Enabling the *CONSTRAINED_INTERPOLATION_SPOTWELD (in detail SPR3) as a general-purpose fastening element

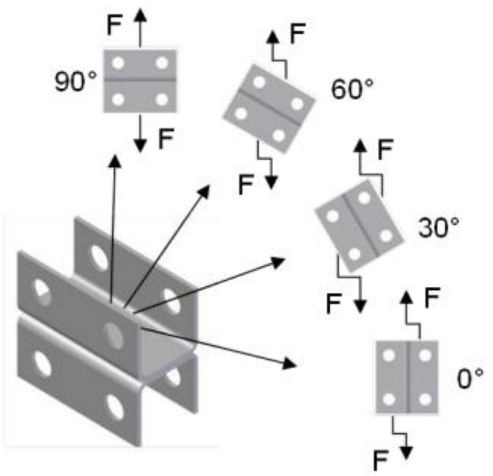
Michal Styrnik[1], Tobias Erhart[2]

[1]BMW AG
[2]Dynamore GmbH

It is known that the simulation of fastening elements can be carried out by using different approaches. One common way is the use a force-displacement based approach for single-point-connections. While enabling LS-DYNA to calculate crash simulations in our crash simulation tool chain it was necessary to make several adjustments to the standard **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** keyword to ensure that the currently available data can be used almost completely. The main improvements on the SPR3 element were accuracy features such as exponential decay, different stiffness factors for different loading conditions, the introduction of torsional moments, mixed-mode dependent hardening, the ability to connect with beam elements and the possibility to switch between different coordinate systems while calculating the rotational behavior of the element. Additionally, we were able to improve the overall stability of the penalty formulation by introducing a new approach to distribute forces and moments on arbitrary meshes, by monitoring the deletion of parent elements and by checking if a proper geometry is defined to proceed with the calculation (**`*CONTROL_CONSTRAINED`**). Considering the evaluation of the elements we introduced the "SPR state variable" to make the whole "life cycle" of a spotweld visible to the user. Due to all these measures the prediction quality of this approach has improved significantly. In order to add the possibility to define the material independently of the element **`*MAT_265`** was introduced. As future challenges we would also like to implement the proper identification of peel moments to switch between different degradation behaviors. And we would like to include the possibility to couple SPR elements properly with solid meshed parts.

1 Introduction

In the last years many different approaches to simulate fastening elements in crash applications have been developed. One promising approach is a force-displacement based approach, which uses specific tests that employ multiple loading directions on fastening elements (e.g., figure 1). From these tests one can identify mechanical properties of the tested connections on a macroscopic scale. This also provides the opportunity to capture several phenomena as a mean or a most probable force-displacement curve. And in the end an efficient calibration process can be applied where the focus lies on the most important features for crash modelling: the proper response and energy consumption of fastening elements.



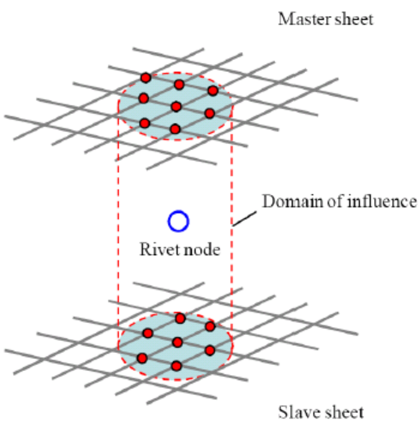


Fig.1: KS-2 test specimen.

*Fig.2: Schematic representation of the **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** element.*

The data from these tests is analyzed to extract the specific stiffness, the maximum forces, and the energy consumption of the element during plastic motion and failure. One possible element in LS-DYNA that could be used to follow this approach would be the ***CONSTRAINED_INTERPOLATION_SPOTWELD**.

This element employs a master/slave penalty constraint on nodes in a defined radius around one node, that represents the element, between two parts. The material response is then calculated on the relative displacements and rotations of the contributing master and slave nodes. This is displayed in figure 2.

To make full use of the data these tests provide and to improve the overall accuracy of the prediction it was necessary to make several adjustments to ***CONSTRAINED_INTERPOLATION_SPOTWELD**. The most important new features are:

- dedicated stiffness factors for different loading conditions
- the introduction of torsional moments
- exponential decay
- fastening of beam elements
- mode mix ratio dependent hardening
- the possibility of switching between different coordinate systems while calculating the rotational behavior of the element
- the introduction of ***MAT_CONSTRAINED_SPR3/MAT_265**

In LS-DYNA R12 further features are planned:

- fastening of solid elements
- identification of peeling and introduction of peel mode specific critical forces and failure behavior
- Use of part sets instead of parts as fastening entities

During the benchmark phase of the solver some stability issues were observed. Due to these findings the formulation to distribute the penalty forces and especially moments was changed. This includes, e.g., the splitting of moments into force pairs on the corresponding nodes. This also secures a stable solution for mesh sizes of 4mm and smaller. Additionally, the possibilities to monitor deleted elements were improved.

