A comparative study of different failure modeling strategies on a laboratory scale test component

Stefan Golling[1], Rickard Östlund[2], Mikael Schill[3], Reimert Sjöblom[4], Kjell Mattiasson[5], Johan Jergeus[5] and Mats Oldenburg[1]

Abstract

Ultra-high strength steel (UHSS) has become a common material in the automotive industry during the last decades. The technique of press hardening allows modifying and tailoring the material properties of the blank in accordance with desired performance.

In the present work, a laboratory scale test component is developed. On basis of tests on the component it is intended to investigate the deformation and fracture behavior of a boron alloyed steel after different heat treatments. The tooling is developed to allow the production of single phase microstructures like martensite and bainite as well as mixed microstructures containing ferrite. Testing of the component is performed in a standard tensile testing machine with additional digital speckle measurements to determine the strain to fracture in the critical cross section. The initial geometry shape introduces bending in the critical cross-section during tensile loading of the specimen.

The aim of this work is to compare different material models on a component like level, including the prediction of failure. A finite element model of a laboratory scale component is analyzed using LS-Dyna. To compare different failure modeling approaches a set of damage models is calibrated to full hardened, martensitic steel. The deformation and fracture behavior of the component is presented in terms of load-displacement, plastic strain-stress triaxiality as well as in principal strain space.

Keyword: Finite element analysis, damage, failure, component simulation

1 Introduction

The global trend of reducing carbon dioxide emissions from vehicles has pointed the focus to weight reduction in automobiles. Due to legislative and customer demands an issue is weight reduction while maintaining safety, i.e. crashworthiness. The use of the finite element method gives the possibility of analyzing components and full vehicles before the first prototype is built. This has a twofold advantage, firstly the lead time is reduced as a large number of variants can be evaluated in short time, secondly the design and manufacture of expensive tooling and prototypes is postponed until an optimal solution is found. The development of automotive components relies on the adequate modelling of material deformation and failure under loading. In the simulation of crash relevant components, it is essential to determine when failure of the material initiates. A variety of different failure models is available, also different assumptions are common in the decision when a component has failed. In sheet metal forming often the onset of necking is considered as the point of failure, in crashworthiness ultimate fracture is of interest.

In the present study five damage models are compared on a virtual component under bending deformation. The commercially available finite element code LS-Dyna is used, GISSMO and the tabulated Johnson-Cook model are available by standard, CrachFEM is module available for many FE codes and is distributed under license of MatFEM. Two additional models are only available using the user defined material subroutines, the first is termed OPTUS and the second VCC. All models are compared on the load-displacement response for two different flange heights of the virtual component. Furthermore, stress state and plastic strain during deformation up to the point of failure are documented.

2 Theoretical background

In the modelling of material failure many models take the stress state into account. It is useful to describe the stress state of different load cases using the invariants of the stress tensor. The present work aims at the analysis of a sheet metal component. For thin material, a common assumption is that the structure is in a plane state of stress. In plane stress one parameter is sufficient to describe the stress state, the triaxiality is defined as the ratio of the mean stress and the effective von Misses stress.

$$\eta = \frac{p}{\sigma_{eff}}$$

Deformation of the material up to the onset of necking is modelled using an elasto-plastic material model. The elasto-plastic description varies between the models but calibration to experimental results causes only minor deviations in the load-displacement response of modelled tensile tests. Post-localization and fracture of the material is modelled using five damage models. The models are briefly summarized in the following. For in detail information on the models the reader is referred to the references.

