Computational Material Models for TSCP Plastics Comparison of the Deformation Behavior with *MAT_24 and *MAT_SAMP-1 with DIEM

M.Dobes[1], J. Navratil[2]

[1,2]Robert Bosch s.r.o. Ceske Budejovice

1 Introducing

We can use a lot of computational material models in LS-DYNA, which are developed directly for plastics. Some models are simple or more sophisticated, with non-symmetrical behavior in tensile/pressure area and advanced with failure and damage techniques. We focused on the comparison between standard computational model *MAT_24 and *MAT_SAMP-1 with DIEM failure and damage model. This article shows results of step by step development during recent years. We started with classic simple model *MAT_24 without strain rate, second version contained strain rate dependence, next one was implementation *MAT_ADD_EROSION card with empirically estimated criterions and at the end *MAT_SAMP-1 with failure and damage model DIEM. The new final computation material model *MAT_SAMP-1 was based on the original experimental data used for *MAT_24 and additional tests (3-point bending test, puncture test, etc.). The strain rate effect is considered based on the experiments in defined range and other strain rate values were calculated based on the analogue with Johnson-Cook constitutive material model. The attention is focused on tensile/pressure definition of computational material model in plasticity. This is very important effect for good prediction of the cracks with using damage material model DIEM. The next characteristic is an accumulation of the damage which leads to rupture and finite elements deletion. This property is depending on the stress triaxiality. The results of deformation and stress response are very different for both approaches and SAMP-1 gave closer results to the experimental reality, but sometimes we obtained non-physical results. This discrepancy is subject of this thesis also.

Note: SAMP-1…Semi-Analytical Model of Polymers
DIEM … Damage Initiation and Evolution Model
TSCP …Typical Semi-Crystal Plastics

***KEYWORDS: SAMP-1, DIEM, Damage, Failure, TSCP, Strain Rate**

2 History of computational models for TSCP

A lot of computational material models were used in development explicit simulations for FSM (Fuel Supply Module). We started in year 2012 with LS-DYNA simulations in our department. We focused on the crash analyses of the FSM, concretely on the failure and damage prediction on the flange, due to the most important part of the passive safety. The material of the flange is TSCP polymer without fibers. The flange is moulded plastic part. The level of the computational material model is very important part of the complete computational approach of failure and damage prediction.

2012
*MAT_24 (Strain Rate Dependence)

2014
*MAT_24 + *MAT_ADD_EROSION

2016
*MAT_SAMP-1

2016, 2017
*MAT_SAMP-1 + DIEM

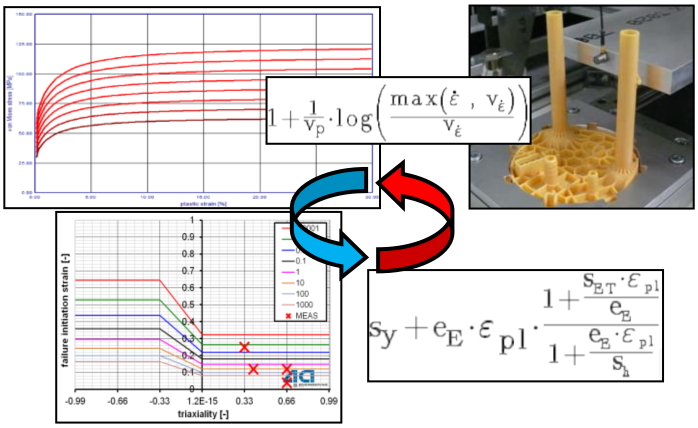

Fig.1: History development of the computational material model for TSCP

2.1 *MAT_24 (*MAT_PIECEWISE_LINEAR_PLASTICITY)

This model was first version of the computational material model for TSCP. The definition of this model is very similar to classic structural static model. The elastic part is defined by Young Modulus with Poisson ratio. The plasticity can be defined with different approaches. We use multi-plasticity curves (Stress vs. Plastic strain) for strain rate range. The failure and damage model weren't considered. The comparison between simulation and experiment shows that this model gives correct direction in next development. Sometimes we have problems with more softening behavior of the computational material model in simulation compared to experiment.

```
*MAT_PIECEWISE_LINEAR_PLASTICITY
$TSCP
$#      mid        ro         e        pr      sigy      etan      fail      tdel
     101000 1.3930E-9    2438.6      0.35
$#        c         p      lcss      lcsr        vp
      0.000     0.000    100000         0  1.000000
$#     eps1      eps2      eps3      eps4      eps5      eps6      eps7      eps8
      0.000     0.000     0.000     0.000     0.000     0.000     0.000     0.000
$#      es1       es2       es3       es4       es5       es6       es7       es8
      0.000     0.000     0.000     0.000     0.000     0.000     0.000     0.000
```

*Fig.2: Text format of the *MAT_24 in basic form*

The basic material experiments were made for validated strain rate range $\dot{\varepsilon} = 0.001 - 50 s^{-1}$. Other curves were aproximated by modified Johnson Cook constitutive model, equation (1), see Fig. 3:

$$f_s = \left(1 + \frac{1}{\nu_p} * \log\left(\frac{\max(\dot{\varepsilon}_{p\_eff}; \dot{\varepsilon}_{p\_eff0})}{\dot{\varepsilon}_{p\_eff0}}\right)\right) \tag{1}$$

Note:

f_s *...strain rate scaling factor,* ν_p *...strain rate dependency factor*

$\dot{\varepsilon}_{p\_eff}$ *...effective plastic strain rate of the scaled curve,* $\dot{\varepsilon}_{p\_eff}$ *...effective plastic strain rate of the base curve*

