

STRUCTURAL ANALYSIS OF INJECTION MOLDED COMPONENTS USING DIGIMAT AS AN INTERFACE SOFTWARE

Eduardo Pedro
eduardo.pedro@ist.utl.pt

Instituto Superior Técnico, Universidade de Lisboa, Portugal

June 2017

Abstract

The possibility of data transfer from injection molding software, as Moldex3D, to structural software, as ABAQUS, can be a valuable asset to computationally represent the behavior of an injection molded part. Resulting from the injection molding process, part residual stress, fiber orientation or mechanical properties can be directly transmitted to the structural software. However to increase structural analysis reliability especially for composite materials such as short fiber reinforced plastics sometimes it is important to resort to an auxiliary CAE software in order to accurately predict the anisotropic properties and the non-linear material behavior of the injected component. In this way the present thesis focuses in studying the integration of the multi-scale modeling tool Digimat as an interface program for data transfer from the injection molding program Moldex3D to ABAQUS. Within this study it will be performed a creation of a material model using a “Reverse Engineering” process based on experimental data.

Keywords: Digimat, Moldex3D, ABAQUS, data transfer, CAE, Reverse Engineering

1. Introduction

Due to increase in demand the technological advance in CAE engineering over the past years allowed the possibility of software information exchange. Particularly in the case of major injection molding software available, such as Moldex3D or Moldflow, it is possible to transfer the resulting data from the injection molding process to structural analysis software like ABAQUS or ANSYS. Therefore data of part residual stress, mechanical properties or fiber orientation can be directly transmitted to the structural program given the right commands.

In the case of short fiber reinforced plastics this data transfer sometimes does not lead to a proper representation of the material behavior in a structural analysis due to the mathematical assumptions made in the injection software when simulating the process. To properly predict the anisotropic properties and non-linear behavior of the component sometimes a multi-scale software is needed such as Digimat or Helius PFA. In this study we will focus on the methodology used for data transfer between injection molding software Moldex3D and structural software ABAQUS, including Digimat as an interface tool and also a material modeler for this process.

2. Injection Process

The injection cycle of a molding process can be divided into five main steps: 1) Mold closing; 2) Filling; 3) Packing; 4) Cooling; 5) Mold opening (Ejection).

The productivity and competitiveness of any manufacturer will be improved by combining a minimum injection molding cycle time without compromising the product quality.

2.1. Material rheology inside mold cavity

During an injection molding process when the molten material is injected to the mold cavity it goes through a number of physical and chemical phenomena that depend not only from its thermal and mechanical properties but also from the mold, the cavity geometry and the parameters defined for the process. The part's final shape can greatly vary from what expected if we don't consider all of these variables during the prediction phase of the project. It is important to understand the material rheology inside the mold cavity to proper comprehend the mechanical behavior during part's cooling. The major points for this matter are related to material's crystallinity, shear stress, shear rate, viscoelastic behavior, viscosity and PvT variations.

2.2. Residual Stresses

Resulting from material rheology and part cooling we can define two major types of residual stresses:

- 1) Induced by material flow
- 2) Induced by temperature

The residual stresses induced by material flow are mainly due to the thermal shock with the cold walls of the mold and also the molecular orientation caused by

shear stress, generally representing only a minor part of the total residual stresses in a molded component.

On the other hand the residual stresses induced by temperature are what normally define part warpage during and after cooling. These changes have to do with material's PvT relations verified during part shrinkage. These factors can be connected to unbalanced cooling, the temperature gradient of the part through thickness and also the pressure profile induced during the process. With all these variations verified during the injection process we will have differential shrinkage which means each material element will tend to shrink in a different way than its neighbors resulting in residual stress formation.

2.3. Warpage

Considering the induced residual stresses verified in a part some of these will tend to be released after its ejection from the mold and cause local and global deformation. The potential causes for warpage are unbalanced cooling, asymmetric geometry, cavity thickness variation [1] and fiber orientation variation.

3. Fiber Reinforced Plastics

With fiber reinforced plastics we also need to consider the influence of fiber orientation in material flow and part warpage. Normally the main aspect that contributes to fiber reinforced plastic warpage is related not to the fiber orientation itself but the variation in orientation from fiber to fiber [2] which induce strong local volumetric shrinkage in the part and consequently creates residual stresses that can result in the referred warpage. These phenomena can mostly be seen in changes of flow direction like part corners, thickness variations, weld lines or transitions between sprue-gate or gate-part, etc. (figure 1).

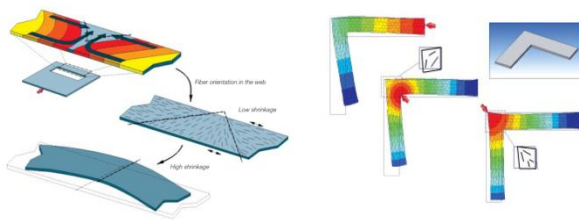


Figure 1. Influence of the change in fiber orientation to part warpage of injected components [2]

3.1. Fiber Orientation Tensor

To simplify the definition of the fiber orientation, major CAE software usually defines the second order tensor a_{ij} that represents the average distribution of the fiber orientation in a volume element.

$$a_{ij} = \begin{bmatrix} a_{11} & a_{12} & a_{13} \\ & a_{22} & a_{23} \\ & & a_{33} \end{bmatrix} \quad (1)$$

The matrix considered is symmetric and the diagonal components define the intensity of fiber orientation to the respective axis in which they are associated with. To easily obtain a representation of the fiber distribution in space it is possible to calculate the eigenvalues (λ) of the matrix.

$$\det[a_{ij} - \lambda I] = 0 \quad (2)$$

Where the solution obtained represents the length of the fiber orientation along each axis. The axis can be defined by the eigenvectors.

