A demonstrator for the experimental assessment of through process modelling of injection moulded parts made of short fibre-reinforced polymers

An example illustrating the greater reliability of results from structural calculations using an advanced design approach and correct anisotropic characterization of the materials

E. Spini, RadiciGroup Performance Plastics, Italy
A. Bernasconi, Politecnico di Milano, Milan, Italy

Abstract

Through process modelling of injection moulded parts is nowadays the industry standard approach to the structural analysis of parts made of short fibre-reinforced polymers. The designer has to face challenging tasks, in order to conduct accurate and reliable analyses. Complex geometries, weld lines and multiple flows make it difficult to transfer results of tests conducted on simple specimens to the assessment of the real parts. We have designed a mould allowing for manufacturing a demonstrator, i.e., a part of sufficiently complex shape, typical of injection-moulded structural parts, allowing for creating different conditions in terms of fibre orientation, presence of weld lines and multiple injection points. The part is designed to be tested under three- or four-point bending conditions. By simulating the test conditions, the designer can compare experimental results with numerical solutions, in order to better understand the potential of the software packages he is using.

Introduction

The transport sector is the driving force of innovation leading to the design and development of new polymers with increasingly higher levels of performance. The need to reduce component weight requires materials capable of performing the same functions that only metals have been capable of achieving up to now. In this field, polymers, and more specifically polyamide polymers, have played a very important role, especially for components operating at high temperatures and subjected to high stress. Novel nylon solutions have been introduced in keeping with the evolution towards smaller, more powerful and longer-lasting engines.

In the field of metal replacement (aluminium alloys, in particular), there are currently numerous mature applications where the replacement rate has reached 80 to 90 per cent. Among these parts are radiator tanks, air intake manifolds, valve covers, fans and air conveyors.

However, there are other components that are still mostly made of metal, but are potential targets for metal substitution: engine mounts, hot side turbo air ducts, engine and transmission oil pans.

There are still other applications that are being studied and for which hybrid solutions have been proposed: plastics with appropriately positioned metal inserts, or continuous fibre-reinforced plastics. Examples of the latter are: brake pedals, suspension parts, front-end parts, car seats parts and dashboard supports.

In this context of change, producers of plastic materials for engineering uses have been investing sizable resources in the production of increasingly more reliable materials with better performance.

Another aspect, comparable in importance to the availability of suitable materials, is the access to data and more reliable design methodologies for more accurately defining the dimensions of plastic components. The traditional design criteria adopted for plastics, e.g., glass or carbon fibre-filled polymers considered as isotropic materials, are the ones inherited from the metal world. Such an approach is likely to introduce severe design errors. In some cases parts may be oversized leading to unnecessary cost increases. In the opposite case, when part sizes are underestimated, additional costs will be incurred due to the need to re-design the components with a greater safety allowance.

As a result, it is necessary to change the approach to the design of engineering plastic components in order to increase the reliability of structural calculations. Such a change represents a very important step, particularly for components expected to operate in increasingly severe environmental conditions and/or plastic parts performing functions that could affect safety.

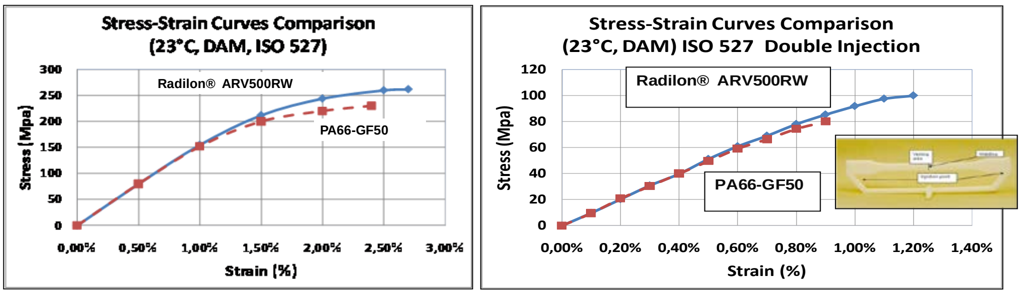
Plastics developed for metal replacement are generally glass or fibre-filled polymers to be processed by injection moulding techniques. These materials, are, for all intents and purposes, anisotropic [1]. Fibre orientation and distribution determine the local characteristics at each point of the moulded object and are dependent on the moulding process.

