ICME SOLUTION TO PREDICT CREEP OF SHORT FIBER REINFORCED THERMOPLASTIC AT PART LEVEL

Maxime A. Melchior, B. Alsteens
e-Xstream engineering, an MSC Software Company

Abstract

Short fiber reinforced thermoplastic part submitted to thermal and mechanical loading during a long period of time will observe creep. Creep depends obviously on the nature of the thermoplastic material but also on the amount of reinforcement, the type of reinforcement and the set-up of the injection process. Many ingredients that made the creep modeling of short fiber reinforced plastics a complex challenge.

Characterize the creep behavior of a reinforced thermoplastic material is a very expensive process. Test must be performed at various level of stress, for various specimen orientations and various temperatures. The test campaign to fully characterize the material can quickly lead to more than 50 tests to do, where each test can run for 1h to 1000h. A huge investment in testing for sure! A nightmare for material supplier, Tiers1 and OEM.

In this paper, ICME solution is going to be implemented to first reduce the test campaign by replacing some physical test by virtual test. Then, ICME solution will help to model accurately the creep behavior of the reinforced material by using a viscoelastic-viscoplastic material model in the frame of a multi-scale material modeling approach. Calibration of the material model is done from a mix of experimental and virtual material data. Finally, the thermo-mechanical response of a part will be predicted by using this advanced material model. ICME Solution will be the backbone of our work to predict the creep behavior of Short Fiber Reinforced plastic part.

Background and Motivation

Short fiber reinforced thermoplastics are used since many years in the automotive industry to replace metal material and lightweight parts. Manufacturing process will be either injection or compression molding. The influence of the manufacturing process must be captured to predict correctly the performance of the final part. Optimization of the design will be not limited to the geometry only but will integrate the manufacturing process too.

The manufacturing process governs the orientation of the fibers in the mold, the weld line position, the residual stress, the warpage… The final part performance is linked to all these characteristics. Depending on the part, different performances has to be evaluated like stiffness, strength, lifetime, creep, damping, …The part efficiency is related to its design but also to the material performance and its microstructure.

In this paper, creep is the performance targeted. Creep is much more critical for plastic material than for metal. Creep performance must be therefore evaluated for all parts made of plastic and submitted to a constant load during a long period of time with or without change of temperature. Oilpan are more and more made of short fiber reinforced thermoplastic to reduce their weight [1]. Oilpan observes a very complex temperature history and must support a given pressure load during a long period of time. The deformation of the oilpan along the time has to be limited to avoid any sealing issue at the connection with the engine block. Creep is therefore a physical effect that must be investigated in the development of this part.

Characterize the creep performance of a short fiber thermoplastic material is a huge investment in time and money. Short fiber reinforced thermoplastic shows an anisotropic behavior due to the fiber orientation. Creep response of the material is different if the load is applied in the direction of the fibers or in the transverse direction. The variation of creep due to the fiber orientation depends on the amount of fibers and the fiber size. This anisotropy in the creep response must be identified for various level of loadings and various temperature. Typically test campaign for a given material requires more than 50 tests where each test can run from 1h to 1000h:

- 4 to 5 load levels
- 2 directions. Specimens are cut out of a plaque in 2 different directions (i.e. 0° and 90°)
- 3 temperatures: room temperature and working temperature (i.e; : 23°C and 80°C)
- 2 to 3 replicates

Reliable results are obtained only if these tests are run in a dedicated room where the temperature and humidity level are maintained perfectly constant.

The number of tests and the associated cost justifies the need for a new approach. This approach will mix information coming from test and information coming from simulation. ICME, Integrated Computational Material Engineering, offers solutions to support us in the replacement of physical test by simulation. In this strategy, two material modeling approaches are combined to predict numerically the creep behavior of the material: the Mean-Field homogenization and the Finite Element homogenization. Both approaches are presented in the next sections and their usages to predict the creep behavior of short fiber reinforced thermoplastic is explained.

Material Modeling approach: Finite Element and Mean-Field homogenization

In both approaches, the short fiber reinforced thermoplastic material is assumed to be made of two major constituents: the resin and the fibers. For the fibers, the material model selected will be elastic isotropic for glass fibers or elastic transversely isotropic for carbon fibers. For the resin, the material model has to capture the viscous behavior of the plastic responsible of the creep. Depending on the load level, specific material models can be selected:

- Viscoelastic model if the load level stays low and don't lead to any plastic deformation.
- Viscoelastic-viscoplastic model if the load history leads to both elastic and plastic deformation.

Besides the selection of the material model for the fibers and the resin, the composite microstructure must is described by specifying the volume or mass fraction of fibers, their orientation and their size.