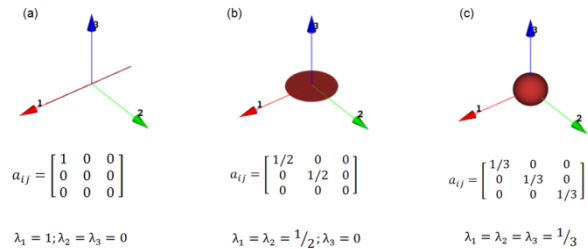


Figure 2. Fiber orientation defined in space: (a) for perfectly aligned fibers, (b) Random 2D and (c) Random 3D

4. Moldex3D Injection Data Output

As we've seen before for injection molded fiber reinforced plastics it is important to know fiber orientation as well as the residual stresses induced in a part to properly model its behavior in a structural software. In this chapter we will discuss what variables we need to export from Moldex3D to properly model our structural component.

The software Moldex3D 14.0 has available a Finite Element Interface (FEA Interface) to allow the data output of the injection molded part. In this interface there's an option available to choose the structural software being used, the element type and also the type of mesh we would like to associate the data being exported.

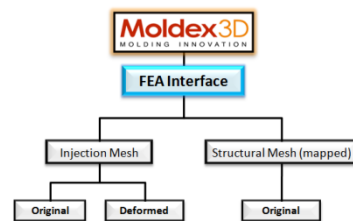


Figure 3. Mesh output options in Moldex3D 14.0 FEA Interface

For the information being exported there's an option to decide whether the data is destined directly to the structural software or firstly to a micromechanics software such as Digimat. Besides this we also have a material option to define the micromechanics model involved in the homogenization process of a composite and additionally if the material created is associated with a single element or a group of elements to reduce the computational time.

4.1. Analyzing the Outputs

Considering the outputs available and the interest data for a structural analysis of a fiber reinforced plastic the following outputs were evaluated:

“*Initial Strain Output*” – can only be exported associated with an original mesh. Elastic anisotropic properties as well as thermal expansion coefficients are assigned to a single or a group of elements of the mesh. These properties are defined based on the homogenization process chosen in the FEA Interface. Besides this a temperature difference is induced into the mesh nodes. The search for thermal equilibrium in a structural software results in part warpage similar to the one obtained in the injection software.

“*Thermal stress output*” – contains the information of part’s final residual stresses induced by temperature. File destined to structural software.

“*Residual stress output*” – contains the information of part’s final residual stresses induced by temperature. File destined to micromechanics software like Digimat.

“*Fiber orientation output*” – contains the information about fiber orientation.

5. Digimat

The multi-scale Digimat software displays two possible interfaces for the user:

- 1) “*Tools*”: set of software focused on material engineering (Digimat-MF and Digimat-FE) and structural engineering (Digimat-MX, Digimat-MAP and Digimat-CAE).
- 2) “*Solutions*”: using the technology present in Digimat Tools this methodology simplifies the sequence of tasks involved in some specific engineering problems, with the help of intuitive and user friendly interfaces (Digimat-RP, Digimat-HC and Digimat-VA).

Only the Digimat tools will be considered in this study.

5.1. RVE

To be able to understand how Digimat works it is important to understand the concept of the representative volume element (RVE). In a structural analysis a RVE is defined as a volume element of the structural mesh and allows the constant interoperability between micro and macro scale [3].

Micro – Macro Transition:

- The RVE contains a finite number of inclusions.
- Constitutive laws are necessary for each phase of the composite material.

- The macro properties of the RVE are determined by a micromechanics homogenization model.
- The mechanical continuity is possible through a constitutive equation calculated for the composite based on its phases.

Macro – Micro Transition:

- At each increment of time the behavior of the constitutive materials are evaluated at the micro scale during a structural analysis

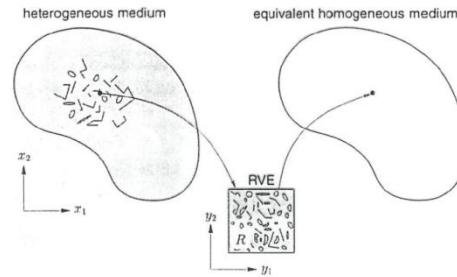


Figure 4. RVE’s multi-scale representation [3]

5.2. Elasto-plastic Model

Material creation is done in Digimat-MF based on each composite’s phase constitutive laws and properties. For short fiber reinforced plastics generally the fiber is considered as an elastic material and the matrix as elasto-plastic. The constitutive model for the plastic behavior of the matrix phase considered in the present thesis was the J_2 model.