2.1 CrachFEM

CrachFEM [1, 2] is the short form for a model combining an elasto-plastic material with a damage model. CrachFEM is a versatile approach for the modelling of failure using phenomenological models. To determine the onset of necking a submodel called ‘Crach’ is used to detailed discretize the neck. The submodel uses a plastic material model with an initial imperfection to allow for a realistic prediction of necking. For the prediction of fracture CrachFEM uses phenomenological models for the phenomena of ‘ductile normal fracture (DNF)’ and ‘ductile shear fracture (DSF)’. Ductile normal fracture is caused by void growth and coalescence with a fracture surface normal to the direction of the first principal strain. Ductile shear fracture accounts for a shear band localization which is followed by fracture. For shell elements a post-instability strain is used. This approach accounts for the additional strain in a shell element, from the onset of necking, until final fracture.

CrachFEM defines a fracture risk ψ for the three fracture modes. The fracture risk is dependent on the equivalent plastic strain ϵ_p and the plastic strain at fracture ϵ_f.

$$\psi = \frac{\epsilon_p}{\epsilon_f}$$

However, the fracture strain ϵ_f depends on the stress state which usually changes during loading. Hence, the fracture risk should be consistent with the change of the stress state.

$$d\psi = \sum_{n}^{N} \frac{\epsilon_p}{\epsilon_{f(\sigma_{ij}^n)}}$$

N denotes the total number of time steps and σ_{ij}^n denotes the stress state in the current time step. If the fracture risk reaches unity failure occur. The difference between the three fracture criteria is their way to determine the fracture strain. For ductile normal fracture the strain to fracture is a function of the triaxiality and the plastic strain rate $\epsilon_{f(\eta,\dot{\epsilon}_p)}$. For ductile shear fracture a shear stress parameter θ is introduced to determine the fracture strain $\epsilon_{f(\theta,\dot{\epsilon}_p)}$

$$\theta = \frac{\sigma_M}{\tau_{max}}(1 - k\eta)$$

The parameter k describes how triaxiality influences the shear fracture, τ_{max} is the maximum shear stress determined from the first and third principal stress. The third criteria and evaluates the ductile instability fracture through the thickness. The equivalent plastic strain at onset of fracture risk is a function of the in-plane deviatoric stress ratio $\alpha = \sigma_2/\sigma_1$ and the plastic strain rate, $\epsilon_{f(\alpha,\dot{\epsilon}_p)}$

2.2 GISSMO

Generalized incremental stress-state dependent damage model (GISSMO) [3, 4], is a phenomenological failure model formulation that allows for an incremental description of damage accumulation, including softening and failure. The initial development aimed at the modeling of forming and crash using the same damage model for both simulation steps and accumulate damage of the forming analysis for further use in crash analysis. The model is intended to provide variability, hence the input parameters are based on tabulated data. Advantage is the input of for example an arbitrary definition of the triaxiality dependent failure strain, this is needed for the use over a wide range of different stress triaxialities and materials. The treatment of post-localization is done through the definition of a mesh size regularization. This is realized through the damage formulation. A mesh size dependent failure strain is formulated for energy regularization. The damage is coupled to the stress tensor in post-localization deformation. The damage variable is described as an exponential function

$$\dot{D} = \frac{n}{\epsilon_f} D^{1-\frac{1}{n}} \dot{\epsilon_p}$$

Where D is the current value of the damage, $\dot{\epsilon_p}$ is the equivalent plastic strain rate, n is the damage exponent and ϵ_f is equivalent plastic strain at failure. For n=1 the exponential function reduces to the Johnson-Cook model. The onset of necking is considered using a forming intensity parameter F, the equivalent plastic strain and a parameter describing the onset of necking in measures of the equivalent plastic strain

$$\dot{F} = \frac{n}{\epsilon_{p,loc}} F^{1-\frac{1}{n}} \dot{\epsilon_p}$$

If n=1 the equation reduces to a linear form like the damage variable. For constant values of $\epsilon_{p,loc}$ the equation can be integrated to a relation of the equivalent plastic strain and the equivalent plastic strain to localization. The forming intensity parameter is accumulated the same way as the damage parameter D. The difference between functions D and F is the input of the limiting strain, ϵ_f or $\epsilon_{p,loc}$, depending on the triaxiality.

