

Structural Analysis of a Polymeric Component Reinforced with Fiber

João Paulo Oliveira Jaulino
joao.jaulino@gmail.com

Instituto Superior Técnico, Universidade de Lisboa, Portugal

November 2017

Abstract

The mechanical project of injected polymers reinforced with fiber, involves a complex co –simulation between *softwares* to achieve the final structural results. In order to include the deformed shape of the component after mold extraction in the structural analysis, it is fundamental to capture the injection mesh, the fiber orientation tensor and the residuals stresses and strains. This can be accomplished by the use of an API (application programming interface), where the parameters from the mold filling simulation are captured and then exported to the structural software. This workflow involves a continue interaction between softwares to accurately predict the mechanical behavior of the component when subjected to service conditions. This work aims to perform a structural analysis of an intercooler plastic part under service conditons, by taking into consideration the deformed shape resultant from the mold filling analysis. The workflow involves an injection molding software, an API and a structural software. The work also includes an experimental characterization under corrosion to simulate the intercooler service conditions.

Keywords: structural analysis, residual stress, polymer reinforced with glass fiber, API

1. Introduction

In the mechanical project of injected polymers reinforced with short fibers, it's necessary to take into account the stresses that are created in the mold cavity during the filling stage, resulting on warpage after extraction from the mold. This causes residual stress fields that need to be accounted on

the structural analysis for a better prediction of the component's behavior under loading conditions. Consequently, there's the need to export the mold filling analysis information to the structural analysis software [1, 2]. The co-simulation helps engineers to capture the information from the mold filling analysis and then export it to the structural analysis, through

the use of an application programming interface (API). Therefore, the workflow involves a mold filling software, an API and a structural software.

In addition to the workflow, it is important to characterize the material that constitutes the component. In mechanical project, the understanding of the material's behavior under service conditions is of the most importance, aiding in the prediction of possible failure.

This work aim to perform a structural analysis of an intercooler plastic part made out of PA6+PA66+GF30 (polyamide blend with 30% glass fiber reinforcement) under loading conditions, taking into account the residual stress resultant from the mold filling analysis. To further discussion the material is also characterized in corrosion conditions, simulating the environment conditions of an intercooler.

2. Experimental material characterization

Understanding the material's behavior is crucial to the mechanical project. Moreover, the real curve of the material can and should be used in the workflow of the structural analysis, providing a more realistic approach to the simulation. Since the material used is reinforced with short glass fibers, it is essential to capture the orientation in the experimental data for further introduction in the workflow.

Simulating the environment conditions of the intercooler is vital to predict the component behavior under service conditions. It is known that the

corrosion of polyamide affects the material properties of a polymer due to weight variation [3]. The experimental tests also include a characterization of the material under a corrosive chemical compound, simulating the *exhaust gas recirculation* (EGR) gases. The chemical compound composition is confidential, but it's known that it is more aggressive than the real conditions.

2.1. Experimental procedure

The characterization of the material was performed with tensile tests according to the norm ISO 527, using the dumb-bells 1BA and a test speed of 1mm/s. The dumb-bells were machined according to three fiber orientations: 0°, 45° and 90°. The tests were conducted in four conditions:

- 0 hours of corrosion;
- 50 hours of corrosion;
- 100 hours of corrosion
- 150 hours of corrosion.

In the corrosion test, the dumb-bells were submersed in a chemical compound simulating the EGR gases, in a controlled temperature at 30°C. After the time of corrosion, the dumb-bells were weighted for further investigation.

2.2. Experimental results and discussion

To analyze the effect of corrosion on the material, the results are presented according to the same fiber orientation under different times of corrosion. Figures 1, 2 and 3 show the results from orientations 0°, 45° and 90°.