Lastly, also the visualization of the SPR3 element in common post-processors was improved. The SPR state variable was introduced to review the current loading phase and potentials of the fastening element.

2 Extensions to the *CONSTRAINED_INTERPOLATION_SPOTWELD keyword.

To make room for all requirements the keyword received a new line which adds a lot of new useful features. Here we will shortly discuss how the options work.

```
*CONSTRAINED_INTERPOLATION_SPOTWELD
$     pid1      pid2      nsid     thick         r     stiff    alpha1     model
         1         2      1000       2.3         7       5.0                   1
$       rn        rs     beta1       lcf     lcupf     lcupr      dens      intp
                                                               7.85E-6         0
$   stiff2    stiff3    stiff4    lcdexp     gamma     sropt     pidvb
```

2.1 New features

2.1.1 Load specific stiffnesses

The update on the keyword adds a lot of new stiffness parameters for several loading types. When applying only the stiff option the resultant force due to elastic behavior was computed as:

$$\bar{\mathbf{f}} = (f_n,\, f_t,\, m_b) = \text{STIFF} \times (\delta_n,\, \delta_t,\, \omega_b)$$

Note that, if STIFF2, STIFF3, STIFF4 are defined STIFF only refers to stiffness in axial direction and STIFF2, STIFF3 and STIFF4 refer to shear, bending and torsional stiffness, respectively.

- STIFF: elastic normal stiffness $f_n = \mathrm{STIFF} \times \delta_n$
- STIFF2: elastic shear stiffness $f_t = \mathrm{STIFF2} \times \delta_t$
- STIFF3: elastic bending stiffness $m_b = \mathrm{STIFF3} \times \omega_b$
- STIFF4: elastic torsional stiffness $m_t = \mathrm{STIFF4} \times \omega_t$

This provides the possibility to use different stiffness factors for different loading conditions e.g., the elastic behavior of fastening elements may be different in axial and in shear direction because the supporting cross-sections of the modelled fastener are different.

2.1.2 Shear rotation option

The option SROPT allows the control of the local coordinate system for evaluation of forces. By default, pure shear does not create a normal component, but this can be changed with SROPT so pure shear does create a normal component. Basically, there is the option for the local coordinate system to be tilted depending on the shear deformation.

Fig.3: Normal direction for SROPT = 0 (Left) and SROPT = 1 (Right)

On the one hand the benefit of this option depends on what the user feels is the right way to calculate forces. On the other hand, experimental results of shear tests might always have some normal component since the sheets will deform when the specimen deforms further. Hence, the answer to the question whether to use this option or not may depend on the test setup that is used to calibrate the models.

2.1.3 Exponential decay

Traditionally linear softening is employed, where damage is defined as:

$$d = \frac{\bar{u}^{pl} - \bar{u}_0^{pl}(\kappa)}{\bar{u}_f^{pl}(\kappa)}, \quad 0 < d < 1$$

with mode mix ratio

$$\kappa = \frac{2}{\pi} \arctan\left(\frac{F_n}{F_s}\right)$$

The nominal force during degradation is calculated as:

$$\mathbf{f} = (1 - d)\,\bar{\mathbf{f}}$$

In the new implementation the option LCDEXP receives a curve where a certain failure exponent m is dependent on the mode mix ratio. Hence the damage d is defined as:

$$d = \frac{1 - \exp(-m\frac{\bar{u}^{pl} - \bar{u}_0^{pl}(\kappa)}{\bar{u}_f^{pl}(\kappa)})}{1 - \exp(-m)}$$

This can improve the prediction of the failure behavior as shown in figure 4. To some extent it is even possible to capture the energy consumption of failure modes of fastening elements that include, e.g., sheet metal failure. This way the overall energy consumption of welded structures can be modeled more realistically although the modelling of the surrounding shells would not cover these failure modes.