This paper presents the results of a collaborative project by RadiciGroup Performance Plastics and the Politecnico di Milano university, in which a comparison is made between real and computed data for a plastic “demonstrator”.

Computations were performed using a method known as Through Process Modelling (TPM), which allows for linking the local properties of the material at each point in a finite element model with the results of the injection process simulation. More specifically, the effect of the fibre orientation is taken into account through the use of appropriate software packages for the interface and the computation of properties as a function of the fibre orientation distribution. Examples of the application of the TPM method to fatigue strength assessment can be found in [2 - 4].

Material characterization and advanced calculation approach

The material under consideration belongs to a PA66-GF50 specialty family featuring enhanced mechanical characteristics compared to standard products. In particular, the base formulation was modified to enhance the tensile strength and deformation at break, especially when weld lines were present. The weld lines are often the weakest place in the moulded part. These improved characteristics are summarized in the graphs in Fig. 1.



+ 12 % tensile strength at break
+ 20 % tensile strain at break

+ 20 % tensile strength at break
+ 30 % tensile strain at break

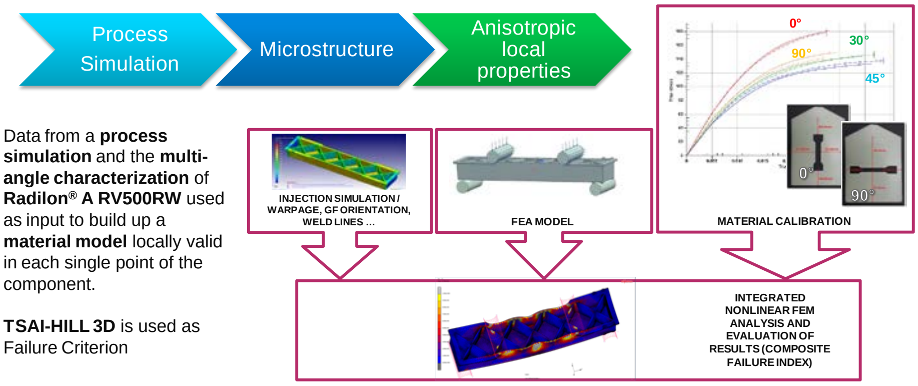
Fig. 1: Stress strain curves of Radilon ARV500RW

In order to perform the advanced structural calculations, measurements of the characteristics of the base polymer material and the short glass-fibre reinforcement were performed, as well as the characteristics of the samples cut out of rectangular plates at different angles with respect to the direction of melt flow. Special care was paid to the quality of the cut and the sample position, which could have a large effect on the results of measurements, based on the numerous tests performed. Material characterization was done taking into consideration the stress-strain curves at 4 cutting angles: 0°, 30°, 45° and 90°. Fig. 2 shows a chart of the approach used, consisting of the following sequence of steps:

Step 1: Simulation of the moulding process (using appropriate software packages), which provides, among other things, the glass-fibre orientation tensor at each point and the position of the weld lines.

Step 2: Generation of a micro-mechanical model of the material with calibration of the model using the experimental results obtained for samples cut from plates at different angles.

Step 3: Mapping of the local properties of the material, using appropriate software packages, taking into account the morphology induced by the simulated moulding process. Mapping is performed for each point of the mesh that will be used later on for the non-linear structural calculations.



Data from a **process simulation** and the **multi-angle characterization** of **Radilon® A RV500RW** used as input to build up a **material model** locally valid in each single point of the component.

TSAI-HILL 3D is used as Failure Criterion

Fig. 2: Flowchart illustrating the material characterization for TPM

Description of the component

The component studied in this work is a beam having an open C section and a pattern of internal reinforcing ribs. The part is shown in Figure 3.