Mean-Field Homogenization

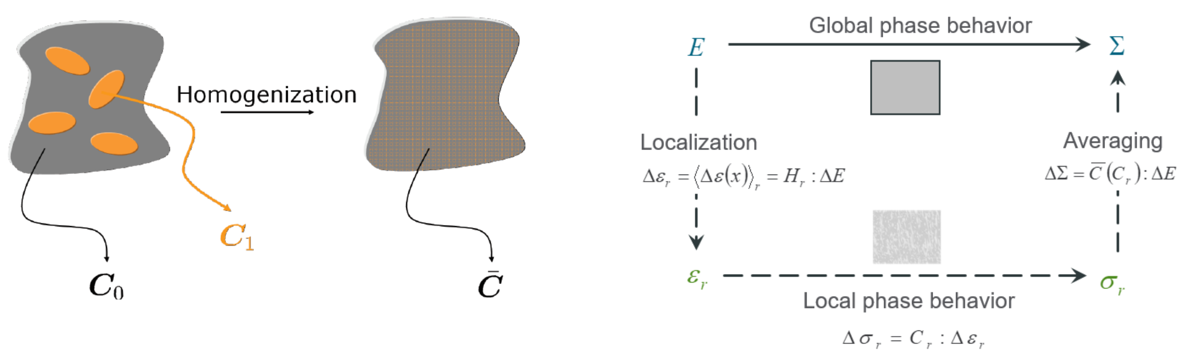
Mean-Field homogenization combines the properties of the underlying constituents of a multi-phase material so that the original heterogeneous material is represented by an equivalent homogeneous one. Implemented in the Digimat software [2], this technology has proven effective for a broad range of materials. This semi-analytical solution shows the advantage to be very fast in computing the composite material behavior. It can be used for various type of performances: stiffness, strength, failure, lifetime, nvh and creep. Nevertheless, it is based on some assumptions that can be sometimes not respected in the real material. Assumptions are:

- Fibers are ellipsoidal
- Fibers are uniformly distributed in the resin, no cluster.
- Fibers stick perfectly to the resin

Mean-Field homogenization is performed in three main steps as illustrated in Figure [1] :

- Step 1: the localization. The per-phase strain is computed from the strain applied on the composite.
- Step 2: the stress computation at micro-level. The stress in each phase is computed from the strain obtained at the step 1.
- Step 3: the averaging of stress. The stress at the macroscopic level is computed through an averaging approach based on the stress at the phases level.

The results provided by the Mean-Field homogenization are limited to the average stress and strain in the composite and in each phase (resin & fibers). This approach does not capture the peak of stress that can be observed locally in the composite at the tip of the fibers for instance.



(a) (b)

Figure 1: (a) Heterogeneous material (left) from which its equivalent stiffness $\bar{C}$ is computed from homogenization. (b) Three steps in the Mean-Field homogenization procedure: localization, per-phase stress computation and averaging.

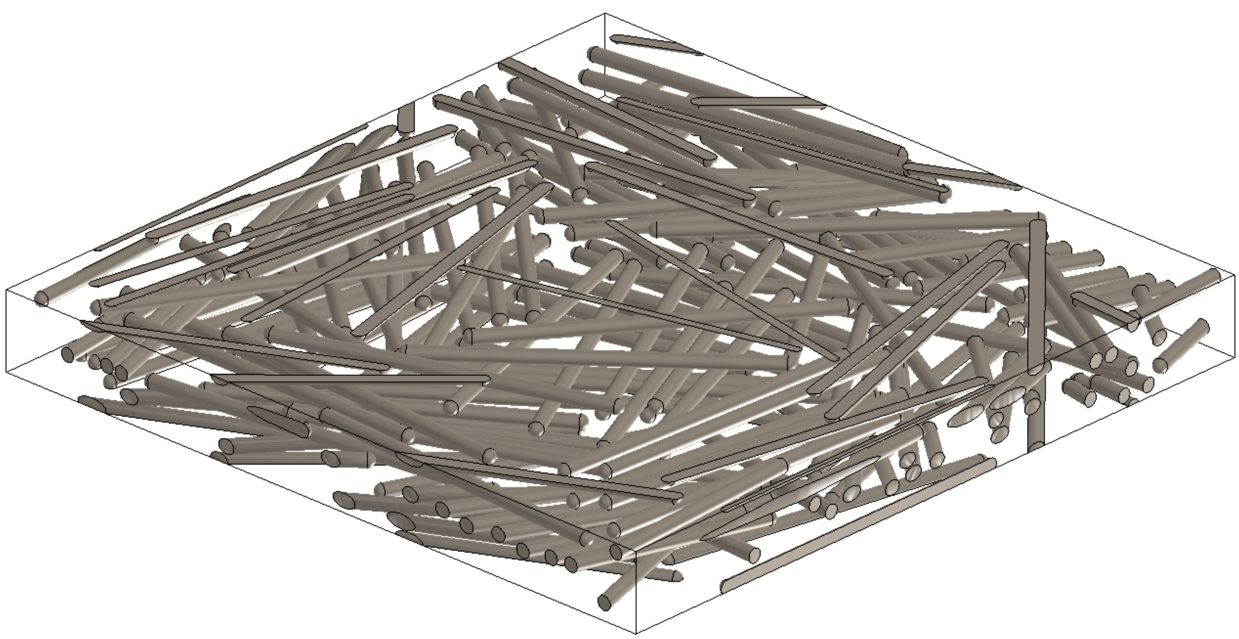
Finite Element Homogenization

The first step in the Finite Element homogenization is to create an RVE (Representative Volume Element) representing the microstructure of the composite. This RVE is meshed, material properties are applied to the different entities (fibers & resin), boundary conditions representative of the physical tests are set and a finite element solver is used to compute the behavior of the composite for this specific loading. Results are the full local stress and strain field in the RVE as well as composite engineering constant and global stress-strain curves. High level of details can be considered in the definition of the RVE like a non-uniform distribution of the fibers, the shape of the fibers that can be non-ellipsoidal, specific cohesion definition between the fibers and the resins, porosity, …

