

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

Helmut Gese, Gernot Oberhofer, Harry Dell

all authors: MATFEM Partnerschaft Dr. Gese & Oberhofer, München, Deutschland

contact: helmut.gese@matfem.de

Abstract:

In automotive industry the need to build light weight structures with increased demands for passenger safety can be significantly supported by numerical simulation. However a correct material description is essential. For a correct prediction of failure behaviour of metallic materials in the crash simulation it can be necessary to consider the complete process chain of forming and crash. Today crash analysis cannot be performed with the same degree of discretization like in forming simulation. In order to transfer results between different meshes a mapping process of the relevant element variables has to be introduced.

The user material model MF GenYld + CrachFEM has been developed at MATFEM as an universal material model which can be combined with finite element code LS-Dyna and other finite element codes with explicit-dynamic time integration scheme. The module MF GenYld describes the elastoplastic material behaviour. It has a modular structure which allows to combine different yield loci with a variety of hardening models. Besides different models for isotropic hardening, it includes advanced models for isotropic-kinematic hardening and anisotropic hardening. The comprehensive failure model CrachFEM allows to predict material failure due to localized necking (for shell discretization), ductile fracture and shear fracture. The models for ductile and shear fracture can be used consistently for shell and solid discretization. As the algorithm Crach for the prediction of localized necking requires information about the complete deformation history of an element an orthotropic mapping is inevitable.

Within this investigation the application of material model MF-GenYld+CrachFEM for forming and crash simulations is discussed. The technical background for the mapping of the deformation history is introduced.

Keywords:

FEM, material model, localized necking, ductile fracture, shear fracture, mapping

1 Introduction

The development of a new body-in-white in the context of the increased requirements for passenger safety and the simultaneous need for lightweight design is one example for a complex CAE based design process. Different groups of advanced materials may have to be modelled:

- Sheets (HSS/AHSS/UHSS/Aluminium)
- Extrusions (Aluminium/Magnesium)
- Cast components (Aluminium/Magnesium)

Clear decisions, which material is suited for certain demands can only be made with a correct representation of the material behaviour including the prediction of possible material failure.

The material model MF-GenYld allows for a combination of different yield loci with different hardening models. The most advanced option is an anisotropic hardening model which covers the real material behaviour of austenitic sheet steel or magnesium extrusions. This model can also be used to approximate the behaviour of polymer materials. CrachFEM is a comprehensive failure model for both material groups. It accounts for all possible failure modes of metallic materials under monotonic deformation (i.e. localized necking, shear fracture and ductile fracture). Applications of MF-GenYld + CrachFEM or CrachFEM in combination with other material models have already been validated for the development of automotive components [3, 4, 6].

In case of crash applications the material behaviour of metals can only be understood when taking into account the production process. For sheets the complete deformation history from the forming process has to be included.

2 Material and Failure Model MF GenYld

2.1 Orthotropic Material Model MF GenYld

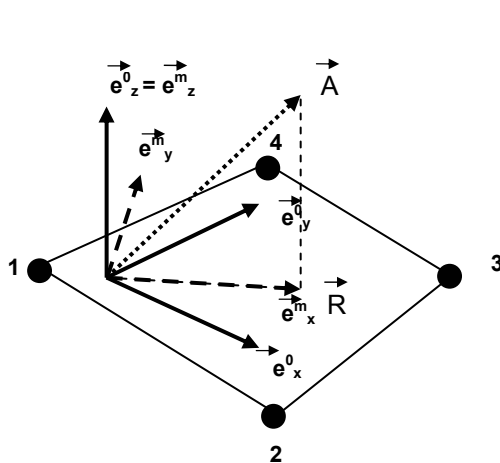
The user of MF GenYld can combine different orthotropic yield loci (plastic orthotropy module) for the plane stress condition (for shell discretization) with different hardening models (plastic hardening module). The available yield loci are:

- Hill - 1948,
- Hill - 1990,
- Barlat-Lian - 1989,
- Barlat-1996,
- Barlat-2000.

In case of orthotropic material models stress and incremental strain tensor is transformed from element coordinate system to a material coordinate system. In LS-Dyna this material coordinate system is defined by projecting a vector onto each shell element. This vector is defined in the material input card through options of 'AOPT'. The transformation matrix between element coordinate system and material coordinate system is stored once at the beginning of the simulation. As shown in Figure 1 for shell elements the transition from element to material coordinate system can be achieved through a simple rotation about element z axis. After calculation of the new stress tensor in the material model the stress tensor is transformed back to element coordinate system.

The available hardening models include analytical hardening laws for metallic and polymer materials. As an alternative, hardening can be defined directly via sampled stress-strain pairs. There is an option to include additional kinematic hardening. The kinematic hardening module extends the plastic hardening module. If it is defined, the plastic hardening module describes the total of isotropic and kinematic hardening. Therefore it is easy to switch between a simulation with pure isotropic hardening or a combined isotropic-kinematic hardening. All hardening models can be defined for different strain rates.