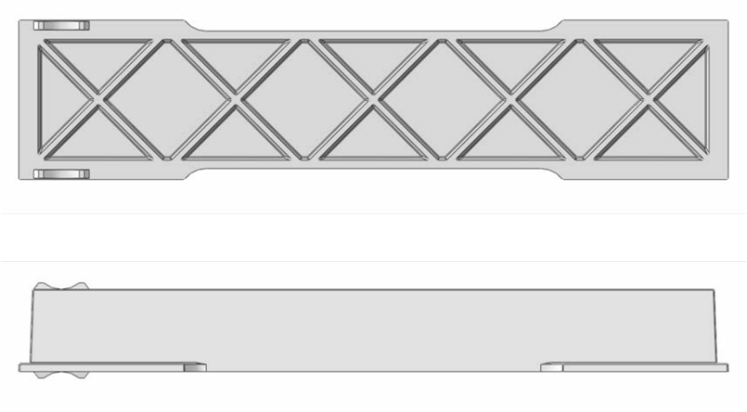


Fig. 3: Shape of the component

The component is manufactured by injection moulding and the mould is designed to use different and multiple injection gates, as shown in Figure 4.

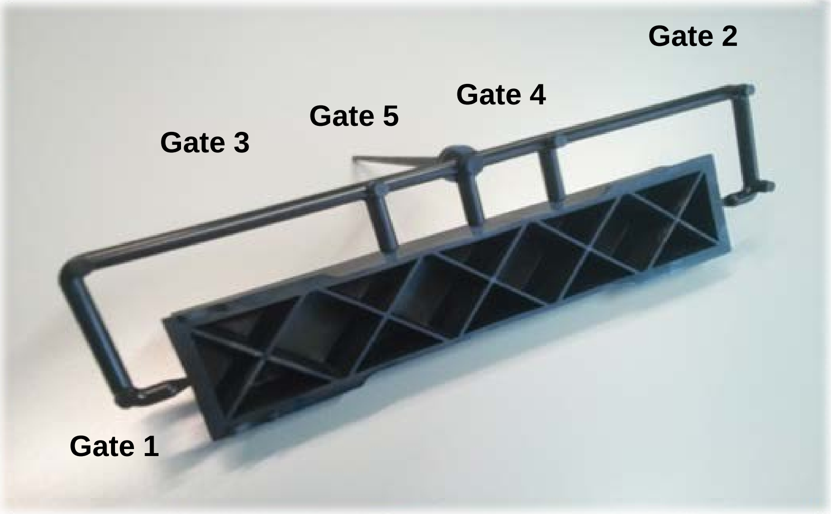


Fig. 4: Configuration of the different possible injection channels and position of the gates

Experimental tests

The part was designed to be subjected to bending tests. A four-point bending configuration was chosen for the first tests presented herein. The setup is shown in Figure 5. It comprises a four-point bending fixture and a deflectometer, which accurately measures the deflection of the part at the centre point (readings of the crosshead displacement transducer would be affected by the compliance of the rig). The fixture is mounted on a MTS RF150 testing machine, equipped with a 150 kN load cell.

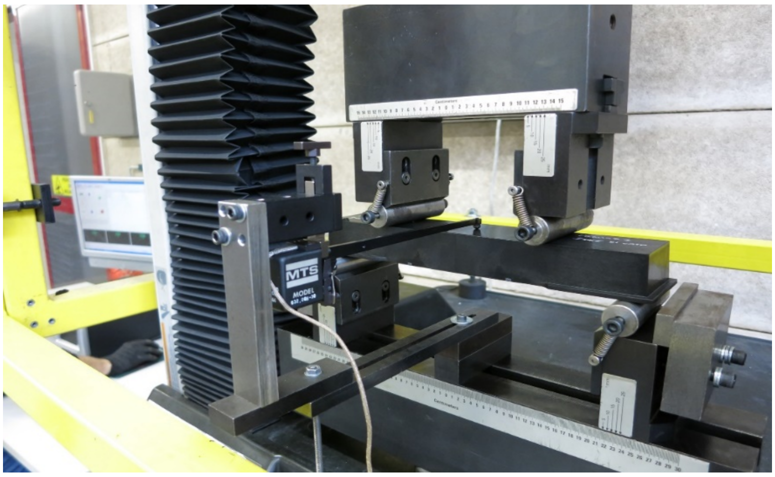


Fig. 5: Four-point bending setup

The test was conducted in displacement control mode at a speed of 5 mm/min up to failure. Failure load and mode were different for different injection configurations.

