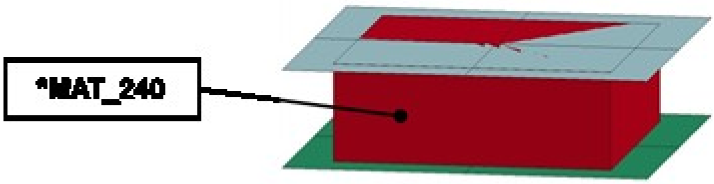

*Figure 1: Modeling with *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE*

3.2.2. *CONSTRAINED_INTERPOLATION_SPOTWELD

The modeling with *CONSTRAINED_INTERPOLATION_SPOTWELD [3] can be characterized as part of the element free connection methods. Hence, no element is defined that represents the rivet joint. Instead one single node is specified which determines the position of the joint and identifies the neighboring nodes of the sheets within a defined radius. I.e. between the nodes of the sheets around this point wise defined position, forces and moments are transferred by the constrained definition.

3.2.3. Rivet-Head-Modeling

The last presented modeling technique for semi tubular self-piercing rivets strives for the goal to build up the occurring failure modes by means of a coarse reproduction of the rivet geometry (rivet head, rivet foot) as done for the constrained-modeling for spot welds. In the context of a first study the model is only investigated for failure prediction on the punch-side.

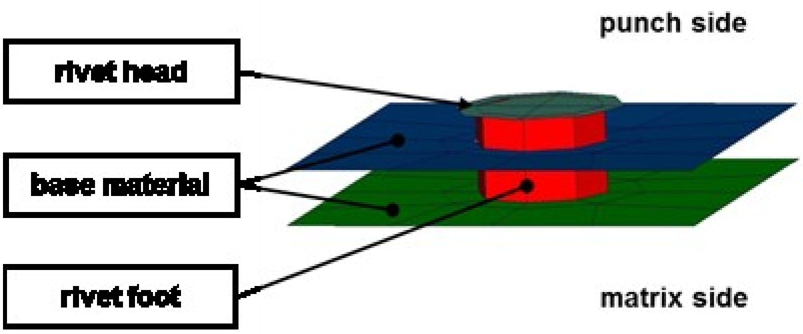

Figure 2: Rivet-Head-Modeling

The mesh of both flanges of the sheet metal is generated in the same way as for the constrained-modeling and is adjusted according to the rivet joint modeled by four solid elements. Hence failure is determined by deletion of this four central elements on the punch side of the connection. The rivet foot is modeled again by four hexahedrons and the rivet head by four additional shell-elements using *MAT_24 each. The connection between the rivet head and the rivet foot is made by tied contact definitions. Likewise is the rivet foot connected to the matrix side base material with another tied contact definition. A connection between the shell-elements of the base material and the rivet head doesn't exist. Here only the standard contact is applied and initial penetration of both parts suppressed.

4. Experimental Evaluation

KS-2 specimen and vehicle-oriented T-joint components are considered in this evaluation. Based on the KS-2 specimen the adjustment of the simplified model is carried out and the specimen are examined with respect to three different load scenarios, namely peeling, tensile and shear loading.

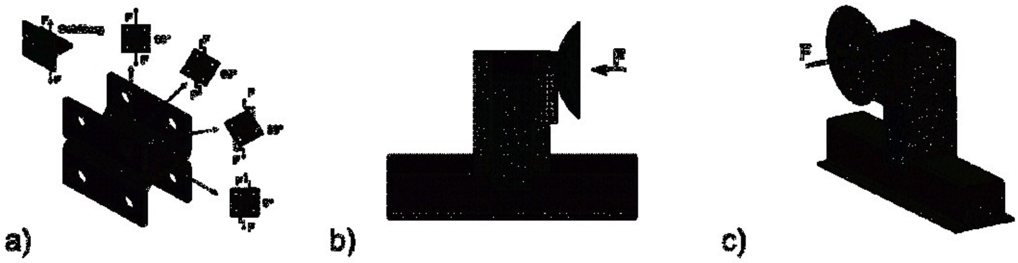

Figure 3: a) KS-2 specimen [5] b) T-joint quasi-static longitudinal loading c) quasi-static transverse load

4.1. Spot weld models - Constrained-Modeling

In a first step the calibration of the model based on the material HT600XD with a sheet thickness of 1.5 mm on both sides is examined. The maximum force and time of failure are sufficiently enough met under peel, tensile and shear load. The distinction, which failure mode occurs, is clearly recognizable in the simulation plot. Particularly in the case of peel load the stepwise separation of the constrained and thereby the pull out of the spot weld is well recognizable by the slump in applied forces after the reach of the maximum force.