When instability is reached, F=1, it is assumed that damage is coupled to the stress tensor using Lemaitre's (1985) effective stress concept.

$$\sigma_{eff} = \sigma \left(1 - \left(\frac{D - D_{crit}}{1 - D_{crit}}\right)^m\right)$$

The value for D_{crit} is determined as the onset of necking is reached. The exponent m is termed fading exponent, it allows for a regularization of fracture strain and the energy consumed during post-instability deformation.

2.3 OPTUS

The OPTUS [5] model originates from the desire to calibrate a failure model solely from measurement of tensile specimens representing different stress states and digital image correlation (DIC). From DIC the strain field on the specimen surface is determined. A post-processing procedure is used to determine stress state and plastic strain up to fracture of the specimen. Parameters for treating post-localization and fracture are determined using least squares fitting. This model does not need inverse modelling to determine necessary parameters. Post instability response is modelled with a localization function which compensates for mesh size effects.

$$L = A\left[exp\left(B\left(\epsilon_p - \epsilon_0\right)\right) - 1\right], \qquad \epsilon_p \geq \epsilon_0$$

The parameters A and B are functions of the analysis length scale i.e. the element size, ϵ_0 is an equivalent localization threshold strain. The localization function is applied in a modified von Mises yield equation, where σ_y is the current yield stress and J2 the second deviatoric stress invariant.

$$f = \sqrt{3J_2} - \sigma_y(1 - L)$$

The L function causes weakening by reduction of the current yield strength. L can be viewed as to be composed of both geometric effects unresolvable by the mesh, and voids induce by plastic damage.

The plastic failure strain is defined as a function of the triaxiality and the analysis length scale l.

$$\epsilon_f = (\epsilon_f^0 - \epsilon_0) exp(-Cl) + \epsilon_0$$

The variable ϵ_f^0 is failure strain at zero l, i.e. ϵ_f =ϵ_f^0 for $l \rightarrow 0$. Failure is postulated to occur when the localization function reaches its critical value L_f.

$$L_f = A\left[exp\left(B(\epsilon_f - \epsilon_0)\right) - 1\right] \;\; and \;\; \int_0^{t_f} \frac{\dot{L}}{L_f} dt = 1$$

The stress state at failure is taken into account by any suitable equation describing the equivalent strain to failure as a function of stress invariants. The model can either be used with solely the localization function causing material degradation until load bearing capacity vanishes or a fracture criteria can be introduced.

2.4 Tabulated Johnson-Cook

The tabulated Johnson-Cook (JC) material model is available in LS-Dyna [6]. This model is based solely on general tabulated input parameters. The plastic failure strain can be defined as a function of triaxiality and element size. The failure criterion of this material model depends on the plastic strain evolution and on plastic failure strain. The damage variable is accumulated over time, where failure is indicated if when unity is reached.

$$D_{JC} = \int \frac{\dot{\epsilon_p}}{\epsilon_f} dt$$

2.5 VCC

The fifth model in this study is termed VCC, it is based on forming limit curves (FLC). The failure model is intended to predict incipient necking as realistic as possible. Neglecting the post-necking region allows a model without necessity of mesh size dependent parameters. The primary source of failure in an FLC is incipient necking. A forming limit curve indicates an upper limit for allowable strains. A FLC can be transformed into the principal stress plane to obtain a forming limit stress diagram (FLSD). Advantage of the FLSD concept is its reduced sensitivity to the strain path, stress based criteria suffer from reduced accuracy in finite element computation compared to strains. It has been shown that the same strain path independence can be obtained in other variable spaces. A limit curve with respect to effective plastic strain ϵ_p and α, the ratio between the minor and major principal plastic strain rates was proposed. In this method, the magnitude of the effective plastic strain together with the plastic flow direction at the last stage of deformation is of importance. For further details on the VCC model the reader is referred to [7]

