Experimental and Theoretical Study of a Short Fibber Filled Injection Molded Part

Pedro Rafael Pinto Abrantes
pedro.abrantes@ist.utl.pt

Instituto Superior Técnico, Universidade de Lisboa, Portugal

June 2017

Abstract

A short fiber filled injection molded plastic part, has properties that cannot be simulated in a normal structural analysis. This happens because the fibers are going to make the material properties very dependent of the direction of the load in relation the fibber orientation, and a normal structural analysis is not capable of simulating this dependence. Also the injection process is going to cause the part to warp and create residual stresses.

The injection simulation programs, in this case Moldflow was used, calculate the fibber orientation tensor, the warping and the residual stresses ate the end of the injection. This data is necessary to correctly represent the behavior of the material in a structural analysis.

Several methods and programs were created to transfer the data from the injection simulation to the structural. This work will use Helius PFA, which allows the mapping of the data from the injection mesh to the structural mesh.

The simplifications and criteria used by Helius PFA to mapped and characterize the material will be studied. Data from tensile tests of the material will be presented, along with the modifications necessary to introduce this data into the analysis.

After performing the structural analysis, using Abaqus, it was concluded that the material characterization criteria used by Helius PFA presents acceptable results, and that the process works and the final results are satisfactory. It´s also established that to correctly represent the material behavior, it's necessary to introduce the residual stresses into the analysis.

Keywords: Injection molding, Autodesk Moldflow, Autodesk Helius PFA, Simulia Abaqus, Fiber orientation tensor, Residual stresses

1 Introduction

When a part is manufactured by injection molding it´s going to warp by the end of the process, and there will be residual stresses present in the final part. Also the fact that the material is reinforced with short fibers means it's anisotropic, which means its properties are very dependent of the local orientation of the fibbers, which can vary throughout the part. This makes it impossible to simulate the material behavior in a normal structural analysis. The data of the fiber orientation and the residual stresses can be obtained from an injection simulation, it's then necessary to transfer this data into the structural analysis.

In this work the method to transfer the data from one simulation to the other using Helius PFA will be studied. The main objective is to ascertain if this method produces satisfactory results. Another objective is to understand how the program works, and everything that is necessary to produce good results.

2 Background

2.1 Injection Molding

Injection molding is a cyclic process, in which molten material is injected into a mold, the material then cools down to a determined temperature before being released from the mold, after being released the part will cool down until it reaches room temperature.

The process can be divided in five stages:

1- Mold closing;
2- Injection – The molten material will be injected into de mold;
3- Packing – In this stage the pressure will be maintained in order to add more material to compensate for the shrinkage of the material as it cools;
4- Cooling – The mold is cooled using water to increase the speed of cooling. This stage ends after the material reaches an established temperate, after which the part has enough stiffness to be removed from the mold;
5- Ejection – The mold opens and the part is ejected. After this a new cycle begins. [1]

The injection simulation programs focus mostly on stages 2-4. However stage 5 is very important because after being released the part is still cooling and it's going to warp, but simulating this is very difficult, at the moment no program can solve this problem using all the variables that exist in the process. [1]

2.2 Warping

Polymers have high thermal expansion coefficients, which is going to cause shrinkage during cooling. This shrinkage leads to the warpage of the part and to the existence of residual stresses. There are several factors that influence the warping and the residual stresses, such has the injection rate and the temperature of the injected material. [2]

2.3 Residual Stress

Residual stresses are induced by the injection process, and will alter the behavior of the material under load. These stresses can be flow-induced or thermal-induced. [3]

2.3.1 Flow-induced residual stress

These stresses result from the fact that when in equilibrium the polymer molecules have a random orientation. However the injection process is going to cause molecular orientation, this causes shrinking and residual stresses. The orientation of the molecules will also cause the material to have anisotropic properties. [3]

2.3.2 Thermal-induced residual stress

These stresses can occur due to several reasons. The material will shrink as the temperature drops until it reaches room temperature. The material behaves differently as it solidifies from the mold to the center. Changes in pressure, temperature, and fiber orientation result in variable material properties. The mold will influence the material shrinkage as it cools. [3]