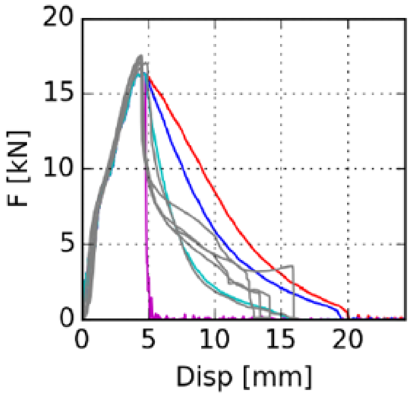


Fig.4: The influence of the failure exponent m on failure behavior. m = 0.1, m = 1.5, m = 5, m = 50.

2.1.4 Mode mix ratio dependent hardening

It is sometimes observed that the behavior of fastening elements after the yield point may vary in dependence of the applied loading conditions. Previously we were able to apply only one hardening curve to model yield behavior. And in order to get a good fit one had to find a compromise where the errors for normal and shear behaviors are minimized. With the mixed mode dependent hardening curves multiple curves can be defined with linear interpolation in between.

To use this feature LCF can be defined as a table, where each curve handles one specific mode mix ratio. For example, the table below will make the hardening for axial loading slightly flatter than for shear loaded connections.

```
*DEFINE_TABLE
       800
0.0
1.0
*DEFINE_CURVE
       801
500.0,0.0
750.0,1.0
1000.0,2.0
*DEFINE_CURVE
       802
400.0,0.0
600.0,2.0
800.0,3.0
1000.0,4.0
```

2.1.5 Gamma factor

The parameter GAMMA adds torsional moments to the shear limits of the failure criterion. The new resulting critical shear force will be calculated as:

$$F_s = f_t + \gamma_1 |m_t|$$

This obviously allows to take into account the influence of torsion on the critical force.

2.1.6 Recommended elements

We tested the *CONSTRAINED_INTERPOLATION_SPOTWELD formulation with ELFORM 2 and 16 shells. It is necessary to apply the drill stiffness constraint to stabilize the solution (DRCPSID on *CONTROL_SHELL).

Besides, it is now possible to connect the end nodes of separate beam elements (tested with ELFORM 1) with this SPR3 connector.

And in the future also ELFORM -2/-1/1/2 hexahedral elements and ELFORM 13/16/17 tetrahedral elements will be available.

2.2 General improvements in stability

To obtain a stable solution it is necessary to carefully choose an appropriate stiffness for the SPR3 element. Especially for large values of the bending stiffness STIFF3 we experienced stability issues. Therefore, one of the most important changes in the implementation of the element was the distribution of penalty forces and especially moments when coupling to a part. The employed strategy is shown here in figure 5.

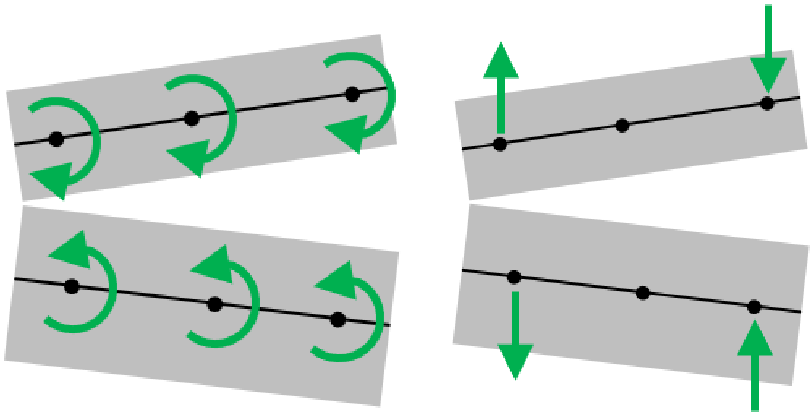


Fig.5: Left picture: old method where the resultant moment is distributed as nodal moments. Right picture: new approach where the resultant moment is distributed as force couples to nodes.

This new approach is based on an elaborate algorithm to apply correct and balanced force distributions for arbitrary mesh situations. Consequently, stability is improved because noise in the involved parts is reduced significantly. The new method is now activated automatically when STIFF3 is used. One can choose to activate the old method by applying a negative STIFF3 value.

To further improve the stability it is recommended to use the SPOTDEL option on **`*CONTRAL_CONTACT`**, which allows the definition of number of shell nodes to fail before the SPR3 is disconnected. To find a proper number of failed elements around the **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** it is necessary to consider how many nodes it will connect. To calculate a stable response with proper force distribution 4-6 nodes should always be present on each side of the connection even when elements have failed. So, the value of SPOTDEL will mostly depend on the used mesh size.