3 Finite element model

The damage models are compared on a test specimen which is not part of the calibration of the material models. A virtual test component is designed where the critical cross-section is under a bending deformation. In Fig. 1the shape of the component is outlined for one flange height. Material throughout this work is the low alloyed boron steel 22MnB5 in fully hardened condition i.e. martensitic microstructure. Elastic and plastic properties as well as data for the failure model calibration are taken from literature, see [8,9]. The specimen is constrained in tensile direction on one end, the opposite end has a prescribed velocity. The element type two of LS-Dyna is used, this is a one point integrated Belytschko-Tsay element, through thickness five integration points are used, shell thickness is 1.5mm. The Lobatto integration rule is used to obtain stress and strain values on the outer surfaces. Models using element erosion, delete the element if the integration point at mid thickness indicates material failure.

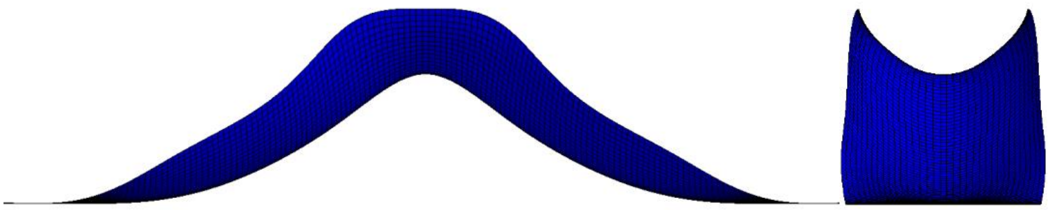


Figure 1: *Front and side view of the virtual test component. In the view to the left, right end has a boundary condition prohibiting displacement in longitudinal direction of the component, left end with prescribed velocity. Deformation of the component in vertical direction.*

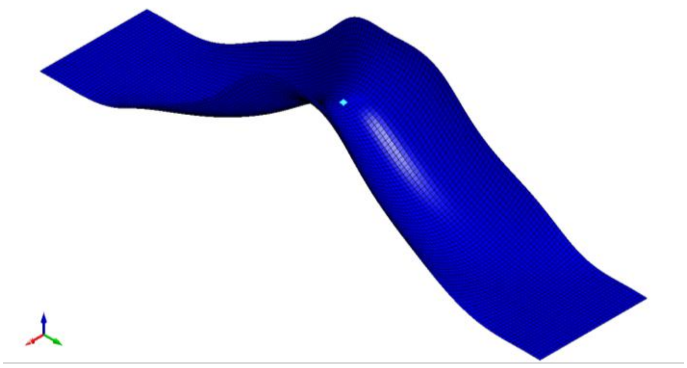


Figure 2: *Geometry of the virtual test component used for the finite element model. Marked in light blue the element used for data extraction.*

4 Results and discussion

The virtual test component is loaded in longitudinal direction and consequently the center cross-section is in bending. Consequently, the two integration points on the surfaces are under compression and tensile loading. The flanges of the component are intended to maintain a high stress triaxiality, i.e. close to the biaxial stress state, in the critical cross-section during loading. The component in its current geometrical definition cannot maintain a biaxial stress state until final fracture.

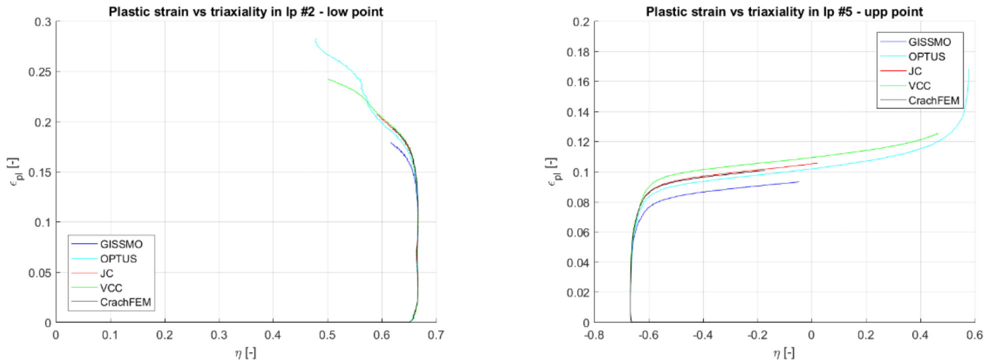
Five failure models are calibrated to fully hardened boron steel and compared on the virtual test component. In Fig. 3 the change of the stress state, i.e. the stress triaxiality, with increasing plastic strain are depicted for two integration points located on the upper and lower surface of an element located at centered position in the critical cross-section, see Fig. 2 for the position of the element on the geometry.

