Composites Modeling Capabilities of Abaqus

Hicham Farid

Aventec Inc. 327 Renfrew Drive, Suite 301 Markham, ON L3R 9S8 Canada

www.aventec.com

Abstract: *Composites occupy a noticeable place in the materials industry. Their characterization during the forming process as well as during their life cycle in different industrial fields impose an advanced knowledge of their mechanical and thermal behavior.*

Different industrial processes are considered in the forming of a composite sheet into a desired shape (blow molding, thermoforming etc.). Generally, the deformation during these processes is very fast, non-uniform, multiaxial and occurs at temperatures above the glass transition temperature. The modeling of this phenomenon remains delicate and creates several difficulties since it involves contact, large deformations, as well the implementation of nonlinear constitutive laws (hyperelastic, viscoelastic, etc.).

The present work summarizes the capabilities of Abaqus for modeling the thermoforming of composites (short and long fibers). A presentation of the Composite Modeler Plugin shows a robust approach to define different plies, layups, and offsets associated with a long fibers composite model. It also shows the great capability of simulating real time draping and the generated shear stresses. The Micromechanics Plugin offers a great tool for subscale modeling in order to simulate the fiber-matrix interaction using an FE-RVE (Finite Element model of a Representative Volume Element) approach, and the generation of the homogenized material parameters.

A presentation of material parameters calibration using Isight coupled with Abaqus illustrates how the two tools can be joined together to solve the classical identification problems associated with new materials or non-linear constative laws. Tosca structure Bead optimization capabilities are shown through the optimization based on natural frequencies. Fatigue analysis has been performed using Fe-safe and shows the fatigue life contours.

Keywords: *Composites, Thermoforming, FE-RVE, Hyperelasticity, Homogenization, Bead optimization, Material calibration, Fatigue.*

1. Introduction

Designing a composite part requires a rigorous knowledge of the material properties, as well as the working constraints and environment. Modeling the mechanical and thermal behavior of composites during the forming processes prove to be useful, even crucial in order to improve the quality of the final products (Michel, Moulinec, & Suquet, 1999).

Modeling the forming of composite materials requires the consideration of various nonlinearities. Often the processes include large deformations, contact between the composite sheet and the mold, without forgetting the nonlinear material models that are usually include hyperelastic (Gong, Peng, Yao, & Guo, 2016) or viscoelastic (Margossian, Bel, & Hinterhoelzl, 2016) material models. The consideration of all, or some of these nonlinearities proves to be challenging.

The present paper summarizes the capabilities of Abaqus as a tool to model mainly the Thermoforming process, considering all the nonlinearities associated with it, i.e. material, contact, and large deformations. Draping simulation of long fiber composites is presented using the Composite Modeler plugin. The tool allows to detection of high shear zone during the draping process.

2. Thermoforming simulation of a laminate composite using Abaqus

Generally, the composites industry requires a large number of experimental tests before reaching the mass production stage, which is usually expensive and demanding. The same challenges are always faced when introducing new products or improving existing products. Therefore, the numerical simulation becomes a go-to tool to improve the forming processes. However, the numerical simulation requires a good understanding of the mechanical behavior of used composites.

In the case of the thermoforming processes, composites are heated up to temperatures between their glass transition temperature and fusion temperature before being formed (Akkerman & Haanappel, 2015). The nature of this process induces various problems such as tearing of the sheet, irregularity of the thickness distribution, the instantaneous cooling of the sheet amid the contact with the cold mold, etc. The introduction of the real physical phenomena involved in the thermoforming process such as wrinkling, failure, material constitutive law, contact between different components, proved to be challenging, but required for a more realistic simulation of the process.

The thermoforming of a woven Carbon-PPS composite sheet is presented in this section. The theoretical development of the equilibrium equations within a continuum mechanics framework has been omitted in this presentation.

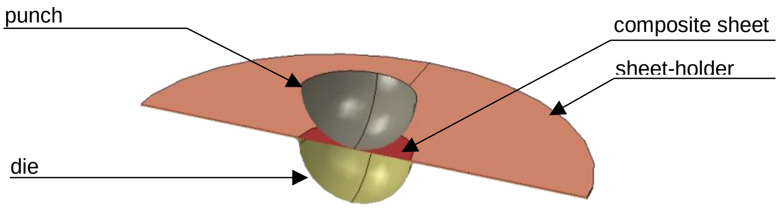


Figure 1. Components of the thermoforming process.