2.3 Post-processing and visualization.

2.3.1 General visualization.

`*CONSTRAINED_INTERPOLATION_SPOTWELD` can be displayed in a post-processor as 2 connected beam elements which are generated during initialization of the model solely for visualization purposes. Normally these beams move according to the average of the translatory motion of the connected nodes and are split when failure occurs. An alternative visualization is available when the option PIDVB is set to a negative number. Here the beams are rotated according to the relative movement of the corresponding sheets and deleted when failure occurs.

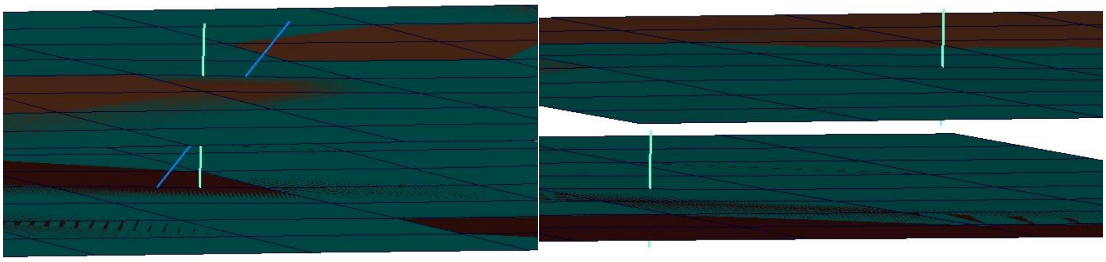


Fig.6: Visualization of SPR-elements with PIDVB < 0 (blue) and PIDVB >= 0 (green). Left: before failure, Right: after failure.

Generally, the option PIDVB allows the user to set the property ID of the visualization beams manually. Otherwise, if the option is not used the property id for the beams is set automatically during initialization.

2.3.2 SPR state variable.

Quick evaluation and assessment of connections is crucial in the development of car body structures. Hence it is very helpful to have as much information as possible condensed in the model when it is viewed in the post processor. For this purpose, we developed the SPR state variable. This variable shows values from 0 to 3 depending on the current state the connection element is in.

Figure 7 shows the phases the element experiences:
0 – 1: Linear elastic behavior / Potential before plastic behavior or damage occurs.
1 – 2: Plastic behavior / Damage initiation
2 – 3: Degradation / Damage evolution

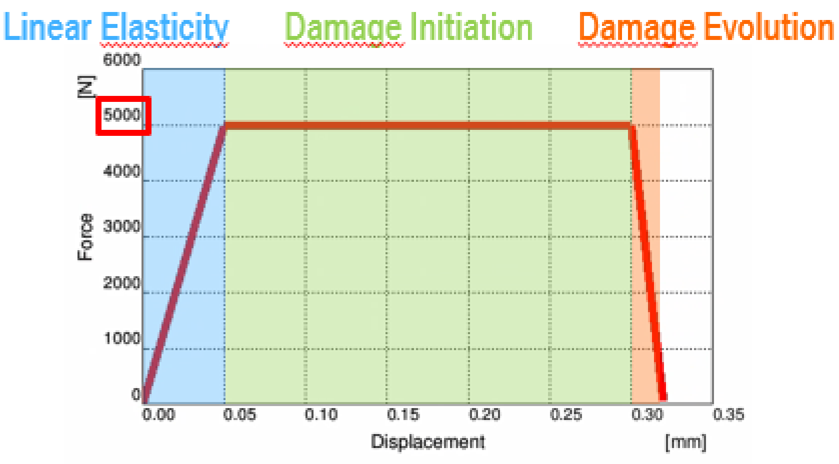


Fig.7: Phases of the SPR element during loading until failure.

For the sake of simplicity this is depicted here with ideal plastic (or constant hardening) behavior.

Figure 8 shows a front rail member where all the different states can be observed on the included spotwelds. This yields a quick overview on how many potentials the investigated structure may have or how robust the result is. Critical areas can also be identified very quickly.

Fig.8: Viewing the SPR state variable in Animator (GNS mbH "Animator" post processor, front rail from Toyota Yaris open model)

2.4 Separate input of material properties: *MAT_265

When applying a negative stiffness for the STIFF option, the value is interpreted as a material id. In that case, the new keyword ***MAT_CONSTRAINED_SPR3/MAT_265** should be used. With this, elements and material can be defined independently further on and multiple elements can share a single material.

The material keyword has the following structure:

```
*MAT_CONSTRAINED_SPR3
$     mid1        ro     model
      1111   7.85E-6         1
$    stiff        rn        rs    alpha1     beta1       lcf     lcupf     lcupr

$   stiff2    stiff3    stiff4    lcdexp     gamma     sropt
```

Thus, the element keyword can be reduced to:

```
*CONSTRAINED_INTERPOLATION_SPOTWELD
$     pid1      pid2      nsid     thick         r     stiff
         1         2      1000       2.3         7     -1111
```

Additionally, there is also an equivalent material interface available for the related SPR2 element: ***MAT_CONSTRAINED_SPR2**.

