EMAS Engineering Modelling, Analysis & Simulation THE NAFEMS JOURNAL
NAFEMS
Vol. 1, Issue 1, 2024

A Mean-field Homogenization Method-based Design of Experiments for Fiber-reinforced Plastics

Y. Shi[a,1], W. J. He[a], H. Ji[a], V. Deshpande[a]

[a]Dassault Systèmes SIMULIA Corp., 1301 Atwood Ave, Ste 101W, Johnston, RI 01929, USA

Abstract

Plastic materials are widely used as lighter alternatives to metals in all industries. Chopped fibers are often added as stiffeners during injection molding to increase the strength and durability of the part. The mechanical properties of fiber-reinforced plastics often exhibit a strong heterogeneity and anisotropy, which largely depends on how fibers are oriented during the injection molding process. The mean-field homogenization method is a widely used multiscale approach for modeling fiber-reinforced plastics. Using this approach, we have developed an integrated workflow of sequentially coupled plastic injection-to-structural analyses, taking into account of the fiber orientation results to better characterize the mechanical properties of the composite. In this study, we conducted two design of experiments analyses on a fully parameterized mounting boss model to study the effects of part thickness and gate location on the mass and maximum stress of the part.

Keywords

short-fiber reinforced plastics, manufacturing, injection molding, FEA, DOE, anisotropy, mean-field homogenization, multiscale

1 Introduction

Fiber-reinforced plastic (FRP) materials are widely used in our daily life, from consumer packages to home appliances, from automobiles to commercial airplanes, just to name a few. FRPs are used as a lighter alternative to metals, with lower costs, comparable durability, better corrosion resistance, and better manufacturing versatility.

To increase the strength and durability of plastic parts, chopped fibers are added to the resin as stiffeners when parts are injection molded. When the injection gates and runner system are not carefully designed, manufacturing defects can be introduced. If parts are not properly cooled down before being ejected out of the mold, significant residual stress can rise and cause parts to warp. In addition, the mechanical properties of FRPs often exhibit a strong anisotropy and temperature-dependency, which largely depends on how fibers are oriented during thermal setting. Understanding the effects of geometric and manufacturing parameters on the material responses of the molded part under service loads is crucial to finding the best design to meet performance requirements. These issues pose unique challenges to the design and manufacturing of FRPs.

Previously, a fully integrated workflow was developed for modeling FRPs with mean-field homogenization (MFH) method-based multiscale models. Automated through a simulation process, this workflow starts with part design and includes an injection molding simulation followed by a structural finite-element analysis (FEA). The fiber orientation distribution predicted by the injection molding

[1]Corresponding author.
E-mail address: yunfei.shi@3ds.com
https://doi.org/10.59972/3w6s4hgz

simulation is used in the MFH-based multiscale models to accurately characterize the mechanical behaviors of FRPs.

In this study, we extended this workflow to design of experiments (DOE) to explore design alternatives. Using a plastic mounting boss as an example, the effects of the part thickness and injection gate location on the maximum von Mises stress in service have been studied. The best design has been obtained by minimizing the stress and weight together among 11 design points.

2 Methods and Computational Models

For this study, a fully parameterized CAD model was created for a mounting boss. First, plastic injection simulation was conducted to assess the manufacturability and obtain the fiber and material orientations in the molded part. A subsequent structural FEA was conducted with composite material properties evaluated using the MFH method by taking account of the material orientation results. Then, a DOE study of the sequentially coupled plastic injection-to-structural analysis was carried out. This end-to-end workflow was created and executed using **3D**EXPERIENCE (R2023x, Dassault Systèmes, Vélizy-Villacoublay, France) applications [1].

The geometry of the boss model includes a hollowed cylinder surrounded by a three-sided wall (Figure 1). For the plastic injection simulation, a small circular surface of a protrusion was specified as a single injection gate. The part geometry was fully parameterized with 15 knowledge-ware parameters, including two key design variables – thickness of the cylinder (*t*) and offset of the injection gate with respect to the axis of the cylinder (*d*).

The part is assumed to be made of Ultramid A3EG10 (PA66-GF50, BASF, Ludwigshafen, Germany), a glass fiber-reinforced polyamide. The material properties for the plastic injection simulation were taken from the material database provided by the platform (Table 1). For the injection molding setting, the default values computed by the application were used directly. Over 7,000 tetrahedron elements with a size of 1.5 mm and two boundary layers were used for the injection simulation for Filling and Packing & Cooling analyses.