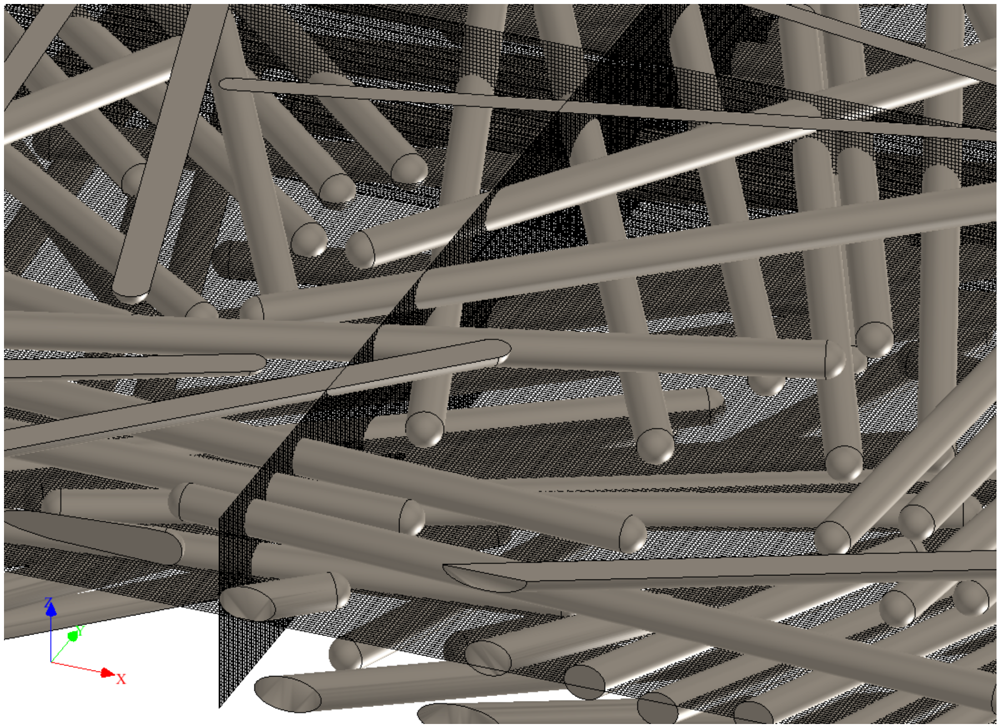
Standard disadvantages of this solution are the complexity of the mesh and the computation time. These weak points have been solved in Digimat by developing a spectral solver based on the Fast Fourier Transform approach. This solver is dedicated to the analysis of periodic RVE discretized with a regular grid of points, following the pioneering work of Moulinec and Suquet (1998) [3][4]. It has several advantages:

- No mesh must be generated
- High speed of resolution
- Smaller memory footprint

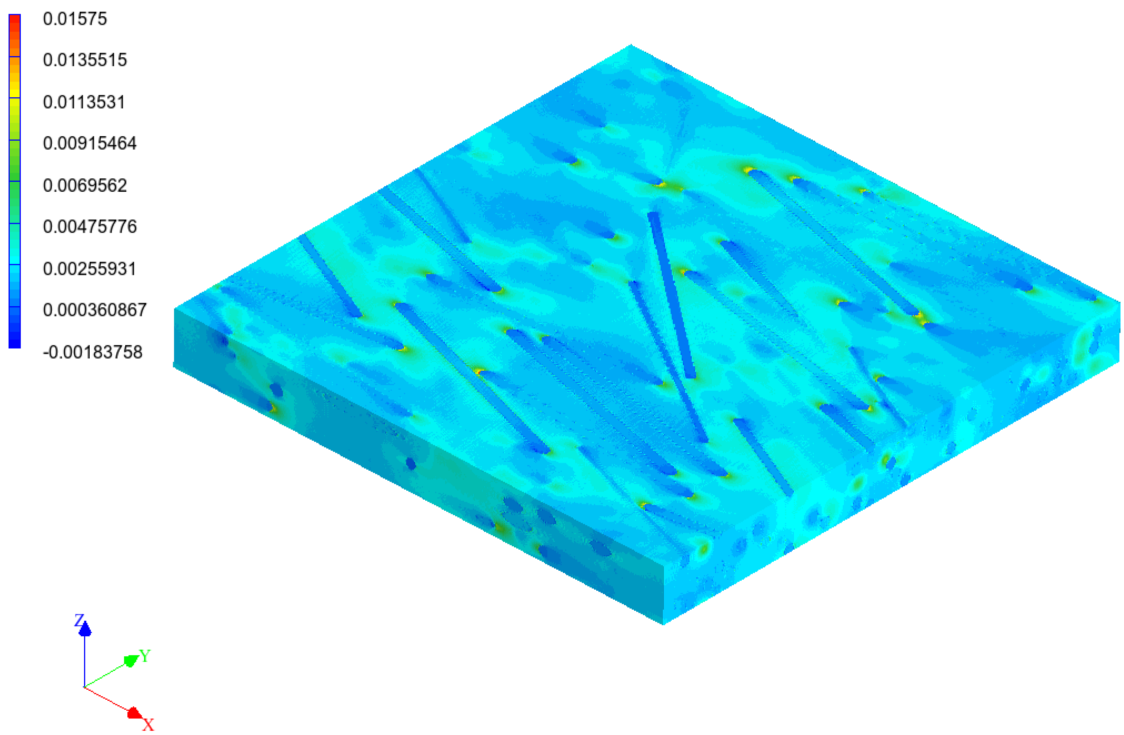
Figure [2] shows a typical RVE with the grid point distribution and the strain field. In Figure [3], the FFT solver results are compared with the results obtained with a classical Finite Element approaches. Stress-strain curves are matching well if enough grid points are used. Even if the number of grid points is 40 times the number of elements used in the traditional finite element model, the FFT solver is still almost 3 times faster than the Finite Element solution as show in the Table [1].



