INFLUENCE OF MANUFACTURING PROCESS IN MECHANICAL BEHAVIOR OF INJECTED PLASTIC COMPONENTS.

Manuel Muniesa[(a)], Isabel Clavería[(b)], Carlos Javierre[(c)], Daniel Elduque[(d)], Ángel Fernández[(e)]

[(a), (b), (c), (d), (e)]Universidad de Zaragoza, Spain

[(a)]mmuniesa@unizar.es , [(b)]iclaver@unizar.es , [(c)]sabicjl@unizar.es
[(d)]danielduque88@gmail.com , [(e)]afernan@unizar.es

ABSTRACT

Mechanical behavior of automotive injection components is usually analyzed without regard the conditions of manufacturing process. Some consequences due to plastic injection process must be taken into account, such as fiber orientation, residual stress or weld line locations. These factors are specially relevant for reinforced materials. However, traditionally CAE tools are limited to analyze stress, strain or displacement results from a isotropic material model.

This paper shows an integrated analysis with CAE tools, including results of manufacturing process in mechanical calculations. Two types of reinforced thermoplastic material are analyzed: PP and PA6. Glass fiber is used for the reinforcement of both materials and fiber orientation tensor due to injection process is taken into account. Also, influence of fiber material percentage is analyzed. Results obtained using integrated methodology show important differences in comparison with traditionally method. Is specially relevant the difference obtained in displacement results.

Keywords: part design, plastic injection process, fiber orientation, mechanical behavior.

1. INTRODUCTION

The development of components made with reinforced thermoplastic materials produced by injection process is on the increase nowadays, and therefore the accuracy of design and the correct calculations involved in making these products has become very important. By using CAE tools we can predict the behavior of these components when in use (Lin et al. 2004). These simulation analyses can save a lot of time and money when used in the design phases of said components. Up to now it has been habitual to make simulations in separate phases of production depending on the types of analysis behavior used. On the one hand analyzing the fabrication process of the components and on the other hand seeing how they behave under conditions of use. However, in the case that concerns us in relation to reinforced materials, these different phases are strongly related. Manufacturing process determines fiber orientation on the matrix of reinforced material (Parveeen et al. 2014), and therefore strongly conditions its later response to the tests which it will undergo (Adam and Assaker 2014).

This article presents a work methodology for the integrated simulation of manufacturing process of reinforced materials and their mechanical responses as well as analyzing the influence of the percentage of fiber in the material and the type of polymer matrix (Doghri and Tinel 2004). For this study a component of a turbo compressor for a vehicle has been used (Ruiz et al. 2008). The said component is subjected to internal pressure and it made with thermoplastic material and reinforced with fiberglass (Bicart et al. 2011). The design of this component is determined, mechanically speaking, by the restrictions of the injection process used in its fabrication, the material used and the strength required (Fernández et al. 2013, Demirer and Deniz 2012). The consequences that the injection process has on the component are very well known, such as the appearance of shrinkage and warpage, residual stress and fiber orientation once it is made (Fernández et al. 2014, Yu et al. 2014, Ozcelik and Sonat 2009). These results should be kept in mind when making a mechanical-structural analysis of the component by increasing the load or applying greater security coefficients.

2. METHODOLOGY

This article presents a work method that objectively integrates the reactions over the component by the injection process as well as the conditions of the load when working with composite materials in the mechanical analysis. More specifically, the fiber orientation tensor of composite material is integrated into the mechanical-structural analysis.

In most cases, only structural analysis with commercial software are made. Start point for a lineal static analysis is a mathematical material model with elastic modulus and Poisson´s ratio coefficients.

For the integrated methodology first step is to create a structural mesh (for example with software Abaqus©) of the part. In this case a component of a turbo compressor. Also, a rheological analysis of manufacturing process is made by Moldflow© software. A different mesh with more elements and nodes is used for calculations along part thickness. Injection time, number of plastic entrance for injection process, material viscosity, process temperature and pressure are different parameters to be considered.

A result obtained of this analysis is the fiber orientation tensor. A file with .xml extension contain all the information about principal directions and elastic modulus in each element of the rheological mesh.

Digimat© software coupled fiber orientation tensor file, structural mesh and a mathematical material model based on experimental analysis. The output of this software is a new structural mesh file with two additional files (.mat, .dof) wich are ready to use in a structural analysis with anisotropic influence by fiber orientation added.

