Digimat Material Model for Short Fiber Reinforced Plastics at Volvo Car Corporation

Mats Landervik[1], Johan Jergeus[2]

[1]Dynamore Nordic AB
[2]Volvo Car Corporation

1 Introduction

The present paper concerns a study of the potential of the Digimat material model [1] for modeling reinforced plastics at Volvo Car Corporation. Fiber reinforced plastics in general may be either thermoplastics or thermosets commonly reinforced with glass or carbon fibers. Depending on the application, different material and production systems are chosen. They all have in common that the mechanical response is highly dependent on the local microstructure. Digimat is a micromechanical model with separate material models for the separate phases along with assumptions of the microstructural topology and homogenizations schemes. In addition to the material models for the inclusion and matrix phases, information regarding the aspect ratio of the inclusions and its orientation is used. This gives a local anisotropic material response that is inherent from the production process. In Digimat, the same microstructure model is used regardless of the fiber length and how the fibers are organized. For continuous fiber composites, either definition of lay-ups or e.g. draping simulations may be the basis for the local fiber orientation. For short fiber thermoplastic composites, being the current case, simulations of the injection molding process are often performed both to adapt the component design for production and adapt the production process itself. Typically, these simulations are carried out in special finite element tools such as Moldflow from which also predictions of fiber orientations to be used in Digimat can be output.

CAE simulations are extensively used in product development to predict the vehicle performance in a wide range of areas. This results in many types of analyses that have different requirements on the material models. Some differences are range of deformation and its rate, loading duration, temperature range and environmental conditions. One benefit of Digimat is that it is available for a number of finite elements codes and different analysis types. Although different matrix material models are applied in difference analysis types, some parts are common and parameters may be reused. The analysis types that have been included in this study are strength, crash, creep and fatigue. However, at the time of writing, the creep analysis is still ongoing as well as a thesis project on fatigue [2]. Thus, only strength and crash analysis are covered in this paper. The aim has been to evaluate the predictability of the simulations, experimental data needs, ease of use and simulation cost. Other studies of LS-DYNA with Digimat simulations are given in [3] and [4].

The process to obtain material parameters for an automotive OEM differs. Sometimes ready-to-run CAE input can be shared by the supplier and sometimes material characterization tests are performed at internal or external labs. In the latter case, the CAE input needs to be prepared which in turn can be done internally or by an external supplier. One recurring difficulty is that the detailed material specification may come late in the product development cycle. Therefore, the time from material specification to CAE input is critical. The required data and process to obtain it may differ for different material models and it is thus something to consider in the choice of models.

2 Component

A pre-production version of a front end carrier (FEC) for the new Volvo XC90 model was chosen to be studied. The component is designed and supplied by Röchling Automotive and is made from Lanxess Durethan BKV 30 which is a PA6 reinforced with 30% glass fiber. The component was chosen due to that its response has a high influence on both pedestrian safety and strength and robustness of the hood. Low stiffness in the longitudinal vehicle direction is of benefit in the first loadcase while the second benefits from high stiffness and strength in the vertical direction. Therefore, the design process benefits from having performance predictions of higher accuracy. Furthermore, material data was available from the material supplier of this component which indeed is a benefit.

© 2015 Copyright by DYNAmore GmbH

3 Material model

For the fiber, linear elastic properties are assumed for all analysis types. For the matrix, different models are used depending on the analysis type. The matrix model is elastoplastic for strength and elastoviscoplastic for crash analysis. The main components of the models are the yield function,

$$f(\boldsymbol{\sigma}, \varepsilon_p) = \left(\frac{3}{2}\boldsymbol{s}:\boldsymbol{s}\right)^{1/2} - \sigma_Y - R(\varepsilon_p) \leq 0, \quad (1)$$

where $\boldsymbol{s}$ is the deviatoric stress tensor, ε_p the accumulated plastic strain and $R(\varepsilon_p)$ the hardening stress given from the exponential and linear hardening law,

$$R(\varepsilon_p) = k\varepsilon_p + R_\infty\big(1 - \exp(-m\varepsilon_p)\big). \quad (2)$$

