Solving Crash Problems of the Fuel Supply Modules in the Fuel tank

Martin Dobeš[1], Jiří Navrátil[2]

[1,2] Robert BOSCH spol. s.r.o., Roberta Bosche 2678, Česke Budějovice 370 04

1 Abstract

This article in the first part deals with the experimental measurement of the material data used for explicit computational FEM analyses. The second part of this paper devotes practical application of the FEM simulation in fuel tank domain and fuel supply modules (FSM). The main focus is on the material computational models, especially material models with strain rate dependence. These computational models are used for polymer materials, like TSCP (Typical Semi-Crystal Polymer). A lot of experiments were conducted for comparison of the simulation results and reality. Some experiments were proposed directly for detailed comparison real deformations in the time of break with a focus on the flange part. The flange is the most important part of the FSM from the view of passive safety.

2 Introduction

Fuel supply module, which consists of the fuel pump, filter system, fuel rail, regulation system and flange belong to main components of the fuel system of the car. The majority of all parts are made of Polymer materials. The TSCP plastics are familiar as plastics with very significant dependence on the velocity of the loading, like strain rate dependence. We have to use more sophisticated material models for modeling of the dynamic loadings and stress responses of plastics in these cases. We can propose a lot of computational material models and approaches for plastics with considering the strain rate effect, but for majority of material models we can’t make experiments for retrieve source measurement data.

The computational analyses generally help to understand some physical principles in different loading situations. We would like to focus on some principles, which are caused by dynamic loading, especially in fuel tank. We are focusing on the deformation behavior FSM inside the fuel tank during non standard situations, like crash, shock, drop tests etc. All simulations are constructed regarding a lot of aspects, simplified geometry model, material model, contact problems, generally nonlinearities. We have to exert force to describe all know boundary conditions for more correct and more real computational analysis. The material computational model is the basic precondition of the appropriate deformation, stress and strain behavior.

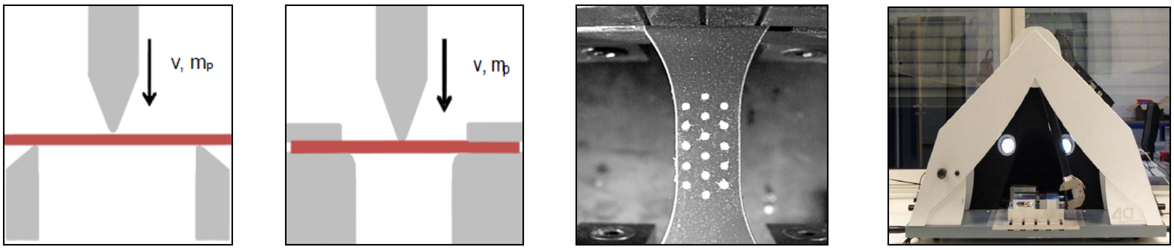

Fig.1: Testing methods for development of computational material model for TSCP (4A engineering)

2.1 Fuel Supply Module (FSM)

The FSM is one part of the fuel system in a car or motorbike. The basic requests on this product are delivery of the fuel from the fuel tank, measuring of the fuel level, filtering of the fuel, regulation of the pressure, flow and the most important request from our view is absolute tightness during nonstandard situations, like front or side impact, overturn of the car etc. The flange is the most important part of the FSM from perspective of passive safety, because flange is the last part which closes fuel tank. The flange and sub-module (fuel pump, reservoir, filters, fuel level sensor) are connected by guiding rods (plastic or steel). The weight of the sub-module is approximately between m_{sub}=0.7-1.2kg, so this weight loads interfaces on the flange almost in all loading cases, primarily in impact situations. The basic customer request is possible cracking of the flange, but the absolute tightness of the closed fuel system. Generally FSM has to fulfill a lot of technical requests and stands in real boundary conditions.

We solve the interface between guiding rods and flange or designed cracking zones on the guiding rods bosses.

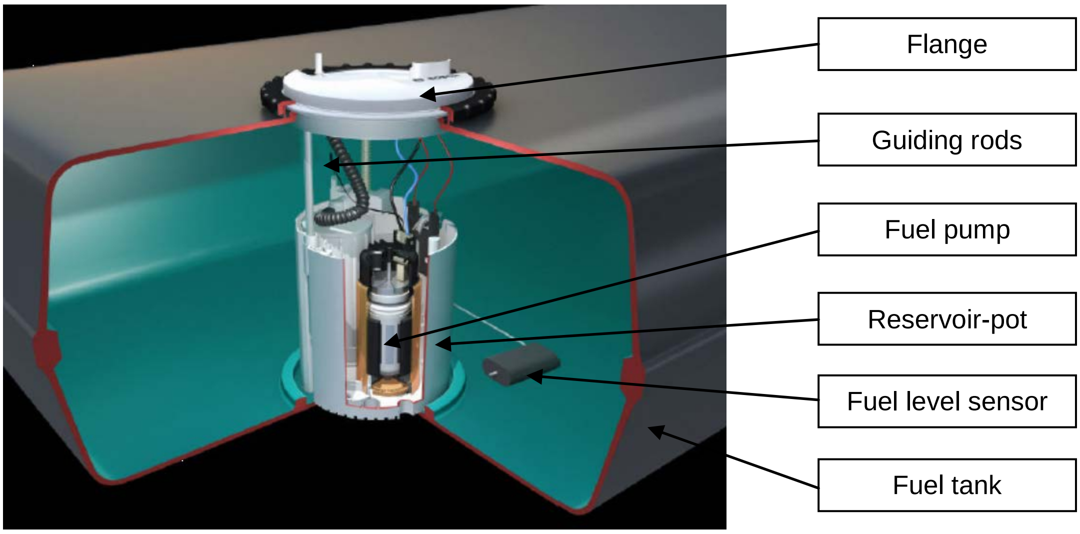

Fig.2: Fuel supply module (FSM) in the Fuel tank

3 Computational material model

The company 4A engineering has necessary experimental equipment for measuring of the material mechanical properties. The material model with respecting strain rate dependence was important for our purposes. This model was fitted for special type of finite elements (FE) 16 fully integrated shell (size 2mm), 10-noded composite tetrahedron (size 0.6mm). The measurement data were validated using other tests for strain range ε=0-15% and strain rate range ε^{-1}=0.001-50s^{-1}. The breaking points weren't founded for all strain rate levels. The stress-strain curves are used to strain $\varepsilon_{=}$15% (measurement range) and after this strain value the curves are approximated. The final stress-strain curves are putted together from a lot of tests. The basic tests were Static bending test, Dynamic bending test, Dynamic clamped bending test, tensile test. The very interesting test is Dynamic 3-point bending test (double pendulum-4A Impetus). The acceleration sensors are placed on the swing hammer. The mass of this swing hammer is m_{pend}=480-1200g (one pendulum system).