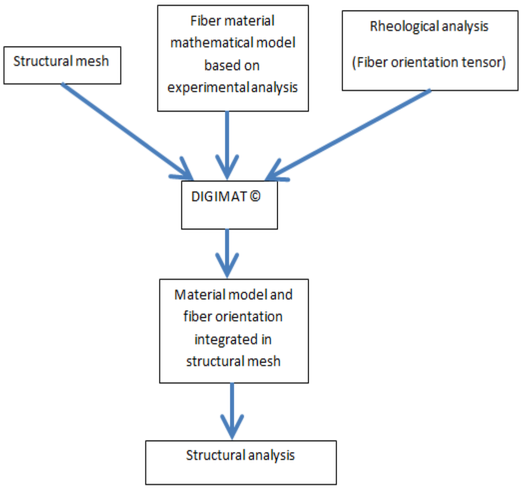


Figure 1: Scheme of integrated methodology applied.

In order to see the influence that the additional procedural and mechanical calculations make, we will analyze different cases with different material compositions. To do this, we will apply the methodology already developed for mechanical analysis of said vehicle component made with: polypropylene 20% fiberglass (Hostacom©G2U02), polypropylene 50% fiberglass (Celstran©PP-GF50-03), polyamide 6 20% fiberglass (Durethan©BKV20FND) and polyamide 6 50% fiberglass (Durethan©BKV50H20).

Material Type	PP+20%GF	PP+50%GF	PA+20%GF	PA+50%GF
Trade Name	Hostacom ©G2U02	Celstran©P P-GF50-03	Durethan© BKV20FND	Durethan ©BKV50H 20
Density	1,05 g/cm^3	1,33 g/cm^3	1,35 g/cm^3	1,57 g/cm^3
Young Modulus	4800 MPa	10300 MPa	6100 MPa	16000MPa
Injection Temp.	220^{o}C	260^{o}C	270^{o}C	290^{o}C
Mold Temp.	40^{o}C	50^{o}C	80^{o}C	80^{o}C

Table 1: Material properties

Moldflow©, Digimat© and Abaqus© CAE software have been used for analysis. In first step, process manufacturing is analyzed with Moldflow© software. Injection parameters like injection time, melt temperature, mold temperature, packing and cooling time are scheduled. Results of injection pressure, bulk temperature, weld line location, air traps or fiber orientation tensor are obtained. Digimat© software is used to couple process analysis with mechanical analysis through a material numerical model including fiber orientation and mechanical properties. Finally, mechanical analysis is made by Abaqus© software using this new material input data.

3. RESULTS

A comparison between integrated methodology and single mechanical analysis has been made. Results from manufacturing process simulation are showed in Figure 1 and Table 1.

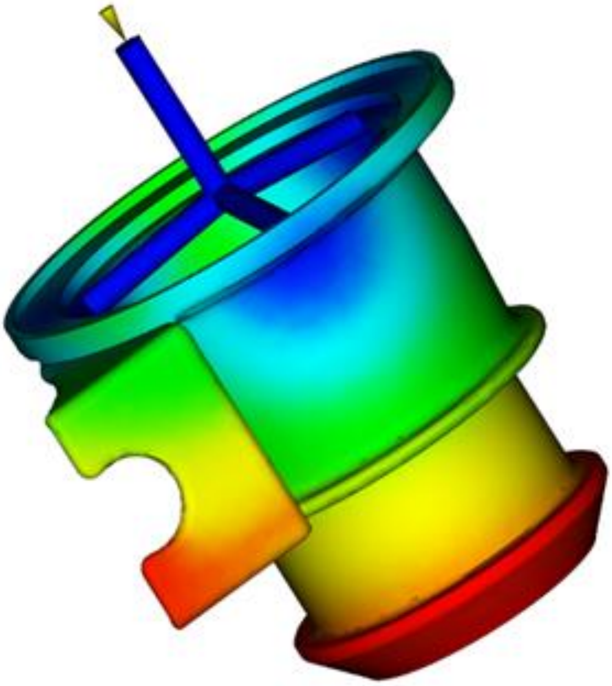


Figure 2:Melt front advance

Material	Max. Injection Pressure	Flow Front Temperature Difference
PP + 20%GF	8 MPa	3^{o}C
PP + 50%GF	14 MPa	4^{o}C
PA6 + 20%GF	16 MPa	3^{o}C
PA6 + 50%GF	22 MPa	6^{o}C

Table 2: Injection results

Higher values of maximum injection pressure is given for polyamide 6 with 50 % glass fiber reinforcement due to viscosity of polymer. Results of fiber orientation tensor are showed in Table 2. This result reveals the importance influence than percentage of fiber reinforcement has in the structural morphology of reinforced material.