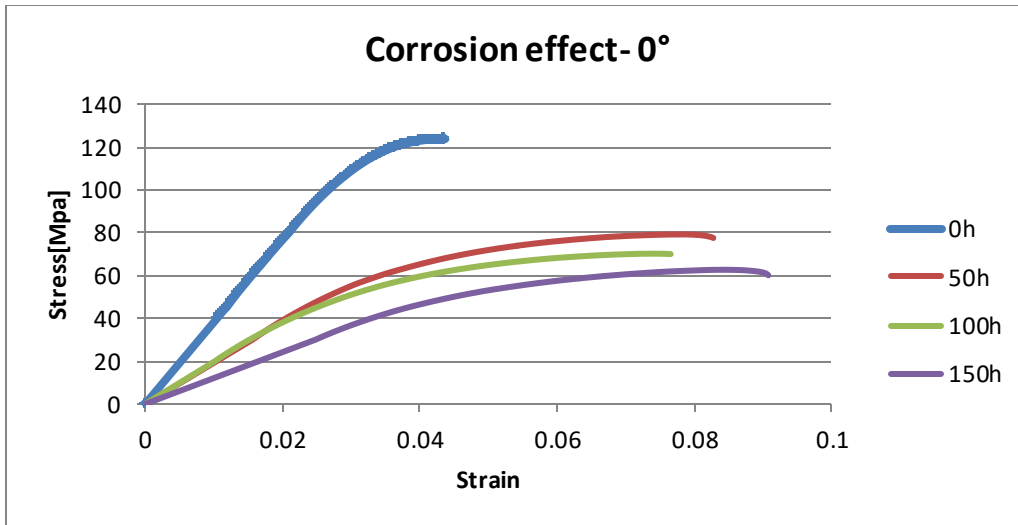


Figure 1 – Effect of corrosion on 0°

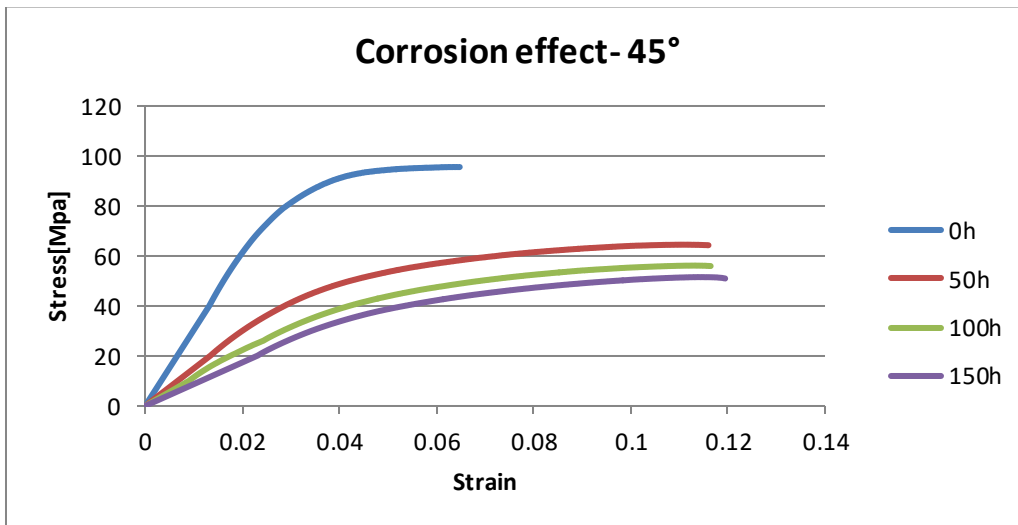


Figure 2 – Effect of corrosion on 45°

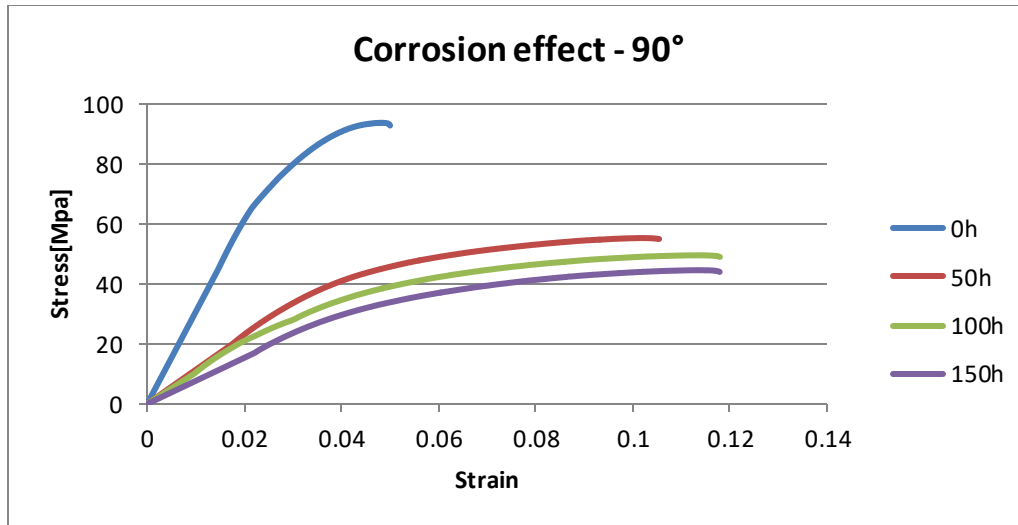


Figure 3 – Effect of corrosion on 90°

It is visible that the mechanical properties of the material are extremely affected in the first hours of corrosions, converging to an asymptotic value on the following hours. Cai *et al.* explore the effect of molecular weight on the mechanical properties of polymers. The authors concluded that, as the weight increases, the stress at rupture decreases and the elongation increases to a maximum value until a certain weight, where the properties start to increase and decrease again [4]. Table 1 is shown to correlate the weight change to the mechanical properties. It is only presented the 0° orientation.