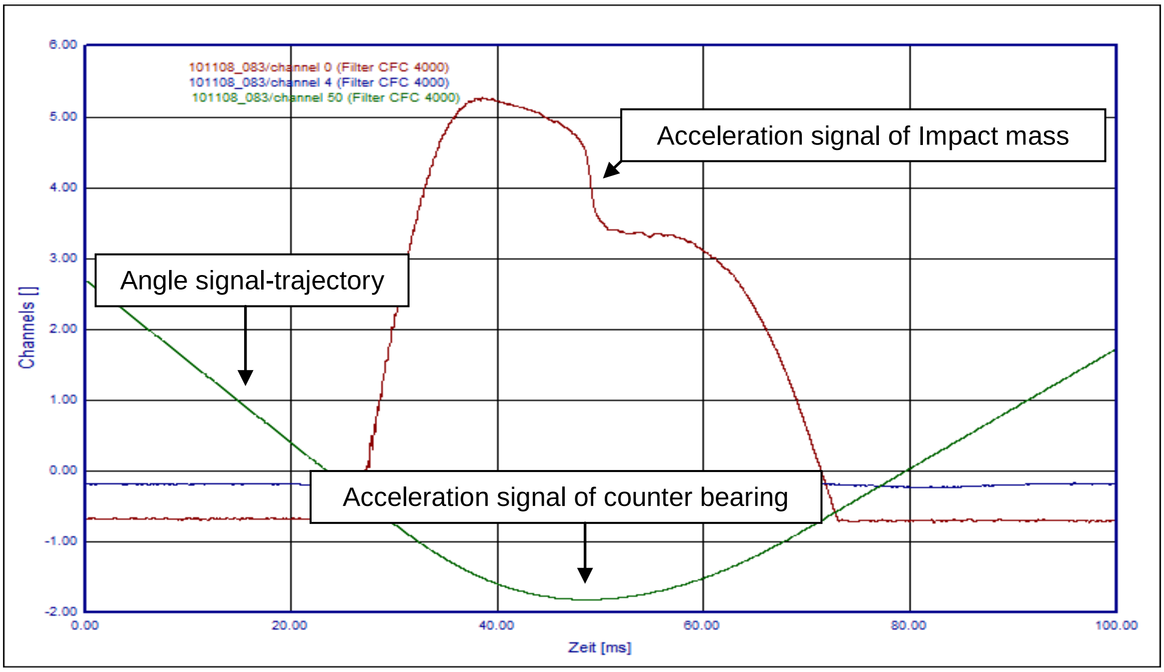

Fig.3: Sensor signal4A Impetus –dynamic bending test

From this signals is evaluated the loading force and impact velocity, see below.

$$F = m_{pend} \cdot a_{pend} \tag{1}$$

Evaluation of the displacements is based on difference in the angle signal.

$$s_i = s_{i-1} + \frac{\Delta\alpha \cdot \pi \cdot l_{pend}}{180} \tag{2}$$

Impact velocity is determinated through the optical signal.