The flow rule in the elastoplastic case is:

$$\dot{\boldsymbol{\varepsilon}}_p = \dot{\varepsilon}_p \frac{\partial f}{\partial \boldsymbol{\sigma}},\ \dot{\varepsilon}_p = \sqrt{\frac{2}{3}\dot{\boldsymbol{\varepsilon}}_p : \dot{\boldsymbol{\varepsilon}}_p}. \quad (3)$$

While in the elastoviscoplastic model the current yield Norton law is applied

$$\dot{\varepsilon}_p = \frac{\sigma_Y}{\eta}\left(\frac{f}{\sigma_Y + R(\varepsilon_p)}\right)^m. \quad (4)$$

3.1 Homogenization

The Mori-Tanaka mean field homogenization scheme [5] is applied to calculate the macroscopic response considering the microstructure of ellipsoid shaped fibers embedded in matrix. An additional parameter, the plastic strain multiplier, is introduced in the modified spectral method used to estimate the strains in both phases [6]. Moreover, using material models for the separate phases, there is an option in Digimat to determine parameters for a macroscopic anisotropic model with its response dependent of the local deformation tensor. This option is called the Hybrid solution and means that the macroscopic parameters are determined before the simulation in a pre-processing step. This option is used for the strength and crash analyses covered in this paper.

3.2 Failure

For failure indication, the Tsai-Hill 3D Transversely Isotropic failure model is applied at the pseudo-grain level. The pseudo-grain level is a pre-step in homogenization. In each material point, the orientation space is discretized into a number of pseudo grains containing only one fiber, having unique direction, embedded in matrix. The volume fraction of each pseudo grain is according to the orientation tensor in the material point. Thus, this pseudo grain is transversely isotropic and this relatively simple model can be applied. The parameter *X* represents the longitudinal (fiber) strength, *Y* for the transverse (matrix) strength and *S* the shear strength. Thus, the difference in strength in tension and compression observed in these types of materials is not present in the model.

$$f = \frac{\varepsilon_{11}^2}{X^2} - \frac{\varepsilon_{11}(\varepsilon_{22}+\varepsilon_{33})}{X^2} + \frac{\varepsilon_{22}^2+\varepsilon_{33}^2}{Y^2} + \left(\frac{1}{X^2} - \frac{2}{Y^2}\right)\varepsilon_{22}\varepsilon_{33} + \frac{(2\varepsilon_{12})^2+(2\varepsilon_{13})^2}{S^2} + \left(\frac{4}{Y^2} - \frac{1}{X^2}\right)(2\varepsilon_{23})^2 \quad (5)$$

The method to realize (progressive) failure in LS-DYNA explicit and implicit simulations is through erosion of elements. The failure of an integration point occurs when the failure indicator (Eq. 5) has reached 1. Next, a criterion for the erosion of an element is needed. The default of Digimat 5.0.1 is to erode the element when the first integration point has failed which is the choice in the current work. For other versions of Digimat, the element will be eroded at the failure of the last integration point. Alternatively, any number of failed integration points for the element to be deleted can be chosen in the LS-DYNA input as an additional failure parameter. In Abaqus implicit simulations, element erosion is not available. Instead it is possible to estimate the point of initial failure by post-processing of the failure indicator.

© 2015 Copyright by DYNAmore GmbH

4 Input data

4.1 Material

4.1.1 Required test data

The material calibration is done through reverse engineering from response curves for test dumbbells cut from injection molded plaques. Since there is no differentiation between tension and compression in either yield surface or failure model, only tensile response is needed. Dumbbells cut at 0, 45 and 90º relative to injection direction are used to distinguish between the fiber and matrix response. The 45º test will induce shear and can thus be used to determine the shear parameter of the failure model.

4.1.2 Available test data

The available data was Abaqus input for crash from dumbbells cut at 0 and 90º. Hardening curves and failure strains were given for several deformation rates together with constant Young's moduli. In addition there was Abaqus input for quasistatic strength analyses ("dpp") based on tension tests on directly molded dumbbells. Hardening curves and Young's modulus was given and tensile failure strain from a picture of a stress–strain curve.