In this article, the results of four configurations are presented:

a) Longitudinal injection (gate 2)
b) Double longitudinal injection (gates 1 and 2 simultaneously)
c) Side injection (gate 5)
d) Double side injection (gates 3 and 4 simultaneously)

Results of the experimental tests

Three specimens for each injection configuration were tested. The results are presented in Figure 6 as superimposed load-displacement curves of one specimen for each configuration. It appears that, due to the different fibre orientation patterns, the stiffness is modified as the injection configuration changes and, additionally, the strength is affected, also because of the presence of weld lines.

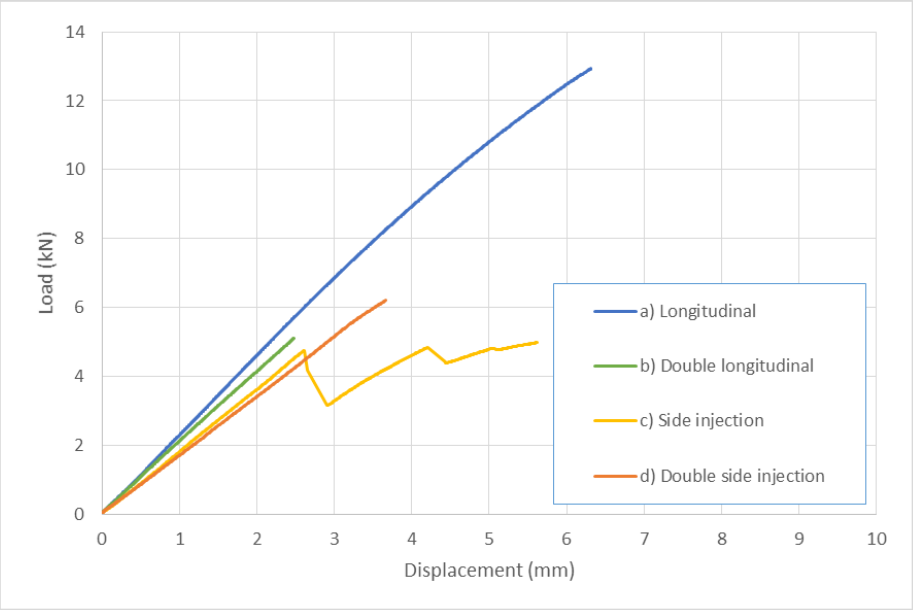


Fig. 6: Force-displacement curves

FE models

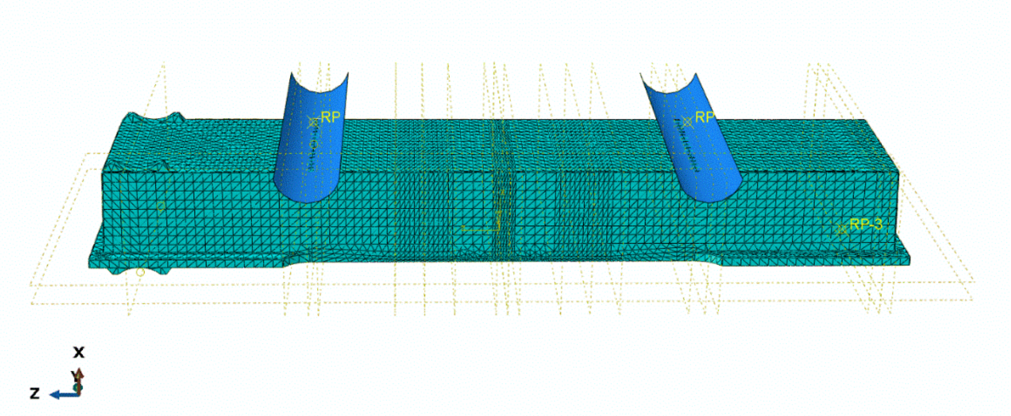


Fig. 7: Structural finite element model

The tests were simulated using Simulia Abaqus 6.14.3 Finite Element (FE) software. The FE model is shown in Figure 7. The elements are quadratic tetrahedra and the mesh size was varied in order to capture local stress fields accurately in the highly stressed regions, i.e., at the intersection of the inner ribs with the outer walls, as shown in Figure 8. Material properties were defined on the basis of process simulation analyses conducted using Moldex 3D software. Local material properties were evaluated using Digimat software, after mapping fibre orientation results from the process simulation mesh to the structural mesh using the Digimat MAP package. The material properties were defined by the material characterization presented in the previous section and are available in the Digimat MX database. The material model is a non-linear model with plasticity and the failure condition is defined by a strain-based Tsai-Hill criterion. A variable FI (Failure Index) is evaluated at each node. A value of FI equal to or greater than 1 indicates local failure.