$$v_0 = \frac{\Delta s_{1-2\,peak}}{\Delta t_{1-2\,peak}} \tag{3}$$

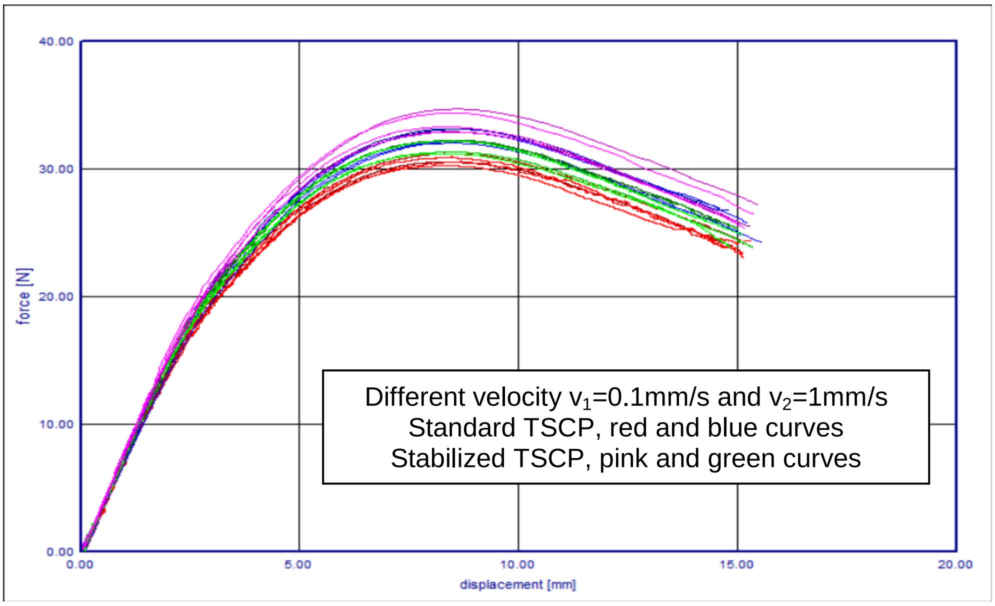

Fig.4: Measurement data from static bending test

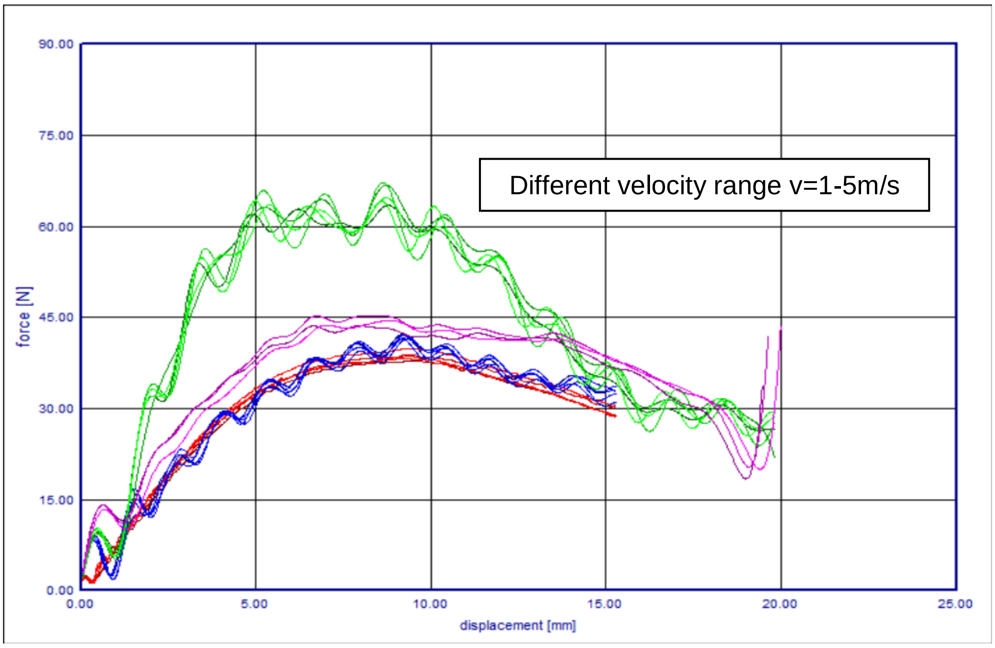

Fig.5: Measurement data from dynamic bending test (standard TSCP)