4.1.3 Model calibration

From the hardening curves, total stress–strain curves were calculated using the Young's moduli. The elastoplastic and failure parameters were calibrated to fit these curves as well as the target failure strains (Fig. 1 and 2). The microstructure in the dumbbells and its variation through the thickness are important parameters in calibration. In this case there were no measurements made. Therefore these were estimated. For the "dpp", average fiber orientation components of [0.9, 0.1, 0.0] constant for the entire thickness was assumed while for the plaques a skin layer with [0.75, 0.2, 0.05] occupying 80% of the thickness and a core layer with [0.2, 0.75, 0.05] was assumed. For the orientation components, the first is the injection direction, second is in-plane and third is thickness direction.

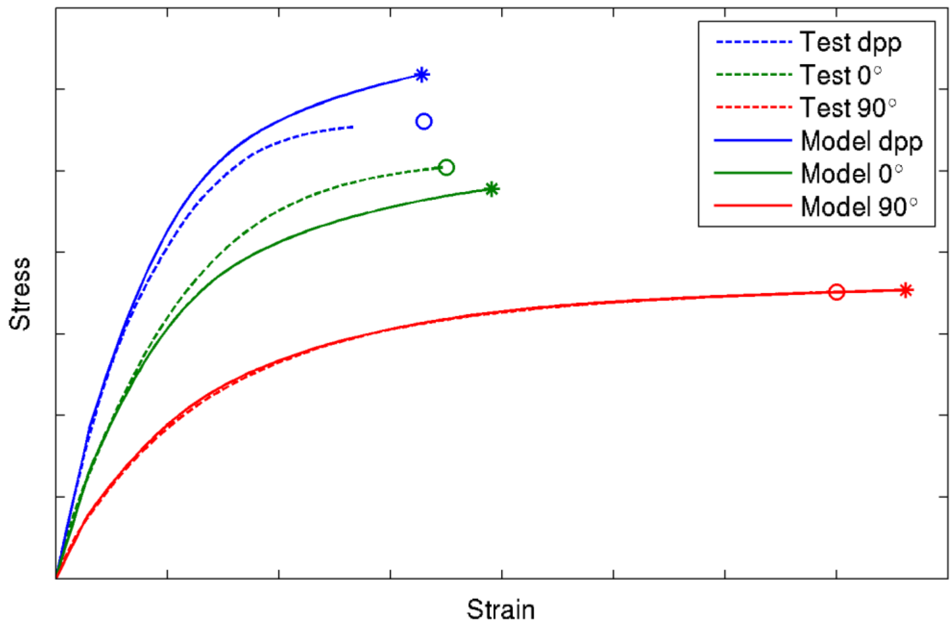

*Fig.1: Model response and test data for quasistatic tension of directly molded dumbbells ("dpp") and dumbbells cut from plaques at 0º and 90º angle relative to injection direction. Point of failure indicated by o and * for test and model, respectively.*