Table 1 – Correlation of weight and mechanical properties

	Corrosion [h]	Weight [g]	Stress [Mpa]	Strain
Orientation 0°	0	1.45	123.761	0.044
	50	1.48	79.195	0.083
	100	1.48	70.204	0.077
	150	1.49	62.689	0.091

The results show that we can correlate the weight change to the mechanical properties of the material. It is exactly at the most amount of weight change, i.e 50 hours of corrosion, that the mechanical properties show their biggest variation.

It is concluded that the corrosion under EGR simulating gases highly affects the mechanical properties of the polymer and should be taken into account in the structural analysis.

3. Mold filling simulation

The mold filling simulation was carried out using the software Autodesk Moldflow Insight. For a better understanding of the output results concerning the residual stress and residual strain after mold ejection, is fundamental to understand the governing equations utilized by Moldflow at warpage analysis. After part ejection, volumetric shrinkage occurs due to the gradient of temperatures. To simulate this phenomenon, Moldflow uses a three-dimensional solid stress theory, using the following equations [5]:

$$\sigma = C(\varepsilon - \varepsilon^0 - \alpha\Delta T) \quad (1)$$

$$\varepsilon = \frac{1}{2}(\nabla u + \nabla u^T) \quad (2)$$

Where σ is the residual stress, C the CLTE tensor, ε the strain, ΔT the temperature gradient and ∇u the displacement tensor.

Capturing this information in the workflow is crucial for the structural analysis in account previous deformation of the component due to residual stress and residual strains. Figure 4 shows an illustrative example of a part subjected to this information in structural analysis.

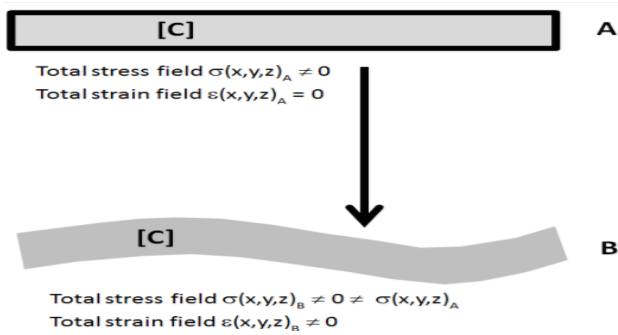


Figure 4 – Residual stress and strain effect on an ejected part [6]

However, the residual stress and strain are calculated assuming that the part is ejected right after the molding cycle ended, which causes an increase in the results. In reality, the part is kept inside the mold while cooling, reducing the stresses and strains that deform the part. This fact needs to be taken into consideration when performing a structural analysis.

4. Data transfer

To transfer the referred data from Moldflow to Abaqus is necessary to use an API. The use of an API allows engineers to capture the information from the injection simulation softwares and use it in the structural analysis. In this study, two methods of

data transfer were discussed: (1) direct transfer data using Abaqus Interface for Moldflow; (2) mapping data using Helius PFA – Advanced Materials Exchanger. The Abaqus Interface for Moldflow revealed itself to be a poor API to transfer the information prevent from Moldflow analysis. In contrast with Helius, it does not allow the user to add nonlinear material data, assuming that the material behavior is linear elastic. Therefore, the method chosen to perform the transfer data was the mapping data using Helius AME.

4.1. Mapping data with Helius AME

Helius AME is an API that allows the map of the residual strains and the fiber orientation tensor from the Moldflow analysis and afterwards exports it to the structural analysis. The API works as a software where the Moldflow mesh and the structural mesh are imported. In it, the user can add the information of the experimental data material in the three orientations of fiber. The data is mapped from the Moldflow mesh to the structural mesh and then exported for structural analysis. Figure 5 shows the workflow using Helius AME.

Helius assumes that the rupture occurs in the matrix of the polymer and not in the fiber. Therefore, it uses a process of homogenization and decomposition based on the Mori Tanaka micromechanical model to calculate the average stress and strain of the material [7]. This improves the accuracy of the analysis.