(a)



(b)



(c)

Figure 2: (a) RVE made of 120 inclusions for a mass fraction of 30% with an aspect ratio of 24.3. Generation time 3 sec. (b) The grid point used in the FFT solver : 14.155.776 points.(c) The strain field in the RVE

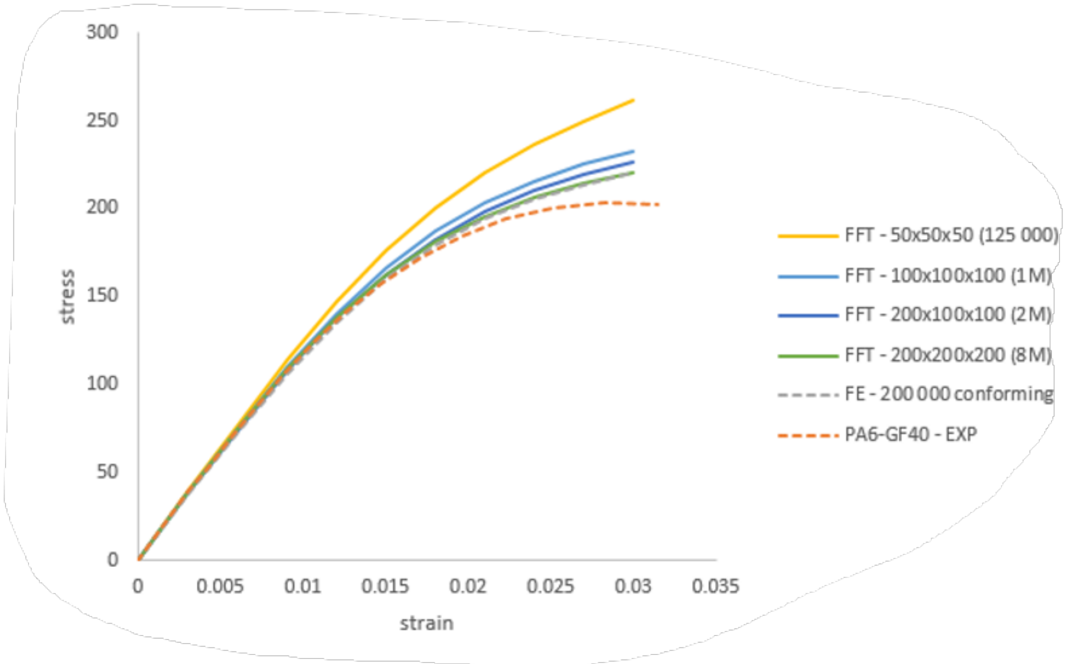


Figure 3: The prediction of the Finite Element Homogenization by using the standard Finite Element solver vs. the results got with the FFT Solver for a Polyamide 6 reinforced by 40% in mass of Glass Fibers. Microstructure is used in this example represent a directly injected specimen.

		Solution Time [s]
FFT Solver - Number of grid points	125000	31
	1000000	417
	2000000	833
	8000000	4993
Finite Element - Number of elements	200000	14000

Table 1: Comparison of computation time of the FFT Solver vs. the traditional Finite Element solver. Results obtained with Digimat 2019.1. Release 2021 will show an even faster computation time by using the GPU solution.

Mean Field and Finite Element homogenization methods are complementary. The Finite Element homogenization can be used to complement the composite test data by providing virtual test data with a high fidelity for both stiffness and failure. The Mean-Field homogenization provides a multi-scale material model that can be calibrated from test data and virtual data. This material model can then be used to predict the creep behavior of a complex part like the oilpan.

Virtual Material Testing

In this section, the creep behavior of short fiber reinforced thermoplastic material is predicted by a direct engineering approach. This approach consists in using the Finite Element homogenization solution to compute the creep behavior of the composite from the properties of the raw resin and the fibers. Focus is made here on the creep observed in the elastic domain.

The inputs for this direct engineering approach are:

- a viscoelastic material model predicting accurately the creep behavior of the raw resin,
- an elastic material model for the glass fibers,
- the microstructure information: fiber length and diameter, mass fraction of fibers and the orientations.

From these inputs, an RVE can be built and the creep behavior can be computed for various level of stress by using the FFT solver. This work can be done at different temperatures. A specific attention must be given to the microstructure to obtain reliable results. The microstructure can be considered as representative of the real composite only if :

- enough fibers are represented in the RVE (>100),
- fibers are correctly distributed inside the RVE : several fibers through the RVE thickness and several fibers fully inside the RVE.