2.3.3 In-Cavity Residual Stress

In-cavity residual stress is the accumulated stress from the process while the part is still inside the mold. After the part is released the stresses will change until it reaches equilibrium and these will be the final process-induced residual stresses. [3] However the calculation of these final stresses is not easy, so Moldflow uses a simplification to calculate the residual strain. It considers that the part cools to room temperature while still inside the mold (in-cavity). [4]

3 Methods

3.1 Injection molding simulation

For this project, injection molding plaques with a dimension of 120x120x2mm were available to study. So a simulation for the injection of a plaque with this dimension was performed. The first step is to create the part using 3D drawing software, in this case SolidWorks. Along with the plaque a fan gate was drawn using SolidWorks, with the same measurements as the gate used for the real part. After creating the geometry, it can be imported into Moldflow, then a mesh can be created first using Dual Domain elements so that It can be repaired until it satisfies all quality requirements, it's then transformed into a mesh of 3D elements. Figure 1 shows the generated mesh for the part.

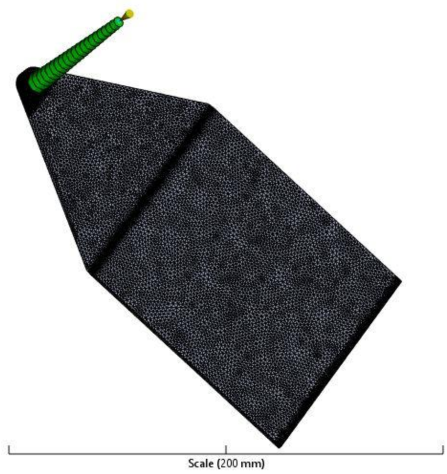

Figure 1: 3D mesh with 302927 elements

The material used was DSM Stanyl TW241F10. The process settings used for the injection of the real plaque were not known, so a molding window analysis was performed, and the recommended values were used.

Table 1: Recommended process settings

Process settings	Values
Mold Temperature	144.62 ^{o}C
Melt Temperature	316.67^{o}C
Injection time	1.0013s

3.1.1 Fill+Pack+Warp

Two Fill+Pack+Warp analysis were performed, one using all elements for the warp calculation, and another one that does not use the elements from the gate in its warp calculation. This leads to different results, especially in the warp in z. Figure 2 presents the results of the z component of the warp for both analyses.

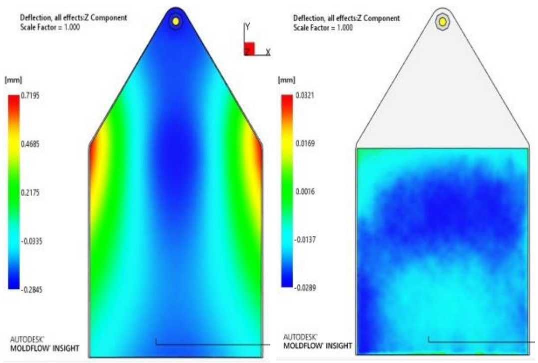

Figure 2: Warp, z component. Calculated using all elements (Left), not using the elements of the gate (right)

3.1.2 Comparison with real plaque

The real plaques were measured using a laser measuring machine, which provides a stl file with the coordinates of the points of the surface. So a MatLab program was created to read the file and generate an image of the surface, it also calculates the displacement in z. This was used to compare with the values obtained in the injection simulation. Figure 3 shows a plaque and its respective displacement in z.

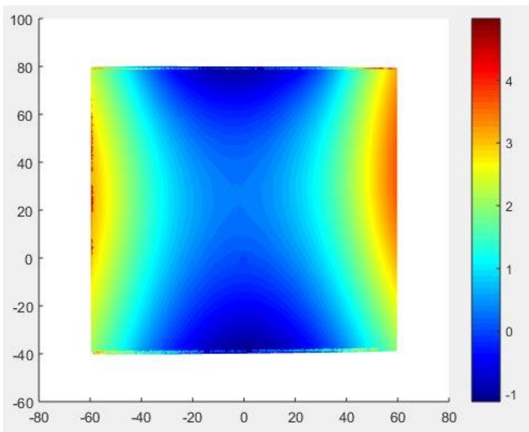

Figure 3: Displacement in z for the real plaque

Three plaques were measured, but the results are similar for all three, so only one is presented here.

