

Modeling of Rivets Using a Cohesive Approach for Crash Simulation of Vehicles in RADIOSS

Niels Pasligh and Robert Schilling
Ford Motor Company

Marian Bulla
Altair Engineering GmbH

ABSTRACT

Rivets, especially self-piercing rivets (SPR), are a primary joining technology used in aluminum bodied vehicles. SPR are mechanical joining elements used to connect sheets to create a body in white (BiW) structure. To ensure the structural performance of a vehicle in crash load cases it is necessary to describe physical occurring failure modes under overloading conditions in simulations. One failure mode which needs to be predicted precisely by a crash simulation is joint separation. Within crash simulations a detailed analysis of a SPR joint would require a very high computational effort. The conflict between a detailed SPR joint and a macroscopic vehicle model needs to be solved by developing an approach that can handle an accurate macroscopic prediction of SPR behavior with a defined strength level with less computational effort. One approach is using a cohesive material model for a SPR connection. The paper describes cohesive element characteristics and calibration effort. Investigated element characteristics are an updated momentum calculation resulting from shear loads. It allows the adjustment of the width-height ratio for a constant meshing approach; handling of lateral distortion of cohesive elements based on sheet deformation or failure of sheet elements and an elemental stabilization if the cohesive element becomes unconnected that comes along with advanced material models and deletion of sheet elements. The updated element characteristics are shown on principle models and the calibration is shown based on coupon level samples. Outlook is given on a vehicle level and application for other joining technologies.

CITATION: Pasligh, N., Schilling, R., and Bulla, M., "Modeling of Rivets Using a Cohesive Approach for Crash Simulation of Vehicles in RADIOSS," *SAE Int. J. Trans. Safety* 5(2):2017, doi:10.4271/2017-01-1472.

INTRODUCTION

Today's and future vehicles are developed with a high emphasis on light weight design. Light weight design is grouped into subcategories e.g. application of light weight materials or high efficiency. Considering light weight material and efficient design, yield exemplarily to an application of aluminum in a well-engineered BiW. For a sophisticated BiW it is important to exactly determine limitations in load capacity to optimize investigated load cases [1].

To avoid extensive and iterative test programs of full vehicles to optimize structures, the Finite-Element-Method (FEM) became a helpful tool. FEM analysis has increased in accuracy and speed over time, increased hardware resources have enabled more detailed meshes to capture geometry and boundary conditions and allows more precise material models and failure criteria for materials. Also, connections of materials can be considered in more detail to measure loading of connections with consideration of failure criteria to show potential overloading conditions.

Relative to discretization of the full vehicle model the local level of discrete detail for a joint connection is today still limited. An accurate modeling of a joint with very fine continuum elements would increase the overall calculation time significantly, for which reason a macroscopic approach based on a substrate level of detail is required. One accurate and runtime friendly approach is to model connections by using cohesive elements [2], [3]. An example of a self-piercing rivet joints for a substrate to substrate connection is shown in Figure 1.

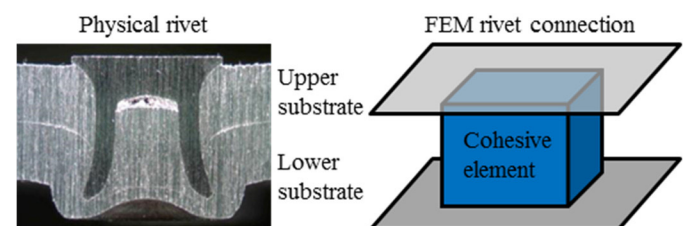


Figure 1. Section cut of physical rivet (left) and FEM rivet connection approach (right)

COHESIVE MODELING APPROACH AND CALIBRATION

The proposed cohesive modeling approach is described for hexahedron and pentahedron elements in RADIOSS. In general, a cohesive element can be differentiated to a continuum element by a reduced stiffness matrix based on reduced sectional stresses, [Figure 2](#).

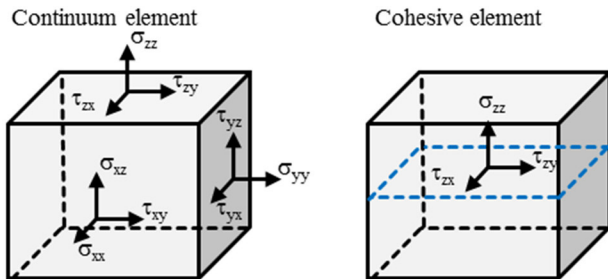


Figure 2. Continuum vs. cohesive element

The mathematical approach of cohesive elements with a reduced stiffness matrix to the z-plane enables the element to be time step independent from its element height because the sectional stresses at cross sections in x- and y-plane are not considered. This results in a lateral stiffness of zero for the element. Based on this theory, an element can be modeled as a very thin element or as a connection between the mid surface layers of shells, [Figure 3](#). If its volume is equal to zero it can be defined by a mass per area without influencing the time-step. This enables a physical modeling of a connection between two or more parts.