Vector transformation



$$T = \begin{bmatrix} \cos^2 \alpha & \sin^2 \alpha & 2 \sin \alpha \cos \alpha \\ 1 - \cos^2 \alpha & 1 - \sin^2 \alpha & -2 \sin \alpha \cos \alpha \\ -\sin \alpha \cos \alpha & \sin \alpha \cos \alpha & \cos^2 \alpha - \sin^2 \alpha \end{bmatrix}$$

$$\begin{bmatrix} \epsilon_x^m \\ \epsilon_y^m \\ \epsilon_{xy}^m \end{bmatrix} = T \begin{bmatrix} \epsilon_x^e \\ \epsilon_y^e \\ \epsilon_{xy}^e \end{bmatrix}$$

- 0 = Element coordinate system**
- m = Material coordinate system**
- A = Vector for definition of principal anisotropy direction**

Figure 1: Components of orthotropy vector R are stored in element coordinate system

The material model MF GenYld allows also for the description of anisotropic hardening. Here, anisotropic refers to anisotropy in the stress space, i.e. the hardening behaviour can depend on the current stress state. Specifically, the basic yield locus may be modified by correction factors for uniaxial compression, equibiaxial tension and compression and shear with respect to uniaxial tension. An overview of the possibilities is given in Figure 2. The correction factors can be a function of the equivalent plastic strain (based on work hardening).

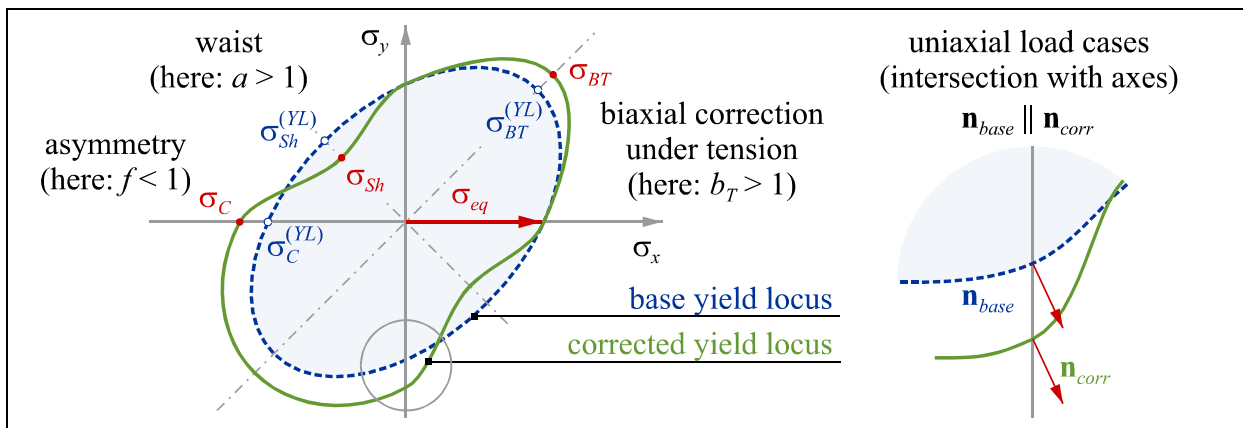


Figure 2: Schematic representation of anisotropic hardening of a yield locus

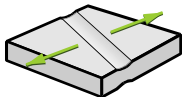
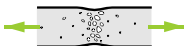

A corrected yield locus that has been modified by the above mentioned correction functions is smooth. For a pronounced “waist” and “asymmetry”, the yield locus might be concave, however. In such cases, the yield locus should not be used as a flow potential. The base yield locus should be used instead (non-associative flow). Experience shows that a yield locus with a moderate concavity does not cause numerical problems in stress calculation with an explicit time integration scheme anyway.

The module for anisotropic hardening in MF GenYld allows to approximate the real behaviour of some metallic materials (e.g. austenitic steels sheets, magnesium sheets and extrusions) and polymeric materials which may show a pronounced anisotropic hardening.

2.2 Comprehensive Failure Model CrachFEM

A correct prediction of a possible sheet metal failure is an essential part of a sheet metal forming or crash simulation. The use of the conventional forming limit curve (FLC) is the standard approach at an industrial level to this problem. The FLC concept is limited to the case of linear strain paths, however. The initial FLC is no longer valid in the case of nonlinear strain paths as described by Dell et al. [1]. The algorithm Crach (cf. Gese and Dell [2]) is included in CrachFEM. It allows for a transient prediction of localized necking in the case of arbitrary strain paths. The standard FLC approach is used for elements with small strains. The Crach algorithm is only introduced for elements with higher strains to avoid a significant increase of CPU time. In some cases also the postcritical strain after onset of necking (elongation of a sheet between initiation of necking and final fracture) can be relevant for a good prediction of a part behaviour in crash. Therefore, the Crach algorithm has been extended by a module for handling the post instability strain. The post instability strain should be nearly independent of the element size of the underlying shell element.