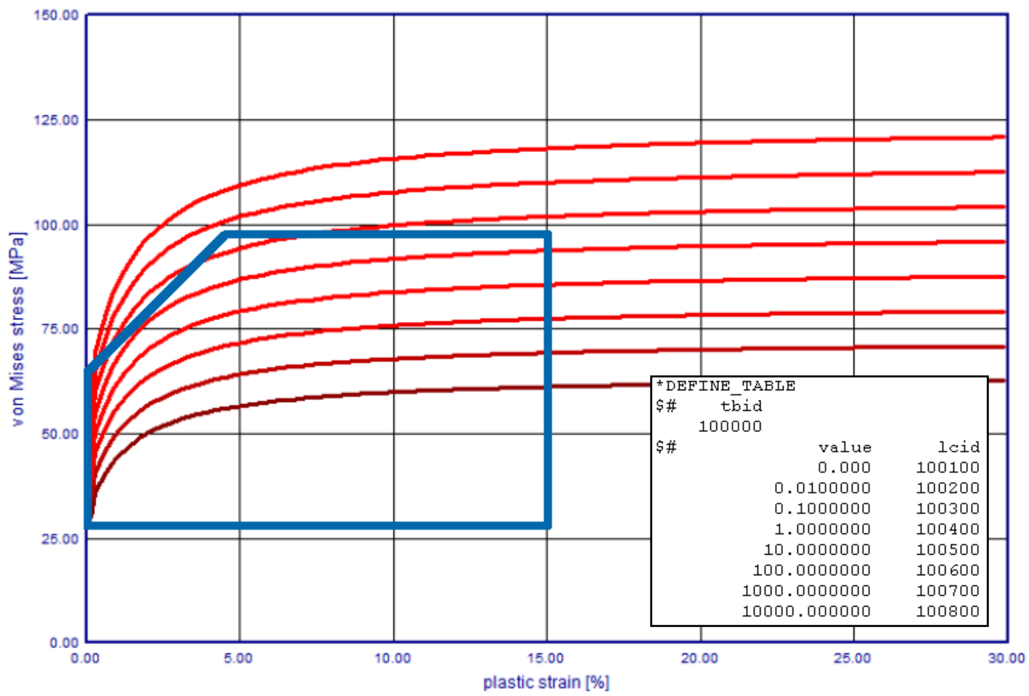

*Fig.3: Definition of the *MAT_24 in plasticity area – strain rate dependency*

2.2 *MAT_24 + *MAT_ADD_EROSION card (empirically estimated)

This computational material model contains finite element erosion based on the defined parameters (stress, strain, etc.). This additional card offers a lot of parameters and almost each new LS-DYNA version has added new erosion parameters. This basic part of this card define only very simple definition of the finite element definition (failure). If you would like to use some damage algorithm then you have to definite additional parameters (GISSMO, DIEM, etc.), but this option will be described in next chapters. This contribution continues on the article from 10th European LS-DYNA Conference 2015-*Solving Crash Problems of the Fuel Supply Modules in the Fuel Tank, Generally Plastics Parts.*

```
*MAT_ADD_EROSION
$#      mid      excl    mxpres     mneps                          numfip
     101000                                                             1         2
$#   mnpres     sigp1     sigvm     mxeps     epssh     sigth   impulse    failtm
      0.000   100.000     0.000       0.1
```

*Fig.4: Definition of the *MAT_ADD_EROSION card – 2 parametrical definitino S1max and Eps1*

The previous model 2.1 was used as basic part of this model. The material card *MAT_ADD_EROSION was added. The parameters of failure were estimated based on the experiment of the real part (plastic guide rods concept). We used two parametrical definition of the element erosion. The first parameter is *Maximum Principal Stress (S1max)* and second is *Maximum Principal Strain (Eps1).* Both values must be fulfilled together. Next definition was number of the integration points, we used tetrahedrons type 16 (4-5 integration points). The defined values of *S1max* and *Eps1* must be reached on 1 integration point for deleting of the finite element. If we use this option then the finite element erosion is very similar to fragile cracking in high strain rate loading like as in crash experiment, but this approach is very simple and it does not consider other aspects which are very important for good crack prediction (tensile, pressure area, damage model, etc.).

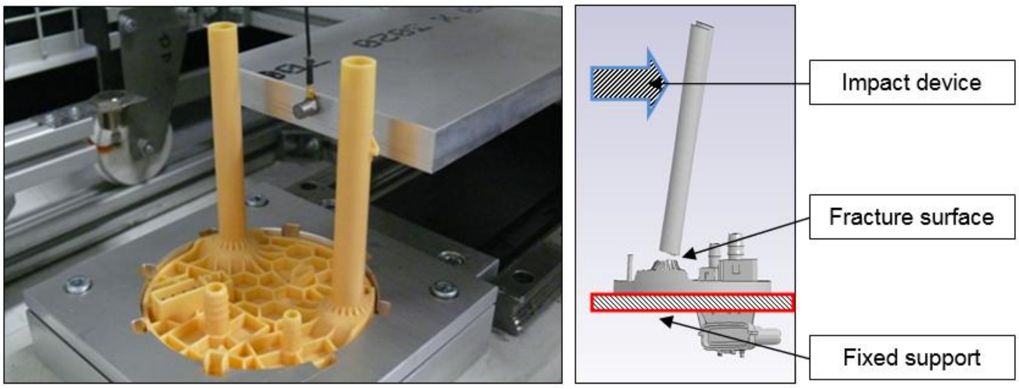

Fig.5: Impact Test – developed for high-speed loading of the FSM flange

The estimation of the finite element erosion parameters was made during internal Impact Test. The recording by high-speed cam was used for showing of the crack initialization. The parameters were fitted based on the starting time of crack. The agreement of the final crack propagation between simulation and experiment is sufficient but for special designed crack zones on the flange is necessary to use more sophisticated computational material model. The important differences could be for example non-symmetrical stiffness behaviour in tensile and pressure area (triaxiality dependence).

*Fig.6: Experiment Bosch Impact Test – estimation of the 2 parameters *MAT_ADD_EROSION card*

2.3 *MAT_SAMP-1

This material model was developed directly for plastics, because sometimes approximation by metallic computational material model is not sufficient for good deformation and stress modeling. In plastics area we obtain often very non-liner deformation behavior, strong strain rate dependence, plastic necking (softening/hardening phase), non-symmetrical behavior in tensile, shear and pressure loading (triaxiality) etc. We cooperated with *4A Engineering* company on development of *MAT_SAMP-1 for our plastic (TSCP). The subjects of this development were a lot of experiments to obtain material data. We continued in the previous experimental measurements from year 2012, where we measured experimental data for *MAT_24. Sometimes there can be problem in moulded plastic, because we receive different material characteristics for two batches of plastics (the same TSCP, but batch 2012 and batch 2016). In our case we made comparison measurement of basic material characteristics (static tensile test). We compared Young modulus, Yield stress, and Strain at break. The comparison shows that this approach and next measurement can be realized. The material characteristics were very similar and reproducible.