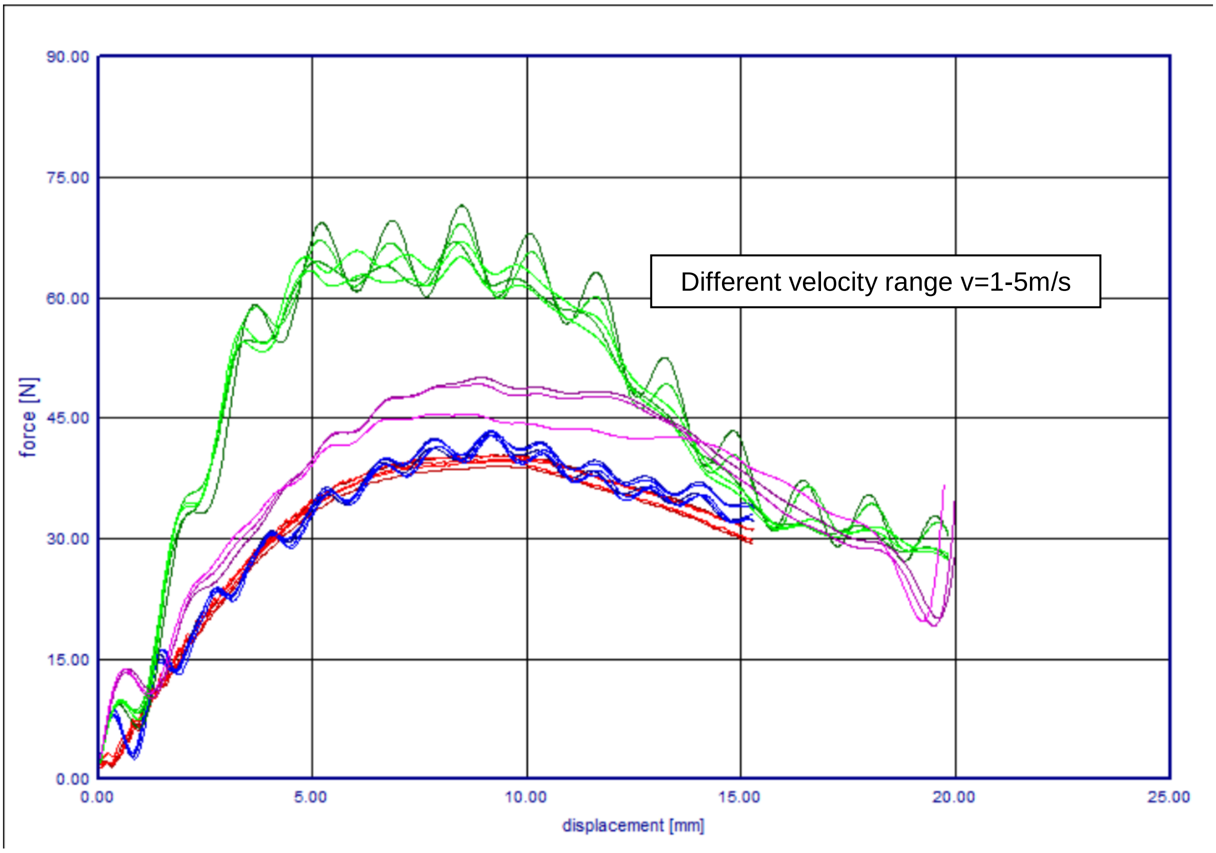

Fig.6: *Measurement data from dynamic bending test (non-stabilized TSCP)*

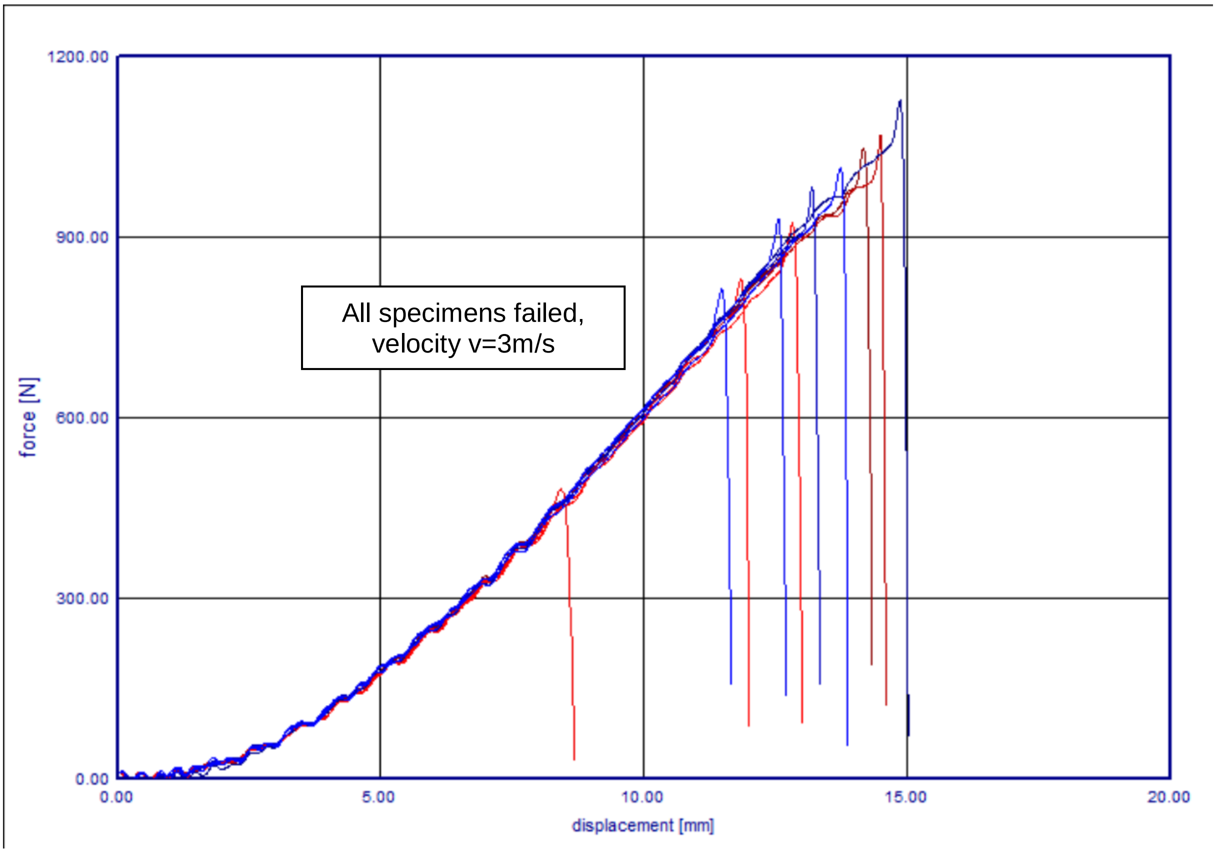

Fig.7: *Measurement data from clamped bending test (red-standard TSCP, blue-stabilized TSCP)*

The last test was classical static tensile test with velocity v=0.2mm/s and clamping 52mm, due to special shape of tensile samples. Two types of the strain were measured on these samples, firstly Technical failure strain (dependence on the measured active length) and second type was True failure strain by high speed cam. The difference between these approaches is very significant. Technical failure strain is about ε_{tech}=22% and True failure strain is about ε_{true}=40-75%, it is depends on the type of TSCP. Considering of this effect is very important, currently for low loading velocity (static).
The final output from these previous experiments and fitting methods of experimental data and founding corresponded parameters for computational model for LS-DYNA are stress-strain curves with dependence on the strain rate. We can use this computational model as source data for implemented computational material model in LS-DYNA material library. These data are possible to use for a lot of material computational models from LS-DYNA library, in our case we choose very famous computational material model suitable for plastics **`*MAT_PIECEWISE_LINEAR_PLASTICITY`.**

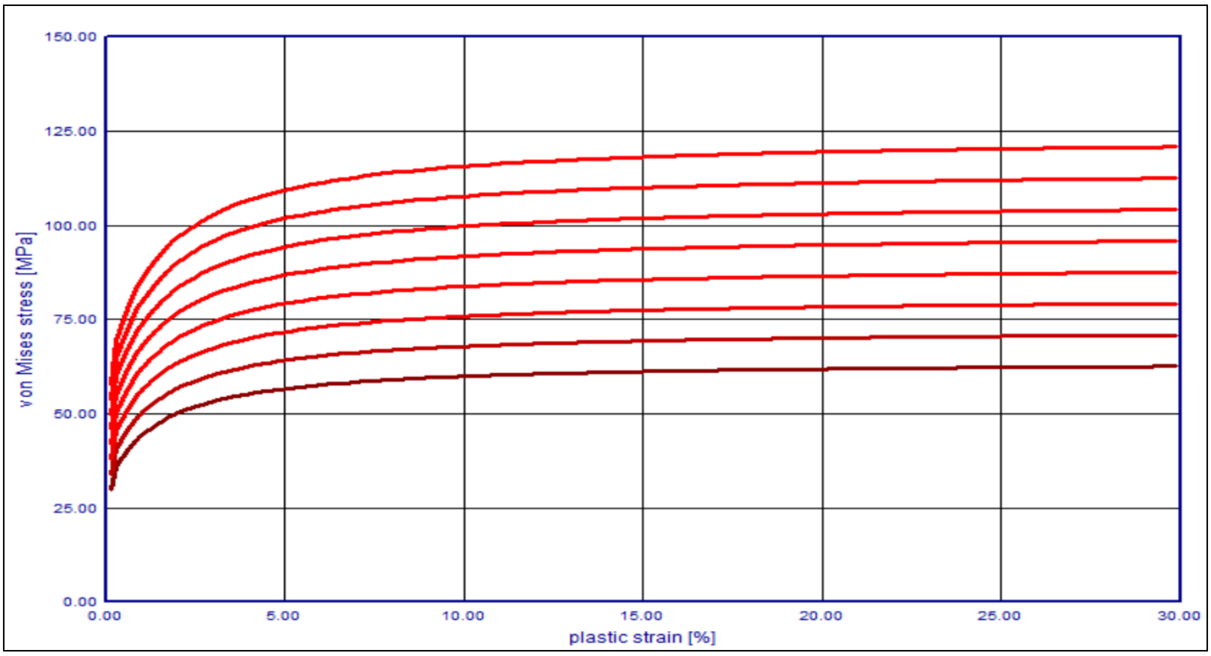

Fig.8: Computational material model of TSCP material for LS-DYNA, strain rate levels 0.1-10000 [s^{-1}]