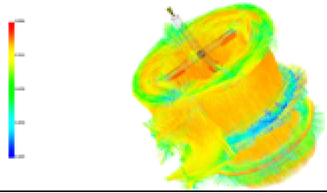
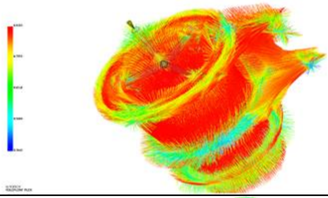
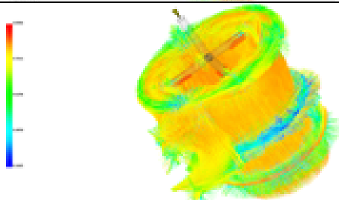
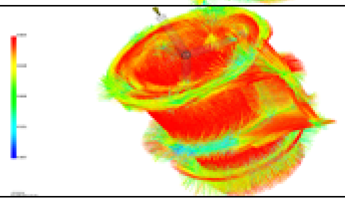
Material	Fiber orientation
PP + 20%GF	
PP + 50%GF	
PA6 + 20%GF	
PA6 + 50%GF	

Table 3: Fiber orientation tensor results

Abaqus© analysis show stress, strain and displacement results under boundary conditions applied, like load or fixation points, Figure 2. In this case, stress results with material model input calculated previously by Digimat© software are showed in Figure 3.