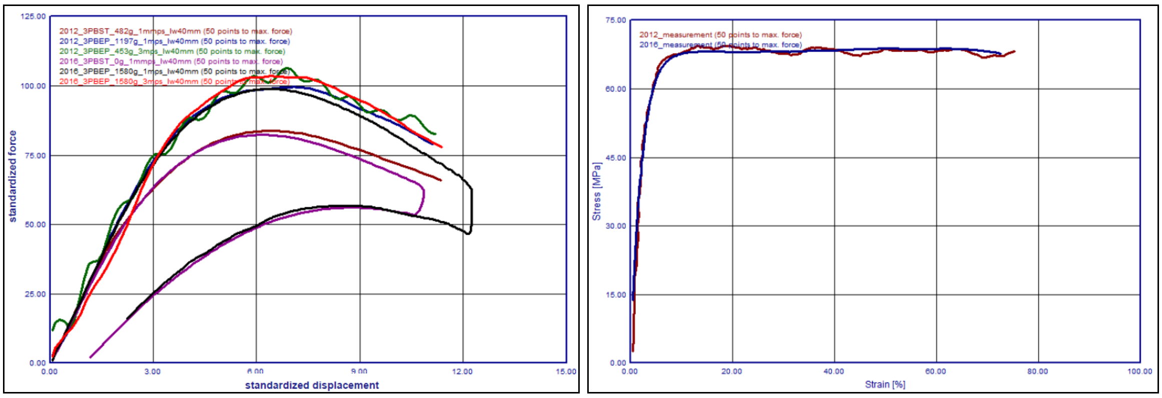

Fig.7: Comparison measurement batch 2012 and batch 2016 (3-point bending test and tensile test)

So, based on these results we continued with additional measurement suitable for *MAT_SAMP-1. The company 4A Engineering use tensile test, 3-point bending test (clamped/free) and puncture test. These tests cover complete important triaxiality area for us. Our computational material model contains pressure/tensile dependency; these phenomena are given by flow curves (stress-plastic strain). These values are represented by true stress and true strain value (DIC evaluation method). The tensile curves are setup as flow curves for different strain rates. The pressure curve is given only for static loading and other pressure curves are generated by amplification factor (dynamic/static ratio) from tensile yield stress. The used model has implemented plastic Poisson ration, which was measured by optical method. The DoE analysis was made to find tensile/compression factor, to better represent displacement-force response, but it can lead to instability.

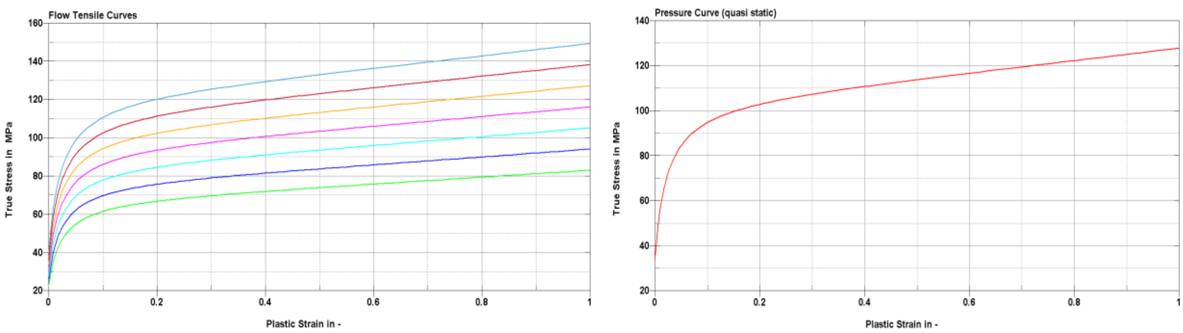

Fig.8: Flow curves of tensile/pressure loading state

The complete fitting process was made internally in 4A Engineering. The computational material model was developed and fitted for 10-nodes tetrahedrons (type16) and element size 0.66-1.5mm. The complete model is compromise for all types of tested loadings, but correspondence between separate tests is sufficient. This basic model of *MAT_SAMP-1 is basis of next advanced model with failure and damage. The *MAT_SAMP-1 mainly deformation part is very important for correct deformation in loading way, which precedes failure and damage model. We can see that the agreement between basic experiments and simulations are very good.

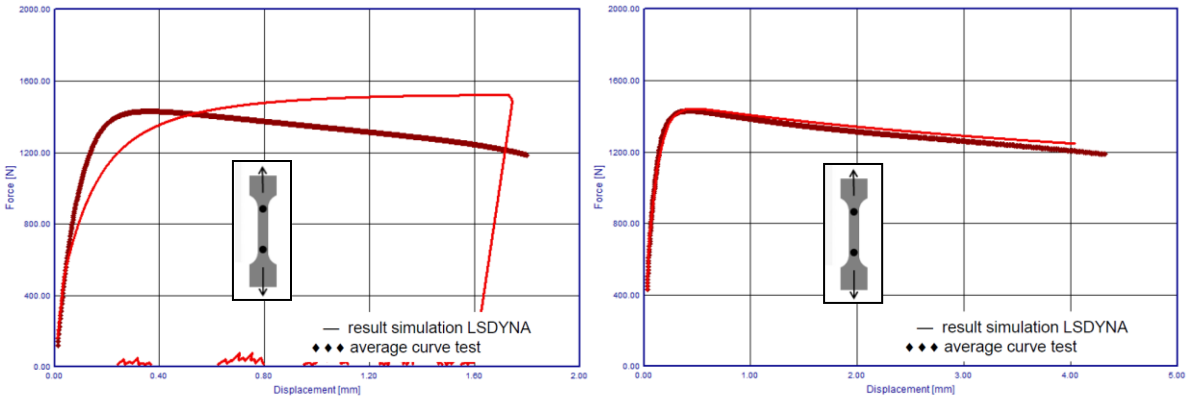

Fig.9: Tensile static test - compression/tensile factor α=1.53 and right figure α=1.18