For the structural FEA, we consider inserting a discrete rigid surface into the cylinder portion of the boss model while clamping its base (Figure 2). The rigid surface, which has a slightly larger diameter than the cylinder, is inserted at a constant speed of 0.5 mm/s for a duration of 0.1 s, causing the cylinder to deform radially and expand circumferentially. The Explicit dynamic procedure was used for solving the problem with a variable element mass scaling that targets a stable time increment of 1e-6 s. Two sets of mesh were used in the DOE analysis: a coarse mesh consists of approximately 2,000 hexahedron solid elements and 2,000 quadrilateral shell elements, and a fine mesh consists of approximately 13,000 hexahedron solid elements and 9,000 quadrilateral shell elements.

To better account for the effects of material orientation and fiber dispersion on the mechanical properties of the composite material, an MFH-based multiscale material model [3] consisting of a matrix and a prolate-shaped inclusion was used for the structural FEA. A second-order material orientation tensor was obtained at each element integration point by mapping the fiber orientation predicted by the injection simulation onto the structural mesh. The Mori-Tanaka formulation was used for homogenizing the strains at the macro- and micro-levels. As a first-order approximation, the matrix and fiber materials are assumed isotropic elastic. For glass fiber, the Young's modulus and Poisson's ratio are known, and its in-situ volume fraction and aspect ratio in PA66-GF50 were measured previously [4] (Table 2). The mechanical properties for the matrix were calibrated by matching the engineering constants of the composite with a 0º fiber orientation (see Table 1) from the material database.

The DOE analysis was done through a simulation process consisting of two physics simulation adapters (Figure 3). Thanks to the sequential multiphysics capability [1] supported recently, the material and fiber orientations given by the plastic injection simulation can be directly used in the downstream structural FEA. Changes in design geometry were driven by the two design parameters (*d* and *t*) as the inputs, and the mass (*m*) and maximum value of the von Mises stress (σ) of the cylinder were chosen as the outputs to be minimized together. Two DOE jobs were executed to explore eleven design points generated using the Latin Hypercube Sampling method (Table 3 and Table 4). The coarse mesh was used in the first DOE analysis for fast execution to validate the workflow, and the fine mesh was used in the second DOE analysis for better accuracy.

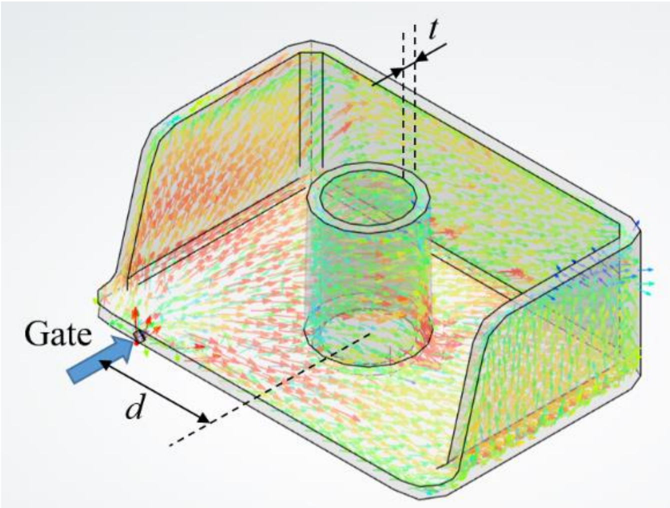

Figure 1. Design parameters (*d* = gate offset, *t* = thickness) and plastic injection simulation result of fiber orientation (arrows).

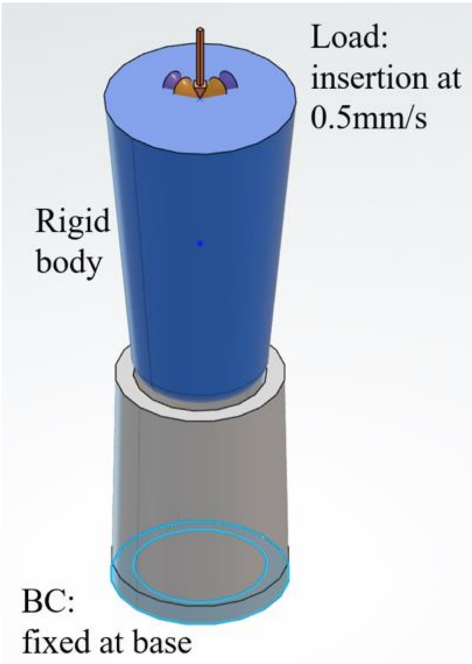

Figure 2. Structure simulation definitions.