Figure 1 presents the model assembly for the thermoforming simulation of a long continuous fiber carbon-PPS composite sheet. The circular sheet has 1 mm thickness and 500 mm diameter. The composite layup was composed of three layers oriented at -45, 0, and +45°. A preload of 300 N was applied to the sheet-holder as compaction force. A displacement of 255 mm was applied to the punch with a strain rate of 1 m/s. The thermoforming temperature was set to 320 °C. The material's properties are presented in Table 1.

Table 1. Material characteristics for carbon-PPS composite

Elastic Properties	**Values**
E11 [GPa]	51.08
E_{22} [GPa]	41.34
ϑ_{12} [-]	0.03
G_{12} [GPa]	4.12
G_{13} [GPa]	3.5
G_{23} [GPa]	3.5

The contact between the sheet and the thermoforming tools (punch, sheet-holder, and the die) was modeled using general contact, other contact techniques and algorithms were investigated, but are not presented in the paper. The composite sheet was meshed using reduced integration shell elements (S4R and S3R), the mesh was refined in the effective thermoformed area for better results. The simulation was performed using the explicit solver of Abaqus (Abaqus/Explicit) using 8 CPUs, the simulation converged after 146.2 s CPU time.

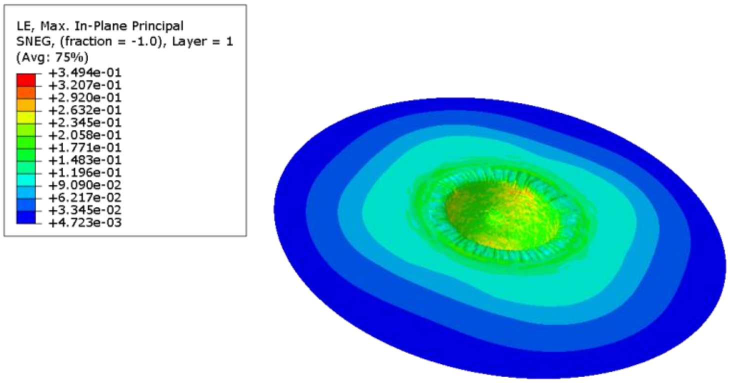


Figure 2. Logarithmic strains of the first layer (-45º)

Figure 2 shows the deformed configuration of the composite sheet. The obtained results are comparable with literature (Abbassi et al., 2011). We note that the strains are more significant in the diagonal direction compared to the median direction, this is typical for diagonally oriented composites.

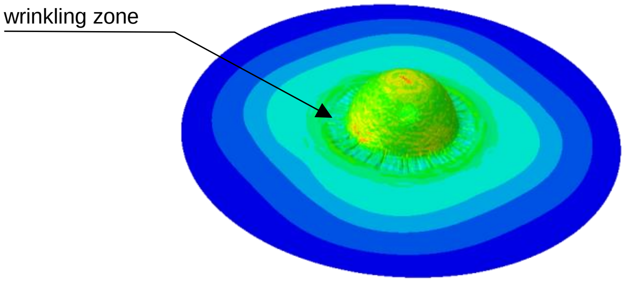


Figure 3. Wrinkling zone due to excessive thinning of the composite sheet

The presence of wrinkles is shown on Figure 3. The wrinkles depend on the initial orientation between plies. The thickness distribution of the composite sheet is shown in Figure 4. This parameter is very important for the forming processes of thin composite structures in order to prevent overstretching or failure. Zones with minimum thicknesses coincide with zones of large strains.