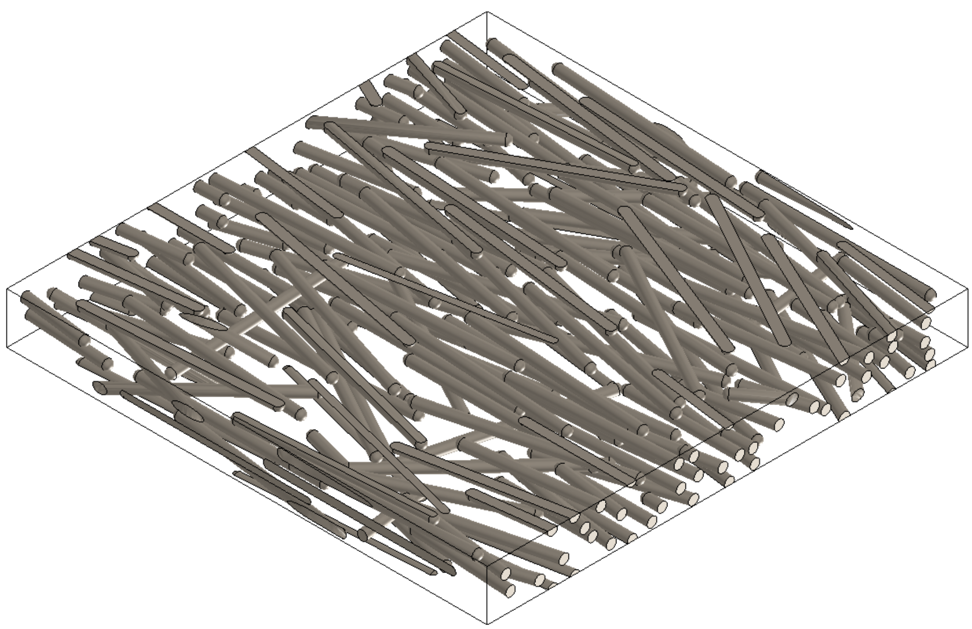


Figure 4: RVE used for the creep prediction of a thermoplastic material reinforced by 30% mass fraction of glass fibers.

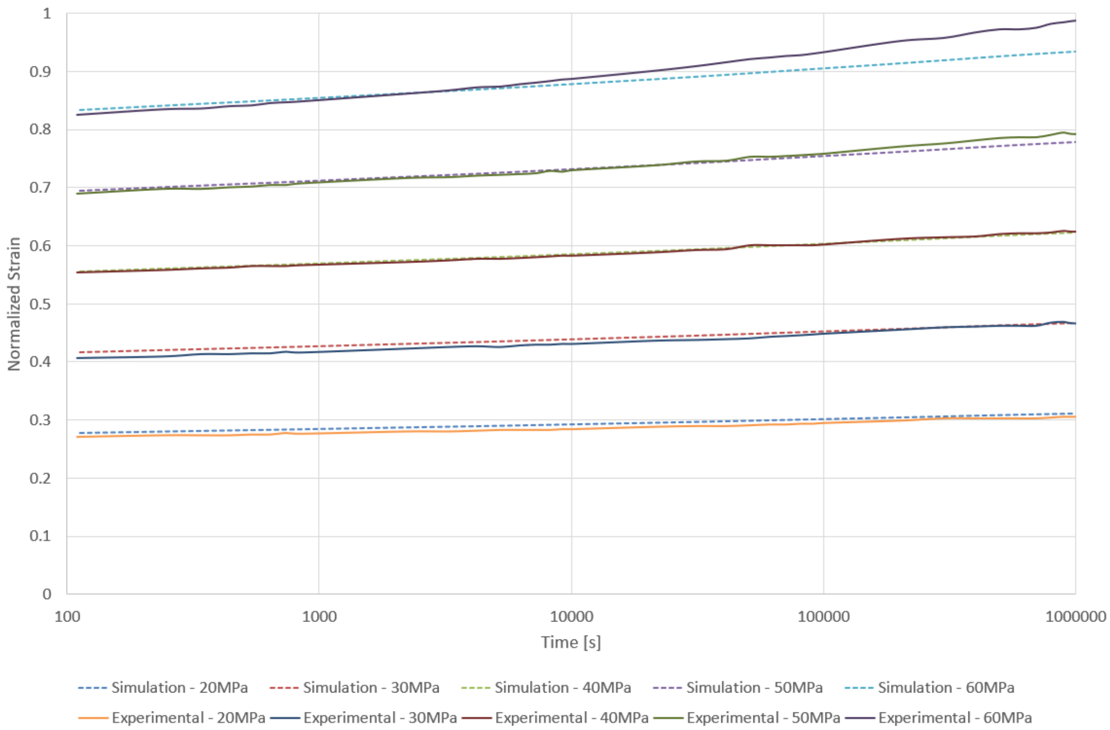


Figure 5: Creep evolution by simulation vs. experimental data for a short glass fiber reinforced thermoplastic material with 30% mass fraction and aspect ratio of 20.