The values from Figure 3 are quite different from the ones obtained in the injection simulation, being more similar with the results calculated using the elements of the gate to predict warpage. The difference appears to be caused by the process of removing the gate in real life, however there was no information as to how the gate was removed, so that was not simulated, this leads to differences between reality and simulation. Also since the process settings used were the ones from the Molding Window and not the ones used to create the real parts, this will lead to bigger differences.

3.2 Helius PFA

Helius PFA (Progressive Failure Analysis) is a program, developed by Autodesk that enhances structural analysis. Advanced Material Exchange (AME) is one of the tools of Helius PFA, that's capable of mapping data from an injection mesh into a structural mesh. [4, 5]

When short fibber filled plastic materials are used AME uses the following simplifications,

- The short reinforcing fibbers do not exhibit any plasticity or rupture, rather the fibbers exhibit a simple linear elastic response;
- The plastic matrix constituent exhibits both plasticity and rupture;
- The idealized model´s matrix plasticity and matrix rupture are intended to account for any fibber/matrix debonding that occurs in the real material;
- All nonlinearity exhibited by the composite material is due to nonlinearity (plasticity and rupture) in the plastic matrix material;
- Plasticity and rupture of the plastic matrix constituent are driven by stress in the plastic matrix constituent, as opposed to being driven by the homogenized stress in the composite material;
- The plasticity and rupture responses of the plastic matrix constituent are strongly dependent on the degree of alignedess of the reinforcing fibbers;
- As the degree of fibber alignedness increases, the plasticity and rupture responses of the plastic matrix constituent become strongly dependent on the direction of loading relative to the average direction of the reinforcing fibbers. [4]

3.2.1 AME Stages

The first step in using AME is to import the injection file and the structural file. When the structural file is important, the units must be specified. After both geometries are imported the program will verify if they are aligned, if not it allows the user to align them using translation and rotation of the structural mesh. [4, 5]

The program has two material models, Linear Elastic and Elastic-Plastic. Linear Elastic only considers a linear material response. Elastic-Plastic allows for a nonlinear material response. [6]

If the Elastic-Plastic model is selected, it's necessary to provide stress-strain data for the material. AME requires three stress-strain curves for three different orientations, 0º, 45º and 90º. The data must be in order of Stress, Strain, Angle, Temperature, Relative Humidity, and Strain Rate. At least 15 points per curve must be provided, with at least one of those points in the elastic range of the material. [6]

Since the injection mesh and the structural mesh are different, it's possible that there will be problems when mapping the data if the meshes are too different. The tool Mapping Suitability Plot will show if there are any areas of the structural mesh that require further refinement. [6]

Having imported all necessary data, it can be mapped into the structural mesh. After the mapping, the results can be exported so they can be used to perform the structural analysis. When exporting the data it's necessary to decide if the residual strains will be exported or not. It's possible to perform the structural analysis using only the fibber orientation. [6]

3.2.2 Material Model

3.2.2.1 Homogenization

In order to simulate the influence of the loads on the material, the program can't apply the load increment on the matrix and the fibber separately, so it needs create a homogenized material that behaves as the composite. To do this the program uses the properties of the matrix and the fibber, and introduces them into the incremental Mori-Tanaka Micromechanical Model, this model is going to create an idealized material with the homogenized properties for a perfectly aligned material, the fibber orientation tensor is then introduced to alter this idealized material, resulting in homogenized properties of the real composite.

The prediction of plasticity and rupture is only made for the matrix, so the program has to decompose the material into its two different constituents, the matrix and the fibber.

