

# Multi-scale Material Modeling Applied from Specimen to Full Car Level using LS-DYNA®

Sylvain Calmels  
*e-Xstream Engineering*

## Abstract

*Tomorrow's vehicles architectures will involve an increasing number of materials. Within this worldwide shared status, fiber reinforced materials are finding their way into not only aesthetic components but also semi-structural and structural components. Today, the question of how to accurately model such parts is not a topic of discussion anymore. All solutions dedicated to these materials proposed on the CAE market are based on multi-scale material modeling techniques. However, even if the modeling strategy has now been adopted, its integration into the whole vehicle design process from the lowest level up to the most complex one on full car is still a challenging task.*

*At specimen level, the main concern is to obtain enough data in order to create an accurate multi-scale material model. In addition, depending on the development phase (early design or advanced design), some additional constraints could complicate the work even more. In early design phases, when the selection of a material is not yet decided, experimental testing is not an option due to the prohibitive cost associated with a large campaign. Instead, in advanced design phases, the best material candidates having been identified, such testing campaigns can now be performed. But which test is required? Is it necessary to cover the entire test matrix physically or is it possible to virtually characterize some of the behaviors? Then comes the application at component level and the need to keep design iteration loops as short as possible, even shorter, if possible, than what was existing for metals. How can the design engineer quickly and efficiently access information regarding fiber orientation distribution through its plastic component in order to account for its effect in his structural analysis? Finally, on full car simulations, model stability and CPU time become the main concern. Is such technique applicable on a full car crash FEA with good robustness and acceptable range of execution time? The present paper addresses how a multi-scale material modeling approach developed through the Digimat platform can be applied from the specimen level up to the full car level with efficiency and a high-level of accuracy. Guidelines recommended by e-Xstream and an overview of the whole CAE chain will be discussed in a practical case study.*

## Introduction : metal replacement projects remain a challenging

In the automotive industry, metal replacement with lighter composite materials is a recurrent theme. The influence of the manufacturing process on reinforced plastics' performance has been a frequent topic of research over the past 15 years. Multi-scale modeling techniques taking into account the fiber orientations resulting from the injection process have been developed and applied in the automotive industry to predict the performance in terms of crash, fatigue, NVH, creep or thermos-mechanical response. Such technique has significantly improved the accuracy of FEA simulations compared to the "black metal" isotropic approaches, by taking into account the influence of local fiber orientation on the material response.

This technique is based on a mean field homogenization scheme aiming at predicting every possible local anisotropic behavior from its constituent's ones plus the local fiber orientation information obtained from the manufacturing process simulation. Figure 1 below describes how a "local macroscopic" behavior can be calculated from constituents properties for all kind of performances ranging from linear and non linear stiffness, failure, strain rate dependency, damping, fatigue failure, creep or coefficient of thermal expansions.

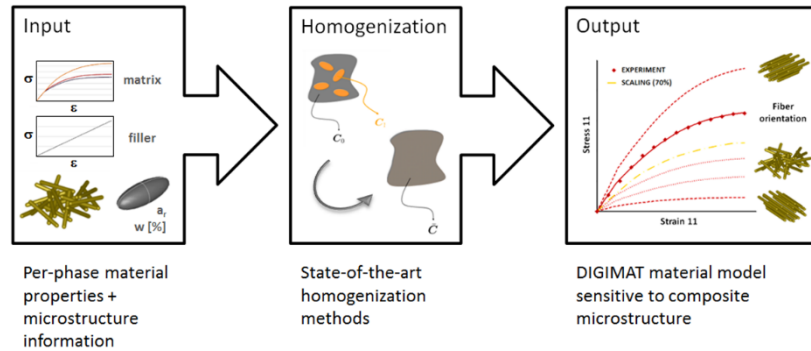


Figure 1: Mean Field Homogenization to predict local anisotropic behaviors from constituent’s properties

Commercial softwares like Digimat offers now the possibility to apply such material models at the structural level. These combines the accuracy of multi-scale material models with mapping capabilities aiming at transferring the local fiber orientation information from the manufacturing simulation mesh onto the structural mesh. Once done, the final FE model contains all required information for a high-fidelity prediction :

- A structural FE model which can be implicit or explicit, notably LS-DYNA
- A material model capable of providing all possible local anisotropic behaviors depending on the local fiber orientation
- The distribution of fiber orientation tensors on all the structural mesh elements