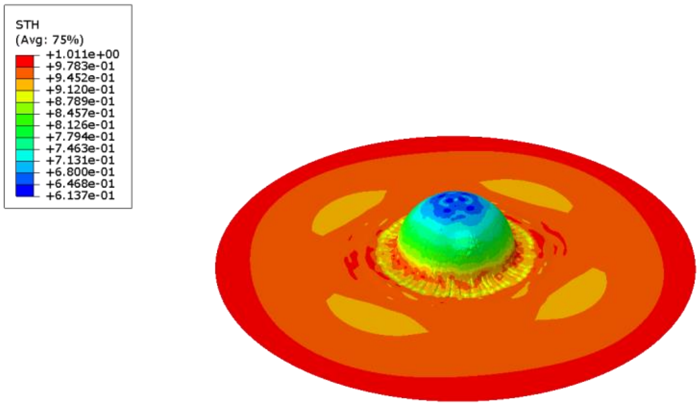
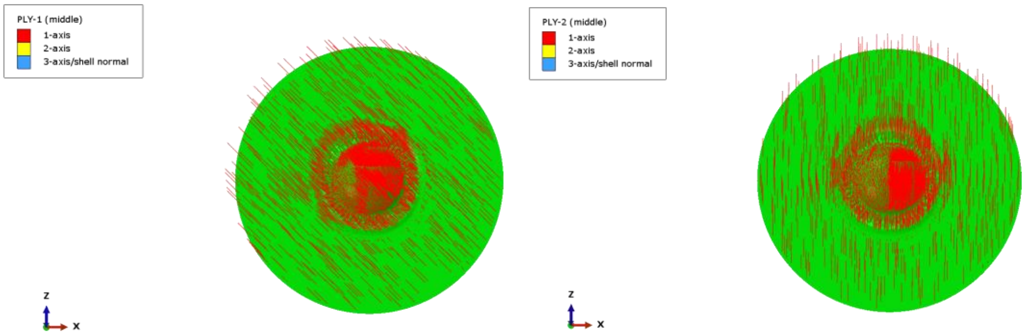


Figure 4. Thickness distribution

One main parameter that engineers seek during long fiber composites forming processes is the material orientation. Figure 5 illustrates the material orientation of two plies after the thermoforming process was performed. Far from the forming zone, the orientation of different plies remains the same, while the plies orientation is obviously affected in the forming zone. This valuable information is necessary in order to improve the quality of the composite final product.



(a) **(b)**

Figure 5. Material orientation: (a) ply oriented at -45º, (b) ply oriented at 0º

3. Layup definition and draping simulation using the Composite Modeler

One of the challenges encountered during the modeling of long fiber composites is the definition of different layups with respect to different plies orientations and materials definition. The Composites Modeler for Abaqus/CAE is an add-on product that allows the creation, definition, and manipulation of plies and layups. The Composites Modeler allows the user to define the accurate fiber angles and ply thicknesses. It helps the user to review and quickly modify the composite model to iteratively improve the design by real time simulations of draping, properties generation, and mapping.

A ply represents a piece of reinforcing fabric which is cut from sheet stock and placed on a mold during the manufacturing process. A ply is fully characterized by the sheet material it is made of, the area it covers, and the way in which it is applied to the surface. The manufacturing procedure is particularly important for non-developable surfaces where there are infinite ways of placing the fabric on a surface.

The Composites Modeler includes comprehensive fiber simulation to predict the manufacture of component effectively. For doubly-curved surfaces, there are infinite ways of covering the surface with reinforcing materials. How the ply is applied makes a difference to ply producibility and resulting fiber orientations. Figure 6 shows a draping simulation of composite layup on a hemispherical shell. The warp and weft fibers color coded by degree of shear in the fabric, where the red areas show above maximum shear strains, while blue areas show acceptable amount of shear.