This means the program is going to create a homogenized material, then applies the strain increment and calculates the deformation based on the stiffness of the homogenized material, and then it will decompose the strain into the average strain in the matrix, in order to predict the plasticity and rupture. [4]

3.2.2.2 Mori-Tanaka

The program uses the micromechanical Mori-Tanaka model for the homogenization process. This model for a high volume of fibbers considers that each fibber is surrounded by a composite and not the matrix. To solve this problem an equivalent isotropic composite is used to create an environment surrounding the fibber, this new environment has a fibber volume fraction of f' between 0 and 1, in which 1 is the fibber volume fraction of the material. The solution for the model is given by,

$$C = C_f + f\left(C_f - C_m\right)A \quad (1)$$

Where C is the stiffness tensor of the composite, C_f and C_m are the stiffness tensor of the fibber and of the matrix, respectively. A represents the strain concentration tensor, given by,

$$A = T[(1-f)I + fT]^{-1} \quad (2)$$

With,

$$T = [I + S_{eq}C_{eq}^{-1}\left(C_f - C_{eq}\right)]^{-1} \quad (3)$$

Where C_{eq} and S_{eq}, are the Stiffness and Eshelby's tensor of the new composite, respectively. [7]

Plasticity Model

The response of the matrix constituent material is calculated using a plasticity model based on the Ramberg-Osgod model. This model was enhanced to take into account the direction of the loading relative to the fibber direction. The effective yield strength of the matrix, σ_y^h, is given by,

$$\sigma_y^h(\varepsilon_{eff}^p) = E^{\frac{1}{n}}(\sigma_0)^{\frac{n-1}{n}}\varphi^{1/n} \quad (4)$$

σ_0 and n are the parameters used in the original Ramberg-Osgood model (isotropic), φ is the effective plastic strain in the matrix. Yielding will occur when the effective yield strength is equal to the effective stress, σ_{eff}, which means,

$$\sigma_y^h(\varphi) = \sigma_{eff}(\varphi) \quad (5)$$

$$\sigma_{eff} = \sqrt{\begin{matrix}((\sigma_{11}-\sigma_{22})^2 + (\sigma_{22}-\sigma_{33})^2 \\ +(\sigma_{33}-\sigma_{11})^2 \\ +6[(\sigma_{12})^2 + (\sigma_{23})^2 + (\sigma_{31})^2])/2\end{matrix}} \quad (6)$$

The stress components represent the average stress in the matrix constituent.

The determination of the plastic evolution during an imposed total strain increment is reduced to solving equation 5 iteratively for φ, using equations 7 and 8.

$$E_{tan} = \frac{EH}{E+H} \quad (7)$$

$$H = \frac{E^{1/n}\sigma_0^{(n-1)/n}\varphi^{(1-n)/n}}{E+H} \quad (8)$$

The model that was described to this point only works for isotropic material, in order to take into account the direction of the loading relative to the fibber direction, the equation for the effective stress is changed to,

$$\sigma_{eff} = \sqrt{\begin{matrix}((\alpha\sigma_{11}-\beta\sigma_{22})^2 + (\beta\sigma_{22}-\beta\sigma_{33})^2 \\ +(\beta\sigma_{33}-\alpha\sigma_{11})^2 \\ +6[(\sigma_{12})^2 + (\sigma_{23})^2 + (\sigma_{31})^2])/2\end{matrix}} \quad (9)$$

α and β are weighting coefficients used to determine the directional dependence.

Since the fibber orientation varies throughout the part, α and β can't be constants. They are a function of the degree of fibber alignedness, which is quantified by the largest eigenvalue of the fibber orientation tensor (λ_i).

$$\alpha(\lambda_i) = \theta + \left(\frac{(\alpha_m-\theta)}{\left(\lambda_{m,I}-\frac{1}{2}\right)}\right)(\lambda_i - \tfrac{1}{2}) \quad (10)$$

$$\beta(\lambda_i) = \theta + \left(\frac{(\beta_m-\theta)}{\left(\lambda_{m,I}-\frac{1}{2}\right)}\right)(\lambda_i - \tfrac{1}{2}) \quad (11)$$