Figure 2 below presents the various possibilities available to run multi-scale material models with LS-DYNA FE solver accounting for the effect of a microstructure state which can be predicted by various manufacturing softwares available on the market :

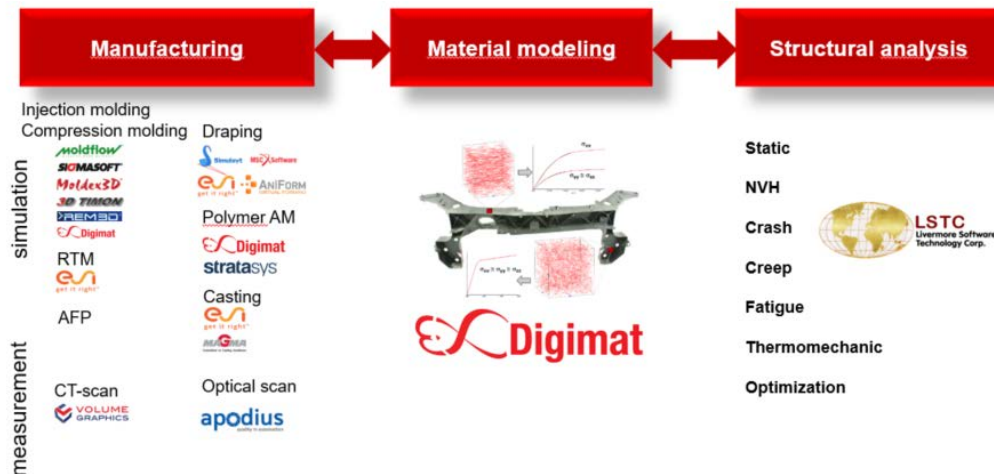


Figure 2: coupling a multi-scale material model with LS-DYNA to account for the effect of local microstructure as predicted by a manufacturing process simulation software

However, many questions remain to be answered when applying such modeling technique in real life scenarios:

- What are the data required to build a multi-scale model, especially in early design phases ?
- Are coupon level tests needed?
- Does a material database exist?
- Who will be performing the manufacturing process simulation?
- What are the computational costs?
- Is it applicable on a full car level?

The present paper will answer those questions. A design workflow will be proposed and applied to a front end carrier. This step-by-step application will demonstrate how this modeling approach can be easily applied from material data level up to the full car level.

### **Proposition : Early Design Workflow**

This section will focus on a proposition of design workflow applicable during early design stages. This design phase is particularly difficult to treat in metal replacement projects due to the fact that everything remains undefined: from the material to be applied to the manufacturing process.

Figure 3 below summarizes the main steps of the proposed workflow :