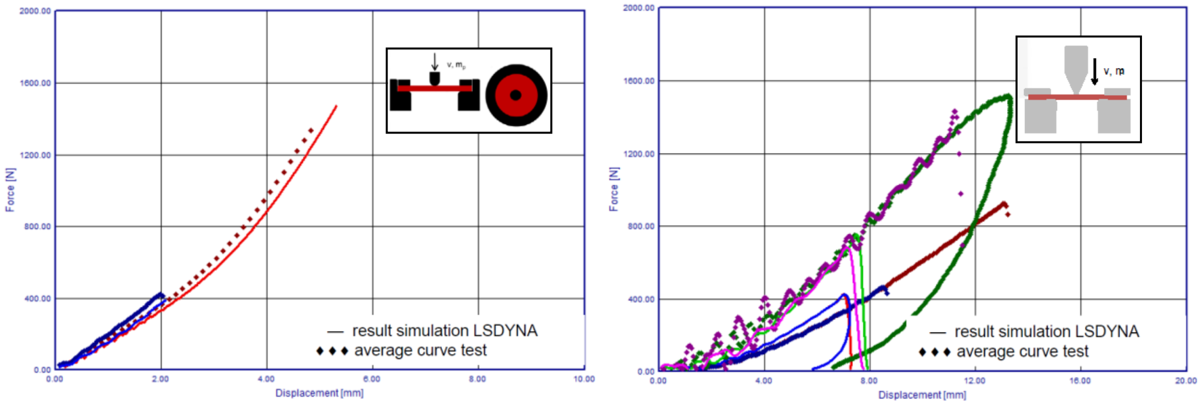

Fig.10: Puncture test and right figure 3-point bending clamped test

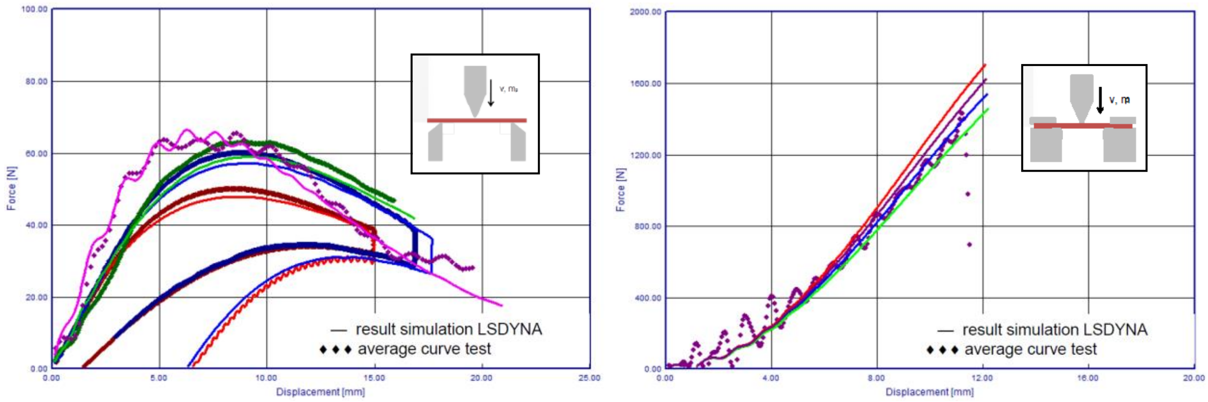

Fig.11: 3-point bending test a) without clamped, b) DoE analysis of compression/tensile factor

This model gives better deformation-force response than *MAT_24, due to higher stiffness in pressure (compression) loading area. The comparison was made on the real practice task and is showed in next part of this contribution. The aim of this study is obtaining of the computational material model for our plastic TSCP, we require failure and damage model mainly. So, based on this request we continued with extended version of the *MAT_SAMP-1. We added failure and damage computational model DIEM. The all theoretical aspects are in detail described in contribution *SAMP-1: A Semi-Analytical Model for the Simulation of Polymers from 4. LS-DYNA Conference, Bamberg 2005.* The development of this model still continues and some bugs are eliminated each year in new release of LS-DYNA version.

2.4 *MAT_SAMP-1 + DIEM

We have sophisticated computational material model without failure and damage from previous chapter. In this moment we applied failure and damage model, in our case DIEM. We need *MAT_ADD_EROSION card for definition DIEM method. We have to defined failure and damage, because between these two notions is difference.

Failure model – the crack occurs when a failure variable reaches a critical value

$$\omega_D = \int_0^{\varepsilon^P} \frac{d\varepsilon^P}{\varepsilon_D^P} \tag{2}$$

Damage – stiffness or strength of material model is reduced as function of damage variable

$$\dot{D} = \frac{\dot{u}^P}{\frac{\partial u_f^P}{\partial D}} \tag{3}$$

The equation (2) is only as example for initiation criteria – ductile. This calculation is based on the accumulation of plastic strain. The equation (3) is definition for evolution criteria – linear damage evolution. The plastic displacement is given by,

$$\dot{u}^P = \begin{cases} 0 & \omega_D < 1 \\ h\,\dot{\varepsilon}^P & \omega_D \geq 1 \end{cases} \tag{4}$$

where h is characteristic length of finite element. The variable is used in our computational material model. The listening of these variables is possible in new version of LS-DYNA, version 9.0.0. We have to define new keyword *DEFINE_MATERIAL_HISTORIES, then we can show parameters, Instability (Initiation) and Plastic Strain Rate, Effective damage D (evolution) etc. The ductile damage initiation is function of triaxiality and plastic strain rate $\varepsilon_D^P = \varepsilon_D^P(\eta, \dot{\varepsilon}_D^P)$. The stress triaxiality is defined as

$$\eta = -p/q \tag{5}$$

, where *p* is pressure and *q* von Mises equivalent stress. The all equations are showed for our case of computational material model. We have dependence of the failure plastic strain on the stress triaxiality for different strain rates. The initiation failure strains in pressure area are two times higher than in tensile area for complete strain rate range.