α_m, β_m and$\lambda_{m,I}$, are the values for a strongly aligned material. θ is the value α and β will reach when the fibber orientation is random. In this model θ=1, which means that when the fibber orientation is random in two dimensions (thickness is not considered because the part is thin walled), and λ_i = 1/2, α and β have the value of 1, turning the model back to its original form. [4, 5]

Helius PFA uses 9 coefficients to characterize the material, Young's Modulus and Poisson ratio for the fibber and the matrix, characterize the elastic behaviour of the material. For the Plastic behaviour the program uses σ_0, n, α and β. The last coefficient is effective stress, S_{eff}, that causes the matrix to rupture. [8]

Rupture Model

The rupture criteria used is the Maximum Effective Stress rupture model, where it is assumed that the weighted Von Mises stress calculated before is enough to define the directional dependency of the material. So this criteria just establishes a maximum value for the effective stress (S_{eff}). [5]

$$S_{eff} \leq \sqrt{\begin{matrix}((\alpha\sigma_{11}-\beta\sigma_{22})^2 + (\beta\sigma_{22}-\beta\sigma_{33})^2 \\ +(\beta\sigma_{33}-\alpha\sigma_{11})^2 \\ +6[(\sigma_{12})^2 + (\sigma_{23})^2 + (\sigma_{31})^2])/2\end{matrix}} \quad (12)$$

3.3 Tensile Tests

As was mentioned before, AME requires the introduction of three stress-strain curves in order to use the elastic-plastic model.

In order to perform the tensile tests, tensile test pieces were cut from the plaques using milling.

The tensile tests were performed at three different test speeds [9], 1mm/min, 50mm/min and 500mm/min.

The tensile test pieces used were very small, type 1BA [10], so there were no strain gauges available that would work in these test pieces. So the calculation of the true stress and true strain was performed using an algorithm developed by prof. Jorge Rodrigues.

Figure 4 shows the results for obtained for 0º,

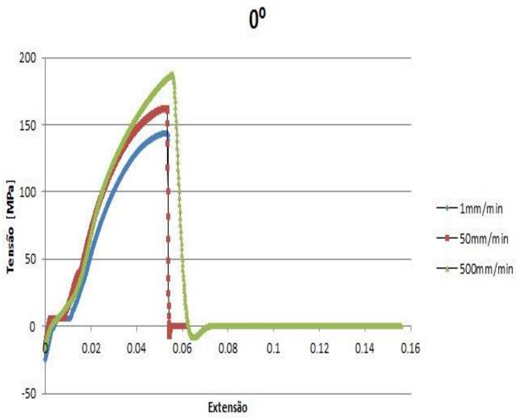

Figure 4: Results of the tensile test for 0º

There are areas of the curves that don't represent the normal elastic-plastic behaviour of the material, because their cause is not related to the tensile test. So the data will have to be modified before it's introduced into AME.

The part to the right where the stress is zero exists because for high speeds the machine has to be stopped manually, so the removal of this area will not alter anything important.

The areas to left that don´t belong in a stress-strain curve are probably caused initially by the pre-stress applied when the test piece is placed in the machine, and the rest is the result of the destruction of the residual stresses.

Figure 5 shows the curves for 0º after the modifications made to the data,

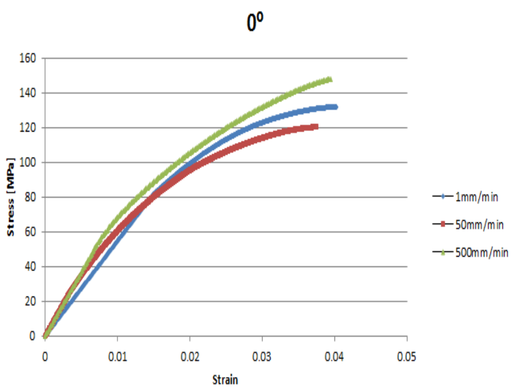

Figure 5: Stress-Strain curves for 0º after the modification of the data

However there are still too many points in each curve, because AME prefers a maximum of 50 points per curve, a larger number will increase the computation time, and might lead to errors.