© 2015 Copyright by DYNAmore GmbH

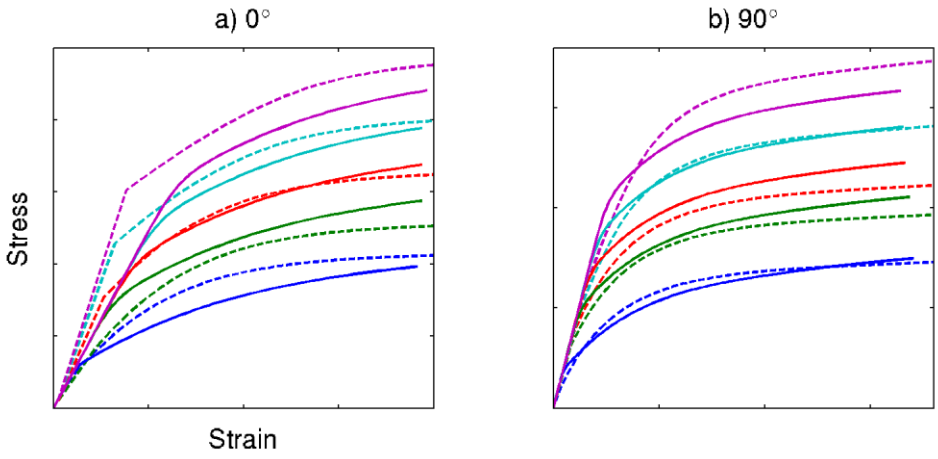

Fig.2: Model response (continuous lines) and test data (dashed lines) for dynamic tension at different rates for dumbbells cut from plaques at a) 0º and b) 90º angle relative to injection direction.

4.1.4 Available CrachFEM data for comparison

The available CrachFEM [7] material data for a PA6 width 30% glass fiber is for BASF ULTRAMID B3WG6 CR (different from the material in the component). The model is isotropic although CrachFEM has the capability to interpolate between two sets of material parameters (from highly aligned and less aligned injection molded plaques) based on input of orientation tensors in each material point. For the current model only the material parameters, not the test data is available. It includes hardening curves for 7 strain rates and an asymmetric (pressure dependent) yield surface. For failure, there are equivalent fracture strain vs. strain rate triaxiality ratio for ductile normal fracture and ductile shear fracture given for two strain rates each.

4.2 Fiber orientation

Since the material response is dependent on fiber orientation, the orientation tensor is required for all integration points. Shell elements are used for the component in all loadcases and it was chosen to use the same midplane geometry for the injection simulation. To provide a prediction of fiber orientation, Moldflow was used to simulate the production process. The maximum number of layers through the thickness of the shells (20) was chosen in the injection simulations while 10 layers were used in the structural simulation to balance resolution of the orientation variation with computational cost. Injection meshes and structural meshes are in general different. Therefor a mapping is done to determine orientation values for the integration points in the structural mesh based on those in the injection mesh.

5 Results

5.1 Quasistatic component tests

In the first quasistatic loadcase, the FEC is subjected to three-point bending at midpoint leading to a progressive failure of the component starting from the bottom side (in tension). At the midpoint, there is a hole close to the bottom edge which combined with few fibers in the loading direction due to the symmetric filling gives a distinct location for the initiation of failure. The failure path is dependent on discretization and the simulations give a reasonable prediction in this aspect. It is clear from Fig. 3 that Digimat gives a very good prediction of the initial stiffness and occurrence of initial failure. Also, the stiffness of the second phase is well predicted. However for the second half of the displacement, failure in compression (particularly close to the supports) is over-predicted causing the decrease in load. The Abaqus simulation was done with a finer mesh which can explain the closer match of stiffness. In Abaqus implicit, it is not possible to remove elements during simulation and therefore failure risk is evaluated by post-processing the failure indicator. The Abaqus isotropic simulation illustrates the current method at VCC: averaged elastoplastic properties and a maximum allowed principal strain. As seen in Fig. 3 it gives an over-estimation of stiffness and conservative design with respect to failure. The available CrachFEM (for BASF ULTRAMID B3WG6 CR, PA6 with 30% glass fiber) model predicts too low stiffness, late initial failure and an early final collapse of the component. Note that this model is for a different material than in the components.