The J_2 plasticity model of Digimat is equivalent to [3],

$$\sigma_{eq} = \sqrt{J_2(\sigma)} \quad (3)$$

$$J_2(\sigma) = \frac{1}{2}[(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2] + 3[\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2] \quad (4)$$

It is assumed linear elastic behavior until the equivalent von Mises stress (σ_{eq}) reaches the yield stress (σ_y).

$$\sigma_{eq} \leq \sigma_y \quad (5)$$

As soon as this limit is reached the hardening of the reference curve is governed by a plasticity law $R(p)$.

$$\sigma_{eq} = \sigma_y + R(p) \quad (6)$$

For the plasticity law there are three possibilities [3],

-Power law:

$$R(p) = kp^m \quad (7)$$

-Exponential law:

$$R(p) = R_\infty[1 - \exp(-mp)] \quad (8)$$

-Linear and exponential law:

$$R(p) = kp + R_\infty[1 - \exp(-mp)] \quad (9)$$

5.3. Homogenization Theory

For the homogenization process Digimat uses semi-analytic models based on Eshelby's single inclusion theory and it is assumed that the strain E and stress Σ verified at a macro level are equivalent to the volume averages of strain $\langle \epsilon \rangle$ and stress $\langle \sigma \rangle$ at the micro level respectively of the RVE's constitutive phases [3].

$$E = \langle \epsilon \rangle_{\text{RVE}} = \frac{1}{V} \int_w \epsilon dV, \quad \Sigma = \langle \sigma \rangle_{\text{RVE}} = \frac{1}{V} \int_w \sigma dV \quad (10)$$

Also for two phase composites such as short fiber reinforced plastics it is assumed a pseudo-grain method to obtain the macro properties of the material element. In this way the anisotropic distribution of a finite number of fibers present in a RVE defined by an orientation tensor can be replaced by an RVE divided into pseudo-grains each one with a well defined fiber orientation. This solution makes possible the application of a semi-analytic model based on Eshelby's theory, such as Mori-Tanaka [4] or Double Inclusion to each grain. After the homogenization performed to all grains a simpler model (Voigt model) is applied to all pseudo-grains resulting in a full homogenized RVE (figure 5).

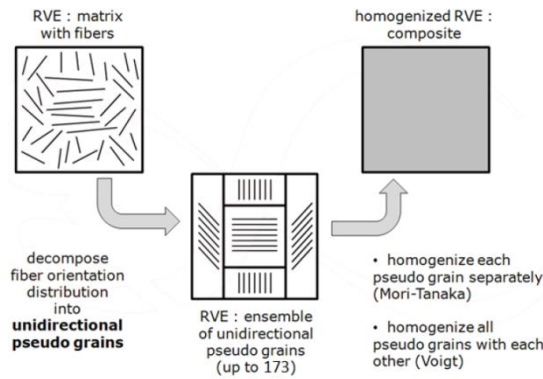


Figure 5. Homogenization process by pseudo-grains [3]

5.4. Coupled Analysis

For non-linear material models Digimat presents two possible solutions to compute variables for a coupled analysis with the structural software. The first is the "MICRO" solution where the material properties are computed along with the structural software for each analysis iteration in order to refresh the tangent stiffness matrix of the composite material. The second is the "HYBRID" solution where material properties are pre-computed by Digimat and communicated to the structural software at each iteration. This last solution is not as precise as the first one but it is significantly faster.

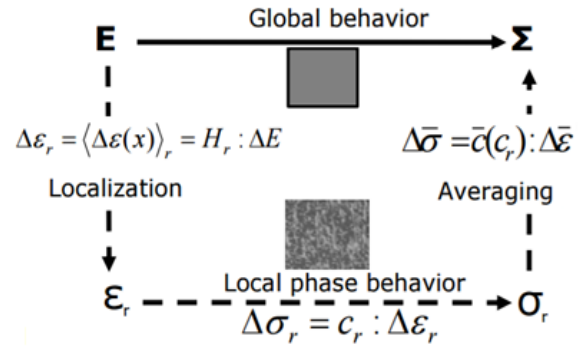


Figure 6. Multi-scalar transition during a coupled analysis [5]

Looking at the computations involved in the analysis (figure 6), at each point we know the macro strain E and we need to obtain the macro stress Σ based on the average micro strains and stresses of the composite phases.

5.5. Failure Indicator During a Coupled Analysis

Digimat gives the option to apply a failure indicator to the pseudo-grain scale level. This method was specifically developed for short fiber reinforced polymer composites and is a progressive failure model that allows the total stiffness of a RVE to gradually decrease [3]. Here the RVE is divided in a finite number of pseudo-grains each one with a weight factor, the same as the homogenization process.

The failure indicator used in the present study will be the Tsai-Hill 3D Transversely Isotropic strain criterion defined by,

$$f_i = \sqrt{F_i(\epsilon)} \quad (11)$$

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

In this criterion it is assumed that the material is isotropic along the transversal plane 2-3 where f_i represents the failure indicator defined by the square root of the failure function $F_i(\epsilon)$. Parameters X, Y and S represent the maximum values of the axial tensile strain, the in-plane tensile strain and transverse shear strain respectively [3]. Before an implicit structural analysis it is possible to define the limit value of the failure indicator and select the option to stop the analysis as soon as the limit value for the failure indicator is achieved. After the analysis in the solution dependant variables (SDV) it is possible to analyze the failure indicator by the critical mean value of failure (PCG) or alternatively by the critical fraction of the pseudo-grain (PGA).

5.6. “Reverse Engineering”

To account the experimental data performed by tensile tests, when modeling the material model, Digimat-MX has available a reverse engineering tool. This tool allows making an adjustment to the mechanical properties of each phase of the composite based on the curves obtained experimentally. The method used for non-viscoelastic models and non-temperature dependant models is the “Direct Tensile” approach [3]. In this method the approximation of curves is made based on relative error. The reverse engineering process can also be applied to failure parameters.