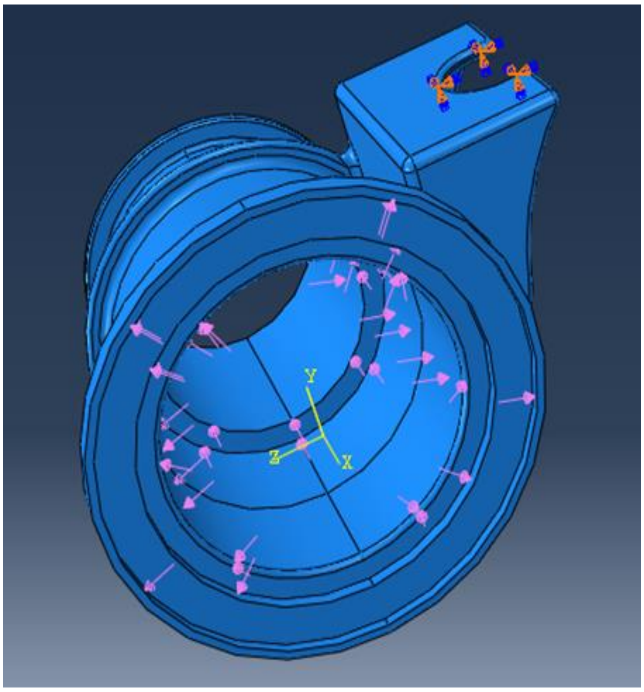


Figure 3: Boundary conditions applied

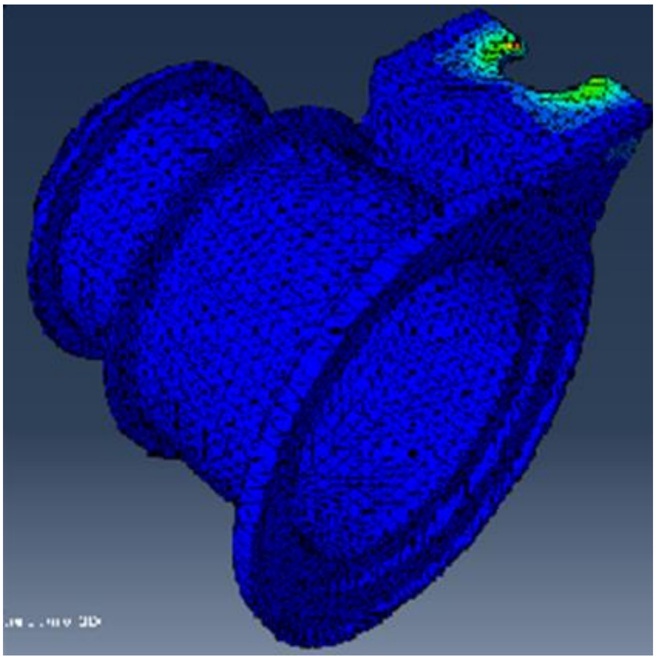


Figure 4: Von Mises stress results for integrated analysis

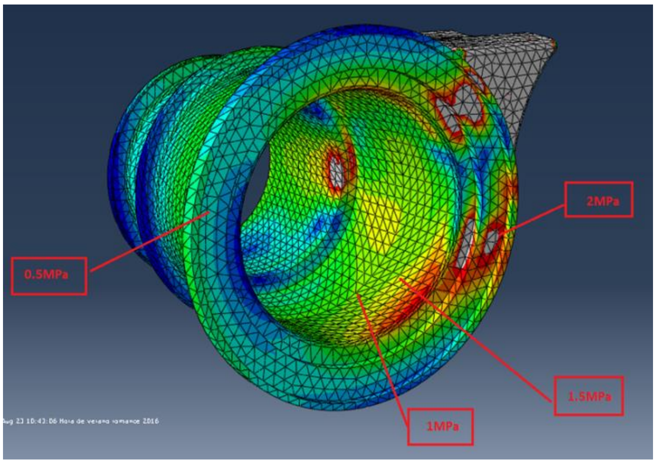


Figure 5: Von Mises stress detail results for integrated analysis in the inner zone of the part.

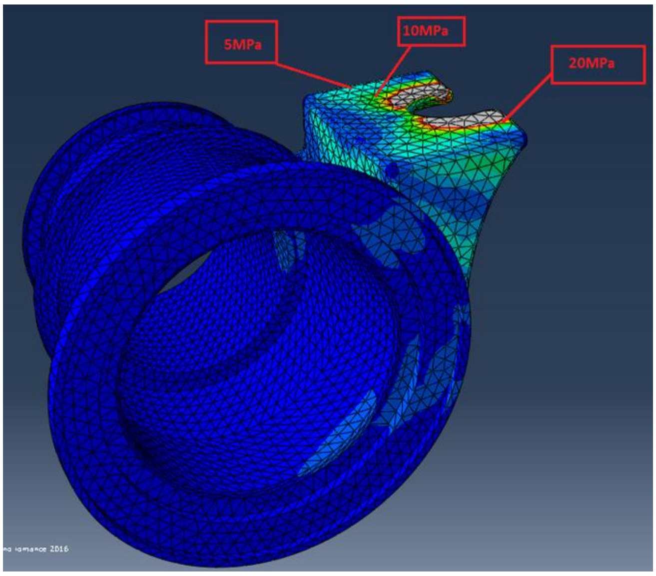


Figure 6: Von Mises stress detail results for integrated analysis in the fixing zone of the part.

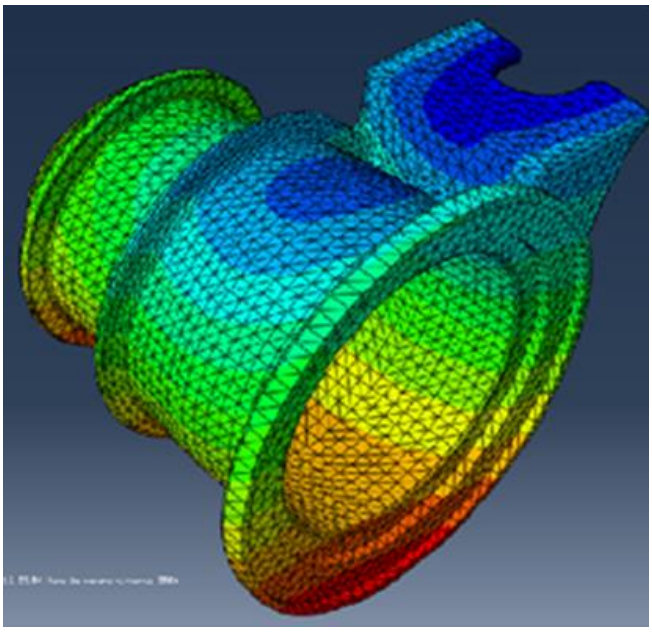


Figure 7: Displacement results for integrated analysis.

One will observe that the resulting displacements nearly double when taking into consideration the anisotropic behavior of the material using the fiber orientation tensor, calculated in the process simulation step. Results with integrated methodology and with traditionally isotropic material assumption are shown in Table 3.

Material	Displacements with integrated methodology	Displacements with isotropic material assumption	Difference (%)
PP + 20%GF	2.73 mm	1.65 mm	64.7 %
PP + 50%GF	1.43 mm	0.62 mm	128.8 %
PA6 + 20%GF	1.51 mm	1.14 mm	32.1 %
PA6 + 50%GF	0.82 mm	0.45 mm	82.8 %

Table 3: Displacements comparation

Displacement result difference is much more noticeable when working with materials with 50% fiber than ones with only 20%. Additionally, the displacements are seen to increase in greater measure when working with polypropylene. If both of these variants occur at the same time the displacement is more than double that which is obtained without applying the integrated methodology.