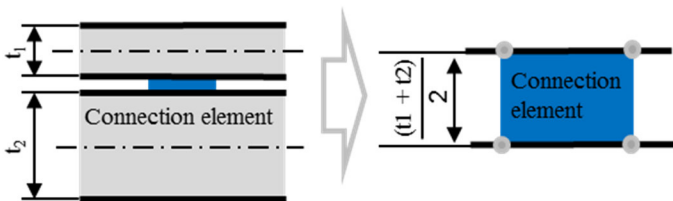


Figure 3. Connection modeling of two parts

Based on the reduced element stiffness, modeling constraints need to be considered. The node numbering of an element is essential to determine the orientation of the stiffness direction. Default node numbering is nodes 1 to 4 belonging to one substrate and nodes 5 to 8 to another substrate. This defines the element normal in 1-5, 2-6, 3-7 and 4-8 direction, [Figure 4](#).

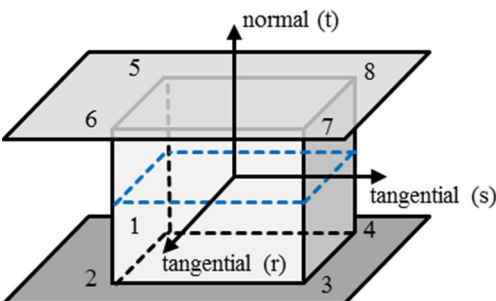


Figure 4. Node numbering and internal elemental directions [4]

Finally, all nodes have to be connected to a substrate structure (direct connection to continuum elements or indirect connection as slave nodes of *Rigid-Bodies*/tied interface). When all nodes of an element become free, the element is deleted to keep the model determined.

For consideration of bending moments, the cohesive element has four Gauss integration points on its mid-plane, [Figure 5](#). Based on the element height and occurring shear forces in r- and s-directions the four integration points allow the calculation of bending moments resulting in normal tensile and compression forces.

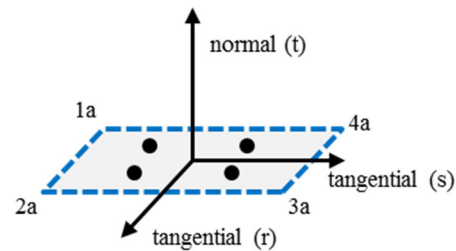


Figure 5. Distribution of the Gauss integration points in the mid plane of a cohesive solid element

RADIOSS currently provides two material models for a cohesive volume element [4]:

- MAT/LAW59 (CONNECT)
- MAT/LAW83 (SNCONNECT)

For each stiffness direction, the material can be subdivided into an elastic part and a plastic elongation when yield stress is reached, [equation 1](#) and [equation 2](#). Based on simplified MAT/LAW59 yield equations of the uncoupled yield stress are calculated based on the deflection in the normal or tangential direction:

$$\text{Yield_normal} = \sigma_{zz} \quad (1)$$

$$\text{Yield_tangential} = \sqrt{\tau_{zx}^2 + \tau_{zy}^2} \quad (2)$$

For an uncoupled yield surface [equation 3](#) and [equation 4](#) are used:

$$\text{normal } \Delta l_{pl} = \text{normal } \Delta l_{total} - \sigma_{true}/E \quad (3)$$

$$\text{shear } \Delta l_{pl} = \text{shear } \Delta l_{total} - \sigma_{true}/G \quad (4)$$

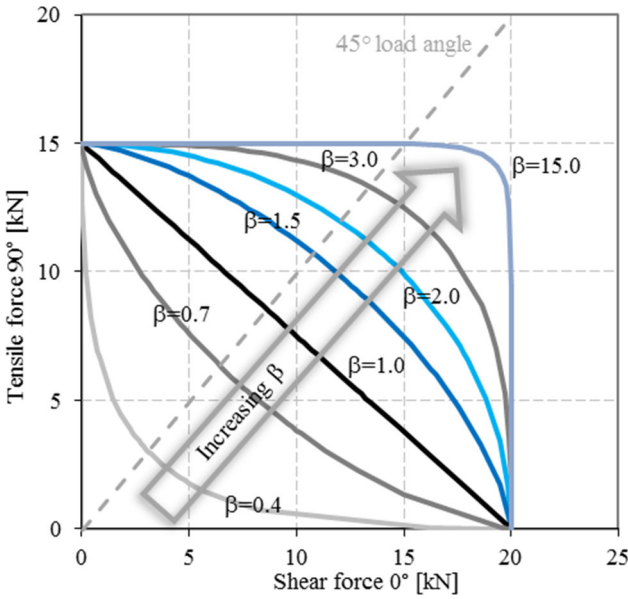
An advantage of the MAT/LAW59 with the uncoupled yield surface is the high calculation performance, because of lower computational effort. For failure detection, coupled behavior based on relative displacements ε_N and ε_T can be used with MAT/LAW59:

$$\left| \frac{\varepsilon_N}{\varepsilon_{maxN}} * \alpha_N * f_N(\dot{\varepsilon}_N) \right|^{expN} + \left| \frac{\varepsilon_T}{\varepsilon_{maxT}} * \alpha_T * f_T(\dot{\varepsilon}_T) \right|^{expT} > 1$$