Figure [4] shows the RVE used in our simulation. Mass fraction of fibers is 30%. Fiber aspect ratio (length/diameter) is 20. Five levels of stress are considered in these simulations: 20MPa, 30MPa, 40MPa, 50MPa and 60MPa. Strain evolution vs time are given in the Figure [5]. Results are normalized by the peak of strain observed at 60MPa. A perfect match is observed with experimental data for all stress levels between 20MPa and 50MPa. At 60MPa, a small deviation is observed when the creep test exceeds 10.000sec. At 60MPa, creep in plasticity becomes important and this speed-up the increase of strain. Our resin material model is here purely viscoelastic and cannot capture this effect. A viscoelastic-viscoplastic model should be used. This correlation between the virtual and experimental creep curves validates our approach and gives us a high confidence in our virtual results.

This model can now be used to generate creep curves for various loading direction: 0°, 45° and 90° to enrich the experimental data already available. RVE used in these different cases are illustrated in Figure [6].

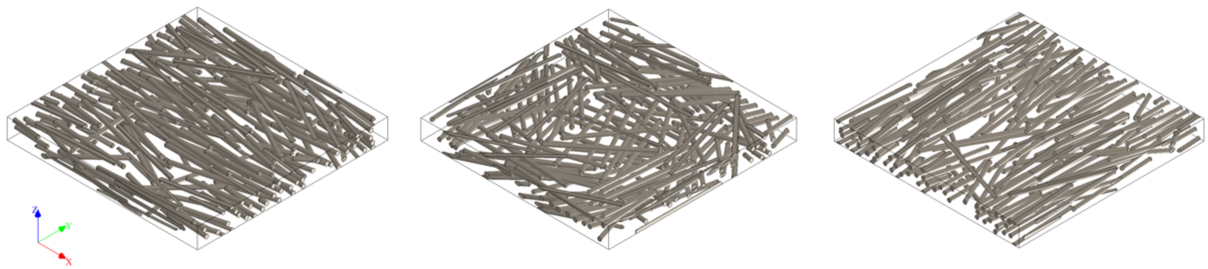


Figure 6: Creep load is applied along the x-axis. From the left to the right: the axial RVE, the Random 2D RVE and the transverse RVE.

Figure [7] illustrates the anisotropic creep response of this composite at 20MPa. Creep is more important in the transverse direction to the fibers than in the axial direction. Transverse response is mainly governed by the resin behavior, this explains why the creep is larger. Random 2D corresponds to a case where there is no preferential direction for the fibers in the plane 1-2. This leads to an intermediate level of creep strain.

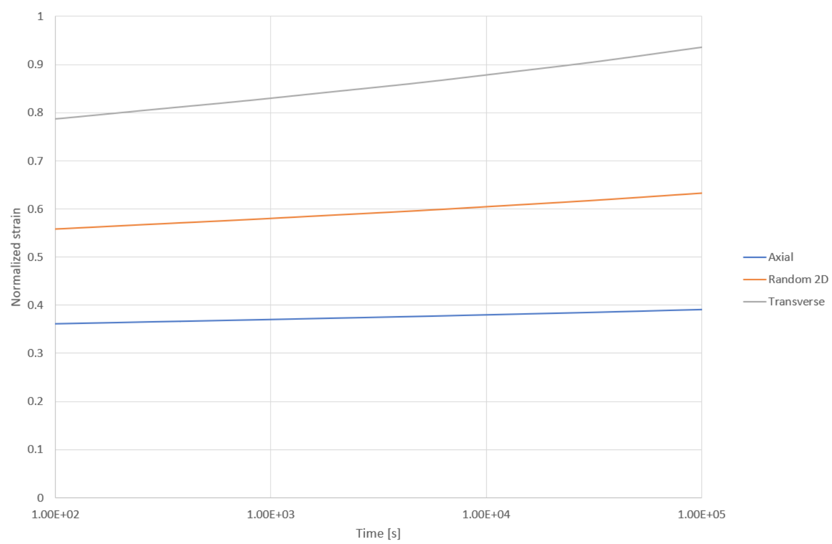


Figure 7: Anisotropic creep behavior predicted by the Finite Element homogenization solution for three fiber configurations: axial, transverse and random 2D at 20MPa.

These virtual test data are used to calibrate a multi-scale material model based on the Mean-Field homogenization solution, Digimat-MF. The calibration is very easy because the same inputs than for the Finite Element homogenization can be used. Without any adjustment, the results showed in Figure [8] are obtained.