For advanced and ultra high strength steels as well as aluminium sheets, there is a risk of fracture without prior localized necking. A wide range of fracture models is compared and discussed by Wierzbicki [7], showing that the equivalent plastic strain at fracture cannot be described by simple one- or two-parameter models for all possible stress states. CrachFEM includes fracture models that account for ductile fracture (caused by void nucleation, void growth and void coalescence) and for shear fracture (caused by shear band localization). The stress triaxiality η is defined as the ratio σ_m/σ_{eq} of hydrostatic stress and von Mises equivalent stress and is typically used as a stress state parameter for ductile fracture. The parameter has been introduced by Rice and Tracy [5]. It was shown by Wierzbicki, however, that this parameter is only unique for the plane stress condition but not for the general case of a 3D stress state. Therefore a new stress state parameter β has been introduced in CrachFEM [8]. It is a function of stress triaxiality η and ratio of first principal stress and von Mises equivalent stress σ_1/σ_{eq} . The shear fracture model presented uses a stress state parameter θ which is a function of τ_{max}/σ_{eq} and the stress triaxiality η . Like β , this stress state parameter is unique for the general 3D stress state. Fracture limit curves of equivalent plastic strain at fracture $\epsilon_{pl}^{**}(\beta)$ and $\epsilon_{pl}^{**}(\theta)$ are determined experimentally and used as a basis for an integral damage accumulation in the sheet metal forming simulation.

		2D	3D
	Local instability (necking)	Initial FLC (approximate)	(1)
		Prediction with Crach	(1)
		Post-critical elongation	(1)
	Ductile normal fracture	$\epsilon_{eq}^{**} = \epsilon_{eq}^{**}(\eta)$	(2)
		$\epsilon_{eq}^{**} = \epsilon_{eq}^{**}(\beta)$	
	Ductile shear fracture	$\epsilon_{eq}^{**} = \epsilon_{eq}^{**}(\theta)$	

■ supported and tested (1) not reasonable
 supported as beta feature (2) not recommended

Figure 3: Schematic representation of failure mechanisms in CrachFEM

The model Crach for the prediction of localized necking and the two fracture models for ductile and shear fracture are combined in the comprehensive failure model CrachFEM. All modules of CrachFEM are summarized in Figure 3.

3 Mapping process for LS-Dyna + MF-GenYld + CrachFEM

3.1 Overview

Mapping is defined in this context as the process of transferring element variables from one FEM mesh to a target mesh. On material model level integration point variables have to be transferred. It has to be distinguished between tensor variables (e.g. stress tensor) and scalar variables (e.g. equivalent plastic strain). Beside integration point variables additional variables belonging to an element like for example the element thickness have to be transferred. An overview about relevant variables is given in Table 1.

	Related coordinate system	Remark
Element related values		
Thickness	n.r.	In case of fully integrated shell elements the thickness can be different
Scalar values at integration point		
Equivalent plastic strain	n.r.	
Scalar values describing membrane deformation history for necking module	values are expressed in material coordinate system	
Failure Risk (ductile normal fracture, ductile shear fracture, instability ...)	n.r.	
Tensors at integration point		
Stress tensor	Global coordinate system	
Backstress tensor	Material coordinate system	Only if kinematic hardening option is used
Damage tensors	Material coordinate system	
Anisotropy direction	Local element coordinate system	

Table 1: Possible Variables for Mapping of a MF-GenYld simulation in LS-Dyna

Typically the element size and orientation changes between the relevant meshes. It might also be the case, that the number of integration points can change. For material model MF-GenYld+CrachFEM it is not allowed to change the number of integration points. If the meshes are identical and the element orientation as well as the number of integration points does not change, mapping simplifies to initialization of the target mesh. For this investigation an equal number of integration points is supposed. The standard mapping functionality in LS-Dyna is used.

3.2 Requirements for a mapping with MF-GenYld+CrachFEM

If material model MF-GenYld + CrachFEM is used throughout the whole process of forming and crash two main requirements have to be fulfilled:

- Orthotropic mapping has to be used;
- All integration point variables have to be mapped.

While using an orthotropic elasto-plastic material model in LS-Dyna the stress tensor and the incremental total strain tensor is transformed into material coordinate system, before stresses are updated in the material model. Afterwards these quantities are transformed back to element coordinate system. If the mesh changes from one process step to another the material coordinate system has to be updated. Then tensor components can then be transferred like scalar integration point variables. For material model MF-GenYld+CrachFEM all integration point variables have to be transferred.

In addition to the stress tensor the material model MF-GenYld+CrachFEM calculates the following tensors with reference to the material coordinate system:

- Damage tensor for ductile normal fracture;
- Damage tensor for ductile shear fracture;
- Deformation history for the instability calculation with algorithm Crach

The deformation history, or strain path, is characterized by three variables in dependency on the equivalent plastic strain. The normalised thickening rate ρ is an indicator for the stress state of the element. Contrary to the ratio of principal strain rates α , which shows a singularity for plane strain compression, the thickening rate ρ is unique and monotonic for all possible stress states. It is defined as:

$$\rho = \frac{\dot{\varepsilon}_1 + \dot{\varepsilon}_2}{\dot{\varepsilon}_{eq}}$$

The second variable is the angle between rolling direction and first principal strain, γ , which can be calculated as

$$\gamma = \frac{1}{2} \cdot \arctan\left(\frac{\dot{\varepsilon}_{xy}}{\dot{\varepsilon}_x - \dot{\varepsilon}_y}\right)$$

and the third variable is the strain rate $\dot{\varepsilon}$. All deformation history variables are evaluated in the neutral fibre, i.e. in the central integration point over the thickness of the shell element.