In Figure 6 shows the curves for 0º with the reduced number of points,

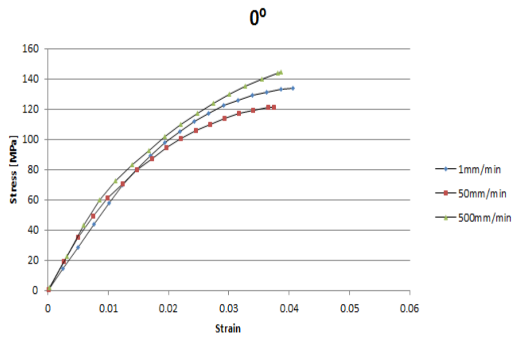

Figure 6: Stress-Strain curves for 0º with a smaller number of points

3.4 Structural Simulation

To use AME it's necessary to create a structural file that contains the structural mesh, all the steps, boundary conditions and loads of the simulation

The mesh is very important because the variation in mesh refinement will lead to a variation in the ultimate stress [6]. Figure 7 shows this correlation between mesh refinement and ultimate stress,

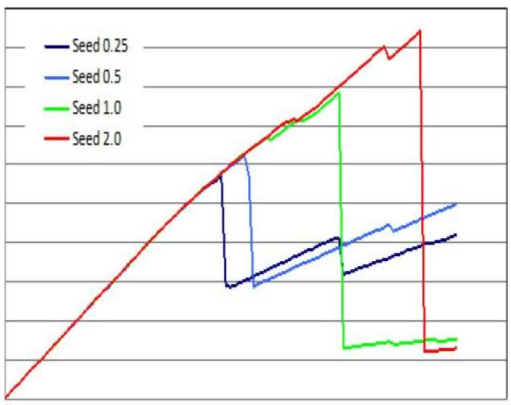

Figure 7: Ultimate stress for different size meshes. [6]

For this work a mesh was created using 10 node tetrahedral elements (C3D10), with 11676 elements. Figure 8 shows the structural mesh,

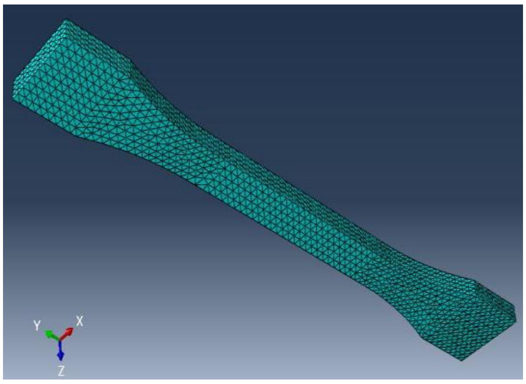

Figure 8: Structural mesh, with 11676 elements

Simulation Steps

For the analysis three different steps were created, the first step (step-0) will perform the automatic stabilization of the part, this is the step that simulates the warping of the part due to the residual stresses created during the injection.

In the second step (step-1) the boundary conditions will be applied, and a compression displacement will be used in an attempt to simulate the pre-stress created with the tightening of the grips in the tensile testing machine.

The last step (step-2) is where the tensile test will be simulated, this is done by applying a displacement of 2mm on the test piece.

It's also necessary to add SDV and STATUS to the Field Output Requests.

4 Results and Discussion

4.1 Idealized stress-strain curves created by AME

AME is going to use the stress-strain curves that were introduced to create its own idealized curve that can then be altered to simulate the behaviour for any direction. After creating its idealized curve it's going to draw the resulting curves for the three directions, so that it can provide a visual representation of the quality of the approximation. Figure 9 shows the curves of the tensile tests and the approximated ones created by AME for the test at 1mm/min,

Figure 9 shows that the approximation is relatively good. The elastic behaviour is almost perfect, however the plastic behaviour is more complicated and there are some differences, especially in the ultimate strain and ultimate stress.

One of the problems with the model is that it considers that the introduced stress-strain curves are for perfectly aligned materials in those directions, which is not true.