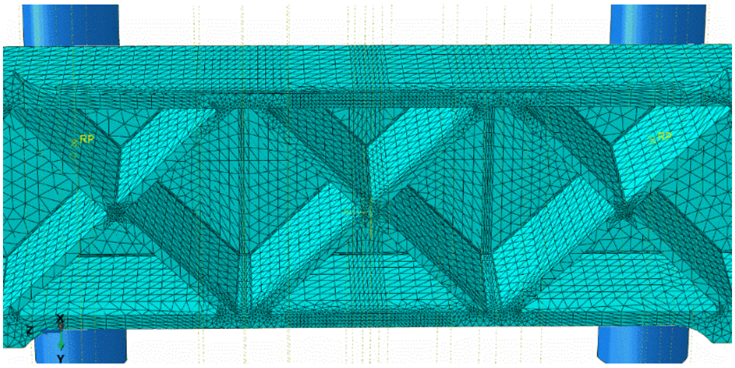


Fig. 8: Detail of the mesh refinement in the highly stressed regions

Incremental displacement of up to 8 mm with a maximum time increment of 0.05 was imposed on the rollers. The rollers were modelled as rigid surfaces, which were constrained to the model of the part through a contact interface. The presence of the lower rollers was simulated by constraining the vertical displacement of a reference point linked through a kinematic coupling to a surface corresponding approximately to the expected contact surface.

Results of the FE analyses

Four different models were built corresponding to the four fibre orientation patterns, as obtained by the four different injection configurations. The results are presented in terms of load-displacement curves, where load is defined as the sum of the reaction forces of the upper rollers, and the displacement was recorded at the node corresponding to the contact point between the part and the arm of the deflectometer.

Loads and displacements are displayed up to the time of failure, defined as the time increment during which a value of the Failure Index of 1 is found at the highly stressed location. A typical map of the failure index is shown in Figure 9.

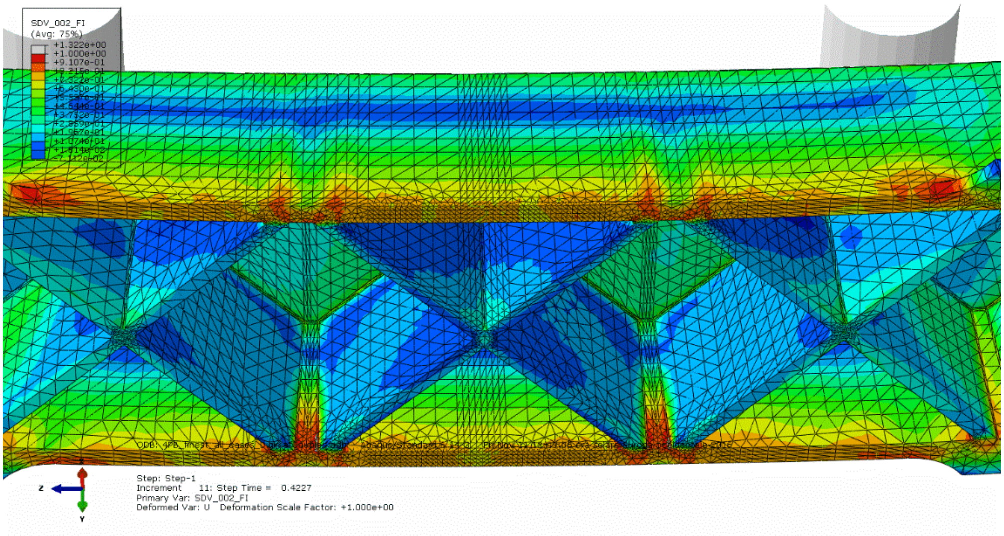


Fig. 9: Failure Index map

Load-displacement curves obtained with the four models are reported in Figure 10. The corresponding experimental load-displacement curves are superimposed on the results of the numerical simulation.