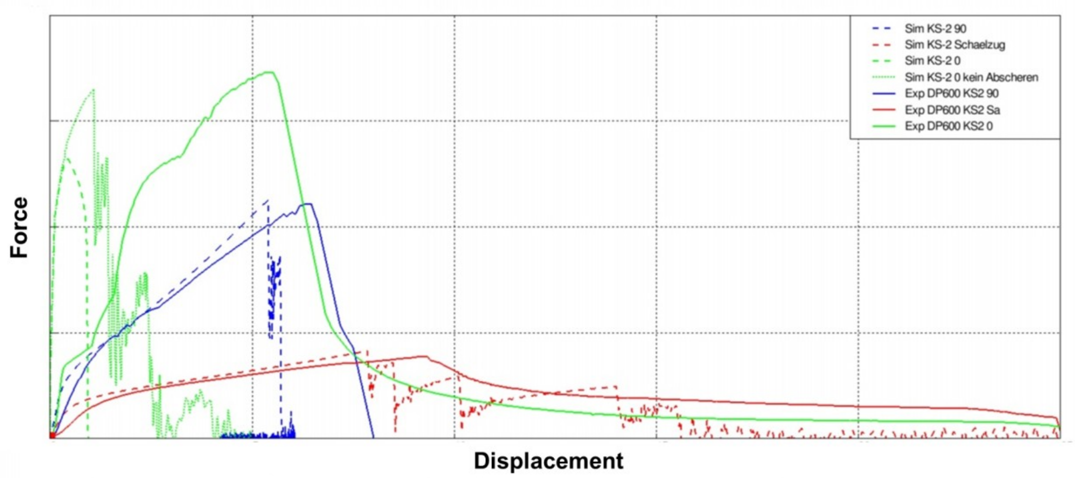

Figure 4: results of tests with KS-2 specimens (material HT600XD)

The parameters gained from the KS-2 specimen are transferred to the simulation of the T-joint experiments. Firstly, the case of quasi-static transverse loads will be examined. Thereby, the simulation follows the experimental results in the characteristic of the strength trend. The examination of the failure characteristics and the order of the failures of the nine individual spot welds relevant for the joint connection shows a good correlation of the simulation results with the experimental results (see Figure 8 and 9).

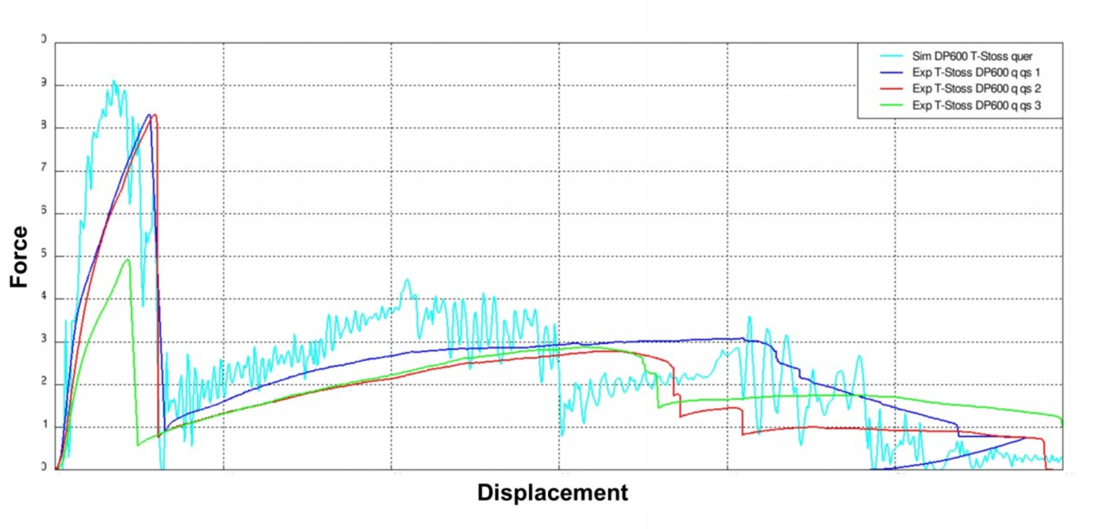

Figure 5: results of T-joint simulations under quasi-static transverse loads (material HT600XD)