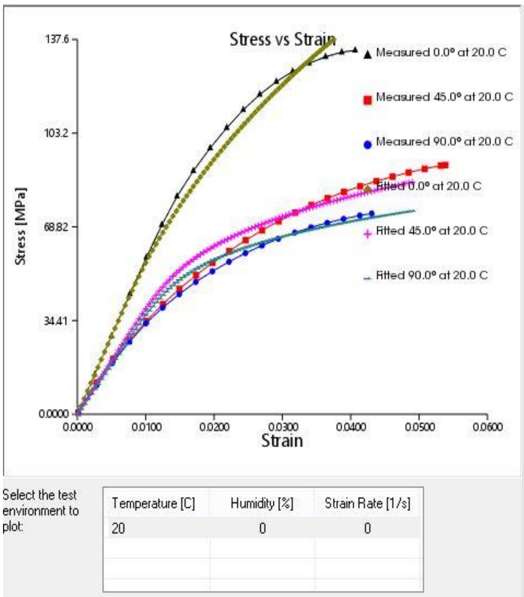

Figure 9: Stress-strain curves from the tensile test and the approximation made by AME, for the test at 1mm/min

4.2 Structural analysis

For every orientation two analyses were performed, one using the residual strains (TR) and one that does not use them (ST). This was done to study the influence of step-0 and the residual strains in the simulation.

Figure 10 shows the results obtained in both simulations for the orientation of 0º,

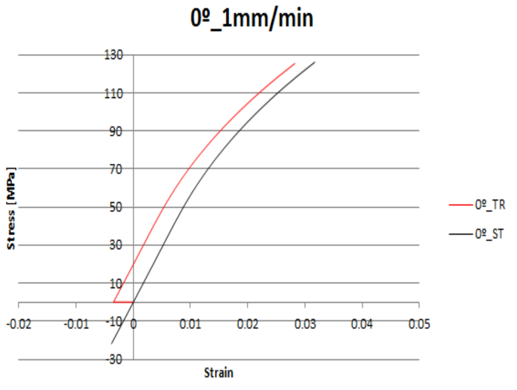

Figure 10: Results of the two simulations for the orientation of 0º

The use of step-0 has a large influence over the final result. Both curves start from (0,0), but ST moves diagonally and ends up with negative stresses and strains, TR only has negative strains, the stresses initially are kept at 0. ST behaves that way because of step-1. TR is different because the use of step-0 is going to cause warping on the part, which means it's going to move without any load from the tensile test, therefore negative strain without change in the stress. The behaviour of TR in step-1 is also different from the behaviour of ST for the same step, because step-0 warps the part in such a way that even a compressive displacement causes an increase of the strain.

Both curves have a very similar behaviour which can be seen in Figure 11 where the influence of step-0 was removed from TR.

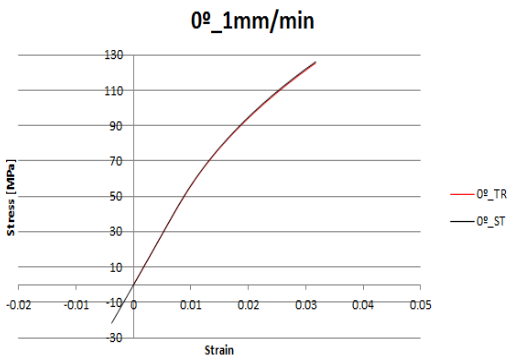

Figure 11: Stress-strain curves where the influence of step-0 was removed from TR

However if we take into consideration that step-1 is not perfect, because in reality the pre-stress causes negative stress but no strain. So if the strain is removed from ST, there will be a bigger difference between the curves.

If the elastic-plastic behaviour of both curves starts at (0,0), where step-2 of ST does not have to work against step-1, the resulting ultimate stress for ST is higher than the ultimate stress of TR, this was to be expected because the existence of residual stresses inside the part before the load is applied, will make the material fracture a lower stresses.