3 Comparing the numerical results with experiments.

All newly introduced features where thoroughly tested and evaluated on how well the overall prediction of point-like connections was possible with this approach. Here we have compiled some results that confirm that the approach provides reasonable results and is able to properly fit the data we have. Figure 9 and 10 show tests that were done with a BMW internal test specimen for resistance spot weld points. Overall, the predictive capability is rated as very good. When validating the model on several component level tests the results were also sufficient to continue the implementation of the model into full vehicle crash applications. See for example figure 11.

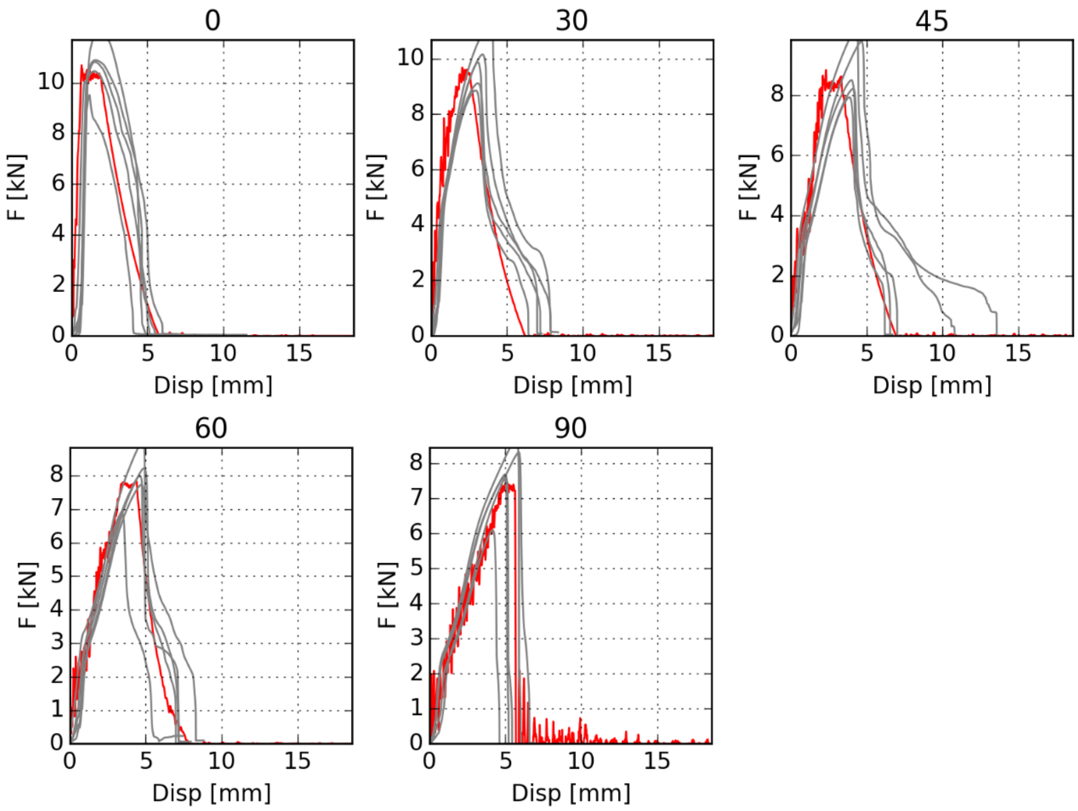


Fig.9: Results of simulations and tests on spotwelds with different loading directions using a BMW internal test specimen.