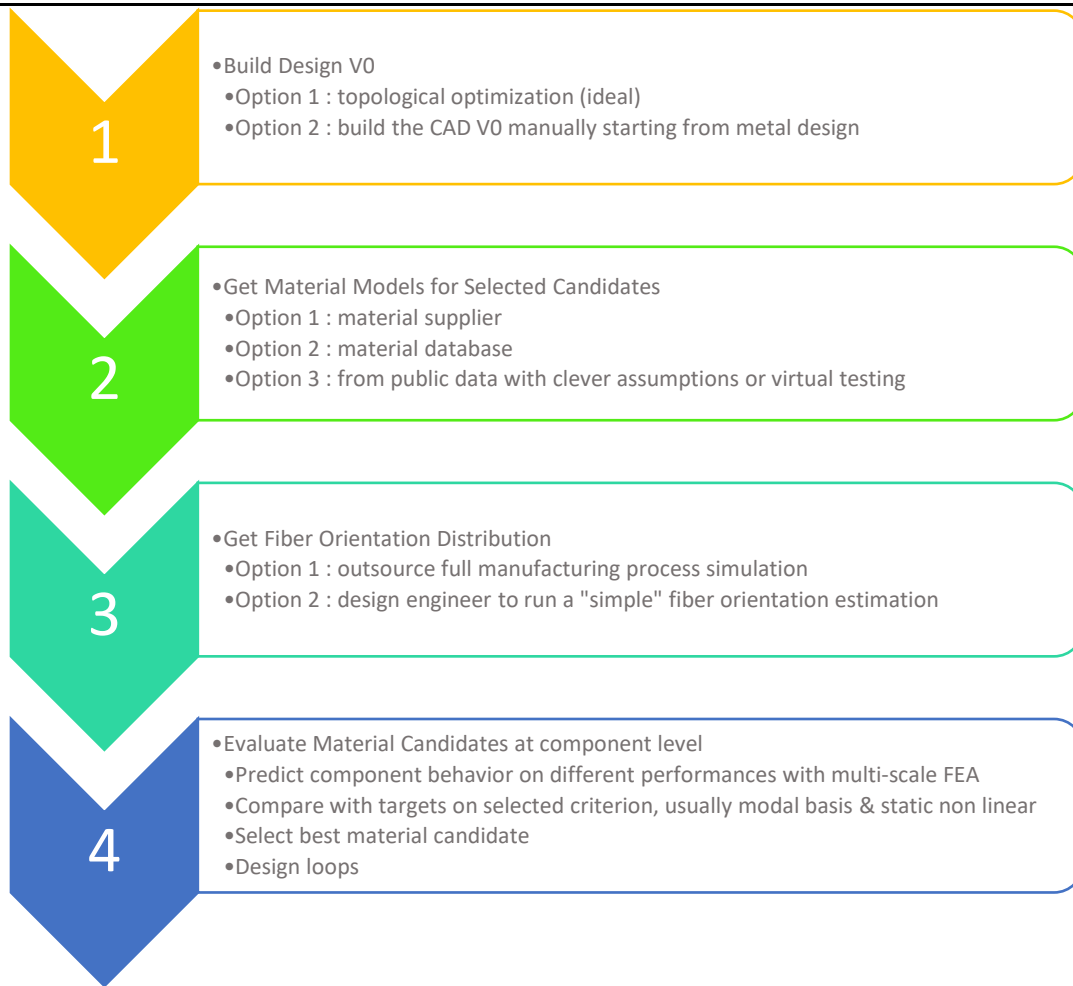


Figure 3 : Early Design Workflow applying a multi-scale material modelling approach

- **Step 1 : Build Design V0**
  - **Goal :** build a first design tailored to the usage of reinforced plastic material and process
  - **Keypoints :**
    - Plastics' main weakness (compared to metals) cannot be compensated by an increase in thickness alone.
    - The advantage of using plastics compared to metals lies in its geometrical inertia which is obtained by adding ribs to the structure.
    - Ribs must be added in such a way that the final design must be unmoldable.
    - For panel type of designs, wall thicknesses are often around 3mm.
    - Bolted areas can require the usage of additional metal inserts to hold the tightening loads correctly.
  - **Option 1 : perform a topological optimization with tools like LS-OPT®**
    - Topological optimization can only be performed with isotropic material models, which means that the microstructure effect on the component behavior cannot be taken into account.
    - At this stage of the design process, as the design itself is not defined, it is impossible to define molding settings and get fiber orientation distribution. Hence, applying a multi-scale material model during the topological optimization is not feasible.

- However, even with an isotropic material model, predictions will lead us towards the proper design. Hence, when possible, this should be considered seriously
- **Option 2 :** build manually the CAD V0 starting from metal design
  - Add ribs on initial metal design, increase local thicknesses, add metal inserts in bolted zones.
- **Step 2 : Get Material Model for Selected Material Candidates**
  - **Goal :** get multi-scale material models capable of capturing the effect of fiber orientations on the local anisotropic behavior for all selected material candidates
  - **Keypoints :**
    - During early design stages, 2, 3 or more materials can be candidates for the final applications
    - Ideally, building a material model for nonlinear static application requires test data at sample level for 3 loading angles with regards to fiber orientations in samples for all conditions to be considered in terms of temperature and moisture level: 0°, 45°, 90°
    - Time and cost constraints during early design phases forbid full characterization campaigns for all possible candidates, hence, the design engineer must be able to work with what is available, not more.
  - **Option 1 : Material Supplier**
    - Several material suppliers are using multi-scale material models like Digimat. When it is the case, a potential future customer of 1 grade can request its material supplier to provide a multi-scale material model which is already existing in their database. This model will probably be encrypted but can be used in a structural FEA.
  - **Option 2 : Digimat Material Database**
    - The Digimat material database contains more than 15000 material models, mostly glass or carbon fiber reinforced thermoplastics.
    - 1 possibility can be to select one of these models for a similar grade to the one targeted. In early design phase, this will allow to assess if a given reinforced plastic with a given design can possibly reach the required mechanical performances.
  - **Option 3 : from public data with clever assumptions or virtual testing**
    - The last possibility can be to gather public data available on the web or by the material suppliers.
    - Usually, only axial data can be found. With clever assumptions on transverse behaviors, a user can build a multi-scale model which, as a first approach, will already provide a very good level of accuracy.
    - The missing transverse behaviors could also possibly be predicted using virtual testing capabilities of such a multi-scale modelling platform like Digimat.
- **Step 3 : Get Fiber Orientation Distribution**
  - **Goal :** predict the fiber orientation distribution throughout the design V0 for each possible manufacturing process settings (e.g. the number, diameter and positions of gates for injection molding) in order to map it onto the LS-DYNA FE model and link it to the multi-scale model obtained in step 2
  - **Keypoints :**
    - Usually such data is obtained thanks to a molding simulation like Moldflow or Moldex3D for injection molding
    - Such calculation aims at predicting much more than fiber orientations alone. These complex calculations are designed to optimize the complete manufacturing process,