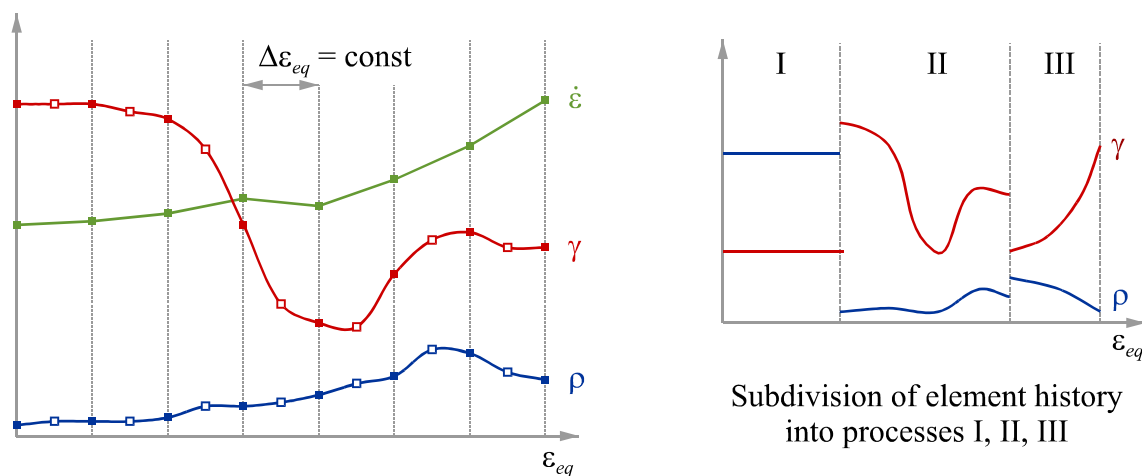


Figure 4: Storage of deformation history (left) and subdivision into processes (right)

The deformation history ($\dot{\varepsilon}$, ρ , γ) is stored at intervals of equivalent plastic strain, $\Delta\varepsilon_{eq}$. The history is considered as smooth, i.e. there are no abrupt changes of the stored history variables within one process. The strain rate $\dot{\varepsilon}$ is sampled with a linear approximation, the thickening rate ρ and the angle γ are sampled with quadratic functions with an additional value at the middle of each interval as shown in figure 4 (left).

The memory for storing the strain path must be allocated beforehand. This means that the number of stored states per element is limited. In order to use the available memory in an effective way, history data is compressed in two ways. First, it is compressed on machine level by splitting one value into two values of lesser resolution. Second, data is compressed by digitalising over the possible range of values. The normalised thickening rate can only have values of

$$-1 \leq \rho \leq 1,$$

and the angle is limited to values of

$$-\frac{\pi}{2} \leq \gamma \leq \frac{\pi}{2} .$$

Using the possible ranges of these two variables, one can normalise them and subdivide them into N intervals. Only the reference to the interval of the actual values of ρ and γ is stored. This allows for storing more data values at the cost of losing resolution. This loss is only minor and the history is still represented reasonably. For example, if $N = 100$, the resolution is $\Delta\rho = 0.02$, and $\Delta\gamma = 0.01 \cdot \pi = 1.8^\circ$. Even with this compression scheme, history storage will eventually run out when high deformations are reached. In this case, a third compression mechanism is used: The abscissa resolution is coarsened for history data in the distant past, i.e. towards zero equivalent strain.

In the course of its deformation history, a part may undergo changes that cannot be reflected by an FE simulation. For example, a deep-drawn part may be annealed before being processed further. The deformation history, however, must be continuous. In order to represent abrupt changes in deformation history and, optionally, in the material properties, the overall history can be subdivided into processes. Between processes, the history need not be smooth; it can change abruptly. Within one process, the history data is continuous, but at the borders between processes, it can change abruptly, as shown in figure 4 (right).

In MF_GenYld+CrachFEM for LS-DYNA the number of processes is currently limited to two, i.e. only one process border may be defined. The definition of processes is not based on simulation runs. One run may comprise several processes, and a process may spread over several runs.

A process border is defined by a time t_p . When this time is reached, a new process begins. Setting t_p to zero considers that simulation run as a new process. Setting t_p to a value greater than the termination time means that the whole run is done in one process. (The input allows for the specification of two process borders.)

When resuming the simulation of one part in a separate run, the data is usually carried over from the previous simulation, either by using a full restart (when the mesh is the same) or by a process called mapping (when the mesh between runs is different).

3.3 Mapping of material coordinate system

Even if elasto-plastic material behaviour is isotropic or transversal-isotropic an orthotropic mapping has to be chosen when using material model MF-GenYld+CrachFEM, as for all failure mechanism the direction of deformation is relevant. In case of instability calculation this is due to the Bauschinger effect (combined isotropic-kinematic hardening). For ductile and shear fracture the directionality results from the internally calculated damage tensor. Here damage is accumulated with respect to orientation of plastic deformation. Currently the standard mapping procedure in LS-Dyna uses all integration point variables from the closest integration point of the primary mesh. From this element also the orthotropy direction is mapped. In Figure 5 one element from the primary mesh and one from the target mesh is shown schematically. The orthotropy direction has to be transformed into the new element coordinate system of target element 'e2'.