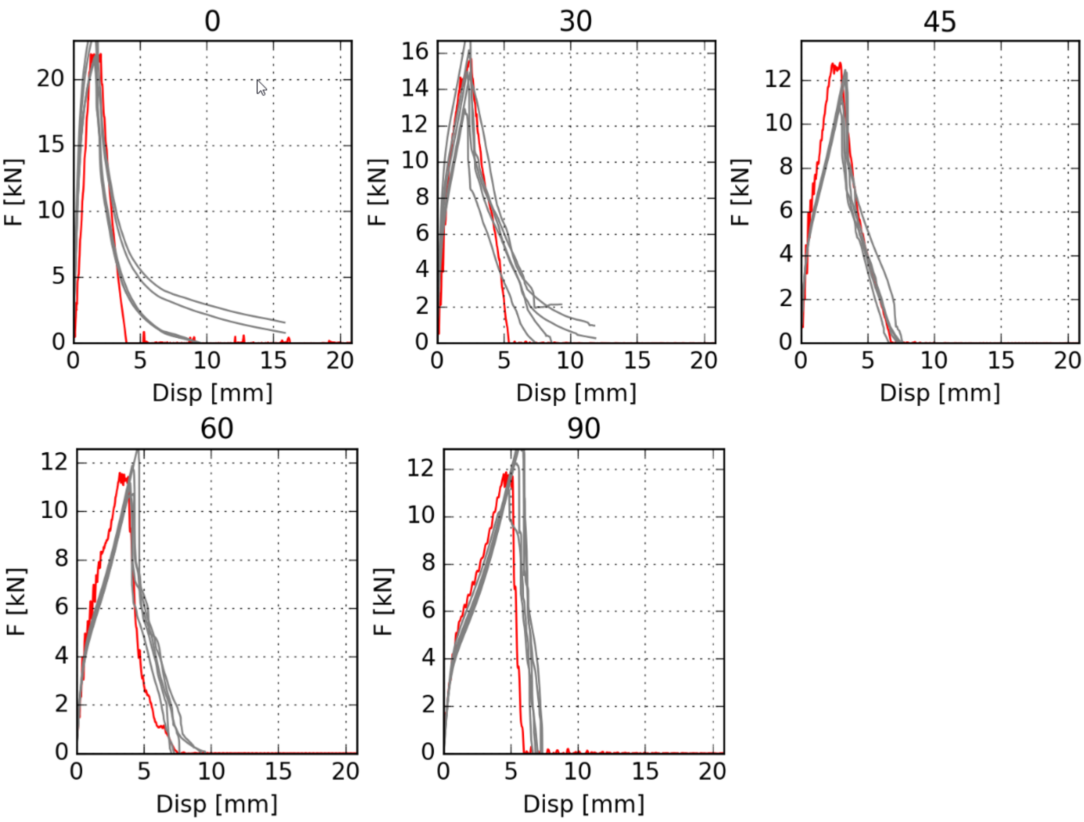


Fig.10: A second set of results of simulations and tests on spotwelds with different loading directions using a BMW internal test specimen. (For higher strength steels and thicknesses.)

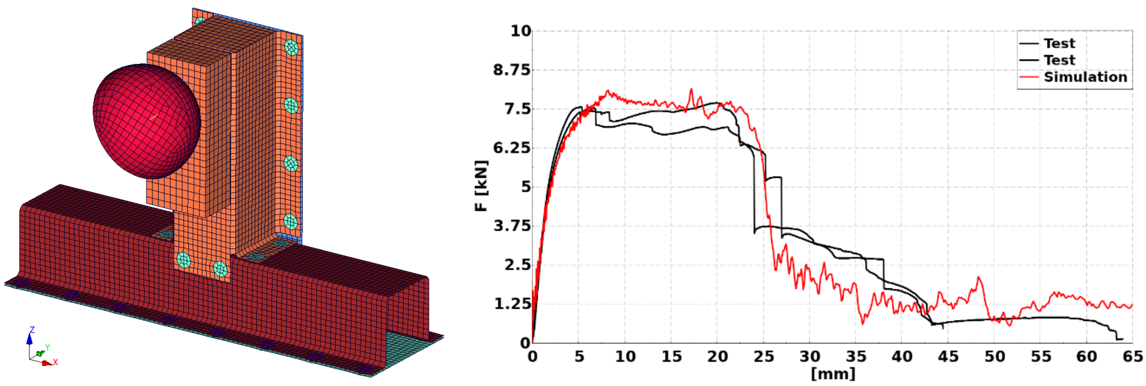


Fig.11: Validation on a test setup from FOSTA project P806 [1].

4 Usage in full vehicle crash applications.

A complete car body can be assembled with over 6000 fastening elements. Our goal was to use one universal modelling technique for as many elements as possible. For all "point shaped" connections this is now possible with the **`*CONSTRAINED_INTERPOLATION_SPOTWELD`**. This includes e.g., spotwelds, different types of rivets, flow-drill screws and generally bolts and screws.

Of course, one has to pay attention to several possible issues if such elements are applied because the SPR3 creates its constraints automatically during the initialization phase of the solver. Some of these issues can be solved if:

- One pays attention to the geometry where the elements are deployed.
- Meshes are sufficiently fine to connect enough nodes and allow the force distributing algorithm to work properly.
- Large gaps between flanges are eliminated to ensure that enough nodes can be found inside of the search radius.