defining the melt and mold ideal temperatures, the filling time, optimize the mold filling, etc...

- A structural FEA with multi-scale material model requires only the fiber orientation distribution as an input. Some solutions exist to perform simpler injection molding simulation, making it easy to set up for a non-specialist and still providing an accurate prediction of the final fiber orientations
- **Option 1 : outsource the manufacturing process simulation**
  - Most usually, tools such as moldflow or moldex3D are not available for directly within design teams. Hence, it is necessary to request to run a calculation by another department or supplier. In the end, a fiber orientation distribution file and the associated manufacturing meshes are provided to the structural engineer who can then map this information on its structural mesh.
- **Option 2 : (for injection molding) the structural engineer to run a simple fiber orientation in Digimat**
  - A simpler solution has been embedded into the Digimat suite which allows a structural engineer to define an injection molding simulation in a few clicks by simply positioning as many gates as required on its structural mesh. Digimat will provide a “fiber estimation” which is nothing more than a simple version of an injection simulation focusing only on the prediction of fiber orientation distributions. At the end of the process, Digimat will automatically map the fiber orientation distribution onto the structural mesh.
- **Step 4 : Evaluate Material Candidates at component level**
  - **Goal** : run structural FEA on design V0 using multi-scale material models linked to fiber orientation distribution mapped onto the structural mesh and compare final component’s performances to selected best candidates and iterate on design.
  - **Keypoints** :
    - During very early design phases, the focus is on “simpler” performances like first eigenfrequency and/or the behavior under normal and exceptional static loadings.
    - Crash can be addressed as well but will require more work to gather enough data and build associated material models. It makes more sense to address this performance once the material is selected. It will be then possible to focus the modeling effort on one material alone.

## Application : metal replacement on a front end carrier

The previously proposed workflow has been applied to a steel front end carrier mounted on a 2012 version of the Toyota CAMRY.

Each of the 4 steps previously presented has been applied on this component in order to build and evaluate the performance of a very first plastic design version.

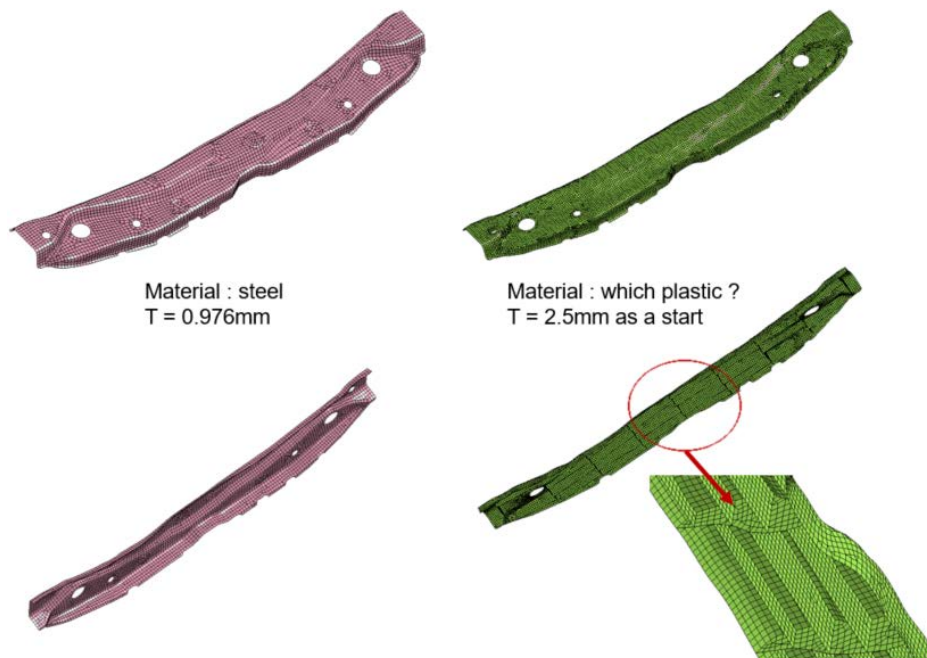
A last step has been added by introducing the plastic component into a full car frontal crash model. The target is here to evaluate the feasibility, the robustness and the impact of considering the effect of the microstructure on this component's behavior on the computational efficiency of a full car crash calculation.