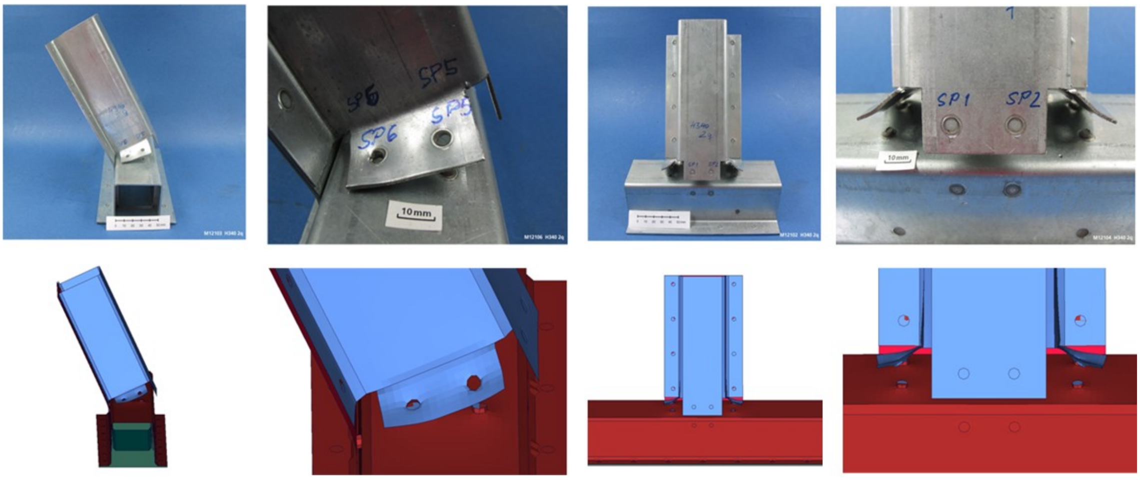

Figure 6: failure characteristics of T-joint simulations under quasi-static transverse loads (material HT600XD)

In addition an investigation on a non-symmetric KS2-speciemen, i.e. two different materials with different thicknesses, was carried out to check the predictability of the present approach. Here, the know parameters of the symmetrical pairing of HT600XD and DX54 was used to generate model parameters for the non-symmetric pairing HT600XD with DX54.

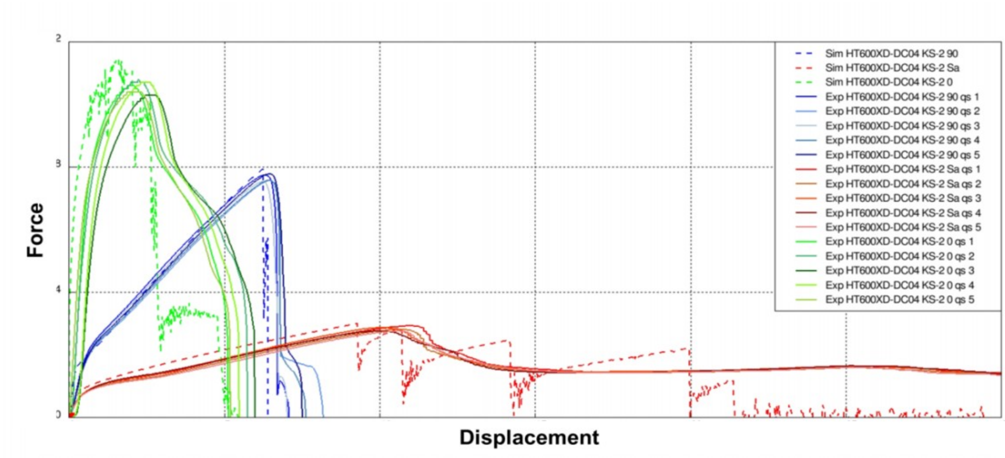

Figure 7: results of tests with KS-2 specimens (material HT600XD-DX54)

It can be seen from Figure 10 that the result is surprisingly accurate and fuels the hope that with present method parameter identification of non-symmetric spot weld connections may be simplified in future. It should be mentioned though, that this result has to be confirmed with other material and sheet thickness combinations in further projects.

4.2. Semi tubular self-piercing rivet models

The proposed connection models are examined for joining of two sheets metals. Here the punch-side was having a sheet thickness of 1.0 mm and the matrix side 2.0 mm. The chosen material was AlMg3,5Mn. Two experiments witch KS-2 specimen were applied.