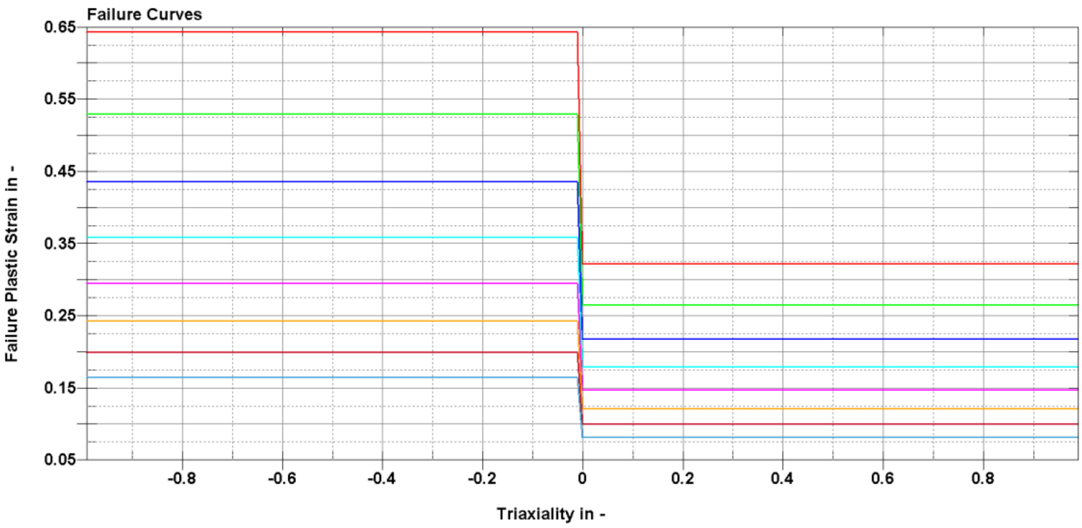

Fig.12: Failure curves in strain rate range (Triaxiality vs. Failure Plastic strain)

The damage function is defined by curve of Triaxiality vs. Plastic displacement, from our purposes this curve has constant character u_f^p *=0.01* for complete triaxiality range. This option secures very brittle crack propagation. We show complete cards for *MAT_SAMP-1 and *MAT_ADD_EROSION with DIEM, because sometimes the practice description is suitable.

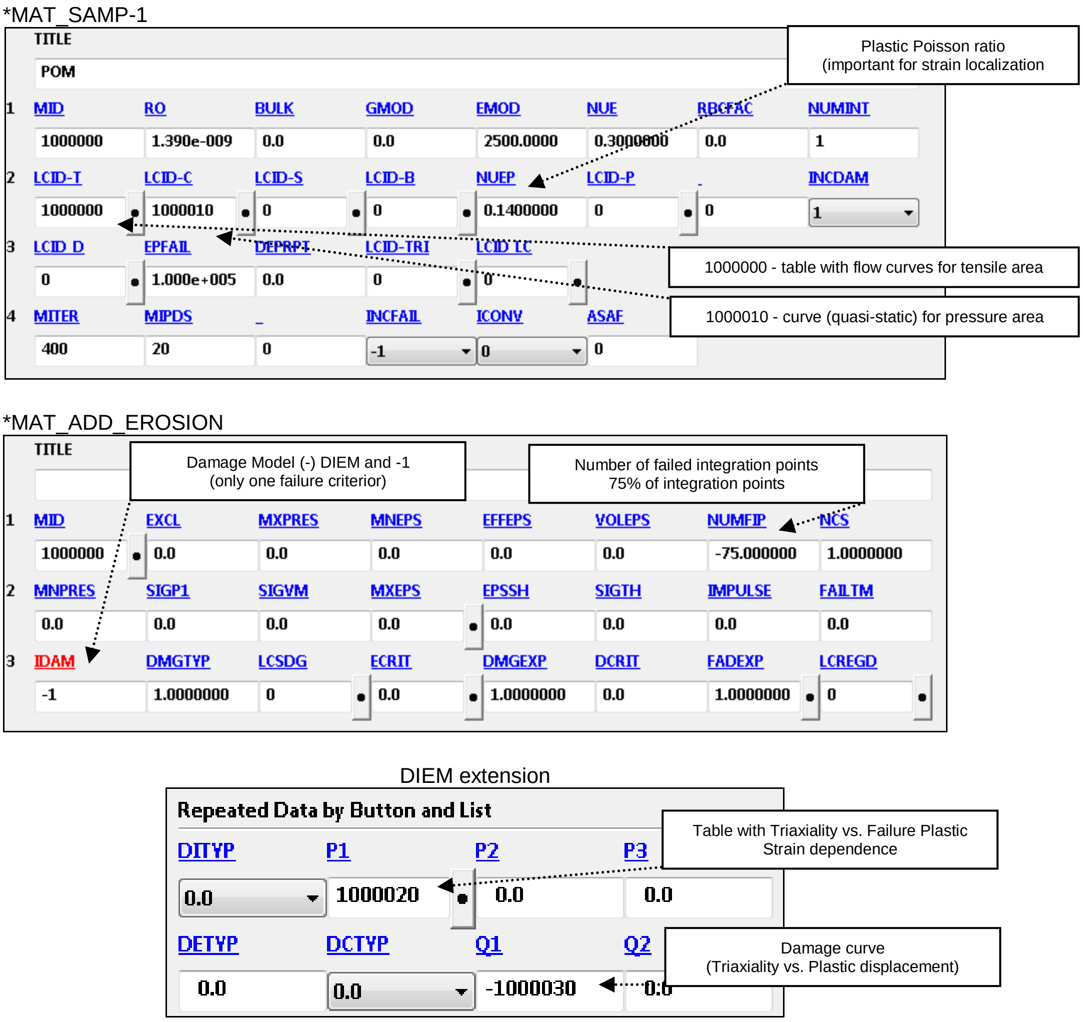

Fig.13: Schema of the computational material model of TSCP in LS-PrePost software

3 Comparison in definition of *MAT_24 and *MAT_SAMP-1

The basis of the both material models are very similar, see equations below, but *MAT_SAMP-1 offers non-symmetrical behaviour in triaxiality space. The using of the additional card *MAT_ADD_EROSION is possible for both models. The quality of complex computational material model is given by basic model and additional failure model. This combination must be fitted together and checked for requirement element size, element type etc. The plastic behaviour is described by using of meta model (7) of general Schmachtenberg equation (6) for both cases,

$$\sigma = E\varepsilon \frac{1-D_1\varepsilon}{1+D_2\varepsilon} \tag{6}$$

$$\sigma = \sigma_y + E\varepsilon_{PL} \frac{1+\dfrac{E_{ET}\varepsilon_{PL}}{E}}{1+\dfrac{E\varepsilon_{PL}}{\sigma_h}} \tag{7}$$

where σ_y is Yield stress, E is Young Modulus, E_{ET} is hardening tangent modulus, σ_h is hardening stress plateau and ε_{PL} is plastic strain.