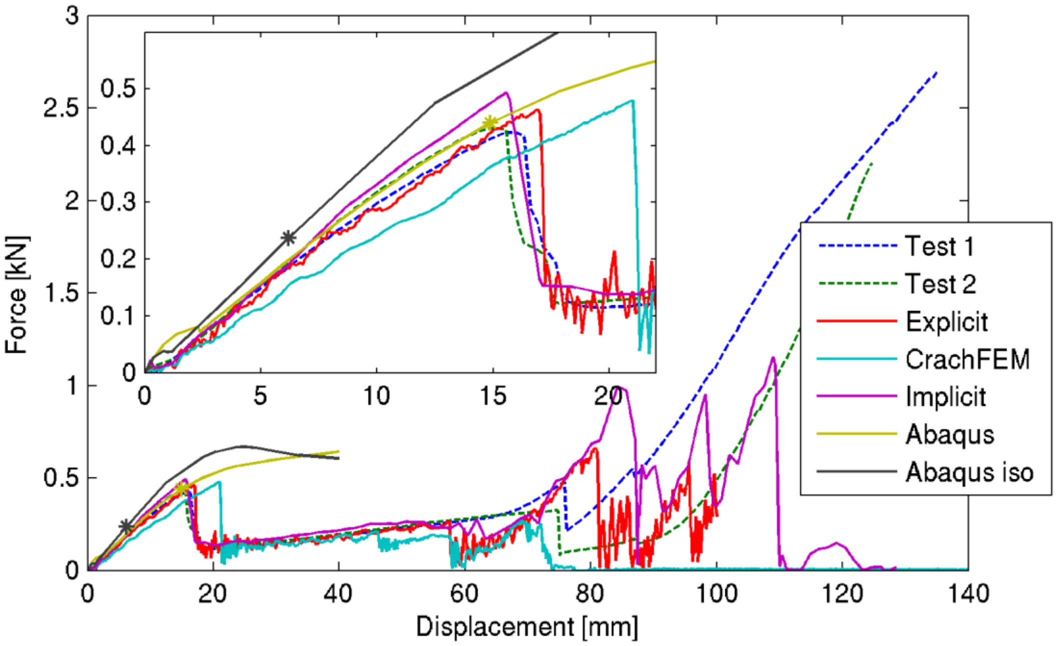

*Fig.3: Force-displacement response of the FEC in quasistatic three-point bending at center point. Result from two tests and the following four simulations are shown: LS-DYNA explicit with Digimat model, LS-DYNA explicit with isotropic CrachFEM model, LS-DYNA implicit with Digimat model, Abaqus implicit with Digimat model and Abaqus implicit with isotropic model. * indicates the predicted point of failure in the Digimat and the isotropic Abaqus simulations.*

The second quasistatic loadcase has the same setup as in the first except that the point of impact has been shifted to an off-center point. A picture of the simulation model is given in Fig.4: 4 and the response curves are given in Fig.5: 5. A similar set of results are presented and observation in line with the first case can be made. Note the two failure locations in Fig. 4 from the simulation with Digimat which occurs as in the test. This shift of failure location from center point to the location below the impactor is not predicted with the CrachFEM model.

© 2015 Copyright by DYNAmore GmbH

Fig.4: LS-DYNA simulation model with Digimat of quasistatic off-center three-point bending of the FEC.