6. Mapping Suitability for Linear Elements

In order to perform a structural analysis with minimum loss of information when transferring data from the injection molding software we need to know if the data transfer is being made correctly. To account for this matter an analysis to the Digimat’s element mapping methods was performed where residual stress mapped data was analyzed. A Matlab program was created in order to compute the difference between the volumes of each mesh for the same value of information mapped.

6.1. Mapping in Digimat

In the Digimat mapping tool (Digimat-MAP) the methods for mapping element based variables such as fiber orientation or residual stresses are [3]:

- 1) From Integration Point to Node of the donor mesh transferred to the Integration Point of the receiving mesh (**IP/N→IP**)
- 2) From Integration Point to Node of the donor mesh transferred to the Node and then Integration Point of the receiving mesh (**IP/N→N/IP**)
- 3) From the Element of the donor mesh transferred to the Integration Point of the receiving mesh (**E→IP**)
- 4) From Element of the donor mesh transferred to the Node and then to the Integration Point of the receiving mesh (**E →N/IP**)

6.2. Mapping Analysis

The geometry considered for this analysis is shown in figure 7.

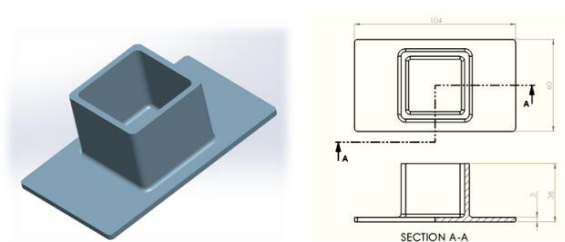


Figure 7. Geometry used in mapping analysis

Also four different meshes were applied to the geometry all with linear elements (figure 8). Two injection meshes and two structural meshes. The information of the four meshes considered can be seen in table 1.

One of the injection meshes is a BLM (Boundary Layer Mesh) composed by prism (C3D6) and tetrahedral (C3D4) elements. The other three meshes are all C3D4 meshes. One of structural mesh has similar characteristics to the injection meshes and the other has less number of elements with the objective to analyze the mapping tool in non-refined receiving meshes.

Mesh Information				
Meshes	Injection		Structural	
	BLM	C3D4	C3D4	C3D4
Elem. Size (Global)	1.2 mm	1.2 mm	1.2 mm	3 mm
Elem. per thickness	7	3	3	1~2
Total Elem.	171 336	181 080	171 734	15 394

Table 1. Mesh information for mapping analysis

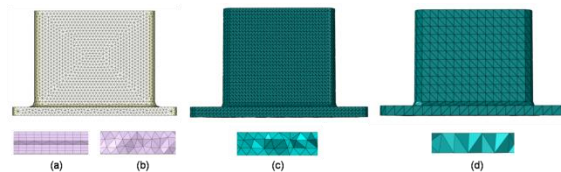


Figure 8. Meshes: (a) injection BLM, (b) injection C3D4, (c) structural C3D4 similar and (d) structural C3D4 non-refined

-Mapping Tetra Injection mesh to Structural meshes:

The data will be analyzed only for the Y direction. For other directions the same conclusions apply and can be seen in thesis attachments.

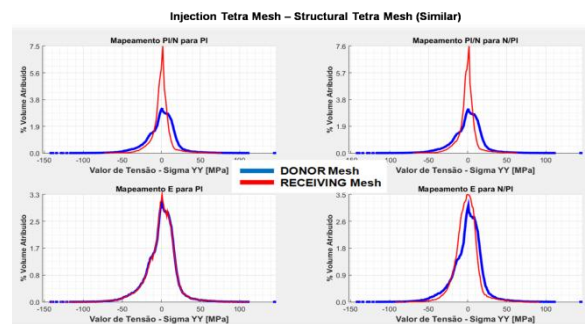


Figure 9. Mapping analysis performed between similar C3D4 injection and structural meshes

We can see in figure 9 that for similar meshes, in the E-PI mapping method the receiving mesh curve (in red) follows accurately the donor mesh curve (in blue).

Average Absolute Relative Error – Similar Mesh				
Method	IP/N→IP	IP/N→N/IP	E→IP	E/N→IP
e_{average}	78.4 %	81.7 %	24.5 %	63.6 %

Table 2. Absolute relative error of the four mapping methods for C3D4 similar meshes

Looking now at the non-refined mesh mapped results in figure 10 and table 3.