Figure 12 shows the difference in ultimate stress between ST and TR, for the orientation of 90º,

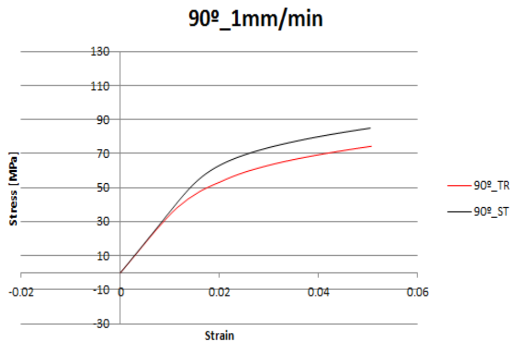

Figure 12: ST and TR curves moved to have the starting point in (0,0), for an orientation of 90º

It is expected for the ST curve to have a very similar behaviour to the experimental curve (after data modification), because both were calculated for a material without residual stresses.

Figure 13 shows the stress-strain curves for ST, TR and the experimental curve,

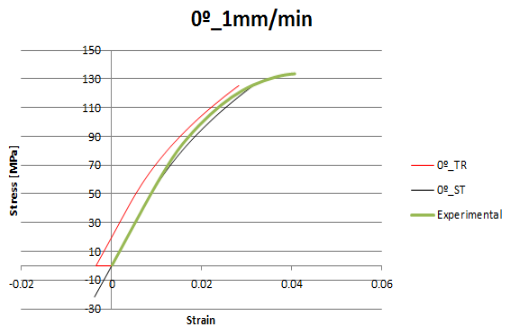

Figure 13: Stress-strain curves for ST, TR and the experimental curve for the orientation of 0º

As expected ST is very similar to the experimental curve, there are some differences. These can be caused by the milling of the test pieces that was not simulated, and the fact that the criteria used in AME are not perfect.

5 Conclusion

The results obtained using AME are not perfect, but they are satisfactory, and it can be used quickly and easily.

It was proven that if a structural analysis is made using only the fibber orientation, it will replicate the behaviour of the stress-strain curves from the tensile test, however this ignores an important part of the problem, the residual strains will change the behaviour of the material, so a correct analysis requires both the fibber orientation and the residual strains.

6 References

[1] Kennedy, P.K. (2008). Practical and Scientific Aspects of Injection Molding Simulation. Eindhoven: Technische Universiteit Eindhoven. DOI: 10.6100/IR634914
[2] M. Medraj. Injection Molding lecture 16. (2016). URL: http://users.encs.concordia.ca/~mmedraj/mech421/lecture%2016%20plastics%203.pdf?q=injection-molding, accessed in May 2017.
[3]Residual Stress, URL: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/physics/b3400001.htm, accessed in May 2017
[4] AUTODESK HELIUS PFA 2017, Advanced Material Exchange Help: Theory Manual. (2017). URL: http://help.autodesk.com/view/ACMPAN/2017/ENU/?guid=GUID-D60A05C0-DD63-47B6-B1CD-248F2BDF1A9D, accessed in May 2017
[5] Kenik, D., Robbins, D., Morrison, A., Gies, J. (2015). Bridging the Gap: As-Manufactured Structural Simulation of Injection Molded Plastics. Automotive Composites Conference. Autodesk, Inc.
[6] AUTODESK HELIUS PFA 2017, Advanced Material Exchange Help: User's Guide. (2017). URL: http://help.autodesk.com/view/ACMPAN/2017/ENU/?guid=GUID-CED5AF9F-EF61-4890-9253-F85DACA3B4A0, accessed in May 2017
[7] Nguyen, B.N. Paquette, J. (2010). EMTA's Evaluation of the Elastic Properties for Fiber Polymer Composites Potentially Used in Hydropower Systems. Pacific Northwest National Laboratory.

[8] Robbins, D. Morrison, A. Dalgarno, R. (2015). Progressive Failure Simulation of As-Manufactured Short Fibre Filled Injection Molded Parts: Validadion for Complex Geometries and Combined Load Conditions. Automotive Composites Conference. Autodesk, Inc.
[9] ISO 527-1. (2012). Plastics – Determination of tensile properties – Part 1: General Principles. International Organization for Standartization
[10] ISO 527-2. (1996). Plastics – Determination of tensile properties – Part 2: Test conditions for moulding and extrusion plastics. British Standards Institution