However, one of the main reasons for choosing **`*CONSTRAINED_INTERPOLATION_SPOTWELD`** as a general-purpose fastening element is the mesh independent coupling that enables us to calculate critical forces for each connection point during the assembly process of the car. Therefore, we are able to manipulate meshes or exchange CAD and CAE data without the need to recreate the modelling of the involved connections. And respectively we can move, delete, or add connections without the need to handle meshes.

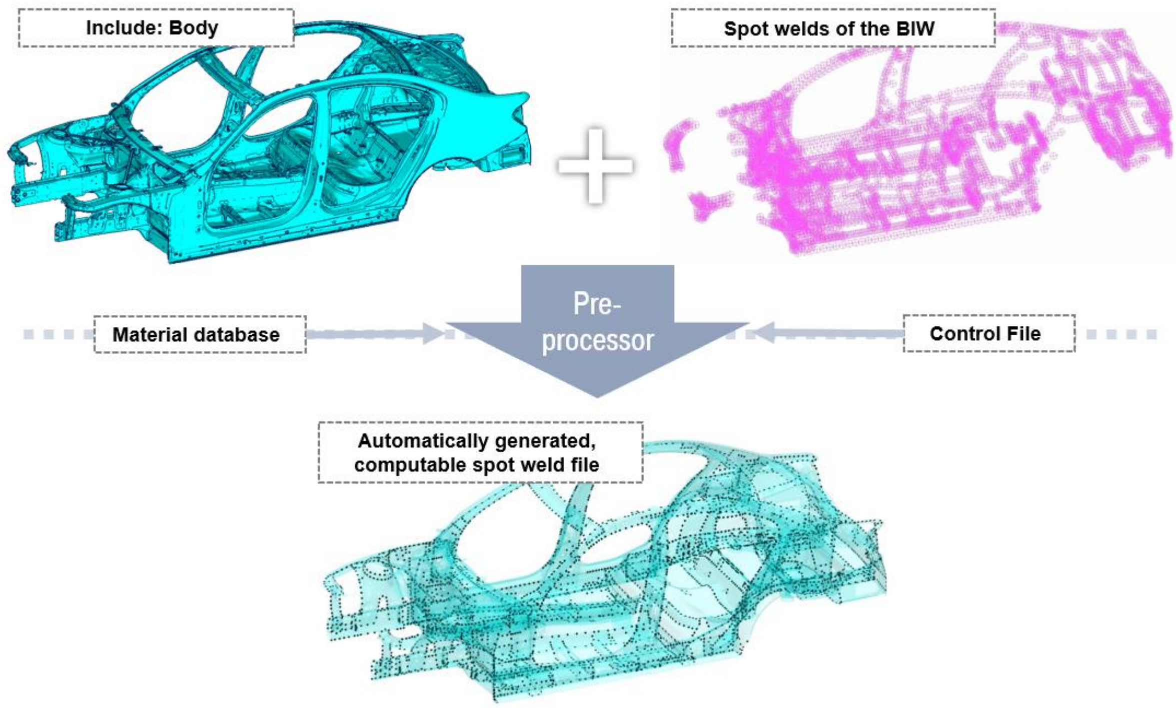


Fig.12: Overview of the assembling process with independent meshes and connections.

5 Future developments

Connections to volume elements:

When modelling aluminum pressure die casting parts sometimes volume elements are used to capture the geometric complexity due to areas that involve very thick structures. Additionally, ribs and injection points might also add a variety of situations that cannot be adequately predicted with shell elements. Thus, the question arose if it is necessary to model the flanges of these parts with shell elements to connect them to steel sheet parts (that are modelled with shells) or if there might be a way to connect a SPR element to the volume elements?

In figure 13 one can see a lap shear test that proves that we already did a step into this direction. However, further testing is needed to fully confirm that the results are of comparable quality to the results when using only shell parts.