4.2.1. *MAT_240 and *CONSTRAINED_INTERPOLATION_SPOTWELD

The adaptation of both models shows a good reproduction of the overall strength level. Oscillations which take place in the *CONSTRAINED_INTERPOLATION_SPOTWELD modeling can be reduced by an additional optimization within the implementation in LS-DYNA. This will be the focus of further development. The modeling with *MAT_240 seems to be a good choice for the modeling of joints with semi tubular self-piercing rivets and is recommend due to its simple application and calibration procedure.

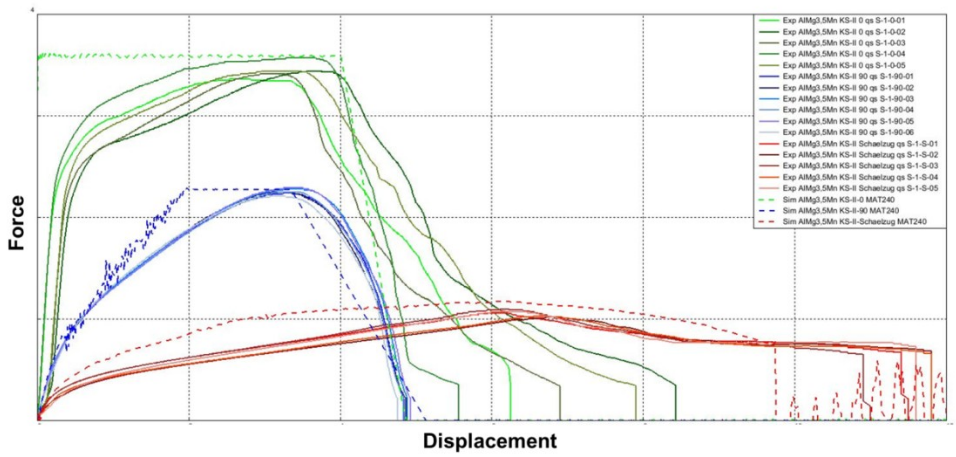

*Figure 8: results of tests with KS-2 specimens for semi tubular self-piercing rivets (*MAT_240)*

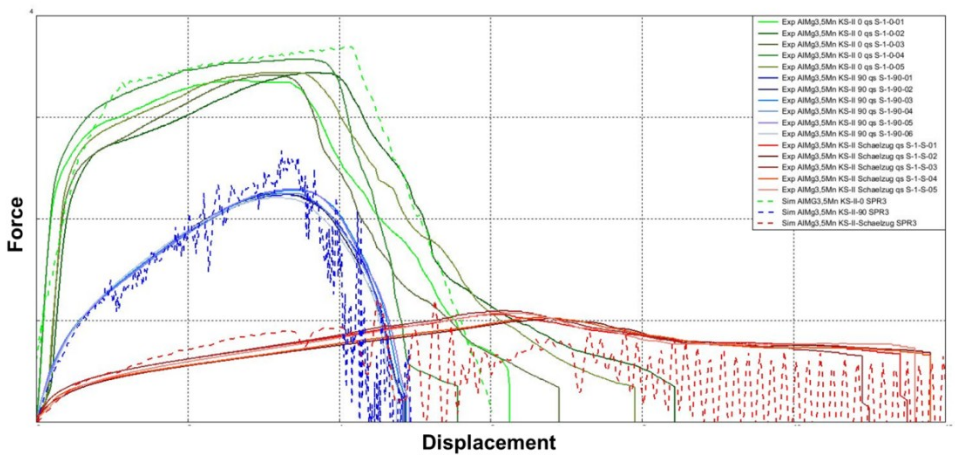

*Figure 9: results of tests with KS-2 specimens for semi tubular self-piercing rivets (*CONSTRAINED_INTERPOLATION_SPOTWELD)*

4.2.2. Rivet-Head-Modeling

The modeling via a coarse representation of the rivet structure leads to sufficiently good results. This is especially the case under tensile loading. Here the punch-side failure is represented in a very physical manner. Under peel and shear load additional cracking appears in the sheets. A reproduction of these cracks is not possible with the present modeling technique and would require failure criteria in the base sheet metal via additional failure criteria like the GISSMO or the Gurson model. This and other effects cause the strength level of the simulation to excess the strength level of the experiments.