A general observation is the good agreement during the first part of the loading. During this loading stage, only plastic deformation occurs. All models are relative easy calibrated to the hardening curve of the fully hardened steel. After a certain amount of plastic deformation is reached the models start to deviate caused by the different formulation of material degradation. For three models fracture in the element is indicated and element deletion is activated. For VCC and OPTUS this approach is possible but instead continued element degradation is chosen. Therefore, final fracture is not displayed for those models.

In Fig. 3a the lower point, the integration point under tensile loading, is displayed. For this case the agreement of all models in the damage region is good. The fracture is indicated at different plastic strain levels, this explained by the parameter calibration. The biaxial stress state on the lower surface of the component cannot be maintained until fracture occurs, instead fracture is indicated closer to a state of plane strain.

In Fig. Figure 3b the result for the integration point of the upper surface is depicted. Commonly, failure models for sheet metal applications are not calibrated in compression. The difference between the models is more pronounced in this type of loading. Models with element erosion reach a triaxiality value close to zero which corresponds to shear. This is though just a transition as it is visible for the models without element erosion.

Two fringe plots taken at the maximum load are shown, in Fig. 4a the triaxiality parameter and in Fig. 4b the plastic strain is visualized. The triaxiality is in large areas of the component in a biaxial stress state with exception of the critical cross-section which shows slighter lower values as already mentioned in the previous section. The plastic deformation of the component is localized in the center.



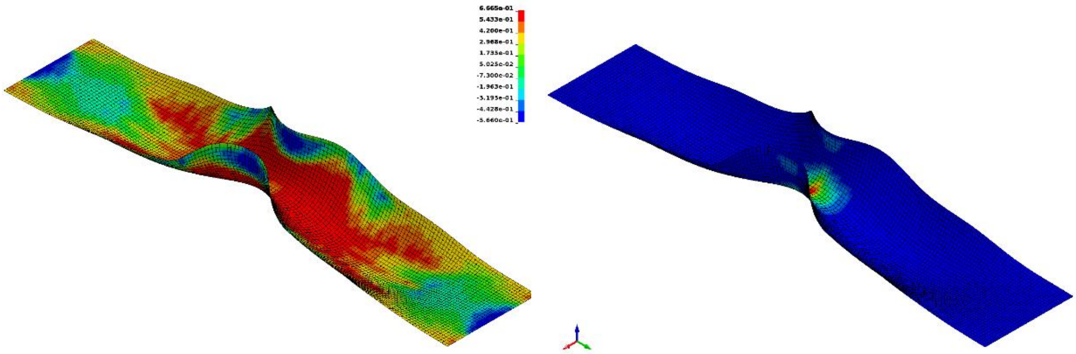
a) ***b)***

Figure 3: *Stress triaxiality in the lower and upper point of the element in the critical cross-section.*

Compared to regular tensile test specimens with notch or hole geometry the present component allows a loading path at the biaxial stress state at the surface under tensile loading. Furthermore, this stress state is rather stable at this level.