The new transformation matrix (for transforming tensor quantities to material system) can be calculated. All tensor quantities used in the material model MF-GenYld are defined in the material coordinate system. Therefore the tensor components can be transferred in the mapping process without modification.

The principal direction of orthotropy in the pre-stretched sheet from above is orientated in 45° -direction toward x-axes as shown in figure 6. The orientation is defined using AOPT definition in material input card.

The results of the pre-stretched sheet are mapped onto the tensile specimen from above. The orientation of the material x-axes has been transferred from the pre-stretched sheet as shown in figure 7.

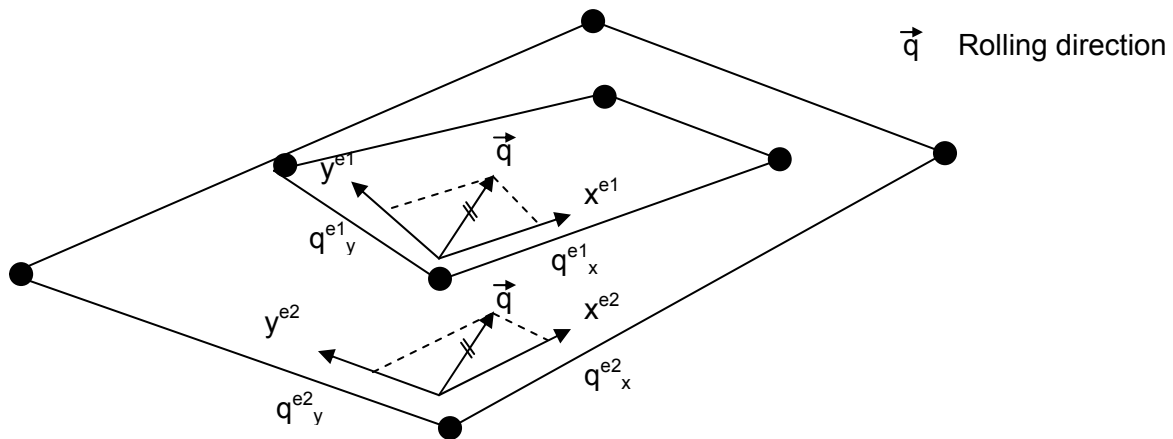


Figure 5: Mapping of Rolling direction to target shell element 'e2'

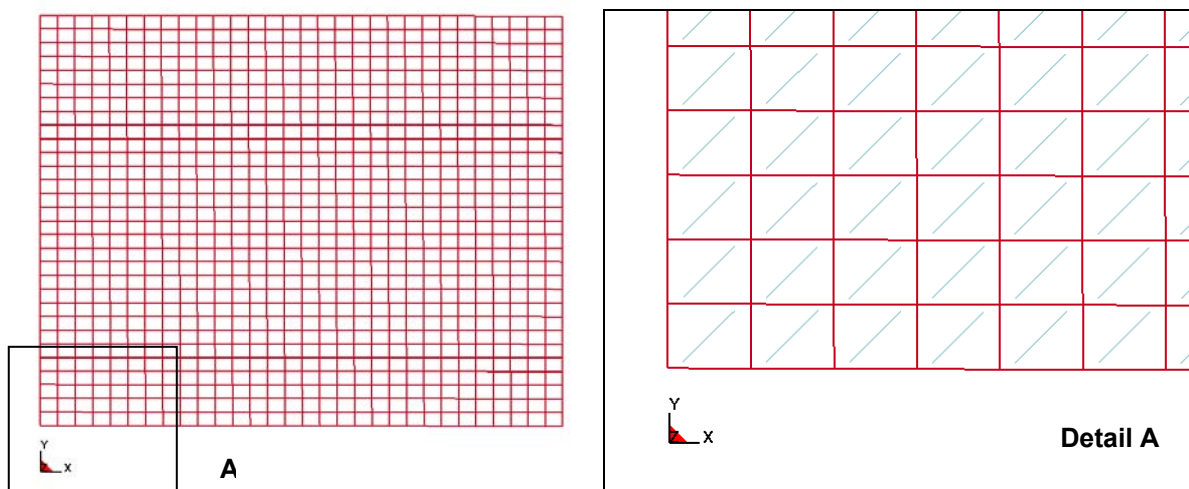


Figure 6: Principal direction of orthotropy; the orthotropy direction is displayed using bar elements, oriented based on the orthotropy definition in the dynain mapping file, written at the end of the simulation (for the visualization a script, developed by MATFEM has been used);