In addition, it implements the material experimental data in a structural analysis by using a modified model of the Ramberg – Osgood and a modified Von Mises stress to predict the plasticity of the material. The yielding occurs when equation (3) equals the equation (4)

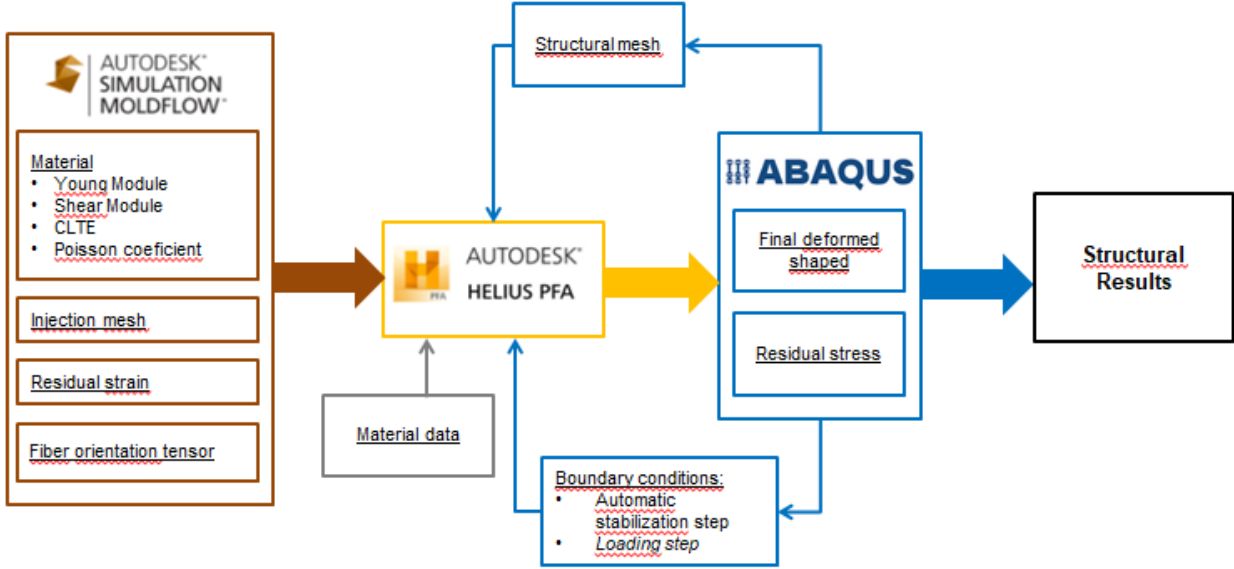


Figure 5 – Workflow chart using Helius AME

$$\sigma_y^h = E \frac{1}{n} (\sigma_0)^{\frac{n-1}{n}} (\epsilon_{eff}^p)^{\frac{1}{n}} \quad (3)$$

$$\sigma_{eff} = \sqrt{\frac{(\alpha \sigma_{11} - \beta \sigma_{22})^2 + (\beta \sigma_{22} - \beta \sigma_{33})^2 + (\beta \sigma_{33} - \alpha \sigma_{11})^2 + 6(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2)}{2}} \quad (4)$$

Where σ_y^h is the yield stress of the polymer matrix, E is the young modulus ϵ_{eff}^p the matrix of plastic strain and α and β are directional dependence coefficients that account the fiber direction in the effective stress equation. They are computed by equations (5) and (6), where θ is the randomness of the fiber parameter, λ is the highest fiber orientation tensor eigenvalue and λ_m , α_m and β_m the values for a strongly aligned fiber orientation [6].

$$\alpha = \theta + \left(\frac{\alpha_m - \theta}{\lambda_m - \frac{1}{2}} \right) \left(\lambda - \frac{1}{2} \right) \quad (5)$$

$$\beta = \theta + \left(\frac{\beta_m - \theta}{\lambda_m - \frac{1}{2}} \right) \left(\lambda - \frac{1}{2} \right) \quad (6)$$

4.2. Preparation of structural analysis

In order for the accountability of the warpage results from Moldflow on the structural analysis, a previous step free of boundary conditions and loadings must be added. This step must eliminate the rigid body movement without any boundary conditions applied. Abaqus offers an artifice to solve this type of problems: the energetic stability.