4.-CONCLUSIONS

A study of the variation of results in mechanical simulations of an automotive part, implementing all of the information gathered in the manufacturing process rather than only the results that do not take into account such information, has been realized. Additionally the influence of the percentage of fiber and the material used has been analyzed.

The results obtained using a composite material with a polypropylene matrix and another with a polyamide matrix, both made of fiberglass, have been compared. A greater influence has been seen in the results of the polypropylene examples, probably due to the different relation between matrix and reinforcement, which induces a better orientation of the fiber on this type of matrix than on the polyamide one.

The percentages of fiber compared, in the polyamide as well as in the polypropylene components, have been 20% and 50%. Upon adding a higher percentage of fiberglass to the material the resistance is increased, as can be seen in the values determined in Young´s model and Poisson´s coefficient. That is why the values of fiber displacement is higher in components with only 20% of said material. However, upon applying the methodology, the values increase much more for the 50% examples that for those of 20%. This contrast is

ISBN 978-88-97999-85-0; Affenzeller, Bruzzone, Jiménez, Longo and Piera Eds.

attributed to the fact that the fiber is better oriented in the case of the 50% samples, but the way of orientation with regard to load applied, make it less efficiency.
To conclude, the mechanical-structural analysis highly depends upon the fabrication process, because that is where the real displacements that the product undergoes are taken into account. On the contrary, if the manufacturing process of the item is not considered, we run the risk of have failed designs if the values of stress, strain or displacements differ much between one case and another.

REFERENCES

Adam, L., Assaker, R., 2014. Integrated nonlinear multi-scale material modelling of fiber reinforced plastics with DIGIMAT RP®: application to short and continuous fiber composites. 11th World Congress On Computational Mechanics; 5th European Conference On Computational Mechanics; 6th European Conference On Computational Fluid Dynamics II – IV, 2322-2333.

Bicart, J., Robert, G., Moulinjeune, O., 2011. Multiscale Modeling of the Orthotropic Behaviour of PA6-6 overmoulded Composites using MMI (TM) Approach. 14th International Conference On Material Forming Esaform, 2011 Proceedings 1353, 726-731.

Demirer, A., Deniz, S., 2012, Investigation of the effects of weld lines on the mechanical properties and energy consumption for injection moulded thermoplastics. Energy Education Science and Technology. Part A-Energy Science and Research 29, 1055-1062

Doghri, I., Tinel, L., 2004. Micromechanical modeling and computation of elasto-plastic materials reinforced with distributed-orientation fibers. International Journal of Plasticity 21, 1919-1940.

Fernández, A., Muniesa, M., González, J., 2013. Characterisation and Processing of Reinforced PA 6 with Halloysite Nanotubes (HNT) for Injection Molding. Journal of Mechanical Engineering 59, 183-192.

Fernández, A., Muniesa, M., Javierre, C., 2014. In-line rheological testing of thermoplastics and a monitored device for an injection moulding machine: Application to raw and recycled polypropylene. Polymer Testing 33, 107-115.

Lin, B.J., Jin, X.S., Zheng, R., Costa, F.S., Fan, Z.L., 2004, 3D fiber orientation simulation for plastic injection molding. Materials Processing and Design Modeling Simulation and Applications, Parts 1 & 2. 712. 282-287

Ozcelik, B., Sonat, I., 2009, Warpage and structural analysis of thin shell plastic in the plastic injection molding Materials & Design 30, 367-375

Parveeen, B., Caton-Rose,P., Costa,F., Jin, X., Hine, P., 2014, Study of injection moulded long glass fibre reinforced polypropylene and the effect on the fibre length and orientation distribution. Proceedings of PPS-29: The 29th International Conference of the Polymer. Conference Papers, 432-435, Jul 15-19,2013, Nuremberg, Germany

Ruiz, I., Jimenez, E., Blanco, J., Pérez, M.,2008. Design and simulation of production of injection pieces in automobile industry. Second UKSIM European Symposium on Computer Modeling and Simulation, 335-340.

Yu, Y., Y., Zhang, B., Tang, Z., Qi, G., 2014. Stress transfer analysis of unidirectional composites with randomly distributed fibers using finite element method. Composites Part B 69, 278-285.