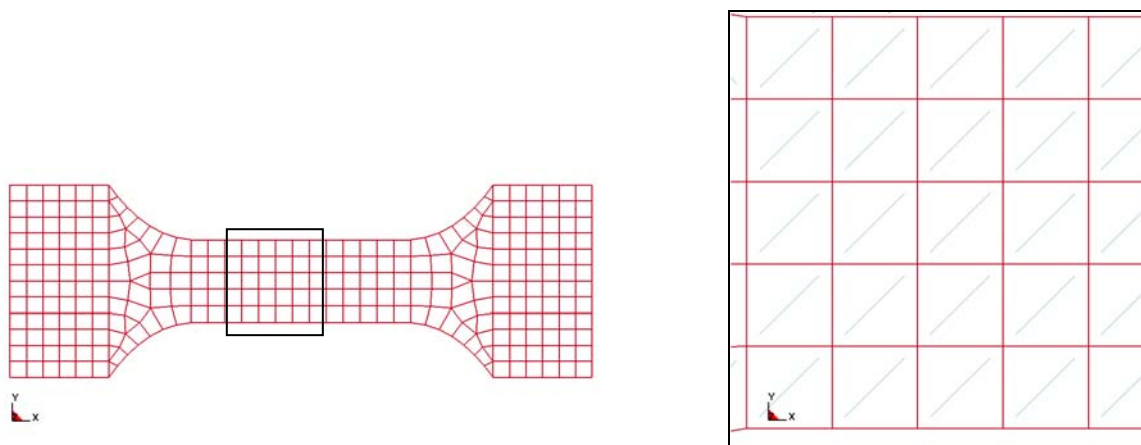


Figure 7: Principal direction of orthotropy; the orthotropy direction is displayed using bar elements, oriented based on the orthotropy definition in the dynain mapping file, written at the end of the simulation (for the visualization a script, developed by MATFEM has been used)

3.4 Example for orthotropic mapping

The use of complex material models throughout the whole process requires a mapping procedure, which is able to transfer all required integration point variables from one mesh to another. Different rules for mapping might be necessary (e.g. a simple rule would be to map the data from the next integration point; another to use weighted values of different integration points). Subsequently results are shown, obtained by using the LS-Dyna internal mapping procedure. For MF_GenYld+CrachFEM an extended version has to be used, as the resolution of 3 valid digits after decimal point in the 'initial stress' definition is not sufficient for CrachFEM.

In figure 8 a sheet is pre-stretched in global x and global y-direction to 24% equivalent plastic strain under plane strain condition. The principal orthotropy direction is directed along global x-axes in both cases.

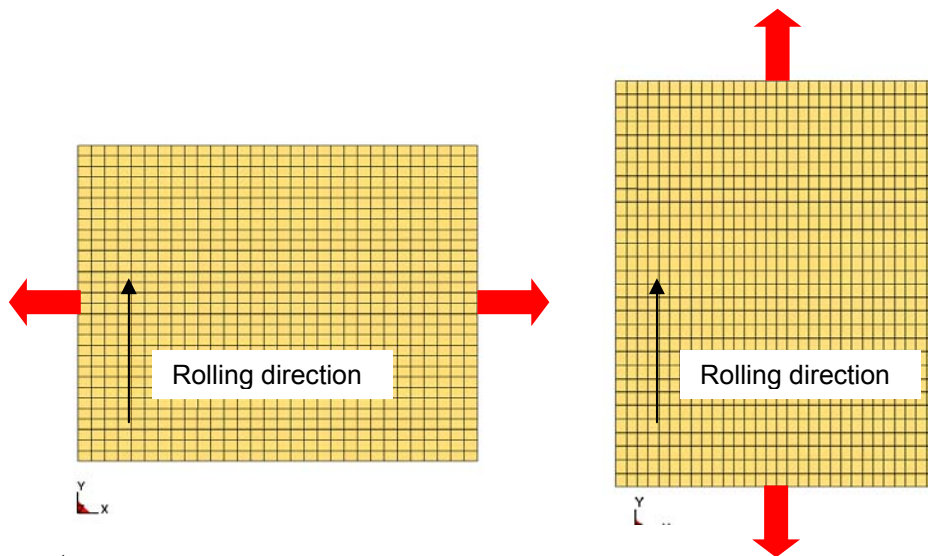


Figure 8: 1st step: Pre-stretch to 24% equivalent plastic strain; the loading condition is plane strain

For the simulation of a second load step a uniaxial tensile specimen is cut out of the sheet (Figure 9). Subsequently two simulations of the tensile specimen are considered using the initialized integration point variables from a pre-deformation in x- and in y-direction. In both cases the loading condition of the pre-deformation is plane strain.