The 2012 Toyota CAMRY full car LS-DYNA explicit model as well as the metal version of the front end carrier model has been developed [1] and validated [2] by the Center for Collision Safety & Analysis at George Mason University and the Federal Highway Administration of the US.

- **Step 1 : Build Design V0**

For the present exercise, the V0 design has been built with an usual FE modeler starting from the initial steel design by adding ribs and increasing global wall thicknesses. Note that at this stage of the process, neither the plastic material nor the global wall thickness are known. In order to perform the first iteration, an initial wall thickness of 2.5mm has been defined on the plastic model.

Figure 4 below illustrates the initial steel made front end carrier and the plastic design V0 :



*Figure 4: overview of the first plastic design V0*

The 2<sup>nd</sup> step will be to select the material candidate and to build a multi-scale model in order to run a FEA on the new plastic component.

In our case, the evaluation of the design was based on 2 criteria:

- 1<sup>st</sup> eigenfrequency
- Force/displacement curve from a 3-point bending test

2 LS-DYNA models have been built on the plastic design V0 :

- Modal analysis on LS-DYNA implicit
- 3-point bending on LS-DYNA explicit

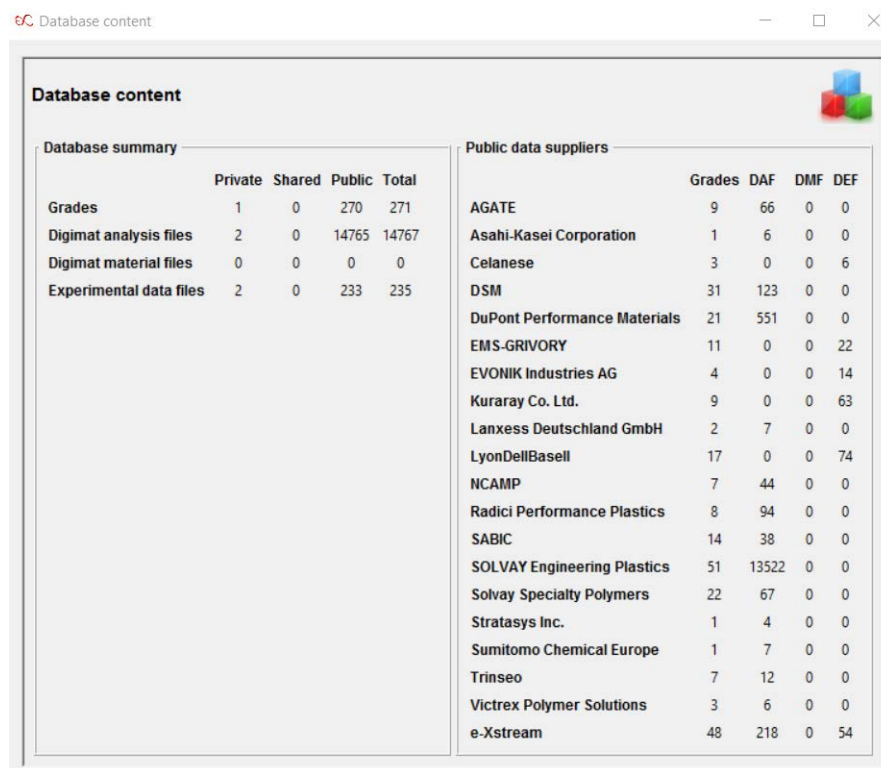
Reference performance will be defined by the performance of the metal design. Final target should be to get a component which provides at least the same level of performance than the metal design.

### • Step 2 : Get Material Model for Selected Material Candidates

The goal of this exercise being to demonstrate a global workflow and not to perform a full design optimization loop, only 1 material candidate will be evaluated which will be a PPGF40. PP resin being a type of resin often chose for under the hood applications thanks to its good capacity to maintain good mechanical performance with temperature increase.