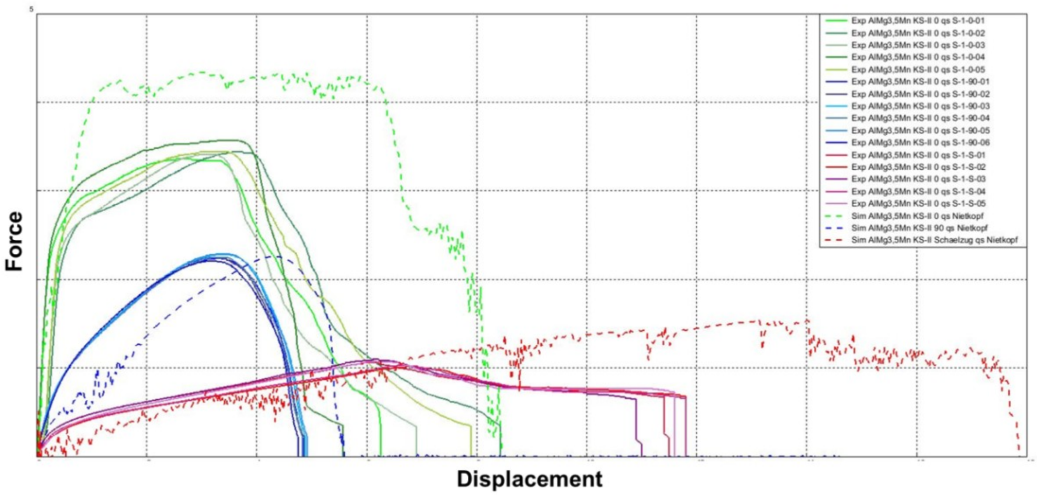

Figure 10: results of tests with KS-2 specimens for semi tubular self-piercing rivets (Rivet-Head-Modeling)

5. Conclusion and Outlook

The attempt to represent the failure modes of the present joints within a coarse modeling approach leads partially to excellent results. Thus, in the case of the simplified model of spot welds, good simulation results could be achieved by separating the reproduction of the pull-out failure and the interfacial failure via a so called constrained-modeling technique. Additionally, this type of modeling seems to offer the possibility to derive simulation parameters for varying combinations of spot weld connections (i.e. having different flange material and thicknesses partners) from only a few experiments on symmetrical connections. This conclusion has to be validated on further connection pairings, though.

Because of the more complex effects which influence the behavior of the joint, the implementation of a semi tubular self-piercing rivet joint in a similar manner induces additional effort. If the model is restricted to a punch-side failure mode, good basic results could also be observed. Further development related to the matrix-side failure and an improvement of the representation of the pull-out behavior has to be performed to obtain an over-all adequate model.

The experience with the implementation of the failure models induces that further effort in the search of simple and fast parameter fitting methods has to be. Additionally, the accounting of local effects via more detailed models of joints in a coarse model of the complete vehicle without violating the time step criteria by reduced element sizes leads to further challenges.

6. Acknowledgment

The experimental data used in the present study was taken from a research project funded by the FAT (Forschungsvereinigung Automobiltechnik), Germany [6]. The support of FAT is hereby greatly appreciated.

7. Literature

[1] Sommer, S.: Modellierung des Verformungs- und Versagensverhaltens von Punktschweiß-verbindungen unter monoton ansteigender Belastung, Fakultät für Maschinenbau der Universität Karlsruhe, Diss., August 2009

[2] R. Porcaro, A.G. Hanssen, A. Aalberg and M. Langseth: The behavior of a self-piercing riveted connection under quasi-static loading conditions. International Journal of Solids and Structures, Vol. 43/17, pp. 5110-5131 (2006)

[3] Livermore Software Technology Corporation (LSTC) (Hrsg.): LS-Dyna Keyword User's Manual. Version 971. Livermore Software Technology Corporation (LSTC), May 2007

[4] Marzi, S. ; Hesebeck o. ; Brede, M. ; Kleiner, F.: A Rate-Dependent, Elasto-Plastic Cohesive Zone Mixed-Mode Model for Crash Analysis of Adhesively Bonded Joints. In: Tagungsband der 7th European LS-DYNA Conference. Salzburg : DYNAmore GmbH, 2009

[5] Hahn, O. ; Wiling, M. ; Klokkers, F.: Ermittlung wahrer Kennwerte für geschraubte und stanzgenietete Blechverbindungen unter schlagartiger Belastung / Europäische Forschungsgesellschaft für Blechverarbeitung e.V. 2009 (EFB-Forschungsbericht Nr. 297).

[6] Oter, M. ; Ozdem, K.: Experimentelle Bestimmung und rechnerische Vorhersage des Tragverhaltens punktgeschweißter Bauteile aus Stahlblechverbindungen unter Crashbelastung mit Hilfe von Ingenieurkonzepten / Forschungsvereinigung Automobiltechnik e.V. (FAT). 2004 (FAT Schriftenreihe Nr. 186).