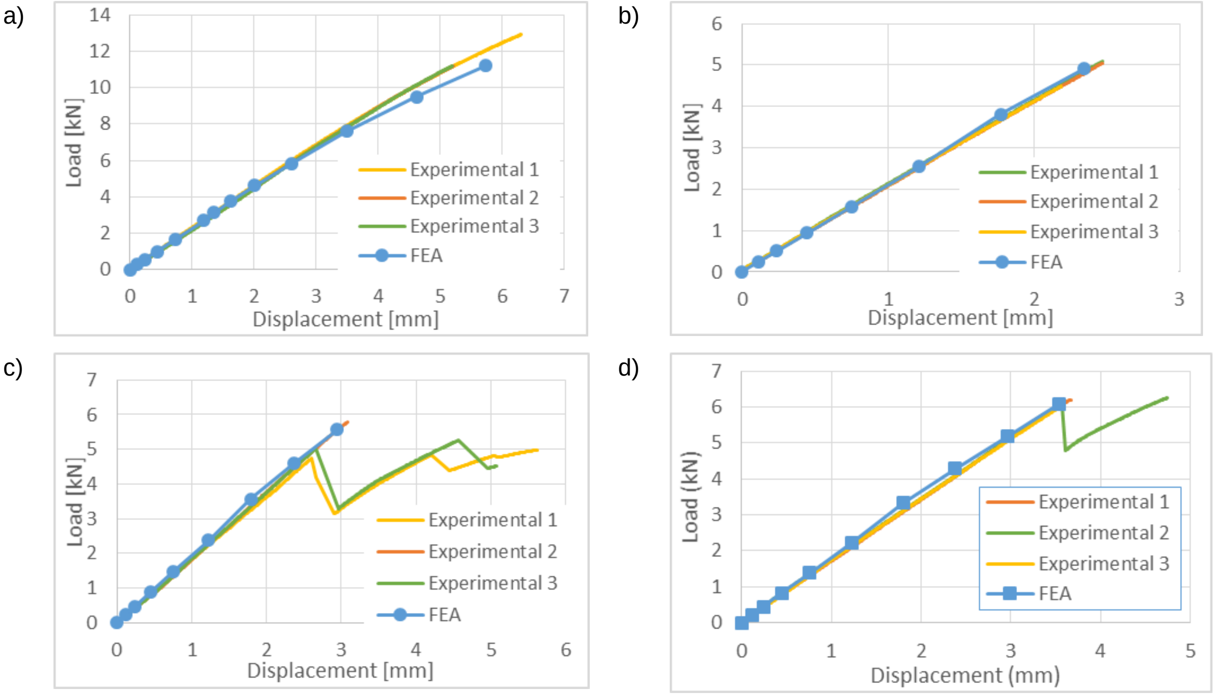


Fig. 10: Comparison of the simulated and the experimental load-displacement curves for all four cases: a) Longitudinal injection; b) Double longitudinal injection; c) Side injection; d) Double side injection

It appears that the FE models allow for correctly capturing the stiffness and the strength of the parts for all four injection configurations. It is interesting to compare the results obtained for the longitudinal configuration by TPM and the results that can be obtained by conducting FE analysis assuming an isotropic material with plasticity, as defined by tensile tests on standard ISO 527-2 specimens. This approach is often adopted for preliminary analysis. In this case, it is often customary to reduce the properties (elastic modulus and the plastic strain-yield stress curve) by a certain empirical reduction factor (e.g., by 1/3), dictated by the experience of the designer, in order to introduce the effect of possible misalignment of the reinforcing fibres with respect to the principal stresses.

The load-displacement curves obtained by modelling the material as isotropic, with and without empirical reduction, are shown in Figure 11, where the experimental curve and the numerical one obtained by TPM are superimposed, as well. The agreement of the TPM results with the experimental results is much better than that of the isotropic models in terms of both stiffness and strength. The isotropic model correctly captures the maximum load but overestimates the stiffness, while the isotropic model with empirical reduction fails to estimate accurately both the stiffness and the strength. By assuming an orthotropic material whose properties are related to the fibre orientation distribution in the part, the stiffness and the strength of the component are better evaluated.