The choice has been made here to prioritize the time of execution. In such situation, the fastest way to proceed is to look into the available Digimat-MX database and pick the available material models.

Figure 5 below shows an overview of all material models available in the Digimat-MX database :



Database content

Database summary					Public data suppliers				
	Private	Shared	Public	Total		Grades	DAF	DMF	DEF
Grades	1	0	270	271	AGATE	9	66	0	0
Digimat analysis files	2	0	14765	14767	Asahi-Kasei Corporation	1	6	0	0
Digimat material files	0	0	0	0	Celanese	3	0	0	6
Experimental data files	2	0	233	235	DSM	31	123	0	0
					DuPont Performance Materials	21	551	0	0
					EMS-GRIVORY	11	0	0	22
					EVONIK Industries AG	4	0	0	14
					Kuraray Co. Ltd.	9	0	0	63
					Lanxess Deutschland GmbH	2	7	0	0
					LyonDellBasell	17	0	0	74
					NCAMP	7	44	0	0
					Radici Performance Plastics	8	94	0	0
					SABIC	14	38	0	0
					SOLVAY Engineering Plastics	51	13522	0	0
					Solvay Specialty Polymers	22	67	0	0
					Stratasys Inc.	1	4	0	0
					Sumitomo Chemical Europe	1	7	0	0
					Trinseo	7	12	0	0
					Victrex Polymer Solutions	3	6	0	0
					e-Xstream	48	218	0	54

Figure 5: overview of material models available in Digimat-MX



Our second criteria are an eigenfrequency and a non linear static calculation result. Hence, two material models are required from the database. An elastoplastic model and an elastic one for PPGF40. Such models are available in our public database among the e-Xstream models :

Grades		Digimat analysis files		
	Trade name	Matrix model	Temp.	RH
1	e-X_GENERIC_PP_GF40	J2_plasticity	23	0
2	e-X_GENERIC_PP_GF40	J2_plasticity	80	0
3	e-X_GENERIC_PP_GF40	elastic	23	0
4	e-X_GENERIC_PP_GF40	elastic	80	0
5	e-X_GENERIC_PP_GF40	J2_thermoplasticity		0
6	e-X_GENERIC_PP_GF40	thermo-elastic		0

Figure 6 : e-Xstream generic models available for PPGF40 at RH0

The elastoplastic model at 23°C / RH0 has been selected and will be used for both the modal analysis and the 3-point bending case to be performed on the front end carrier.

Figure 7 illustrates the stress/strain curves describing the various behaviors that the multi-scale model will be capable of providing to the FEA model depending on the local fiber orientation tensors :

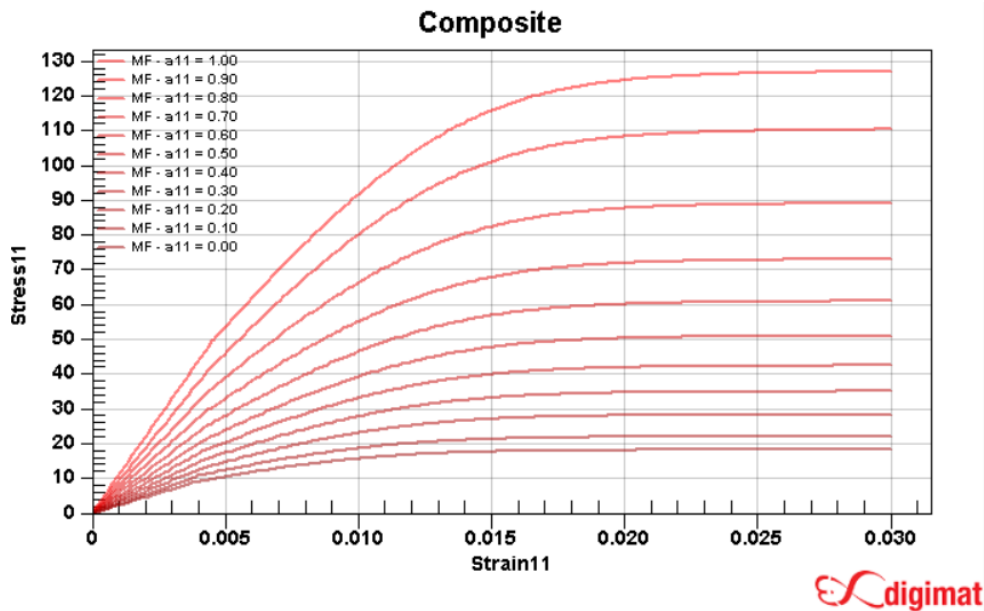


Figure 7: overview of all axial behaviors provided by the multi-scale material model depending on the a11 coefficient of the orientation tensor ; model does not contain a failure model