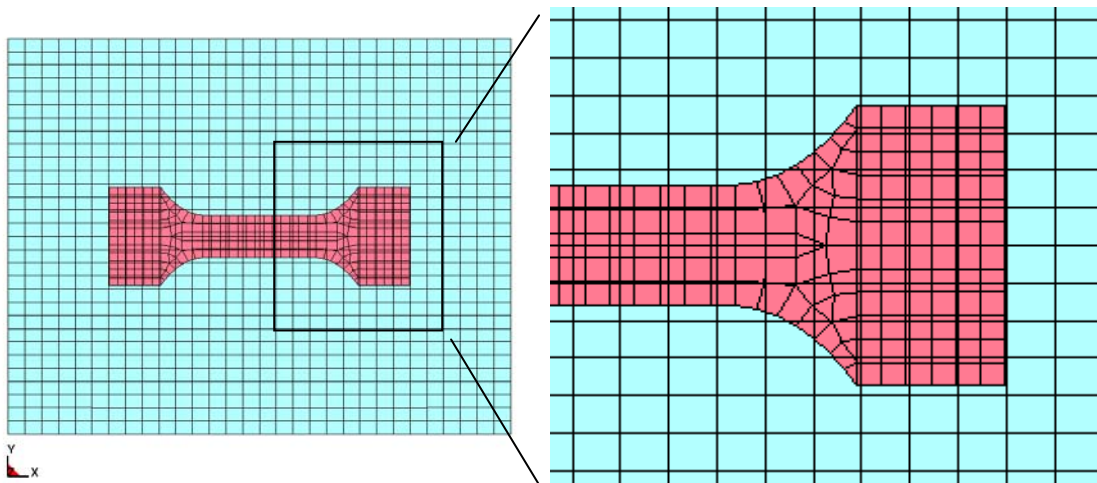


Figure 9: Incongruent meshes between 1st and 2nd step

Subsequent tensile loading leads to failure due to instability as shown in figure 10. Elements with a failure risk for instability over 1.0 are deleted. The possible deformation up to necking for the specimen with rotation of the loading direction in the 2nd deformation step is higher due to the increased kinematic hardening after load reversal. The kinematic hardening is accounted for in the algorithm Crach even if there is only isotropic hardening in the FEM material model. Without orthotropic mapping it would be not able to express the two different deformation histories. Both histories are plane strain tension followed by uniaxial tension. The only difference is the orientation of the uniaxial tension in the 2nd deformation step relative to the pre-strain. This rotation can be expressed only by the use of a reference coordinate system (defined by the rolling direction).

The influence of non-linear strain paths onto failure behaviour is not restricted to instability. Also ductile normal fracture and ductile shear fracture can be sensitive for nonlinear load paths. This material behaviour can be also described with CrachFEM, but has not been discussed here.

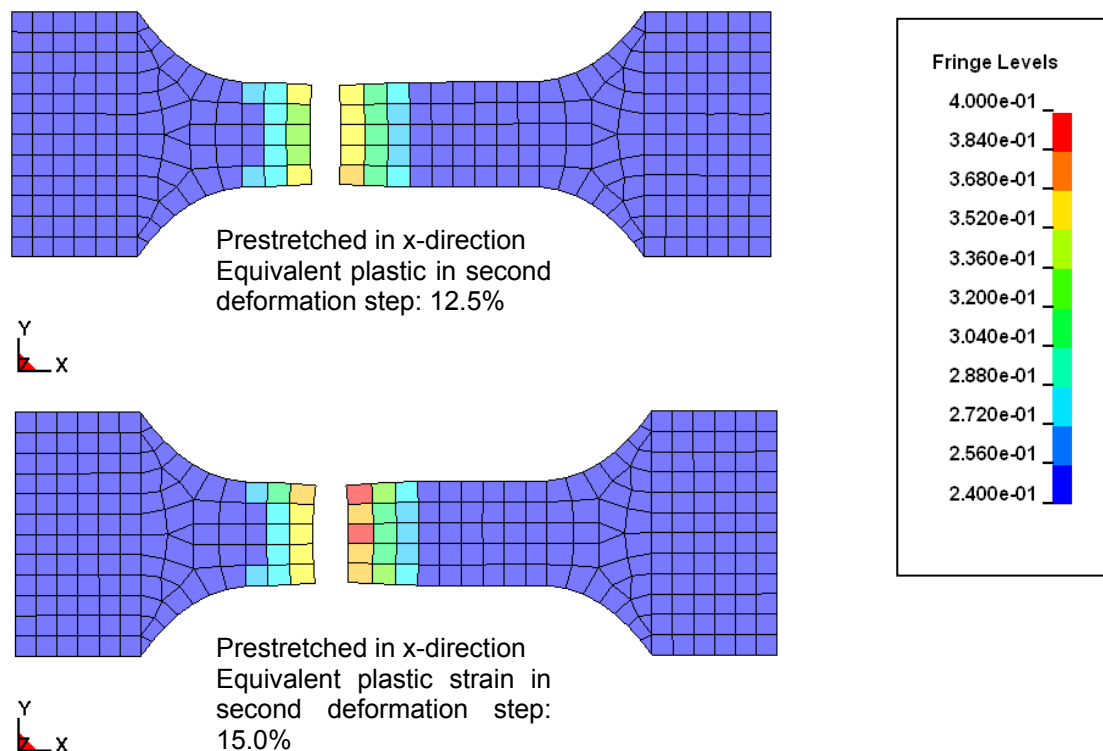


Figure 10: Distribution of equivalent plastic strain after fracture

In figure 11 the force – elongation curve of the tensile specimen is shown. The different amount of plastic strain at failure due to instability leads to different global displacement at fracture. Remaining stresses at the end of the pre-deformation lead to oscillation of the forces in the 2nd deformation step.