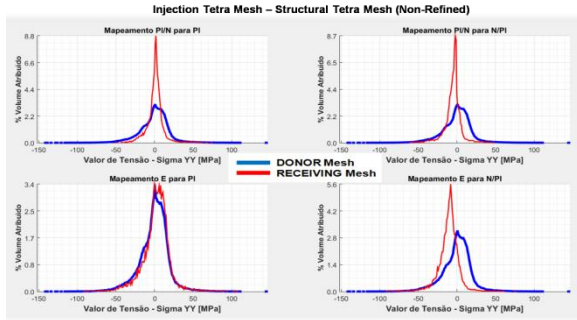


Figure 10. Mapping analysis performed between C3D4 injection mesh and C3D4 non-refined mesh

Average Absolute Relative Error – Non-refined Mesh				
Method	IP/N→IP	IP/N→N/IP	E→IP	E/N→IP
e_{average}	80.2 %	85.2 %	70.7 %	85.4 %

Table 3. Absolute relative error of the four mapping methods for C3D4 injection mesh and C3D4 non-refined structural mesh

The results are slightly worse. In fact the average relative error values show us that particular conclusion for the E-IP method. This can be better seen if we plot the non-absolute error values between the donor and mapped volumes. If the error is negative it means the mapped information is associated with a bigger volume ($V_{Mapped} > V_{Donor}$). If the error is positive or equal to 1 it means that the mapped information is associated with a lower or equal volume respectively to the donor one. The equation used for plotting this error is equation 13. Where i represents the value of information being mapped (stress or fiber orientation).

$$e_i = \frac{V_{Donor} - i - V_{Mapped} - i}{V_{Donor} - i} \quad (13)$$

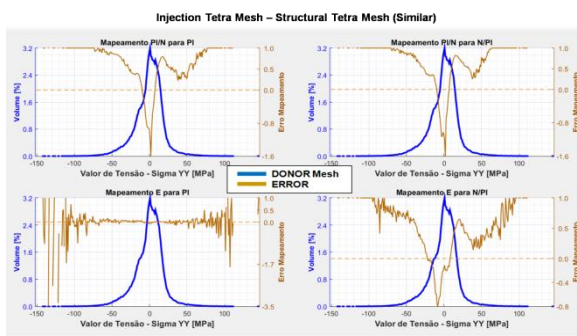


Figure 1. Relative error between C3D4 injection mesh and C3D4 non-refined mesh

In figure 13 we can see that the cause of error increase in the E-IP method occurs for high absolute values of residual stress. For the other methods especially the IP/N-IP and IP/N-N/IP method, after a certain absolute value of stress, mapping does not occur (relative error equal to 1) indicating a softening in the mapped residual stresses.

-Mapping BLM Injection to Structural meshes:

For the BLM donor mesh the same conclusions can be taken into account for similar meshes knowing that with the increase of elements through thickness gives us less mapping error especially in IP/N-IP, IP/N-N/IP and E/N-IP methods.

Average Absolute Relative Error – Similar Mesh				
Method	IP/N→IP	IP/N→N/IP	E→IP	E/N→IP
e_{average}	52.1 %	63.5 %	24.4 %	58.2 %

Table 4. Absolute relative error of the four mapping methods for BLM injection mesh and C3D4 similar structural mesh

For a non-refined mesh we increase the mapping error.

Average Absolute Relative Error – Non-refined Mesh				
Method	IP/N→IP	IP/N→N/IP	E→IP	E/N→IP
e_{average}	70.3 %	95.7 %	63.6 %	85.5 %

Table 5. Absolute relative error of the four mapping methods for BLM injection mesh and C3D4 non-refined structural mesh

-Considerations about the mapping methods:

After all the analysis made some conclusions can be taken in account relative to this matter:

-For mapping between similar meshes the E-IP method is revealed to better represent the data coming from the injection mesh. However for non-refined structural meshes this method can lead to some errors because we have elements with low volume in the donor mesh. This means the information can be amplified in the receiving mesh due to the bigger structural mesh volume.

-For IP/N-IP, IP/N-N/IP and E/N-IP donor mesh zones that have information associated with a low volume are smoothed with the contribution of the information present in of neighbor elements. Before going to the receiving mesh element the information first as to go to the nodes which means the information will also come from elements that share the same node. This phenomena is amplified in the IP/N-N/IP where the softening occurs both in the donor and receiving meshes therefore softening low volume information zones in the mesh.

Seen as the fiber orientation data is also associated with mesh elements the same conclusions can be taken for the different mapping methods.

7. Data Transfer Methodology using Digimat

Considering the need to use Digimat as an interface and material modeler software for the data transfer between the injection and structural programs, the process can be divided into five distinctive phases.

Phase 1: Creation of injection and structural meshes based on part geometry created in CAD software. Export of injection process simulation results. The data to output is:

- “Initial strain output” on the structural mapped mesh.
- “Residual stress output” or “Thermal stress output”
- “Fiber orientation output”

Phase 2: In Digimat-MAP map the fiber orientation and residual stress data to the structural mesh using the original geometries.

Phase 3: Associate the mapped fiber orientation file with the Digimat material model that will be used for the analysis. This can be done through Digimat-CAE or via ABAQUS’s Digimat plug-in.

Phase 4: In ABAQUS execute an initial step (*Step-0*) to the “Initial strain” file exported onto the structural mesh. This will allow to obtain the deformed part similar to the one obtained in the injection simulation. Export the deformed mesh file (*.inp).

Phase 5: Associate the residual stress file to the deformed structural mesh (this can be done directly by modifying the “*.inp” file) and also the material model containing the fiber orientation information.