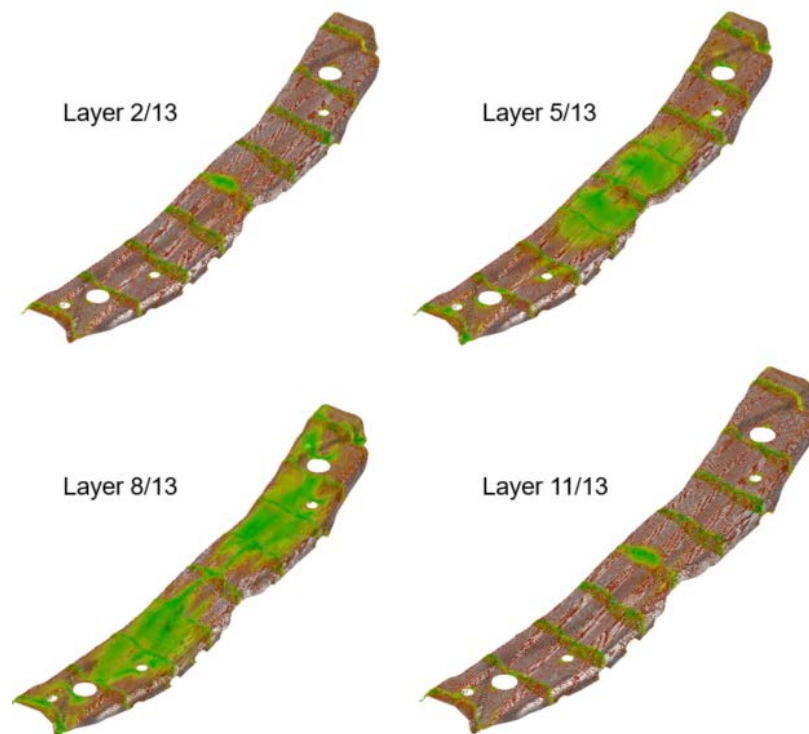
- **Step 3 : Get Fiber Orientation Distribution**

The process targeted here is injection molding. At this stage, a molding specialist should be involved in order to provide guidelines to the structural engineer on how gates should be positioned, which diameters they should have, and how many could be used.

In our case, one gate has been positioned on top of the front end right in the middle of the top surface. This has been defined directly in our Digimat-RP solution dedicated to the structural applications using Digimat material models for chopped fiber reinforced plastics in combination to fiber orientation distribution in FEA. Once the injection setting is defined, an injection simulation is performed directly from the Digimat-RP solution. Finally, at the end of the injection simulation calculation, the fiber orientation distribution is mapped automatically onto the LS-DYNA structural mesh.

An assumption which can be made to accelerate the very first iteration is to assume a relatively constant fiber orientation with wall thickness change. Hence, if necessary to iterate on the thickness between 2 and 3mm, it will be reasonable to consider the same fiber orientation distribution and not rerun new injection simulations for all thicknesses to be tested.

Figure 8 below shows the result of fiber orientation mapping on different layers through the thickness of the plastic shell model:



*Figure 8 : fiber orientations mapped onto the LS-DYNA shell mesh on 13 layers*

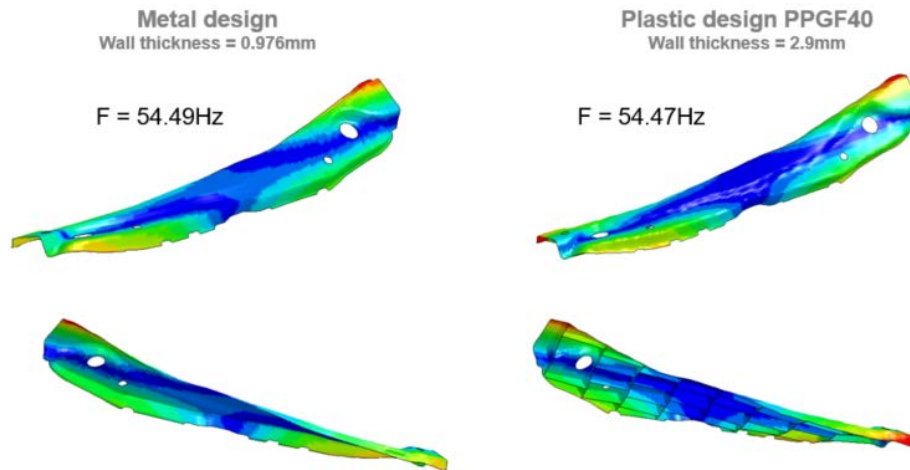
- **Step 4 : Evaluate Material Candidates at component level**