4 Crack prediction on the real part

Material erosion setup is very important for crack initiation prediction. Correct place and time of the crack is necessary for design preparation. Precise crack prediction in the simulations leads to the positive test results. Every design change of the real part is very expensive (tool correction). Commonly used *MAT_PIECEWISE_LINEAR_PLASTICITY with *MAT_ADD_EROSION is thus being replaced by *MAT_SAMP-1 with *MAT_ADD_EROSION (DIEM). Primary validation was performed on defined tests with test samples in 4A Engineering. Final validation on the real parts had to be done before material release in Bosch.

4.1 *MAT_SAMP-1 validation with steel guiding rods concept

The Bosch Impact Test was used for the material validation. Real impact test was performed together simultaneously with calculations. Left guiding rod boss was broken and guiding rod flow away. Right guiding rod was bent and its boss remained without damage.

Fig.14: Experiment of Bosch Impact Test (steel rods)

The same situation was calculated with *MAT_24 and *MAT_SAMP-1. The *MAT_24 which was commonly used had similar result with the real experiment. Crack mode of the left guiding rod boss was slightly different. One of the crack propagation way was the same, but mainly longitudinal crack wasn’t considered in simulation. This is main difference between simulation and experimental results. We have very similar experiences from other projects too. The reason could be following: erosion parameters were not strain-rate dependent, triaxiality was not taken into account, etc. → crack propagation cannot match with reality. Right guiding rod behavior (bending) was similar to the experiment.

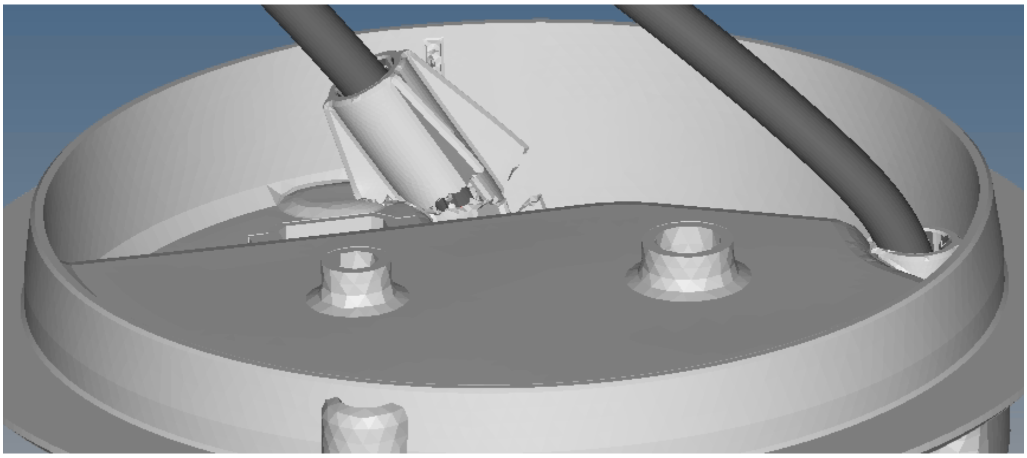

*Fig.15: Calculation of Bosch Impact Test *MAT_24 (steel rods)*

Calculation with *MAT_SAMP-1 stopped with error termination caused by the exceeded mass increase. Elements of the right guiding rod boss exploded when first erosion occurred. This erosion occurred in the pressure area. This behavior was not expected ant it is not physical. On the real part is only print of contact face, but any failures. On the second side the left guiding rod boss has crack initiation in longitudinal direction on the wall. Based on the results we have suspicion that *MAT_SAMP-1 doesn't work correctly in pressure/contact areas. This fact is confirmed in next validation with different concept 4.2.

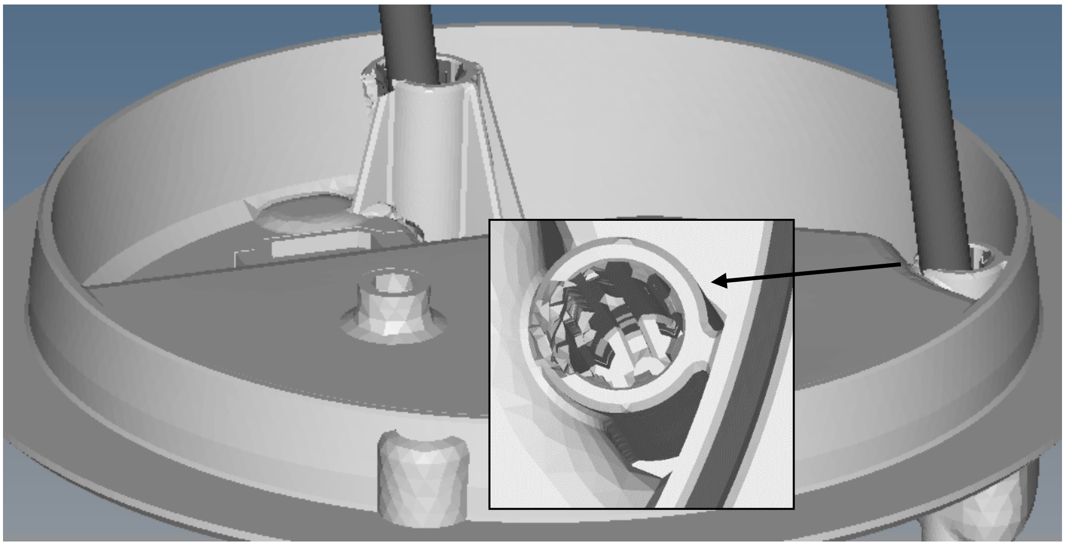

*Fig.16: Calculation of Bosch Impact Test *MAT_SAMP-1 (steel rods)*

4.2 *MAT_SAMP-1 validation with plastic guiding rods concept

Real impact test was performed followed by calculations again. Both guiding rods were broken in the specified area after the test. There were no additional cracks on the flange and guiding rods.