Alternatively, the coupled yield surface in MAT/LAW83 can be used, [equation 6](#). The development was done by the Fraunhofer-Institute IWM, [5], [6]. The exponent β controls the coupling rate between the yield stress in normal n and tangential s direction.

$$\sigma_y = \left[\left(\frac{f_n}{R_N(1-sym\alpha)} \right)^\beta + \left(\frac{f_s}{R_S} \right)^\beta \right]^{\frac{1}{\beta}} \quad (6)$$

[Figure 6](#) shows the dependency. With $\beta = 1$, the yield forces between tensile and shear load change in a linear way. With increasing exponent β the yield surface becomes more convex. For some very seldom observed experimental results, the shape of the yield surface can even be concave. This can be achieved by $\beta < 1$.



[Figure 6](#). Development of yield force depending on exponent β [5]

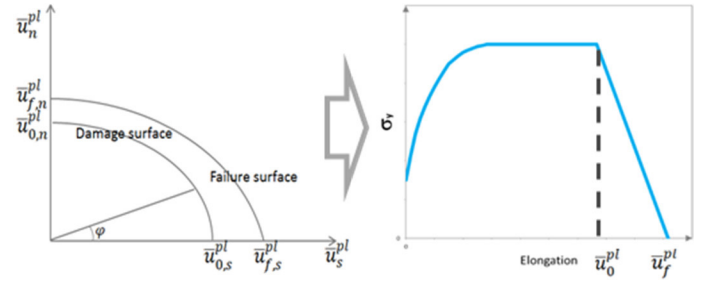
For the damage initialization, [equation 7](#), the same approach as for the yield-surface, [equation 6](#), is used:

$$\sigma_y = \left[\left(\frac{\bar{u}_{0,n}^{pl}}{Fct_{0N}(1-sym\alpha_0)} \right)^{\beta_0} + \left(\frac{\bar{u}_{0,s}^{pl}}{Fct_{0S}} \right)^{\beta_0} \right]^{\frac{1}{\beta_0}} \quad (7)$$

The failure surface, [equation 8](#), for coupled normal and shear load is:

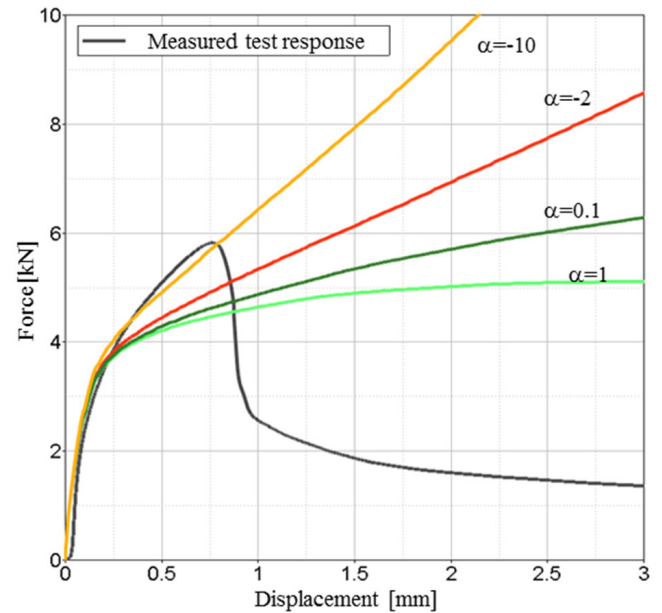
$$\sigma_y = \left[\left(\frac{\bar{u}_{f,n}^{pl}}{Fct_{fN}(1-sym\alpha_f)} \right)^{\beta_f} + \left(\frac{\bar{u}_{f,s}^{pl}}{Fct_{fS}} \right)^{\beta_f} \right]^{\frac{1}{\beta_f}} \quad (8)$$

[Figure 7](#) shows the coupled normal and shear behavior of MAT/LAW83. Therefore, all three limits of yield point, damage initialization points and failure need to be defined by using [equation 6](#) to [equation 8](#).



[Figure 7](#). Coupled normal and shear behavior in MAT/LAW83

The implemented equations allow the normal force to be scaled according to the load angle. It allows a separate adjustment of the maximum peel force by remaining tensile strength. [Figure 8](#) shows that the force-displacement curve in a peel load case is influenced by parameter α . The parameter sym is the angle between the top and bottom faces of the cohesive zone element. It is changing with time if a constant deformation is assumed.



[Figure 8](#). Modification of the force depending on the peel load [7]

If peel forces are differing by a factor of 0.5 with respect to tension forces a modified resultant peel resistance can be considered by MAT/LAW83. The factor of 0.5 results from momentum calculation to the middle axis of the element. For physical connections with a ratio of ~ 0.5 between maximum peel force divided by maximum tension force a separate adjustment of the peel resistance is not needed.

For the following investigation, the MAT/LAW59 is used to describe the principle calibration procedure of a cohesive material law. An appendant material card is shown in [Figure 9](#).