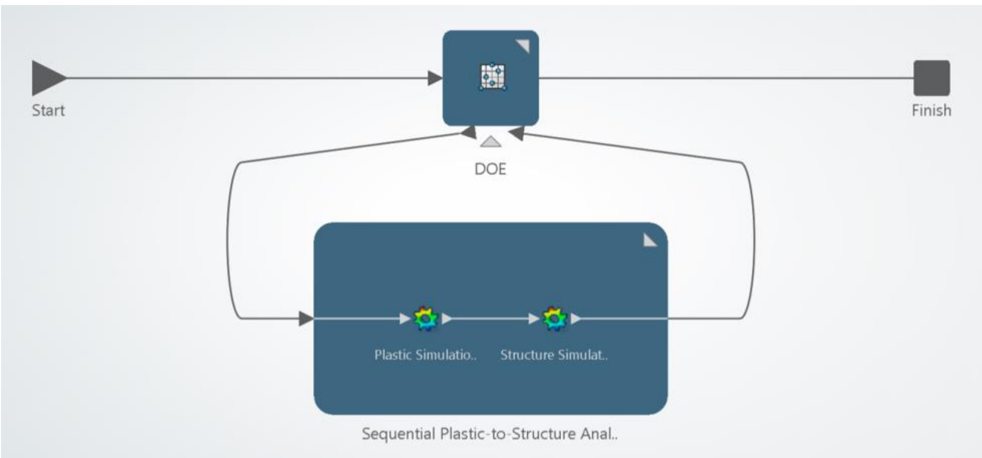

Figure 3. Simulation process for DOE analysis.

Table 1. Material Properties for PA66-GF50 in Plastic Injection Simulation.

General	density		1510	kg/m^3
Thermal	conductivity		0.323	W/(m·K)
	specific heat		2.956	kJ/(kg·K)
	coefficient of thermal expansion	(parallel)	1.2e-5	K^{-1}
		(normal)	5.6e-5	K^{-1}
Solid	Young's modulus	E_1	20.0	GPa
		E_2	5.8	GPa
	shear modulus	G_{12}	4.6	GPa
		G_{23}	1.9	GPa
	Poisson's ratio	v_{12}	0.38	
		v_{23}	0.57	
Fluid	melt temperature		573.15	K
	mold temperature		363.15	K
	glass transition temperature		523.15	K
	part ejection temperature		498.15	K
	viscosity (cross-WLF model) [2]	D_1	3.44E+15	Pa·s
		D_2	323.1	K
		D_3	0	K/Pa
		A_1	37.07	
		A_2	51.6	K
		τ	1.83E+5	Pa
		n	0.258	
	max. shear rate		6.0E+4	s^{-1}
	max. shear stress		4.95E+5	Pa
	PVT (modified Tait model) [2]	b_{1m}	6.973E-4	m^3/kg
		b_{2m}	6.968E-7	$m^3/(kg·K)$
		b_{3m}	3.366E+8	Pa
		b_{4m}	0.003	K^{-1}
		b_{1s}	6.973E-4	m^3/kg
		b_{2s}	1.704E-7	$m^3/(kg·K)$
		b_{3s}	2.452E+8	Pa
		b_{4s}	0.003	K^{-1}
		b_5	496.059	K
		b_6	4.671	K/Pa
		b_7	0	m^3/kg
		b_8	0	K^{-1}
		b_9	0	Pa

Table 2. Constituents' Properties in Multiscale Material Model.

Matrix		Fiber			
E_m	v_m	E_f	v_f	volume fraction	aspect ratio
3.68 GPa	0.33	72.0 GPa	0.22	0.31	18.0

Table 3. First DOE Analysis with Coarse Mesh.

Design	Gate offset d (mm)	Thickness t (mm)	Mass m (g)	Max. stress σ (GPa)	Execution time (min)
1	0.	1.16	0.386	1.400	6.4
2	1.	0.92	0.299	0.882	4.4
3	2.	0.88	0.285	0.867	4.3
4	3.	1.12	0.372	1.282	5.2
5	4.	1.04	0.343	1.096	5.2
6	5.	1.20	0.401	1.243	4.8
7	6.	0.84	0.271	0.788	4.4
8	7.	0.96	0.314	0.750	5.5
9	8.	0.80	0.256	0.782	5.0
10	9.	1.08	0.357	1.103	5.3
11	10.	1.00	0.328	0.940	5.7

Table 4. Second DOE Analysis with Fine Mesh.