The last step of this evaluation is to run a modal analysis and a 3-point bending case on the metal and plastic designs.

The multi-scale plastic models are composed of three files to be submitted all together for final calculations :

- The LS-DYNA input deck including a User material card calling the Digimat material model
- A Digimat material file containing the multi-scale model
- A fiber orientation file containing all fiber orientation tensors on each element of the LS-DYNA model

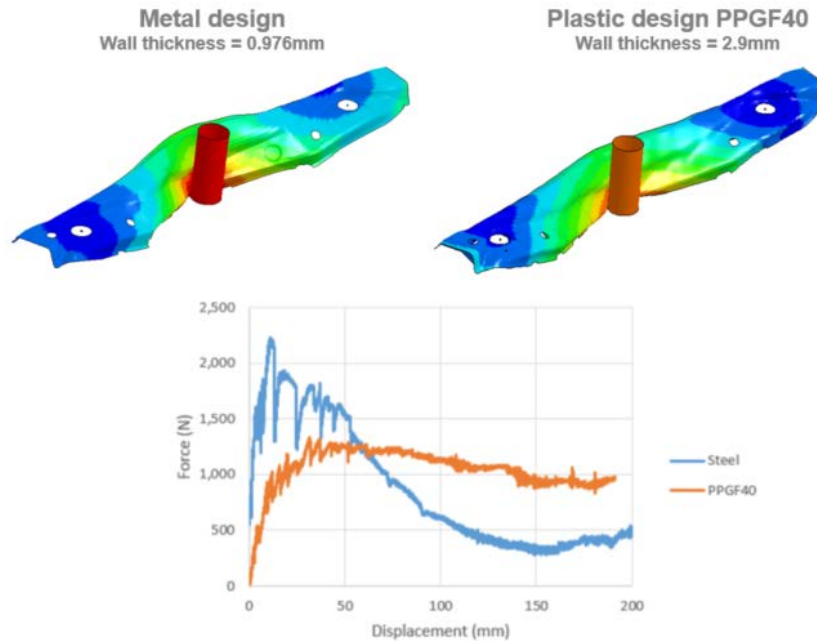
First, 2 to 3 iterations are performed on the modal analysis case in order to reach a similar first eigenfrequency as shown in Figure 9 below :



*Figure 9: with wall thickness of 2.9mm and PPGF40 material, the plastic design V0 reaches the target by yielding the same first torsional eigenmode than the metal design at the same frequency*

In a 2<sup>nd</sup> sub-step, a 3-point bending is performed on plastic design with wall thickness 2.9 mm and force displacement curves are compared between plastic and metal designs. Note: as failure was not incorporated into the steel model, it has not been considered for plastic model as well.

Figure 10 below shows the comparison of deformations and force/displacement curves.



*Figure10 : (no failure models) from the stiffness point of view, the plastic design takes a lower maximum load than metal design (1250N vs 2200N) but holds it better for higher deformations (1000N vs 300N at 150mm displacement) thanks to its better geometrical inertia.*

This last result must be considered carefully as both the steel and plastic models do not capture failure.

- **Additional step : Full Car Level**

As a very last step, the plastic front end carrier has been integrated into the full car model of the Toyota Camry and computational efficiencies have been compared between the run with metal front end and the one with the plastic front end.

Table 1 below shows a reasonable increase of 1.5% induced by the use of a multi-scale model at full car scale making the technique viable at the various required scales of application.

Total CPU Time on 10 cpus		
Full Car with PPGF40 FEC	Full Car with Steel FEC	delta
20h30min	20h11min	+1.5%

### References

[1] R. Reichert, D. Marzougui, S. Kan ; “Development & Validation of a Finite Element Model for the 2012 Toyota Camry Passenger Sedan”  
 [2] R. Reichert, D. Marzougui, S. Kan ; “2012 Toyota Camry Detailed Model v5 – Validation”