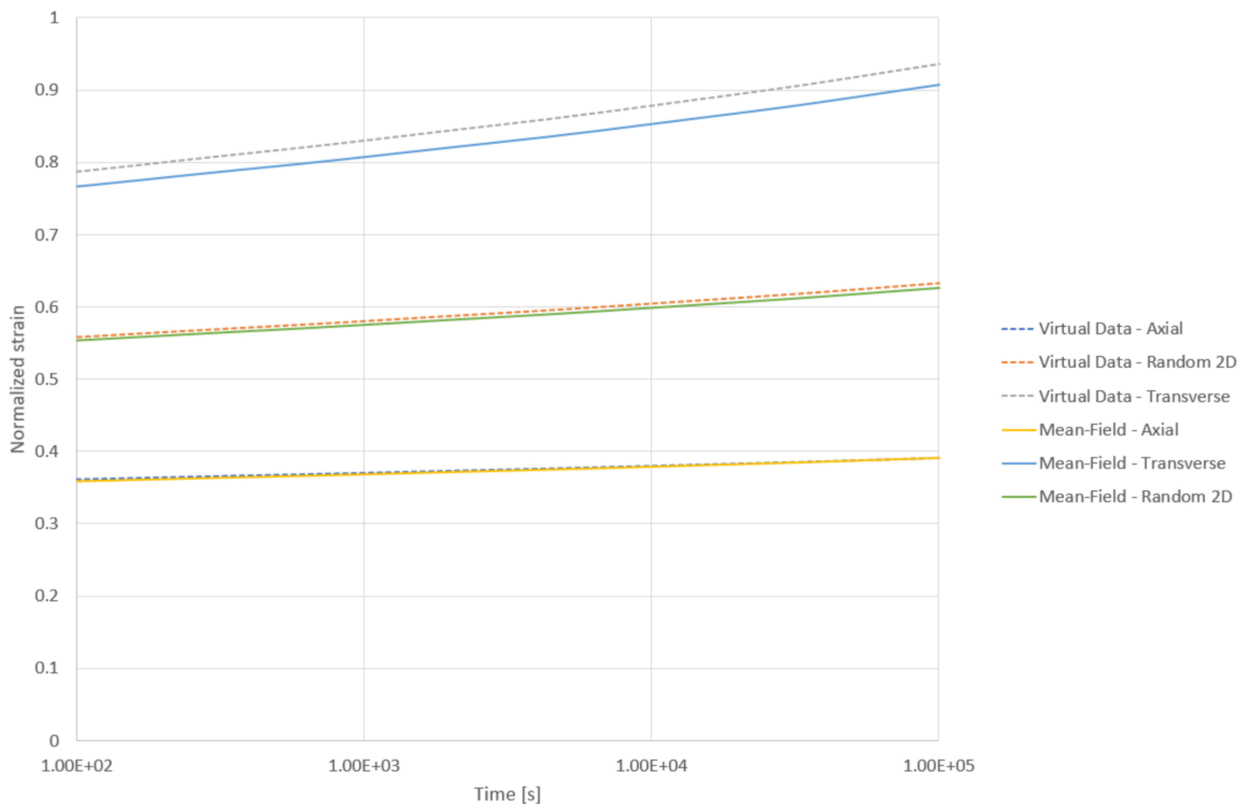


Figure 8: Mean-Field homogenization solution prediction vs. Finite Element homogenization solution.

The small difference between the two solutions comes mainly from the aspect ratio. In the Mean-Field homogenization, the fibers are supposed to be ellipsoid. Even if the aspect ratio is the same between the two approaches, the volume of the fibers is going to be different. For the same aspect ratio, the volume of the sphero-cylinders fibers used in the Finite Element homogenization is larger than the volume of the ellipsoid. A correction factor must be therefore applied to compensate this difference of volume. By changing slightly, the aspect ratio in the Mean-Field material model, the same creep behavior is then obtained between the two solutions.

Prediction of creep behavior at part level

This multi-scale material model obtained in Digimat-MF can be used to predict creep at the part level. To do so, few inputs are needed:

- FEA model of the part with the right loading and boundary conditions.
- Manufacturing data: fiber orientations in the part depending on the injection process and weldline position

With this information a coupled Digimat – FEA analysis can be set-up by using the Digimat-RP solution. In few clicks, the Digimat material model obtained at the previous step can be assigned on the part and the link with the manufacturing data can be done as illustrated in Figure [9]. If the model used to compute the fiber orientation in the part is different than the model used in the structural analysis, a mapping of the fiber orientation and the weldlines can be done to have data compatible with the structural mesh.

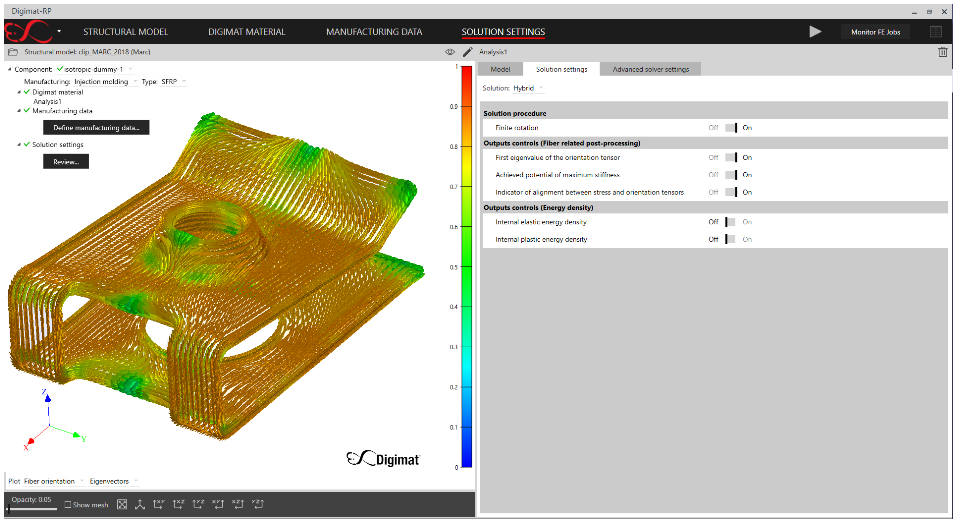


Figure 9: Digimat-RP is the solution used to create the bridge between the part performance and the manufacturing process by using the hybrid solution based on the Mean-Field material model.