In unstable nonlinear problems, strain energy is transferred from one location of the component to another. This strain energy transferred can be minimized using the energy stabilization. This artifice introduces viscous forces on the equilibrium equation in the problem solving process of Abaqus. The viscous forces are dependent on the damping coefficient by the equation:

$$F_V = cM^*v - I = 0 \quad (7)$$

Where c is the damping coefficient, M^* an artificial matrix of mass with unitary density, and v the local velocity.

The damping coefficient can be specified directly or by the use of the dissipation of energy. If the damping coefficient is specified, it will be use in all the problem solving process of Abaqus. Otherwise, by using the dissipation of energy, the coefficient will be adapted to each problem solving process.

In case of directly specifying the damping coefficient, two considerations must be accounted:

- 1) The warpage results from Moldflow must be similar to the results of the structural analysis with only the stabilization step;
- 2) The intensity of the dissipated energy must be smaller than the internal strain energy.

In an attempt to directly specifying the damping coefficient it was made an interactive process based on the study of Pedro [8]. The comparing results from Moldflow and the structural analysis with only the stabilization step are shown in Table 2.

As the damping coefficient increases, the viscous forces introduced in the equilibrium equation are stronger, approximating the results from the stabilization step to Moldflow. Damping coefficients of 0.1 and 0.01 produce similar results to Moldflow, however, they do not respect the energy consideration. So, to eliminate the rigid body movement it was used the dissipation of energy in the step of the energetic stabilization.

Table 2 – Comparing Moldflow results to the structural analysis using dumping coefficient

	U[mm]		U _x (mm)		U _y (mm)		U _z (mm)		
	U _{máx}	U _{mín}	U _{máx}	U _{mín}	U _{máx}	U _{mín}	U _{máx}	U _{mín}	
Moldflow	1.554	0.054	0.7743	-1.036	1.367	-1.301	1.435	-0.9598	
Abaqus	C = 1e ⁻¹⁰	4.545	0.04685	1.177	-1.049	1.714	-1.418	2.899	-4.46
	C = 1e ⁻⁴	3.438	0.4527	1.211	-2.116	2.119	0.1562	3.363	-3.091
	C = 1e ⁻²	1.42	0.09959	0.8797	-0.8348	1.246	-1.125	1.368	-0.7975
	C = 1e ⁻¹	1.454	0.1026	0.6643	-0.9678	1.099	-1.187	1.343	-0.7916

5. Case Study

The aim of the case study was to perform a structural analysis to a intercooler plastic part, taking into account all the workflow explained before. Two

situations were analyzed: Material under no corrosion and material under 50 hours of corrosion. To simulate the service conditions it is introduced another step, which takes into account the loading

(Figure 6 a)) and the boundary conditions (Figure 6b)) of the model.

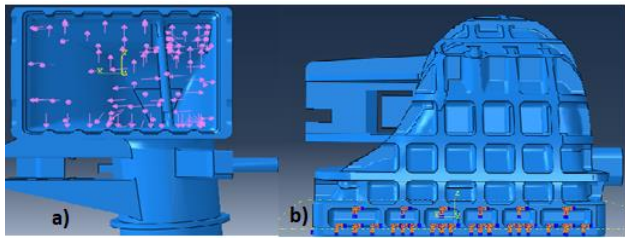


Figure 6 – Step of service conditions: a) Loading ; b) Boundary Conditions

5.1. Structural analysis results

The structural analysis is dependent of the material introduced in Helius. Based on the three fiber orientations curves, Helius creates a curve that characterizes the material matrix of the polymer, in which is established a limit stress that must not be exceed, in order to avoid fracture of the component. The interaction between Helius and Abaqus allows the use of solution dependent variables (SDV) that offers output related to the fracture of the component. To determine if the component supports the conditions applied it was analyzed SDV1, which allows the user to conclude if the component fails. If SDV1 takes a value of 1 there is no fracture, and if it takes a value of 2 the limit stress of the material matrix is exceed and the material will fracture, causing the failure of the component. In figure 7 are shown the final SDV1 for both cases. The red spectrum shows the location where the material matrix fractures, meaning that both situations show failure where the boundary conditions are applied. Although, for the situation where there is no corrosion of the material, there should be no failure. This can happen due to the boundary conditions

applied. In reality, the boundary conditions are applied only in the lateral and inferior faces of the orifices and allow the degree of freedom of θ_x , θ_y and θ_z . In the structural analysis performed, the boundary conditions are fully fixed on all faces of the orifices, raising the reaction forces and leading to the failure of the component.