Fig.17: Experiment of Bosch Impact Test (plastic rods)

In the case of simulation with *MAT_24, result of the simulation was almost the same like the experiment. The guiding rod was broken and there were no additional cracks. We use damping in the simulation and we obtain very good agreement between crack initiation time in simulation and in experiment. It is necessary to say that this concept has very significant crack zone (symmetrical – full circle), but in previous simulation we had sometimes problems with non-physical cracks in simulation, if we didn’t use damping.

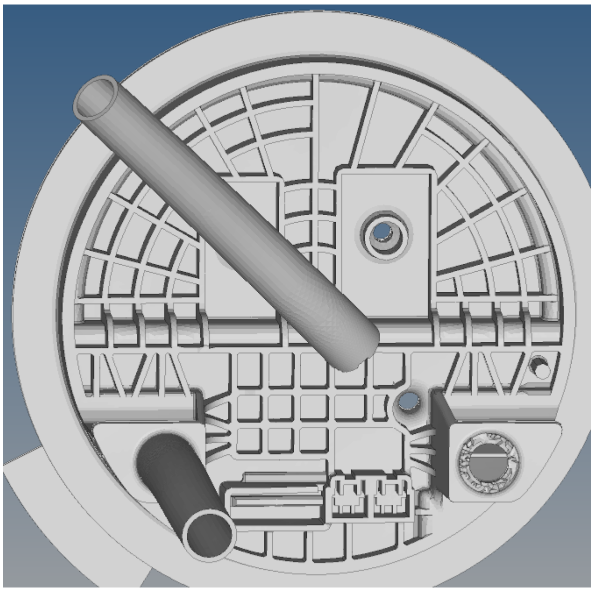

*Fig. 15: Calculation of Bosch Impact Test *MAT_24 (plastic rods)*

In the case of simulation with *MAT_SAMP-1, result of the simulation was not suitable. There is artificial crack caused by the pressure load in the middle of the rod (contact between rod and steel hammer). The guiding rod was broken onto two parts after the crack propagation between rod and pedestal. There were several additional cracks after the spring back. This behavior was not physical in comparison with experiment.

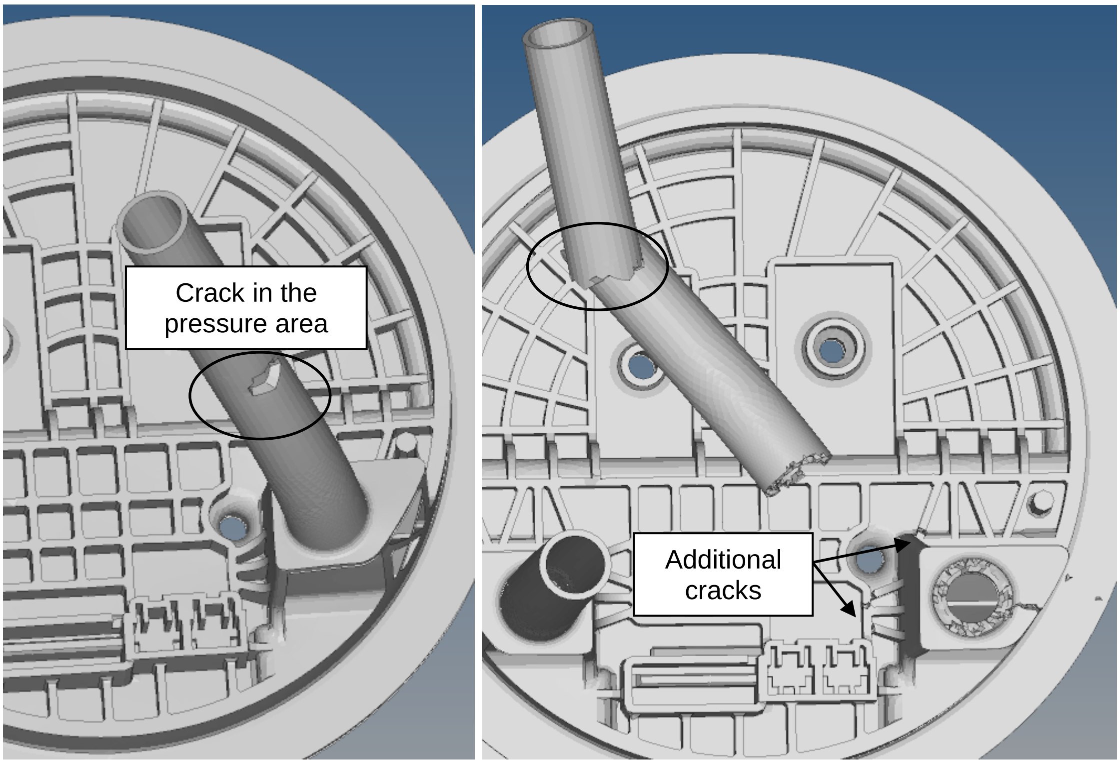

*Fig. 16 Calculation of Bosch Impact Test *MAT_SAMP-1 (plastic rods)*

4.3 *MAT_SAMP-1; mismatch finding (plastic guiding rods)

When the first calculations did not match with experiment, several attempts for improvement were performed. The problem was discussed with 4A Engineering and Ls-Dyna support. Some recommendations were obtained, but no one leaded to the positive result. One of those tips was magnifying of the erosion limit in the pressure area 10 times. This change had no influence for the crack on the pressure area.