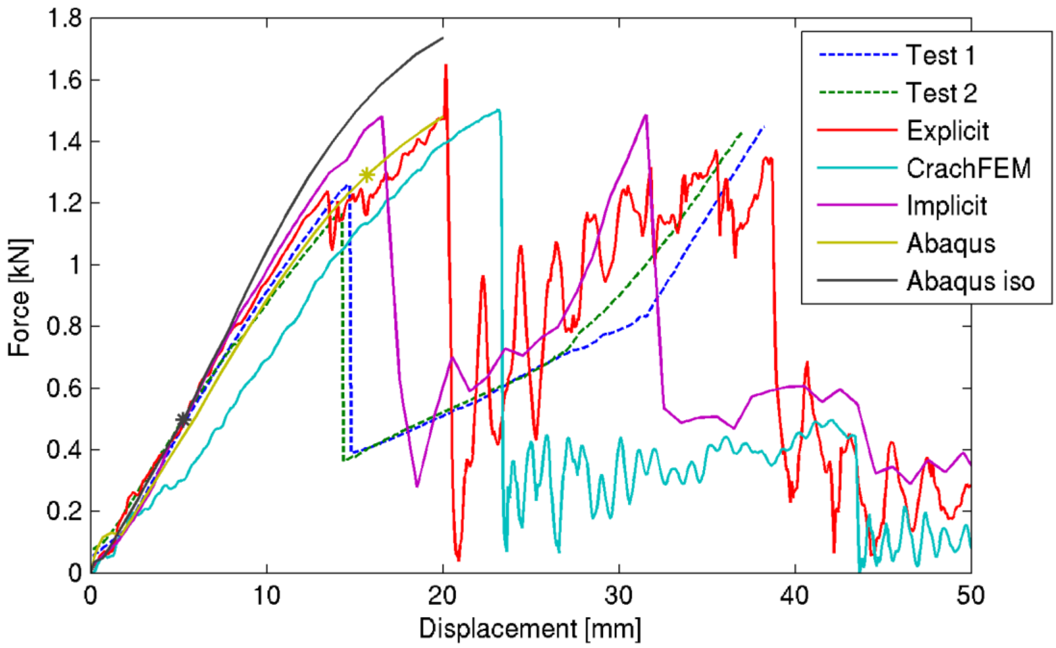

*Fig.5: Force-displacement response of the FEC in quasistatic three-point bending at an off-center point. Result from two tests and the following four simulations are shown: LS-DYNA explicit with Digimat model, LS-DYNA explicit with isotropic CrachFEM model, LS-DYNA implicit with Digimat model, Abaqus implicit with Digimat model and Abaqus implicit with isotropic model. * indicates the predicted point of failure in the Digimat and the isotropic Abaqus simulations.*

5.2 Dynamic component tests

The dynamic component tests were done in a droptower rig. Two cases were considered: on-center and off-center impact as in the quasistatic tests. A sequence of tests was done for each case with variation of the height from which the impactor was dropped to find the lowest impact velocity that would break the FEC into two parts. The same impact velocities were used in simulations to be compared to the tests. It is noted that the Digimat simulation very well predicts the initial stiffness while the failure strains are too high. In the Digimat simulation of the first loadcase, the FEC is not split all the way through. Instead the impactor bounces back upwards. It is possible that rate dependent failure could improve the failure prediction. For the CrachFEM prediction, it is initially too weak and the final collapse is too early. See Fig. 6.

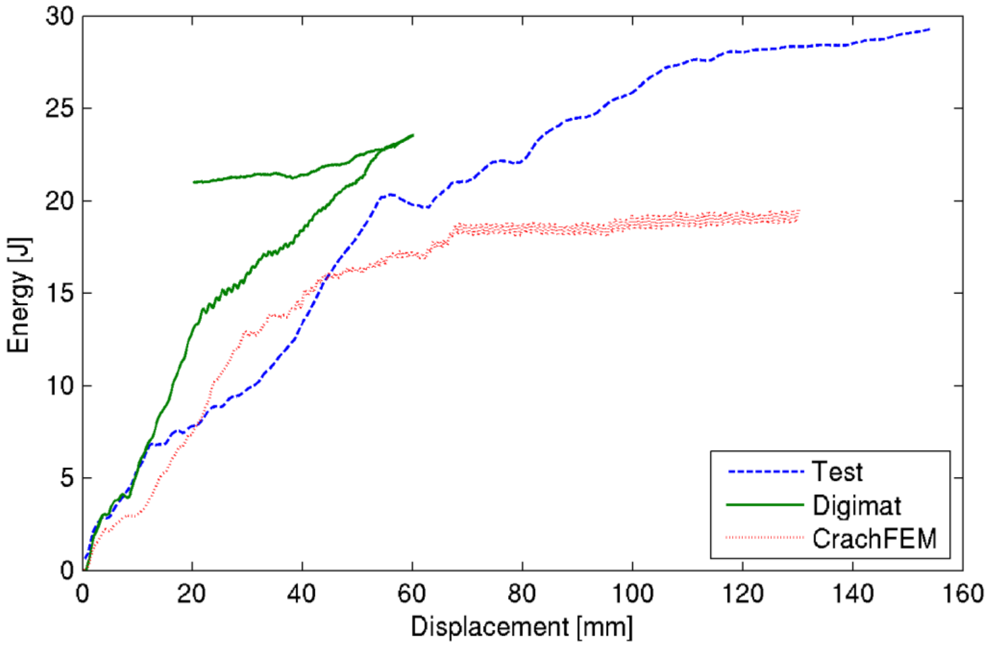

Fig.6: Energy–displacement response of FEC subjected to drop test at center point. Test results and results from LS-DYNA simulations with Digimat and CrachFEM models are included.