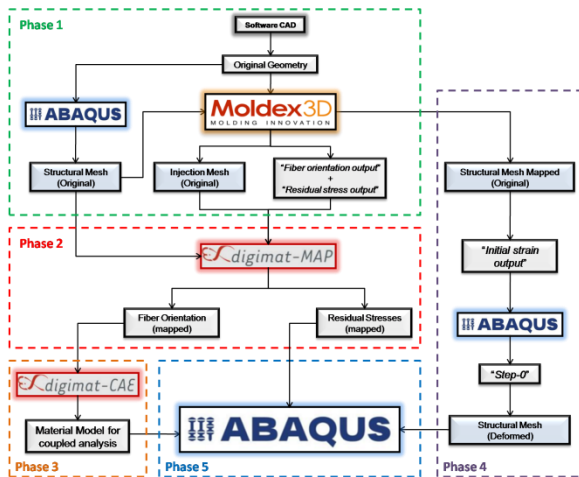


Figure 2. Data transfer between software using the “Initial Strain” option to obtain the deformed structural mesh

After this we only need to apply the correct loading and boundary condition to the part to perform the structural analysis.

7.1. Step-0

When releasing the initial strain energy from the part there is the need to apply boundary conditions to prevent rigid body motion. However this sometimes can lead to unnatural deformation comparing to the deformation obtained in the injection simulation process. Another way to look at the problem is to apply a damping energy to the part in order to stabilize it and prevent the rigid motion. In ABAQUS this is done by adding viscous forces to the global equilibrium equation.

$$F_v = c * M * v \quad (14)$$

Where M represents the artificial mass matrix with unit density and $v = \Delta u / \Delta t$ the nodal velocity vector, being Δt the time increment defined in the step. The variable c represents the damping coefficient and controls the intensity of the viscous forces to be applied [6]. To see the influence of the stabilization energy used by the viscous forces we can look at the total energy of the system (ETOTAL), the internal energy of deformation (ALLIE) and the energy used to stabilize the part (ALLSD) that are computed at each time increment during the computational step.

$$ETOTAL = ALLIE + ALLSD \quad (15)$$

For the “initial strain” release we want the minimum damping influence possible in the system to not compromise part’s natural deformation. Considering the part used in chapter 6 and looking at two different values for the damping coefficient when performing a “Step-0” we have,

ALLSD contribution to ETOTAL		
$c [s^{-1}]$	$c = 1e^{-10}$	$c = 1e^{-2}$
ALLSD [%]	1.34e-7	11.37

Table 6. Stabilization energy (ALLSD) contribution to the total energy of the part (ETOTAL)

With the results for a damping of $1e-2$ we can see the bigger influence of the ALLSD in ETOTAL. Increasing the damping coefficient we have,

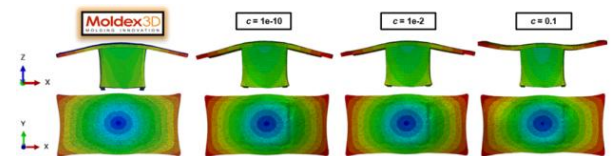


Figure 3. Influence of the damping coefficient in *Step-0*

Notice that for a damping of 0.1 it was used too much stabilization energy for the system compromising part’s natural deformation. In the extreme case of a much higher damping value the part wouldn’t be able to deform. Therefore the damping coefficient has to be big enough to prevent the rigid body motion of the

part and small enough to not compromise part's natural deformation when searching for equilibrium. The value used for the damping coefficient in the case study of chapter 8 was $1e-10 s^{-1}$.

8. Case Study

For this chapter we will consider the modeling of a non-linear Digimat material to perform a structural analysis in ABAQUS. Experimental results will be included in a reverse engineering process to improve simulation results. Two types of analysis will be conducted:

- 1) Specimens with original shape with no influence of injection molding phenomena
- 2) Deformed specimens with thermal residual stress resulting from the injection molding process.

To correctly predict a composite's material behavior it is important to have information about its anisotropic data. Therefore several tensile tests were conducted for specimens in three different directions (0° , 45° and 90°). The specimens considered refer to the ones present in ISO 527-2 Standards [7] (type 1BA). They are obtained from an injection molded plate (figure 15).

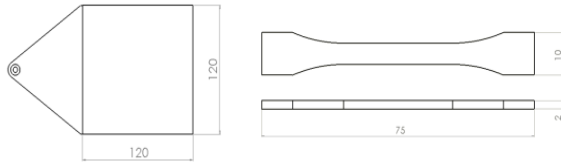


Figure 4. Geometries considered for injection and structural analysis

Computationally the analogy to obtain the specimens is made by simulating the injection process of the plate and mapping the results to a structural mesh respectively to the specimen. The information of both meshes can be seen in table 7.

Mesh information		
Mesh	Injection BLM	Structural C3D4
Global Elem. Size	1 mm	1~1.2 mm
Elem. per thickness	11	2~3
Total Elem.	390 087	9 616

Table 7. Mesh information for injection and structural analysis

8.1. Experimental Data

The material tested is a short fiber reinforced plastic, namely a PA46 GF50 (TW241F10) and the strain rates used for testing were: 1mm/min, 50mm/min and 500mm/min.

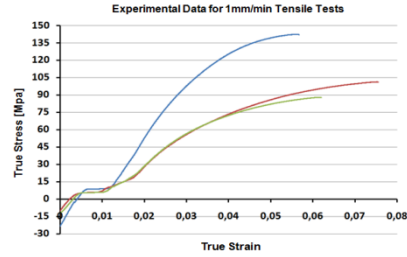


Figure 5. Average of the experimental curves from the tensile test performed (1mm/min)

In this text we will only consider the 1mm/min tests (figure 16). It can be seen that for the initial phase the material shows a strange behavior. This initial profile is assumed to be related to two different phenomena. The first, related to the release of the compression force applied by the grips of the tensile machine. And the second, to the gradual release of material's residual stresses resulting from the injection molding phenomena (horizontal profile). A more detailed explanation to this can be seen in the full text.