Design	Gate offset *d* (mm)	Thickness *t* (mm)	Mass *m* (g)	Max. stress *σ* (GPa)	Execution time (min)
1	0.	1.16	0.388	1.230	29.6
2	1.	0.88	0.287	0.947	19.3
3	2.	0.96	0.316	0.932	25.9
4	3.	1.20	0.402	1.222	29.2
5	4.	0.84	0.273	1.058	20.9
6	5.	0.92	0.301	1.020	23.5
7	6.	1.12	0.374	1.045	28.5
8	7.	1.08	0.359	1.060	26.7
9	8.	0.80	0.258	1.084	19.2
10	9.	1.00	0.330	0.933	25.3
11	10.	1.04	0.345	0.994	24.0

3 Results and Discussion

Table 3 and Table 4 include the inputs, outputs, and execution time for each job in the two DOE analyses. The gate offset *d* changes between 0 and 10 mm with an increment of 1 mm, and the thickness *t* changes between 0.80 mm and 1.20 mm with an increment of 0.04 mm. Because design points are generated independently during each execution, the design points for these two DOE analyses are not identical.

With a four-cored CPU (Intel® i7-8850H, 2.60 GHz), the first DOE jobs took 5.1±0.6 min on average to complete, whereas the second DOE jobs took 24.7±3.8 min, almost as five times long. In both analyses, the mass *m* increases with the thickness *t* because the height of the cylinder is kept constant. The maximum von Mises stress *σ*, on the other hand, depends on both *t* and *d*. For isotropic linear elastic material, *σ* is expected to increase with the thickness *t* (i.e., a decrease in inner radius while keeping the outer radius constant). However, this is not necessarily true when fiber orientation results are used for characterization of the anisotropic behaviors of the composite.

To understand the effects of *d* and *t* on the fiber orientation and stress distributions, results of the second DOE analysis were studied closely. Figure 4 includes the results for three of the eleven design points. With the second smallest mass and the third smallest stress, design 2 is ranked as the best design; with the largest mass and the second largest stress, design 4 is ranked the worst design; and design 9 is chosen as an intermediate design.

As shown in the fill time plots, the plastic flow starts from the injection gate and propagates toward the far end of the wall. When the flow passes across the cylinder, it initially separates into two, and then the two flow fronts quickly merge at the back side of the cylinder. As a result, the plastic flow leaves air traps at the edge of the cylinder as well as at one corner of the wall that is filled the last. It can also form weld lines at the back side of the cylinder, creating potential weak spots in the part. Near the injection gate, the fibers are dispersed in every direction. On the bottom plate and side walls where the flow is fully developed, the fibers are strongly aligned with the local flow direction. In the cylinder, the fibers are primarily along the circumferential direction. In regions where two flow fronts collide, such as the back side of the cylinder and corners of the wall, fiber orientation becomes relatively random. The degrees of fiber alignment are indicated by the colors in the fiber orientation plot, where red represents perfect alignment and blue represents random orientation in 3D.

In the structure analysis, the mapped first principal directions are qualitatively consistent with the fiber directions predicted by the plastic injection simulation. The maximum von Mises stress occurs at the edge of the cylinder’s inner surface, where the rigid surface first contacts the cylinder. The fiber orientation plot for the design 4 shows a higher degree of circumferential alignments compared with the other two designs. This can contribute to the higher value for the peak stress observed in the design 4.

It is worth noting that the peak stress in design 2 coincides with a weld line, increasing the risk of material failure. Future work of DOE studies to include more realistic and complex material models for the composite, such as plasticity, damage, and failure, is warranted.