For the second dynamic case, there is more energy required to break the FEC. Thus the impact velocity is higher. In this case, Digimat slightly overpredicts the failure while the total energy sustained by the FEC is close to the test result. It should again be emphasized that the fracture path and thus failure levels are dependent on discretization. For the CrachFEM model, initial failure energy is close to the test result while the total collapse is too early. See Fig. 7.

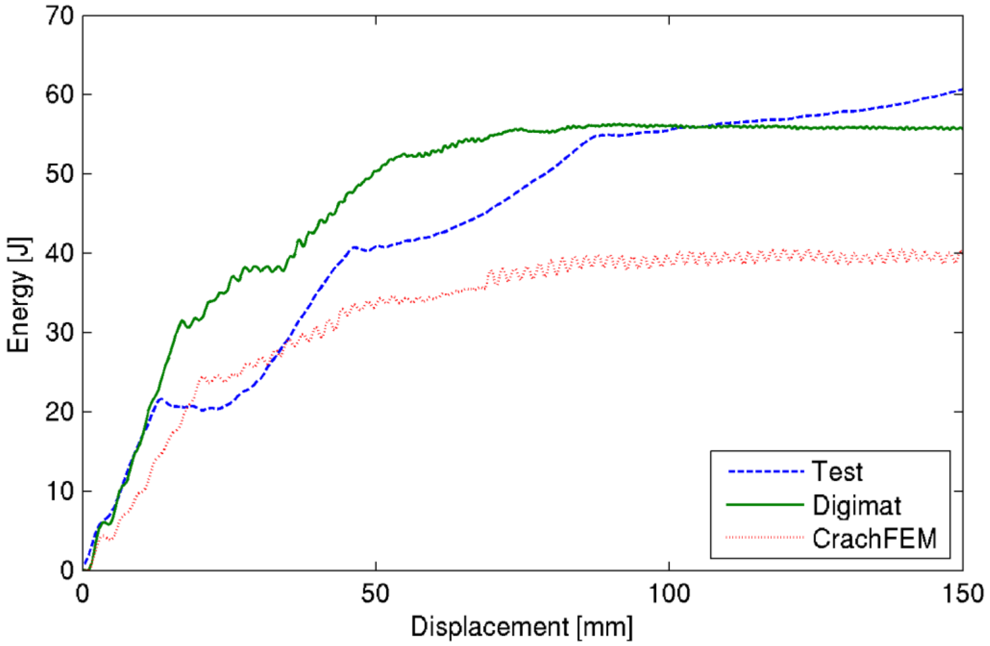

Fig.7: Energy–displacement response of FEC subjected to drop test at off-center point. Test results and results from LS-DYNA simulations with Digimat and CrachFEM models are included.

© 2015 Copyright by DYNAmore GmbH

5.3 Strength loadcase

In the “fat farmer” loadcase, an impactor is pushed onto the edge of the hood at different locations. For the current study of how Digimat performs in an Abaqus product development simulation, the location in the car centerline, with the impactor overlapping the hood and the grill, is chosen because load will be transferred through the FEC. However, there should be no permanent damage, so the FEC loading is small and well inside the elastic region. Thus, there are only small differences in results using the Digimat model compared to the isotropic model (see Section 5.1). Comparing the simulation cost using the two models, each iteration adds about 5% simulation time using the Digimat model. However, the convergence rate with the Digimat model is higher so that 9% total calculation time is saved using the Digimat model (Tab. 1).

Material Model	Elapsed time [s]	Number of iterations	Time/iteration [s]
Abaqus isotropic	10780	101	106.7
Digimat Hybrid	9814	88	111.5

Table 1: Computational cost comparisons for “Fat Farmer” Abaqus simulations with Digimat model and isotropic model.