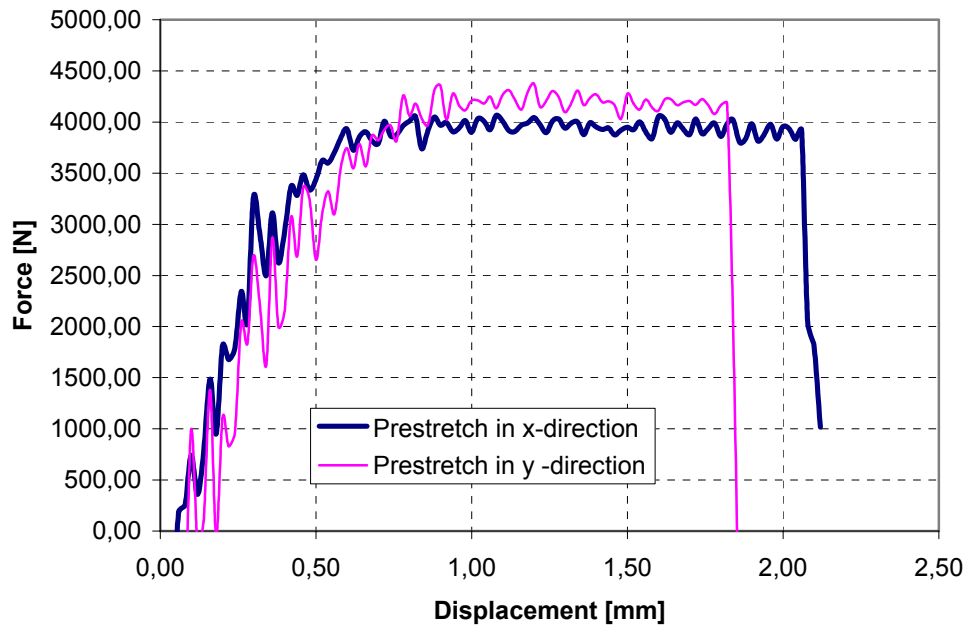


Figure 11: Force – displacement diagram for pre-deformation in x- and in y direction

4 Forming to Crash Mapping for Automotive Components

Different examples for the mapping between forming simulation and crash simulation of automotive components using material model MF-GenYld+CrachFEM will be demonstrated in the oral presentation.

5 Summary

Within this study the orthotropic mapping functionality in LS-Dyna has been used together with user material model MF-GenYld+CrachFEM. In case of an orthotropic material model the local orthotropy direction is transferred to the target mesh. This orthotropy direction is the basis for the calculation of the transformation matrix from element to material coordinate system. On material model level all tensor/vector components used in the material model are transferred from element to local material coordinate system. If the same material model is used in the first and second process step all integration point variables can be transferred directly. It has to be mentioned, that the best results can be obtained, if the geometrical difference between both meshes is small. Large deviation in the mesh size can lead to significant errors. In case of material model MF-GenYld+CrachFEM the same number of integration points over the thickness has to be used, if mapping is necessary.

As an example the failure behaviour of a tensile specimen, which is cut out of a pre-stretched sheet, has been shown. The tensile specimen shows a different elongation at failure, dependent on the direction of pre-deformation. A pre-stretching in cross direction towards the loading direction of the tensile specimen leads to a larger elongation at failure due to instability, compared to a pre-stretching in loading direction.

6 References

- [1] Dell, H.; Gese, H.; Keßler, L.; Werner, H.; Hooputra, H.: Continuous Failure Prediction Model for Nonlinear Load Paths in Successive Stamping and Crash Processes; in: *New Sheet Steel Products and Sheet Metal Stamping. SAE Proceedings SP-1614*, paper 2001-01-1131, pp. 113–122. SAE, 2001 World Congress, Detroit, Michigan, March 5–8, 2001
- [2] Gese, H.; Dell, H.: Numerical Prediction of FLC with the Program CRACH, *Proceedings of FLC Zurich 2006*, 15th – 16th March 2006, IVP, ETH Zurich, Switzerland
- [3] Keßler, L., Werner, H., Horstkott, D., Dell, H., Gese, H.: Material Selection for an Ultra High Strength Steel Component Based on the Failure Criteria of CrachFEM, *Proceedings of Numisheet 2005*, Detroit
- [4] Lanzerath, H., Bach, A. Schilling, R., Oberhofer, G. und Gese, H.: Failure Prediction of Hot-Formed Steels in Crash, *Tagungsband der 13. VDI-Tagung „Berechnung und Simulation im Fahrzeugbau“ in Würzburg* (18.10.2006)
- [5] Rice, J.R.; Tracy, D.M.: *J. Mech. Phys. Solids*, 17 (1969) pp. 201-217
- [6] Werner, H.; Dell, H.; Metzmacher, G.; Kessler, L.; Heath, A: Methodology, Validation and Application of a Failure Model Based on Transient Forming Limit Curves for Coupled Stamping and Crash Processes as Part of the IMPACT Project, *Proceedings of Europam 2003*, October 16th to 17th, Mainz, Germany
- [7] Wierzbicki, T.; Bao, Y.; Lee, Y.-W.; Bai, Y.: Calibration and evaluation of seven fracture models, *International Journal of mechanical Sciences*, 47 (2005) 719-743
- [8] Dell, H.; Gese, H. and Oberhofer, G.: CrachFEM – A Comprehensive Approach For The Prediction Of Sheet Metal Failure, *NUMIFORM '07, Materials Processing and Design: Modeling, Simulation and Applications*, Porto, 18 - 21 June 2007, American Institute of Physics, CP908, pp. 165-170