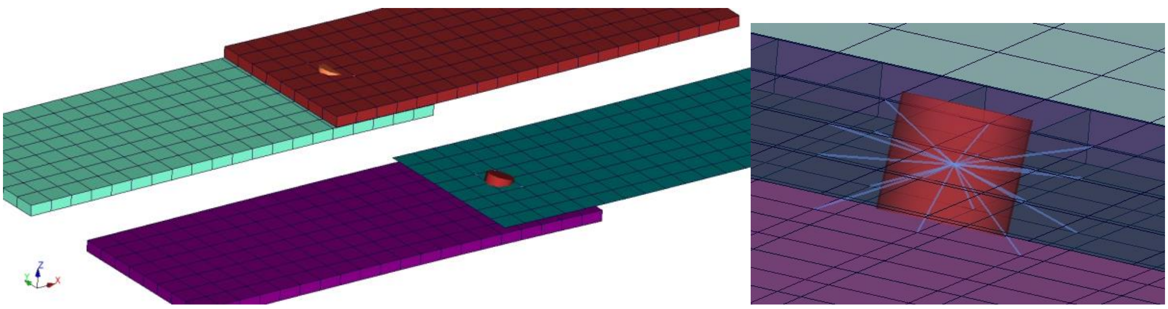


Fig.13: Lap shear test with mixed solid and shell elements. Right picture: Coupled nodes for mixed shell/solid situation.

Peel strength:

Spotwelds where the bending moment has a significant influence on the overall critical force may fail in different modes than when stressed with other loading conditions. The idea of our approach is that when these situations are properly identified the `*CONSTRAINED_INTERPOLATION_SPOTWELD` may be able to switch into different post failure modes. Because the attempt to capture the crack initiation and especially the propagation of such cracks during peeling in the surrounding mesh would require very fine meshes and very complex material models, we tried to make the SPR element consider the energy consumption when peeling occurs.

We are currently investigating this approach further because the first results on dedicated peeling test specimen looked very promising.

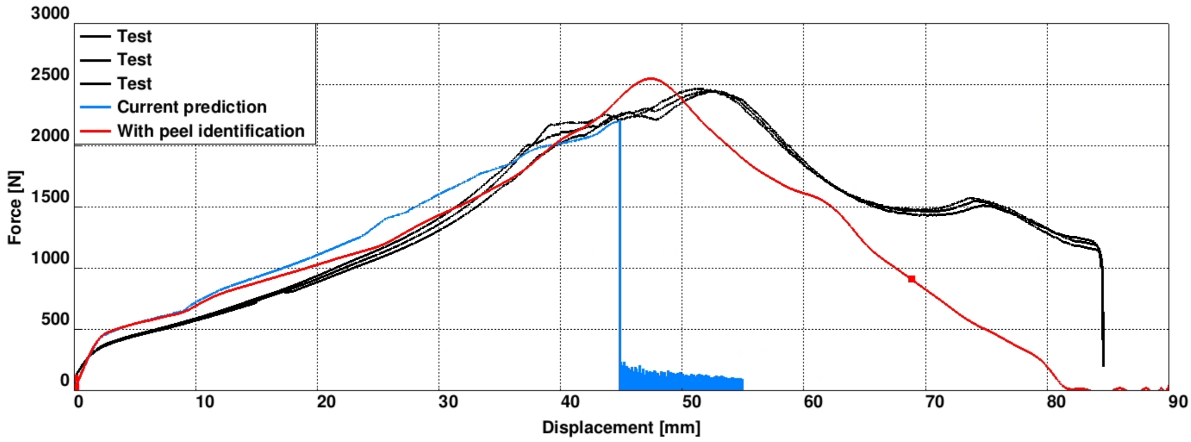


Fig.14: Result of peel tests and corresponding simulations. Red curve shows that it is possible to capture the energy consumption of hybrid spotweld and sheet failure with the ***CONSTRAINED_INTERPOLATION_SPOTWELD***

6 Summary

At the beginning of our last LS-DYNA solver benchmark we were looking for a force-displacement based approach for spot weld modelling as this approach fits well the potentialities of our data base. While the SPR element was a very promising candidate at this point we still had to improve its capabilities. After introducing many new features and improving the stability to make it work in full vehicle crash applications, we verified the feasibility of the approach on calibration tests and component tests. Now we make use of the high flexibility of the modelling approach when employed in full vehicle crash models. Especially the opportunity to have mesh independent fasteners (the user only defines what kind of connection he wants, where it is and what it connects) in combination with an automated pre-processing tool yields great benefits.

Finally, as the approach is now feasible as a standard modelling approach further investigations are done to extend the capabilities of the ***CONSTRAINED_INTERPOLATION_SPOTWELD** to its limits.

7 Literature

[1] Sommer S., Burget S., Klokkers F., Wink H.-J., Krätschmer, D., Charakterisierung und Ersatzmodellierung des Bruchverhaltens von Punktschweißverbindungen aus ultrahochfesten Stählen für die Crashsimulation unter Berücksichtigung der Auswirkung der Verbindung auf das Bauteilverhalten, Schlussbericht zu AVIF-FOSTA-Vorhaben P806 / A262, ISBN 978-3-942541-27-5, 2013.