Introducing these experimental curves to Digimat without any sort of treatment it would make almost impossible the reverse engineering process due to the constitutive laws present in the software for material modeling. Therefore we can only consider experimental data without any influence in material's natural behavior this means without the initial part of the tests. For this a linear regression was made to the elastic behavior of the material after it was free of the referred phenomena, in order to replace the non-considered data points of the curves. After data treatment the tests performed at 1mm/min were chosen for the reverse engineering process in Digimat and can be seen in figure 17 and table 8.

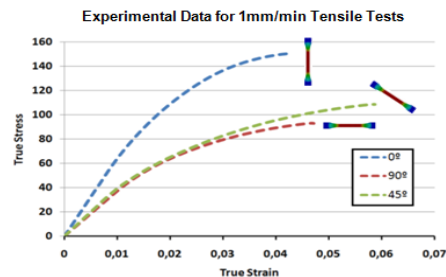


Figure 6. Experimental data after treatment (1mm/min)

Experimental data – 1mm/min – Young Modulus			
Specimen Orientation	0°	45°	90°
E [MPa]	6482	4005	3736

Table 8. Young's Modulus for the three curves considered in figure 17

8.2. Reverse Engineering

In this process we first need to define an orientation tensor for the curves we are going to use. This means

the reference curve from the created material model in Digimat and also the experimental data curves. Since we don't have any information about fiber orientation of the specimens tested we resort to the injection software where we reproduced the injection molding of the plate. Based on the results obtained, a second order fiber tensor was created and assigned to the experimental curves for the reverse engineering process.

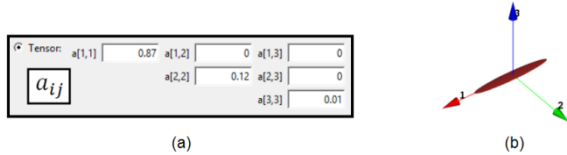


Figure 7. (a) Orientation tensor used in the reverse engineering process and (b) its representation in space

Now the analysis can be conducted where we “reverse engineer” the new mechanical properties of the matrix and/or fiber and therefore the composite (figure 19).

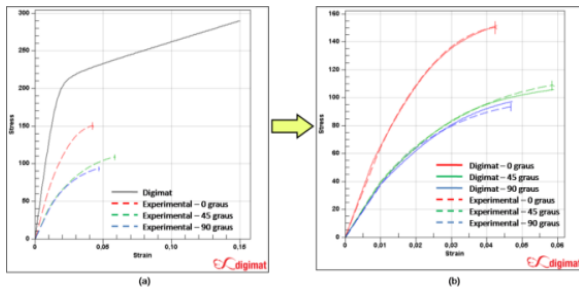


Figure 8. “Reverse Engineering” (a) before and (b) after

This process must be divided in several stages. First we need to adjust the elastic part of the curves. Second the yield stress and at last the plastic behavior. After a satisfactory adjustment the Digimat material model behavior is updated creating a new reference curve with the information of the experimental data.

The reverse engineering process can also be done for the failure indicator. This process relies on some experience in previous results of fractured components to see whether it is possible to increase or restrain the strain limit. Besides that, the approval of the reverse engineering process to the failure criterion can be supplemented by the tensile responses for specific curves defined by specific fiber orientation tensors.

After process completion the Digimat material model has now all the experimental data information. The next step is to associate the mapped fiber orientation of the structural mesh to the material created before the structural analysis. This is done in Digimat-CAE or by the Digimat plug-in included in ABAQUS.

8.3. Tensile Tests in ABAQUS

As said before the tensile tests in ABAQUS are divided in two phases. First we analyze three

specimens of different directions (0°, 45°, 90°) without any influence of the injection molding process. The results are displayed in figure 20.

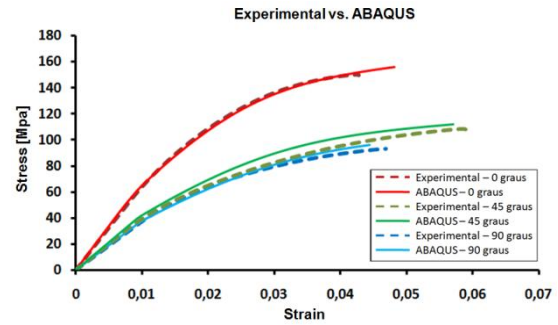


Figure 9. ABAQUS results for specimens without the influence of injection molding phenomena (original geometry and no residual stresses considered)

Analyzing all the curves profile it is possible to conclude that the fiber orientation tensor (a_{ij}) used in the reverse engineering process reproduces well the real fiber orientation of the specimens tested. From the ABAQUS analysis made we obtain a good approximation to the experimental data. The evaluation made for the fiber orientation tensor of the reversed engineering experimental data was based on injection results from Moldex3D. The definition of this tensor is extremely important to correctly construct a real approximation to the material behavior from the material model created in Digimat. To further see this importance it is shown, in figure 21 (b), results obtained in ABAQUS with a different fiber orientation tensor used in the reverse engineering process.