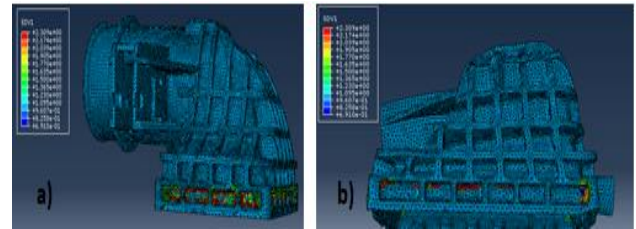


Figure 7 – SDV1 results for: a) Material with no corrosion; b) Material with 50 hours of corrosion

5.2. Discussion

After concluding that the component will fracture in both situations, is now fundamental to explain an inherent error in the structural analysis while using the material under corrosion.

The deformation of the component due to the residual strains in the molding process creates residuals strains that are accounted on the final structural analysis. These residual strains are exported from Moldflow to Abaqus using Helius, which uses the experimental material introduced. By introducing the corroded material in Helius, the residual strains will correspond to a different value of residual stress that occurs in reality. Because Helius doesn't allow the introduction of two different materials for the different steps of the structural analysis, an error is generated in the calculus of the residual stress. Table 3 shows the residual stress and the displacement for both situations.

Table 3 – Residual stress and displacement of both situations

	U [mm]		S ₁₁ [Mpa]		S ₂₂ [Mpa]		S ₃₃ [Mpa]		S Von Mises	
	Máx	Mín	Máx	Mín	Máx	Mín	Máx	Mín	Máx	Mín
Non corroded material	1.7	0.104	64.26	-0.0112	66.13	-0.0101	66.18	-0.0135	115.5	0.4799
Corroded material	1.69	0.1112	21.12	-0.4285	18.96	-0.3963	20.64	-0.6034	55.85	0.1591

Since the residual strains used in both cases are the same (exported from Moldflow to Helius), the displacement shows similarity. However, the residual stresses are different, mostly because of the Young's modulus in each curve. For a material in a corroded condition, the Young's modulus will be smaller compared to the Young's modulus of the non-corroded material, lowering the residual stress at the end of the molding process. This creates an error in the results generated on the structural analysis.

6. Conclusions

The objective of the workflow is to provide a more accurate prediction of the results in the structural analysis of injected components reinforced with fiber. The methodology used is able to capture the residuals stress and strain from the mold filling analysis and use this information to perform a structural analysis, taking into account the warpage created by the cooling of the component.

Material characterization is crucial for the workflow. It can and should be used in the structural analysis and helps in the prediction of the component behavior.

The data transfer must be made by an API that can map the experimental data into the structural analysis. It gives a more realistic approach of the structural analysis, and consequently, allows the

prediction of more accurate results. Helius shown itself to be a powerful tool, aiding the structural software in the prediction of the residual stresses created in the molding process, however for this case study it was proven insufficient because it cannot simulate the corroded conditions.

7. References

1. Peng, A.Y., E. Wu, and C. Huang, *INTEGRATING PLASTICS MOLDING AND STRUCTURE DYNAMICS ANALYSIS BY LEVERAGING LS-DYNA, MOLDEX3D AND PRESYS*.
2. Kumar, P.S., et al., *Structural analysis of fiber-filled plastics with moulding process induced anisotropy*. SASTECH, 2010. **9**(2).
3. Sakai, T., et al., *Study on permeation behavior and chemical degradation of PA66 in acid solution*. *Polymer Degradation and Stability*, 2007. **92**(3): p. 379-388.
4. Cai, J., et al., *Effects of molecular weight on poly (ω -pentadecalactone) mechanical and thermal properties*. *Polymer*, 2010. **51**(5): p. 1088-1099.
5. Yang, W.-H., et al., *Integrated numerical simulation of injection molding using true 3D approach*. *transition*, 2008. **5**(5): p. 6.
6. Autodesk Helius PFA. *Theory Manual*. 2017 Setembro 2017]; Available from: <https://knowledge.autodesk.com/support/helius-pfa/learn-explore/caas/CloudHelp/cloudhelp/2018/ENU/ACMPAN-AME/files/GUID-D60A05C0-DD63-47B6-B1CD-248F2BDF1A9D-htm.html>.
7. Tucker III, C.L. and E. Liang, *Stiffness predictions for unidirectional short-fiber composites: review and evaluation*. *Composites science and technology*, 1999. **59**(5): p. 655-671.
8. Pedro, E., *Análise Estrutural de Componentes Obtidos por Moldação por Injeção com o auxílio do Digimat*. 2017: IST.