4 Computational FE model of the Fuel Supply Module

Now, we have more sophisticated computational material model for simulation of the impacts, generally high speed problems. We can use these models for a lot of types of simulation, but our focus is mainly cracking of the plastic parts and presumable prediction of the cracking direction. For this estimation with using CAE simulations we need some computational model using erosion of the finite elements or computational model of the damage based on some theory for cracking. We use only approach using erosion of the finite elements, based on fulfilling some mechanical criteria for erosion of the finite elements. We can make erosion of the finite elements in card **`*MAT_ADD_EROSION`.** This card is possible to use for a lot of materials (steels, plastics, foams, etc.), due to many parameters for set up. However, we solve only plastics, concretely TSCP type of plastics. Our type of the plastics has a typical character of the cracking. Almost all failures are in tensile loading, this type of the loading is common for TSCP and this material has very significant ductility in static loading, of course increasing of the loading velocity causes degreasing of the failure limit of the strain.
Our aim is estimation of material characteristics on practice simulations; the reason is non-failure samples in dynamic bending test (4A Impetus). The failures create frequently in our tests. Especially for verification of the design of the plastic parts like flanges in non-standard situations defined in previous text, like crash, impact etc.
For comparison we use experiment of the rigid impact to the guiding rods and subsequent cracking of the flange due to the effect of the pick-up of the guiding rods from the flange. FE model is created from shell and solid finite elements with using of the rigid beam finite elements for necessary connections between parts. Our products are generally very small in range *mm* to *cm* and design elements, like reinforcement ribs, electrical sockets and etc. are very small. Using of the size of the finite elements about 0.3-0.5mm is necessary (at least 3-5 finite elements per thickness of the wall)), so due to this

fact we have problem with critical time step. Our experiences show, that the sizing and sufficient number of the finite elements per thickness of the wall is very important for correct propagation of the cracking in the wall. Another problem is with contacts for these small finite elements. We can use very small finite elements for plastic parts, due to the very low Young's modulus used for calculation of the critical time step, but for steel (guiding rods) this benefit is missing. So the quality and sizing of the mesh is very important. We can execute some tips and tricks in situation, where we have problems with high penetration and non-stability of the contacts. The first possibility is the change of the set up parameters in the contact card.

Second type of the solving of problems with contacts is using of the shell finite elements on the outside surfaces of solid finite elements. These elements have the same nodes as solids finite elements and behavior of contacts between shell finite elements is more stable then solid finite elements vs. shell finite elements. We use contact type **`*CONTACT_AUTOMATIC_SINGLE_SURFACE`** for definition contact between guiding rods (shell FE) and guiding rods bosses (solid FEs with surface shell FEs). This option is more stable and penetrations are very small, the smallest from all the tested options.

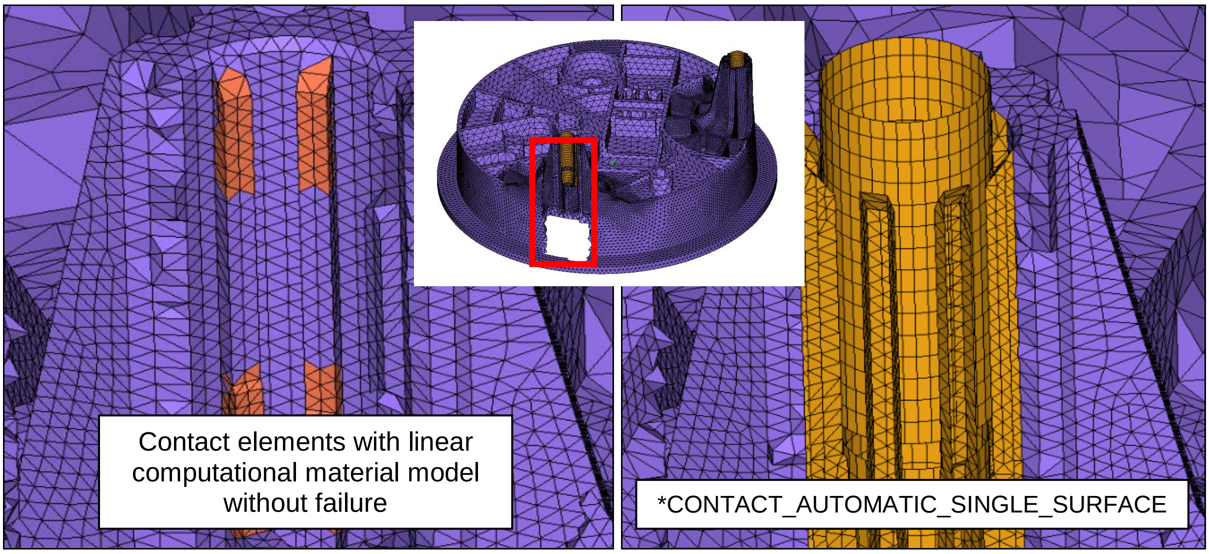

Fig.9: Detail of the contact faces: left-contact by solid FE, right-contact by shell FE

Next important simplification of the computational model is using of the rigid bodies (RBs) as connection between shell finite elements envelope of the guide rods and beam finite elements. The guiding rod bosses are modeled by combination of the beam finite elements and shell finite elements (shell envelope), which were created for contacting with inside ribs of the guide rod bosses, see (Fig.10a). This solution offers saving of time to solve the simulation task, more transparent behavior and options of the contact. The important set up is sufficient mesh size of the beam finite elements and shell finite elements, due to the correct bending stiffness. Similar solution is used for modeling of the fuel pump in holder (mass point in the center of gravity with mass of the fuel pump). Connection between fuel pump and holder is by RBs, see (Fig.10b)

Generally we use finite elements 10-nodes tetrahedron with quadratic base functions (type 16, fully integrated), solid finite elements (type 16, fully integrated) and beam finite elements (type 1, Hughes-Lie with cross section integration).