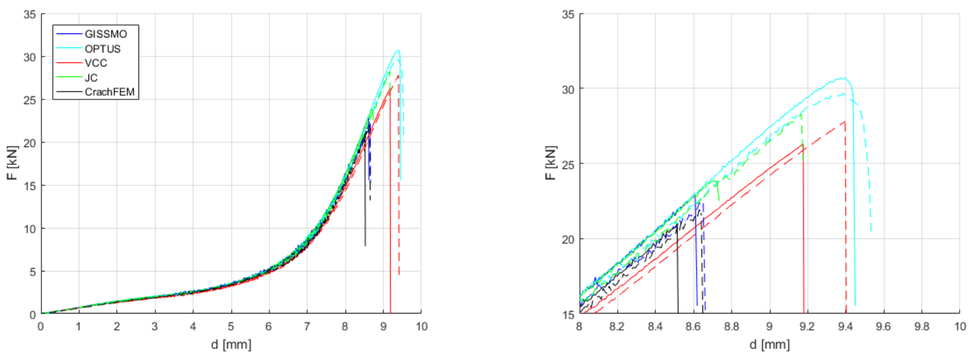
All models allow for a compensation of mesh size relate effects. Hence, two different mesh sizes are compared in the figures. To overcome post-necking mesh size dependency the GISSMO, Johnson-Cook and CrachFEM use regularization functions, the OPTUS model relies on parameters regularized to the element size. The VCC has no explicit mesh size compensation as its main purpose is to predict the onset of incipient necking.

The load-displacement response of the virtual component is shown in Fig. 5. The overall response is similar but with significant differences in peak load and maximum elongation. This is to be expected when applying different damage and fracture models with different assumptions concerning the onset of failure. The reduction of the mesh size sensitivity shows good compensation for all models. For the OPTUS model a slightly different unloading can be seen in Fig. 5b, this is explained by the mesh size dependent localization function.



a) *b)*

Figure 4: *Fringe plot of a) stress triaxiality, and b) plastic strain.*



a) *b)*

Figure 5: *Load-displacement response of the virtual test component compared for five models.*

References

[1] Oberhofer, G., Dell, H., Gese H.: MF GenYld+CrachFEM 4.2 User's Manual (2014)

[2] Gese H., Oberhofer G., Dell H.: Consistent modelling of plasticity and failure in the process chain of deep drawing and crash with user material model MF-GenYld+CrachFEM for LS-Dyna. In: 6th LS-Dyna Anwenderforum, Frankenthal (2007)

[3] Effelsberger J., Haufe A., Feucth M., Neukamm F., DuBois P.: On parameter identification for the GISSMO damage model. In: 12th International LS-Dyna Users Conference

[4] Neukamm F., Feucht M., Haufe A.: Considering damage history in crashworthiness simulations. In: 7th European LS-Dyna Conference

[5] Östlund, R.: Evaluation of localization and failure of boron alloyed steels with different microstructure compositions. In: Journal of Materials Processing Technology 214 (2014), pp. 592-598.

[6] LS-Dyna Keyword User's Manual, Volume II – Material Models, (2016)

[7] Mattiasson K., Jergéus J., DuBois P.: On the prediction of failure in metal sheets with special reference to strain path dependence. In: International Journal of Mechanical Sciences 88 (2014), pp. 175-191.

[8] Golling, S., Östlund , R., Oldenburg, M.: Characterization of ductile fracture properties of quench-hardenable boron steel: Influence of microstructure and processing conditions. In: Materials Science and Engineering: A 658 (2016), pp. 472-483.

[9] Östlund , R., Golling, S., Oldenburg, M.: Microstructure based modelling of ductile fracture initiation in press-hardened sheet metal structures. In: Computer Methods in Applied Mechanics and Engineering 302 (2016), pp. 90-108.

Affiliation

[1] Division of Solid Mechanics, Luleå University of Technology, 971 87 Luleå, stefan.golling@ltu.se

[2] Gestamp HardTech, Luleå

[3] DYNAmore, Linköping

[4] Scania AB, Södertälje

[5] Volvo Car Corporation, Trollhättan