5.4 Safety loadcase

To evaluate the performance of Digimat in a safety loadcase, an upper leg simulation with impact on the centerline of the vehicle front was run. The original 5 layer shell properties of the FEC are changed to 10 layers to provide a higher resolution of the orientation tensor through the thickness. In the particular case, the total simulation time was increased by 7%. Depending on the original decomposition of elements on the CPUs (cores), it is possible to spread parts on all CPUs using *CONTROL_MPP_PARTSET_DISTRIBUTE. Doing this in the current case lowers the increase of simulation time to 6%. See Tab. 2.

Elapsed time [s]	Total parts	Standard parts	CrachFEM parts	Digimat parts	Decomposition
17775	1554	1339	215		Slices in x-z plane
19016	1554	1339	210	5	Slices in x-z plane
18929	1554	1339	210	5	Even distribution of FEC

Table 2: Simulation times for an existing pedestrian model, with a mix of parts having standard LS-DYNA materials and CrachFEM materials, and a model where the isotropic CrachFEM model in the FEC parts were replaced by Digimat anisotropic model. In the third simulation the Digimat parts were distributed on all CPU cores.

© 2015 Copyright by DYNAmore GmbH

6 Conclusions

Simulation of two quasistatic loadcases and two dynamic loadcases with a Digimat model has been done yielding a good correlation with experimental results. There is potential to improve the confidence in the material parameters and possibly the predictions with a more complete set of test data and measurement of the orientation in the material test specimens. Furthermore, there is a need to enhance the failure model to account for different failure levels in tension and compression which would be accompanied by the need of additional test data. Furthermore, there has been no validation of the used Moldflow predictions of fiber orientations in the component. Material imperfections and the possible decrease of strength in weldlines have not been considered. For example microphotographs of the fractured surfaces in the components show the presence of pores that has not been accounted for in the modeling. For the dynamic testing, tests have not been repeated for each impact velocity. Also, measurement of displacement is done by integrating accelerometer data and the accuracy may be less than for direct measurement of displacement. Only one component and one material has been considered in this study. The planned step towards a full implementation of the material model is to test it in the daily product development simulations with additional components and materials.

Comparisons to alternative models have been done. For strength analysis, the improvement of the accuracy of the predictions of stiffness and failure load is substantial. There is also a decrease in simulation time when using the Digimat model in the particular case. However, there is demand for additional material and process data for an anisotropic Digimat simulation compared to standard isotropic Abaqus material. Additionally, anisotropic simulations in general require predictions of fiber orientations. For crash analysis, a direct comparison of Digimat and CrachFEM was not possible since the available CrachFEM model is isotropic and represents a different material than the one in the component. One difference between the models is that for a CrachFEM model usually more advanced experiments are used to calibrate the parameters.

7 Acknowledgements

Magnus Andreasson, MSC Software Sweden is acknowledged for his contributions including all the Abaqus simulations and analysis of results. Several persons at Durability and Safety departments at VCC are acknowledged for their contributions. Thierry Malo and colleagues at e-Xstream Engineering are acknowledged for their support. Lanxess is acknowledged for supplying material data.

8 Literature

[1] DIGIMAT: www.e-xstream.com

[2] Lindhult, J-A, Ljungberg, M: Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymer by Use of Digimat and nCode DesignLife (*In preparation*), Chalmers University of Technology, 2015

[3] Foss, P: “Prediction of the Drop Impact Performance of a Glass Reinforced Nylon Oil Pan”, 13th International LS-DYNA Users Conference, 2014

[4] Norman, O: “Anisotropic material modeling and impact simulation of a brush cutter casing made of a short fiber reinforced plastic”, Linköping University, 2014

[5] Mori, T, Tanaka, K: "Average Stress in the Matrix and Average Elastic Energy of Materials with Misfitting Inclusions", Acta Metallurgica, 21, 1973, 571–574

[6] Selmi, A, Doghri, I, Adam, L: “Micromechanical simulations of biaxial yield, hardening and plastic flow in short glass fiber reinforced polyamide”, International Journal of Mechanical Sciences, 53, 2011, 696-706

[7] CrachFEM: www.matfem.de

© 2015 Copyright by DYNAmore GmbH