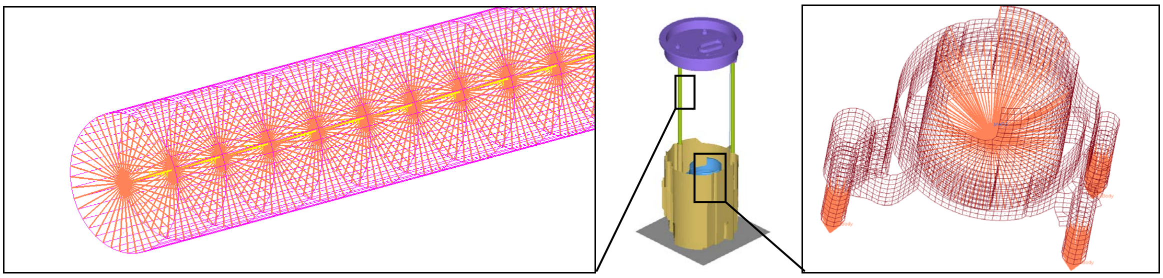

Fig.10: Using of the RBs, beam and shell FE- a) FE model of the guide rod, b) FE model of the holder

5 Character of the loading profile

Generally in automotive we know a lot of type of the loading profiles. All parts are exposed to some boundary conditions with different loading, for example temperature, pressure, forces, fatigue cycles etc. In our case we consider only loading profiles for area of the crash tests, especially with focus on the fuel tank and components inside the tank. Typical loading profile in crash problematic is dependence of the acceleration (deceleration) in the time. The shape of this curve is depending on the stiffness of the car body and deformation zones of the car in real. In crash test area this signal is specified by norms or customer specifics.
The curve of this signal is envelope curve and closes the area below this curve. The area is Impulse of the force in physics, if we match some mass to the acceleration. Very different types of the loading profiles are in customer specifications, we can use some examples of the signals for crash test of the fuel tank and inside components or only fuel supply module.

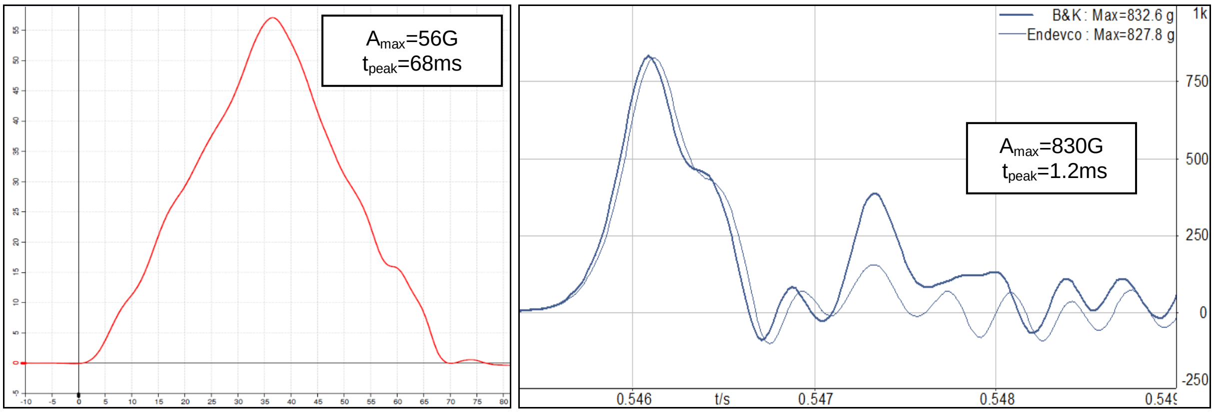

Fig.11: Examples of the crash loading profiles, customer specifications

We use very specific test machines for this type of the tests. A lot of principles exist for causing the sufficient impact force. It is possible to use hydraulic, pneumatic principles, drop tower or linear electric motors. The last technique is the most precise test machine, due to the correct set up loading profile and subsequent observation. We can state some typical test machines for better explaining of this problematic.

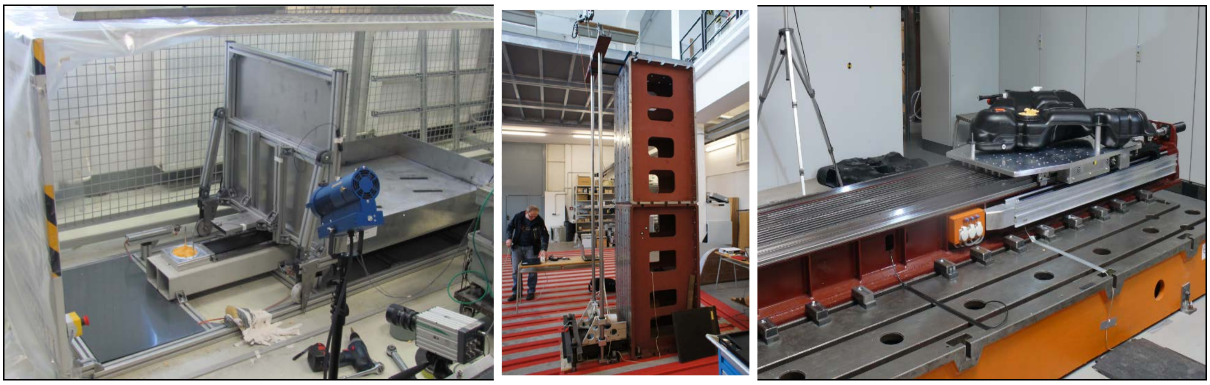

Fig.12: Examples of the test machine(TM) for FSM: a) Intern TM, b) Drop tower, c) SLED test

We can divide these tests to two groups:

a) With requirement on the breakaway (propagation of the cracking vs. 100% closeness)
b) Tests with prescribed loading profile (only 100% closeness, but cracking is not necessary)

These tests help to us with verification and comparison of our FE simulation with real experiments. These tests are very transparent and measuring of the deformation behavior with high-speed cam is possible. We use record from high-speed cam and accelerometers for comparison analysis of our results from FE simulations with real test samples. The records purvey detail information about the time of the initialization of the cracking in the time and then we can study crack propagation on the real part and on the corresponding computational model in FE simulation.