In this structural analysis, the Digimat Hybrid solution is used to predict the material behavior in every integration point of the part. The Hybrid solution is a robust, fast and easy reduced model built on the top of the Mean-Field homogenization which is designed specifically for advanced non-linear applications in order to provide accurate predictions in an industrial time frame. The load history applied on this part is given in the Figure [10].

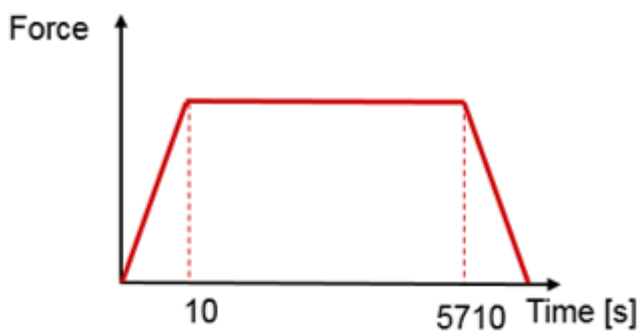


Figure 10: Constant load is applied on the part during 5700 leading to some creep deformation.

This constant load applied on the part during 1h35min leads to an increase of deformation of the part. After the first loading step (10s), the magnitude of displacement is 2.040mm. This displacement increases progressively to reach the maximum value of 2.240mm. An increase of 10% due to the creep behavior of the material as shown in Figure [11].

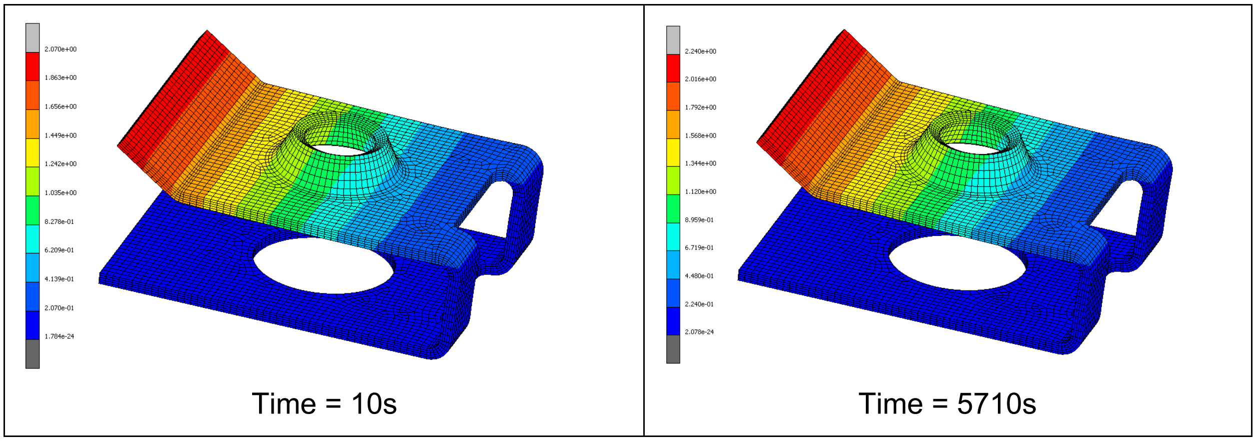


Figure 11: The magnitude of displacement increases by 10% in 1h35min, time during which the load is maintained constant.

Perspectives

Our goal for the conference is to extend this approach to the most complex material model: the viscoelastic-viscoplastic. This will allow us to predict the creep behavior of the material in elastic and plastic regime simultaneously. This extension of our solution is an on-going work. Few results have been already obtained at the material level. Work at the part level will be done for the conference by using the same hybrid solution.

Conclusions

The ICME, Integrated Computational Material Engineering, offers a great environment to investigate the behavior of composite material, to predict the behavior of the material in various condition and to represent the material as accurately as possible at the part and assembly level.

In this paper, ICME has been used to predict the creep behavior of short fiber reinforced thermoplastic material from a limited set of inputs: the raw resin behavior, the fiber behavior and the microstructure information. Prediction has been successfully validated against experimental data. A Mean-Field Digimat material model has been then calibrated from this virtual data and used to predict the creep at part level.

The next steps will be to move from the viscoelastic material model to a viscoelastic-viscoplastic model to predict the creep behavior on the full range of deformation that can be supported by the short fiber reinforced thermoplastic. This on-going work will be done at material and part level.

References

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

[2] e-Xstream engineering, "Digimat documentation Release 2020.0," 2020.

[3] Moulinec H., Suquet P., “A numerical method for computing the overall response of nonlinear composites with complex microstructure”, Compu. Methods Appl. Mech Engrg 157, pp. 69-94, 1998.

[4] Zeman J., de Geus T. W.J., Vondrejc J., Peerlings R. H. J., “A finite element perspective on non-linear FFT-based micromechanical simulations”, International Journal for Numerical Methods in Engineering 111(10), pp. 903--926 · September 2017