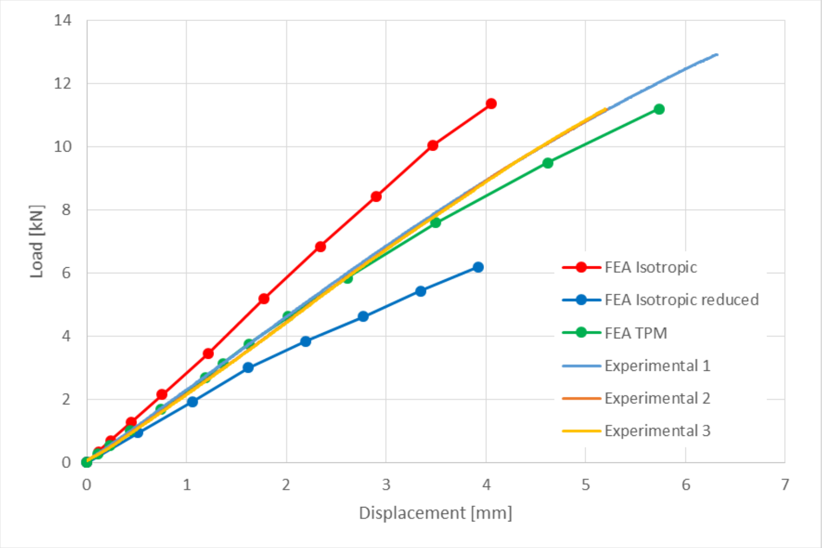


Fig. 11: Comparison of the experimental load-displacement curve for longitudinal injection with results obtained by FE with an isotropic material and FE with TPM

Comparison of isotropic and anisotropic models is not meaningful in the case of the other injection moulding configurations, as the isotropic models are insensitive to variations in the orientation patterns and to the presence of weld lines. In spite of the excellent agreement between TPM simulations and experimental results presented herein, the analysis of injection moulded parts containing weld lines is still an open issue. In fact, for similar simulations conducted with other fibre-reinforced materials, we observed larger differences between numerical and experimental results. More investigation in the simulation of the mechanical behaviour of weld lines, based on accurate observations of their microstructure as in [5], is needed. We believe that the component presented herein could be used as a demonstrator for different possible solution techniques in the future.

Conclusion

In this paper we presented the design of an injection moulded ribbed beam to be used as a demonstrator for different modelling techniques. The demonstrator is designed to be injection moulded using different combinations of injection channels and gates, thus allowing for obtaining different fibre orientation patterns and varying the position of weld lines.

The beam was tested under four-point bending and the tests were simulated by TPM, i.e., by combining process simulation with structural analysis. The results of the simulations were in good agreement with experimental results and constitute an improvement over simplified modelling techniques, such as those assuming an isotropic material.

Bibliographic references

[1] Bernasconi A, Davoli P, Basile A, Filippi A. Effect of fibre orientation on the fatigue behaviour of a short glass fibre reinforced polyamide-6. International Journal of Fatigue 2007, 29: 199–208.

[2] Castagnet S, Nadot-Martin C, Bernasconi A, Lainé E, Conrado E, Caton-Rose P. Through Process Modeling for the fatigue life assessment of notched injection-molded specimens, 16th European Conference on Composite Materials (ECCM 16), Seville, Spain, (2014).

[3] Moesenbacher A, Brunbauer J, Pichler P, Guster C, Pinter G. Modelling and validation of fatigue life calculation method for short fibre reinforced injection moulded parts. In: ECCM 16; 2014; Seville.

[4] Guster C, Pinter G, Mösenbacher A, Eichlseder. Evaluation of a simulation process for fatigue life calculation of short fibre reinforced plastic components. Procedia Eng 2011;10:2104-2109.

[5] Ayadi A, Nouri H, Guessasma S, Roger F. Large-Scale X-Ray Microtomography Analysis of Fiber Orientation in Weld Line of Short Glass Fiber Reinforced Thermoplastic and Related Elasticity Behavior. Macromol. Mater. Eng. 2016, DOI: 10.1002/mame.201500463.