6 Comparison analysis of the Experiments and FE simulations

We chose three types of the design of the flanges and FSM for the comparison of the computational FE model of the FSM and deformation behavior of the real FSM. In each design concept we utilize different principle of the deformation during crash situation (crash test). These different principles are given by customer or platform requirements.

6.1 Concept 1 – Plastic guiding rods

The problem with design of the plastic guiding rods is first practical task, where it was used explicit analysis with possible prediction of the crack propagation. This analysis serves for verification of the proposed cracking zones of this concept. The Bosch Impact Test (internal RBCB test) was used for verification, see (Fig.13).

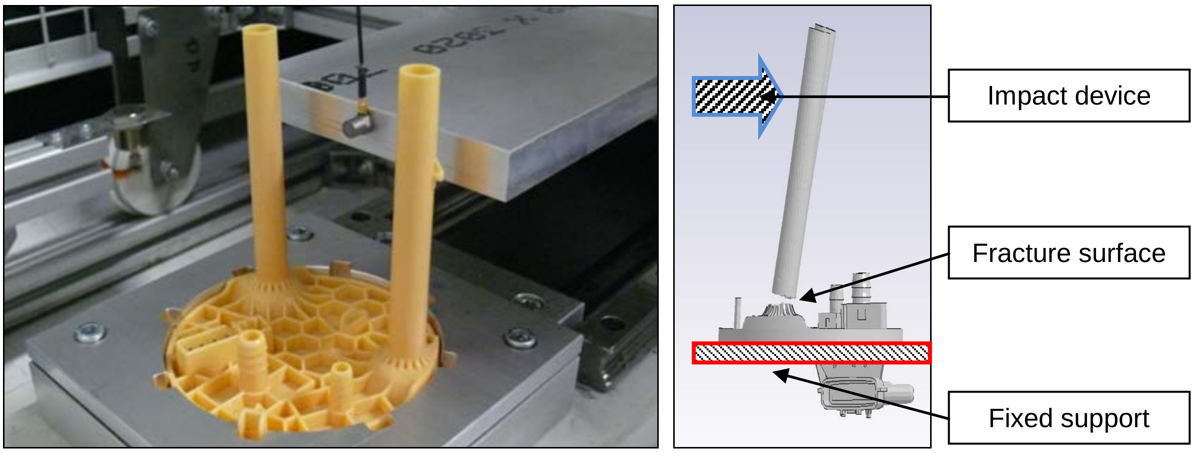

Fig.13: Schema of the Bosch Impact Test and Test boundary conditions

In this analysis material card **`*MAT_ADD_EROSION`** was used with set up only one parameter for deleting of the FE elements, Maximum principal stress. This parameter assures only cracking in tensile hot spots. The main advantage of this criterion was not considering of the ductility of the plastic material. The suitable time for deleting was set up only by value of the Maximum principal stress and number of the integration points, where the condition must be fulfilled. The dependence on the quality of the mesh is very significant and sensitive. We usually use 3 integration points for fulfilling of the condition on the element, due to the good correspondence between deformation behavior in FE simulation and experiment. This approach was very simple and non-sophisticated for more correct options of the FE simulation. Generally, results from comparison of fracture surfaces were sufficient. The more sophisticated material model will be necessary for better simulating of the deformation behavior of plastic parts in real test. Next potential is considering of more parameters during erosion process of finite elements. The more correct and sophisticated is evaluating of plastic material ductility as an additional parameter for erosion of FE. So, in next experiments and designs we use two parameters erosion of the FE, which reflects Plasticity (Stress at Break) and ductility simultaneously, see next chapters.

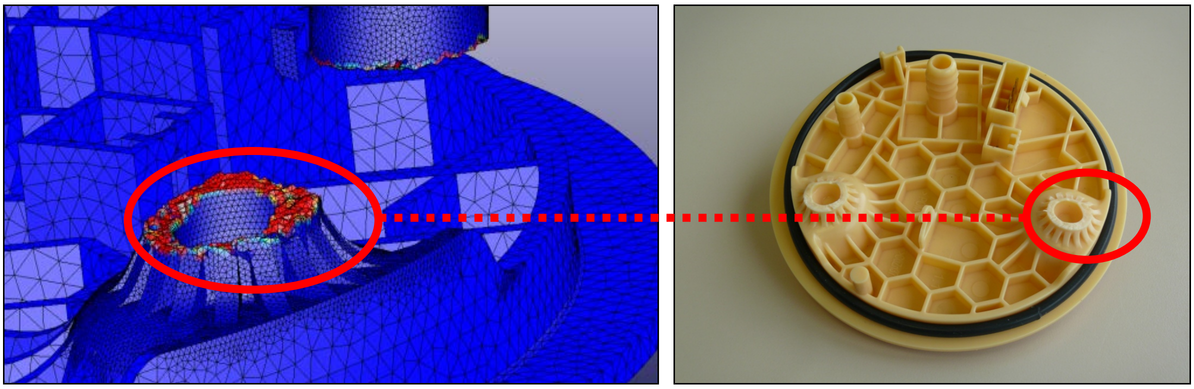

Fig.14: Fracture surfaces after the impact of rigid body, FE simulation vs. Experiment

6.2 Concept 2 – Cracking zones on the guiding rod bosses

Steel rod pressed into the rod housing is used in this concept. This connection must ensure endurance during whole lifetime of FSM. On the other side, rod housing must ensure correct crack propagation in defined position during ultimate load (crash). Steel rod is designed to stay straight. Shape of rod housing ribs is the most important area for breakaway strategy. Rib cranking and radii have significant influence for crack initialization. More sophisticated material model with element eroding based on maximum principal stress and ductility is used for calculation. Bosch Impact Test (internal RBCB test) was used for verification also, see (Fig.15).