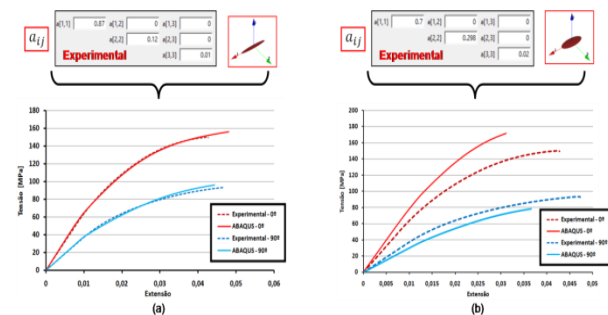


Figure 10. ABAQUS results with (a) an estimated orientation tensor from Moldex3D simulation and (b) a less aligned orientation tensor

The results shown are clearly different. Associating a less aligned orientation tensor we obtain higher stress values (+15MPa) in ABAQUS analysis for the 0° specimen and lower stress values for the 90° specimen. When associating this less aligned tensor to the experimental curves in the reverse engineering process we are assuming that, in the space between the 0° and 90° experimental curves, there are going to be less fiber orientation curves. This means we are restraining the number of possible outcome curves for that interval. If the opposite was done, i.e., a more

aligned tensor was used in reverse engineering, we would be increasing the number of Digimat curves inside the space between the 0° and 90° experimental ones. This concludes the importance of knowing beforehand the orientation tensor to use for the reverse engineering process because the mechanical properties of the material will be estimated based on that.

Considering now the specimens influenced by the injection molding phenomena (with deformation and residual stresses) we have the following results of figure 22.

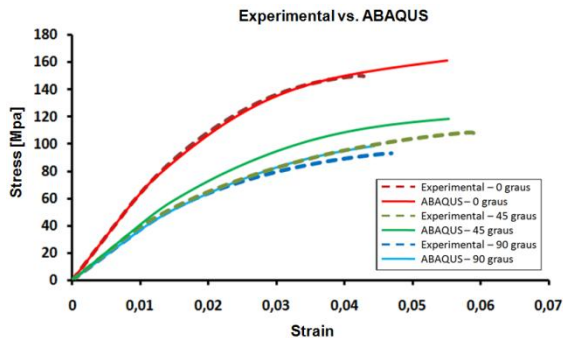


Figure 11. ABAQUS results for specimens with the influence of injection molding phenomena (deformed geometry and part residual stresses)

We can see a slight variation in results especially for the limit value for the 0° curve and the hardening increase on the 45° and 90° ones. To better understand this behavior, four elements of the mesh were analyzed in both cases considered.

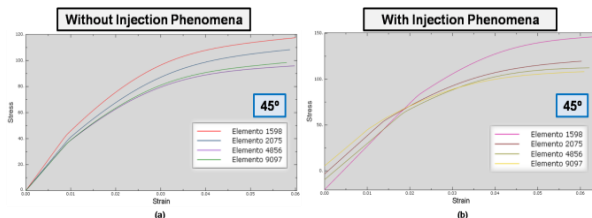


Figure 12. Behavior of the four elements considered: (a) without injection molding phenomena and (b) with injection molding phenomena.

It is possible to see in the case without the injection phenomena, figure 23 (a), all the curves start from the origin. The same doesn't happen for the case with injection phenomena (b), where they already start with a pre-defined stress value equivalent to the residual stress associated with each element. Considering all the elements in the mesh contributing to this type of variation we will have a different material behavior in the part. This influence can be low if the residual stresses present in the part are low, and high for high absolute values of residual stress.

9. Conclusions

The present work allowed to perform an extended study to the data transfer between Moldex3D and

ABAQUS using Digimat as an auxiliary software. With this study some conclusions can be made:

- The study of the mathematical assumptions and functionalities of the Digimat software for fiber reinforced plastics is important to understand the material engineering processes involved and also the multi-scalar interactions during a coupled analysis with ABAQUS.

- Considering the information exported from Moldex3D a data transfer methodology was created between the three CAE software considered.

- Associated with the “Initial Strain” deformation, the energy stabilization method used to prevent rigid body motion allows to obtain similar deformation results to the injection molding software.

- The reverse engineering process allied with the methodology proposed for data transfer between software allowed to obtain good approximations to the experimental data.

In conclusion, this work allowed to create a material and structural engineering guide for the processes involved in data transfer between the three CAE software considered. This study allows the possibility for future development of this matter and possibly a practical application to an injection molded part. Digimat is a vast software that requires knowledge and experience but shows itself as a powerful tool to solve some of the engineering problems of present day industry.

References

- [1] “Residual Stress – A Culprit in Shrinkage and Warpage Problems” [Online] - URL: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/physics/b3400001.htm - accessed in March 2017
- [2] “Warpage characteristics of fiber-reinforced injection-molded parts”, *BASF Plastics*, 2000
- [3] Digimat User’s Manual, Version 2016.0, *e-Xstream engineering*, 2016
- [4] T. Mori, K. Tanaka, “Average stress in the matrix and average elastic energy of materials with misfitting inclusions”, *Acta Metallurgica*, Vol. 21, ELSEVIER, 1973
- [5] T. Malo, et al., “Nonlinear Multi-Scale Modeling of Reinforced Plastic Parts with Digimat to RADIOSS”, *e-Xstream engineering*, 2009
- [6] ABAQUS User’s Manual, Version 6.14, Dassault Systems SIMULIA Corp., 2014