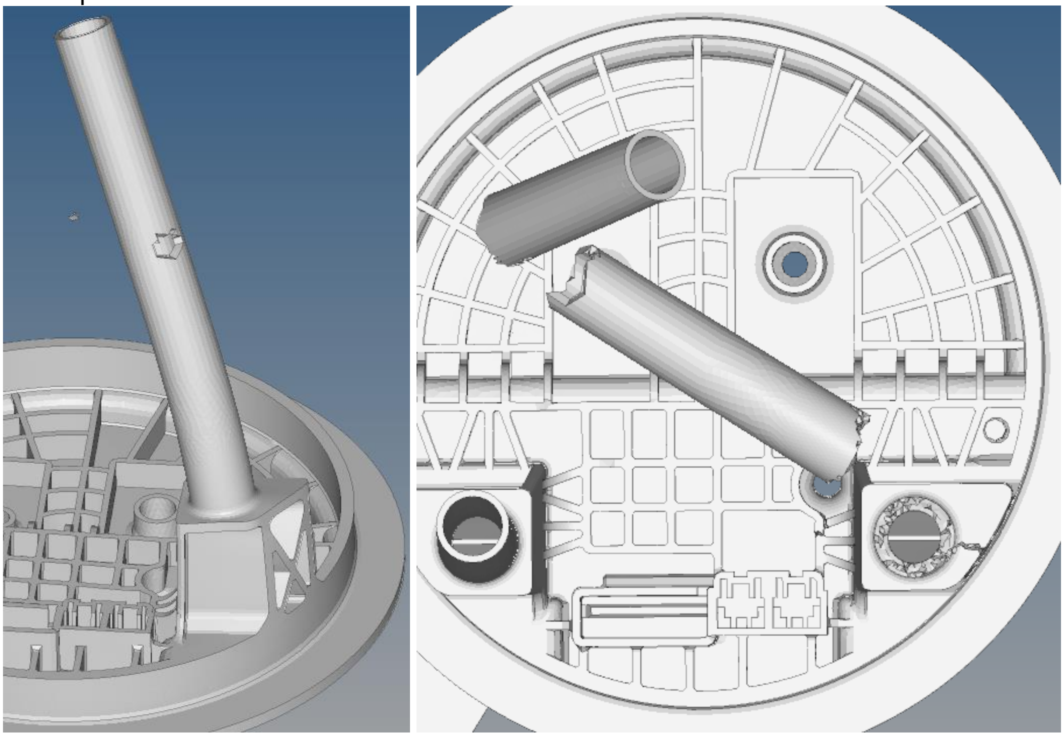

*Fig. 17 *MAT_SAMP-1, increased pressure limit (plastic rods)*

Crack in the pressure area was prevented with tetrahedron ELFORM = 4 (S/R quadratic element with nodal rotations) instead of tetrahedron ELFORM = 16 (10 – noded tetrahedron) which *MAT_SAMP-1 and DIEM were fitted for. Artificial cracks were omitted with ELFORM = 4, but this element is less precise and material parameters were not fitted for that.

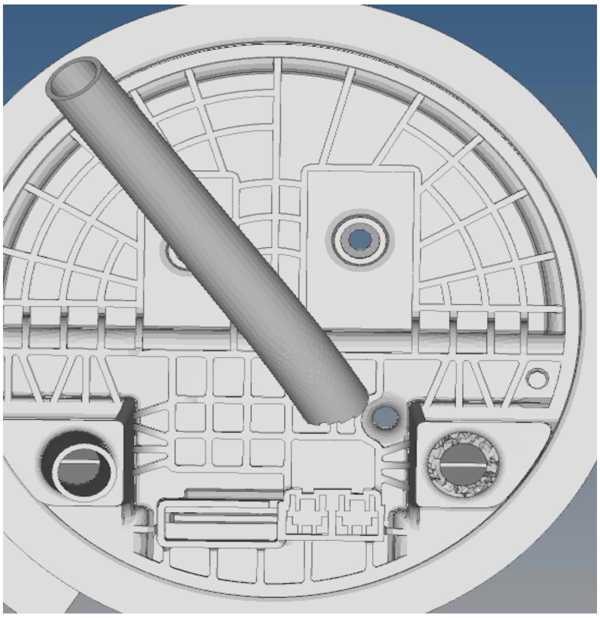

*Fig. 18 *MAT_SAMP-1, ELFORM = 4*

Summary

During the development of the simulation methods for LS-DYNA since 2012, there was continual development of the material model for TSCP. The main effort was to get precise material data for the crack prediction during the crash event. From the simple material model with basic erosion criteria we made a progress to the sophisticated material model with accumulation damage function. Additional necessary experiments were made to obtain missing data and mat. *MAT_SAMP-1 with DIEM erosion was created (4A Engineering). Material properties were adjusted for test samples. Correct behavior was expected with real parts as well. New material model was confronted with the previous one and with the experiment of Bosch Impact Test. Results did not fulfill expectations. Behavior of model with *MAT_SAMP-1 and DIEM erosion was not better than old one model. Nonphysical cracks occurred. There were no cracks in the pressure area in the experiment, but DIEM erosion generated cracks in the contact area (pressure load). Material model cannot be released due to this artificial behavior. We would like to use *MAT_SAMP-1 with DIEM because we know that the dependence of the material characteristics on the triaxiality stress has important effect on the deformation and force response of the simulated problems, but we have to solve the problem with non-correct behavior in failure and damage model. The *MAT_SAMP-1 model shows more stiffener behavior, which is caused by non-symmetrical behavior in compression/tensile areas. The *MAT_24 gives softener deformation and force response due using of the same material characteristics in compression/tensile areas. The future steps are elimination of the non-physical behavior of *MAT_SAMP-1 in problematic situations.

Several attempts were performed to find the reason of this mismatch. The problem was discussed with 4A Engineering and Ls-Dyna support, but no advice (recommendation) had demanded effect. There is some phenomenon in mat. *MAT_SAMP-1 with DIEM erosion which is to be observed and fixed. Otherwise it is not usable for serial service. So, this article is appeal for LS-DYNA users and developers to propose of solving of this non-physical behavior *MAT_SAMP-1 in specific situation, like as contact, compression loading area.

5 Literature

[1] Brepta, R; Prokopec, M: Šíření napěťových vln v tělesech, Academia Praha, t.ST05, 1972

[2] Schwer, L: Optional Strain-Rate Forms for the Johnson Cook Consitutive Model and Role of the Parameter Epsilon_0[1], 6. LS-DYNA Anwenderforum, Frankenthal 2007, EI2-EI14,

[3] Reithofer, P; Haufe, A: Validation and Material Modelling of Plastics, 8. LS-DYNA European Users Conference, Strasbourg 2011

[4] Kolling, S; Haufe, A: Material Models of Polymers for Crash Simulation, Dynamore Infotag, 30.11.2009

[5] Andrade, F; Feucht, M; Haufe, A: On the Prediciton of Material Failure in LS-DYNA: A comparison between GISSMO and DIEM, LS-DYNA forum Bamberg, 2014

[6] Kolling, S; Haufe, A; Feucht, M; Du Bois, P.A: SAMP-1: A Semi-Analytical Model for the Simulation of Polymers