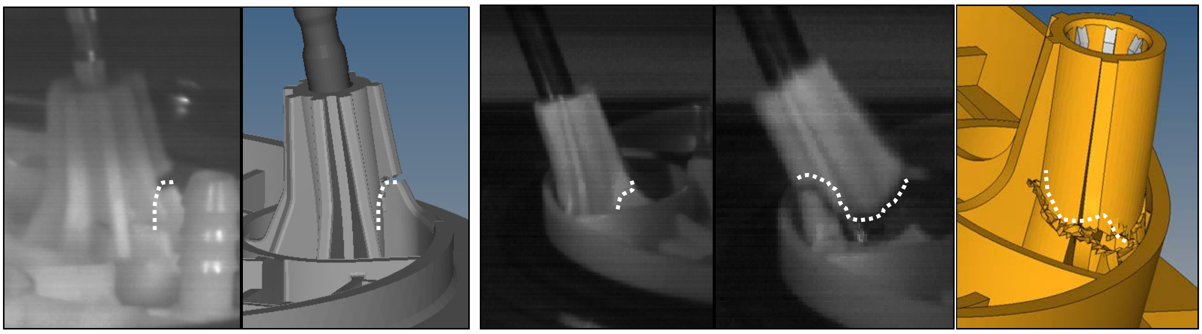

Fig.15: Record of the High-speed cam and Simulation results – initialization of the crack

6.3 Concept 3 – Cracking zones on the guiding rods

Steel rod is pressed into the rod housing again. Main difference is in the stiffness ratio between housing and rod. Rod housing is supposed to be tight and steel rod must be bended or fractured. Rod housing ribs have been reinforced and rod notch has been improved. Both materials must be precisely measured and defined for correct calculation result. Element eroding of flange, verified on the previous concept, is applied. Element eroding of steel rod is omitted due to the large steel ductility. Proposed design is tested again with positive response.

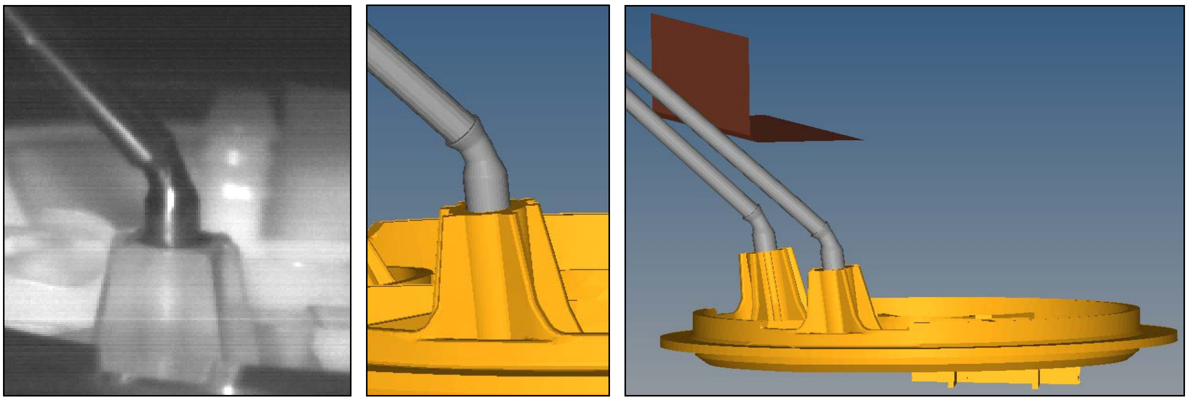

Fig.16: Record of the High-speed cam and Simulation results – steel rod bending

All simulations are validated by the simple Bosch Impact Test. This verified material models and proposed designs are consequently used for calculation of whole FSM in the fuel tank. The next FE simulations with considering fuel tank include other aspects, which are very important for detailed description of the deformation, stress and strain response on the FSM. The effect of the fixing of the FSM in the fuel tank is very important, due to significant difference between stiffness of the fuel tank (HDPE material) and JIG (fixing device, Steel or Aluminum). The study of this effect was included in previous works.

7 Summary

In this paper are showed aspects of founding correct material models for explicit analysis using experiments. The second part of paper deals the results from the experiments and explicit simulations, on practice task are demonstrated important information about set up of the analysis, sensitivity on the different parameters in material cards etc. The main aim of this work was finding of the correct parameters for computational material model and following eroding of elements. For this we use rigid impact test for FSM and compare deformation behavior of the real parts and computational model. Second test for verification is Underside Impact test (Drop test tower). It is possible to say, that our computational model with found material parameters has very good agreement. The time step of initial breaking is almost the same as in real condition and propagation of the cracking is very similar too. The comparison analysis shows, that we have the same initialization points of the cracking and final fracture surface are almost the same as in experiments, but propagation of the cracking is not overly good in some loading directions and boundary conditions.

Next work will be focused on the more sophisticated computational model of the propagation of the crack in material during Impact and considering other aspects, which are important for change of the deformation behavior during nonstandard situation, as fuel effect, ageing etc. For this modeling we would like to use ALE FE elements or combination of Euler and Lagrange meshes, possible in LS-DYNA.

8 Literature

[1] Dobes, M., Navratil, J.: Application of the different computational material models of Polymer material for solution of FEM in LS-DYNA, 22nd SVSFEM ANSYS Users' Group Meeting and Conference 2014, Slovakia, pages 10-27

[2] Hubert, F., Hiermaier, S., Neumann, M.: Materials models for Polymers under crash loads – existing LS-DYNA models and perspective, Proceedings o the 4th LS-DYNA forum, Bamberg, Germany, 2005

[3] Internal specification of the Loading crash profiles BOSCH (GS-FS/ENG1-Bj)

[4] Material characteristics of the TSCP, http://www.basf.com/group/corporate/en/

[5] Measurement of strain rate dependency of TSCP, Project RBCB and 4A engineering 2013

[6] Kolling, S. Polymers in Crash and Impact Simulation. In: Laboratory of Mechanics, Giessen University of Applied Sciences, Technologietag. Schladming, Austria, 2010.

[7] Mulliken, A.D., M.C. Boyce A G.T. Gray. Mechanics of the rate-dependent elastic–plastic deformation of glassy polymers from low to high strain rates. International Journal of Solids and Structures. 2006, vol. 43, issue 5, s. 1331-1356