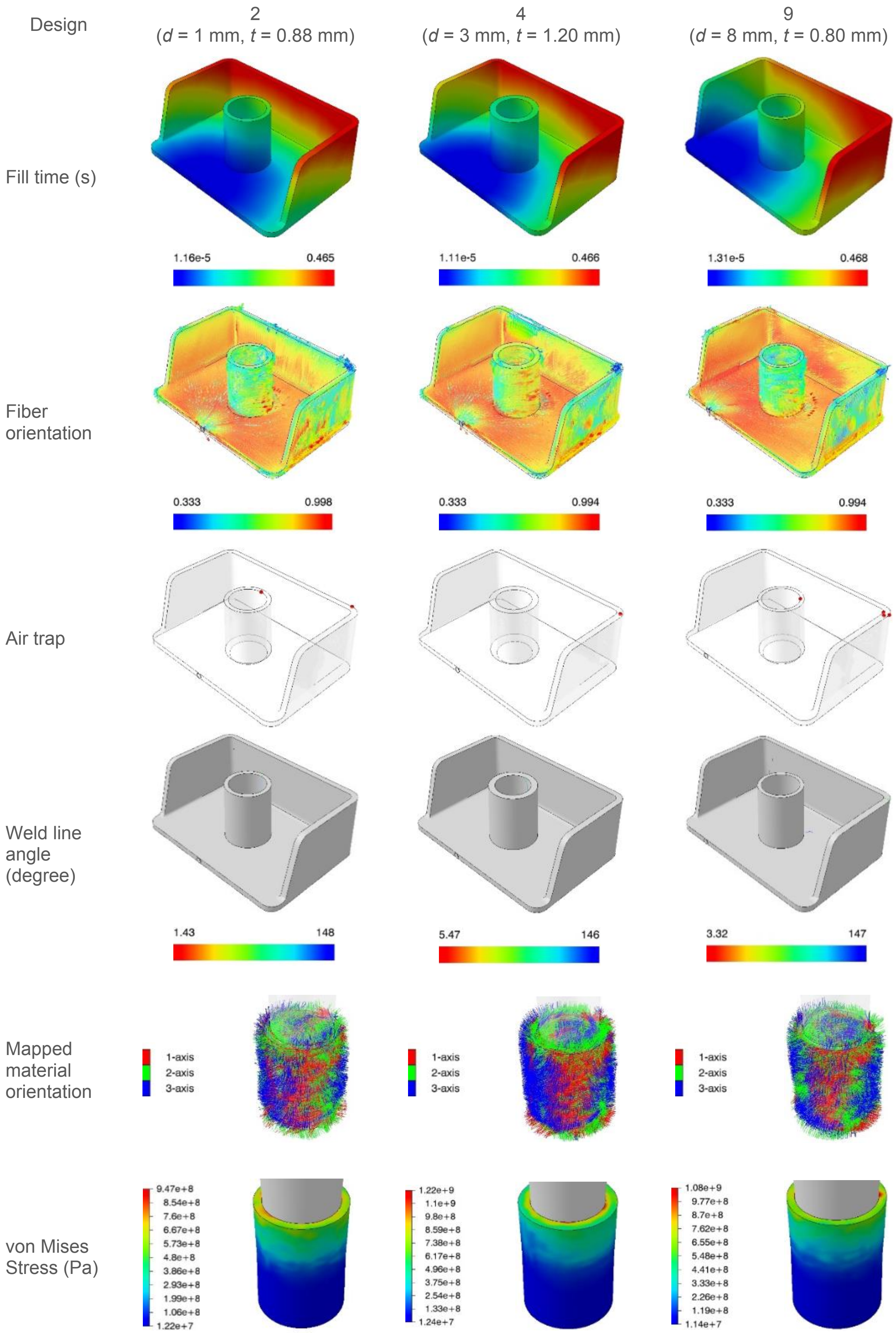

Figure 4. Plastic injection and structural simulation results of second DOE analysis.

4 Conclusion

We have developed a DOE workflow of sequentially coupled plastic injection-to-structural analyses of FRPs, leveraging the fiber orientation results and the MFH-based multiscale material model to better characterize the behaviors of the molded part. We have conducted two DOE analyses for a fully parameterized mounting boss model to evaluate the mass and structural performance for 11 design alternatives.

5 References

[1] Dassault Systèmes. ***3D**EXPERIENCE User Assistance R2023x* (2022). Available: https://help.3ds.com/2023x/English/DSDoc/FrontmatterMap/DSDocHome.htm

[2] Autodesk. *Moldflow Insight 2023*. Available: https://help.autodesk.com/view/MFIA/2023/ENU/

[3] Dassault Systèmes. *SIMULIA User Assistance R2023x* (2023). Available: https://help.3ds.com/2023/English/DSSIMULIA_Established/SIMACAEMATRefMap/simamat-c-meanfieldhomogenization.htm

[4] C. Xue, M. Chu, Y. Yang, J. Yang, and H. Hamada, "A Study on the Analysis and Prediction of the Anisotropy of Injection Molded GF/PA66," *Journal of Physics: Conference Series*, vol. 1790, no. 1, p. 012055, Feb. 2021, doi: 10.1088/1742-6596/1790/1/012055.