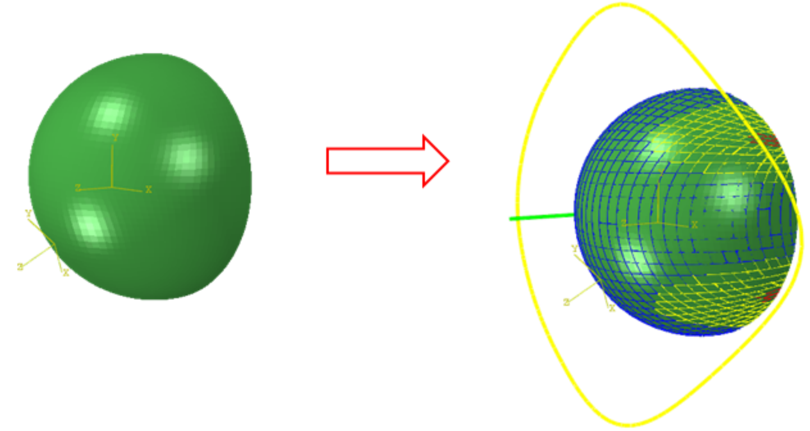


Figure 6. Draping simulation

In order to decrease the amount of shear, the user can use different techniques in order to control boundaries, splits, order of drape, seed curve, etc. these options allow the users to improve the quality of the final product even before running any stress analysis. Figure 7 illustrates a draping simulation with and without splits, the improvement of the shear zone is shown as well.

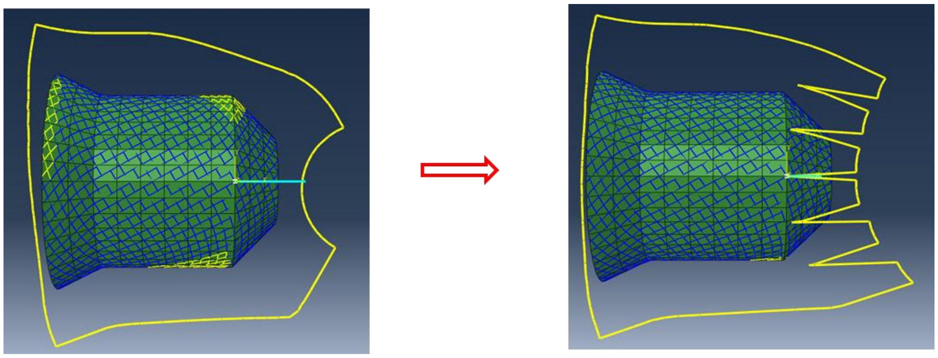


Figure 7. Draping simulation and improvement of shear strains using splits

The defined composite layups can be easily exported to Abaqus/CAE in order to perform any type of analysis (Stress, Thermal, Coupled thermal-stress, etc.). The layups are defined as sections, and could easily be manipulated within Abaqus/CAE.

4. Micromechanics and mean field homogenization

Composites are highly heterogeneous materials, even when modeled using linear material models, the interaction between fibers and the matrix prove to be challenging in terms of modeling and simulation, especially during forming processes where various nonlinearities are considered.

Multi-scale modeling provides an effective alternative to predict the composites behavior based on the microstructure influence (Wu, Noels, Adam, & Doghri, n.d.). The macroscopic properties can be extracted in order to provide a more realistic homogenous response of the material, a schematic of the method is illustrated in Figure 8. Therefore, a finite element model of a representative volume element (FE-RVE) of a material's microstructure can be used to predict the localized stress/strain fields between the fiber and the matrix. This approach is more realistic and not subjected to assumptions used in mean-field homogenization. Properties such as elasticity and thermal expansion can be obtained directly from the RVE and extrapolated to a larger scale (Ji, McLendon, Hurtado, Oancea, & Bi, 2017).

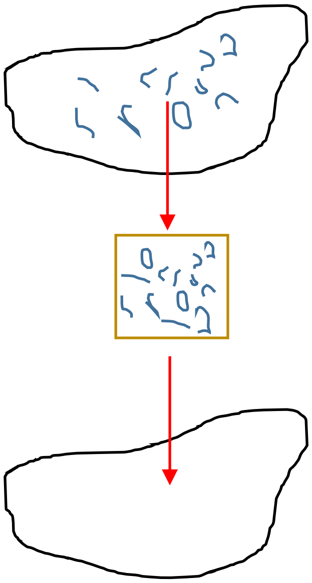


Figure 8. Micromechanics approach

An RVE is a volume of a composite that is large enough to yield the aggregate composite response. FE-RVE is loaded based on the far-field solution; the local solution field is obtained from finite elements. The FE-RVE plug-in for Abaqus/CAE allows for users to create microstructures and extract the homogenized properties of the composite material. These homogenized material properties can then be used to run simulations at larger scales. Figure 9 shows the Von Mises contour plot for a long fiber composite RVE.

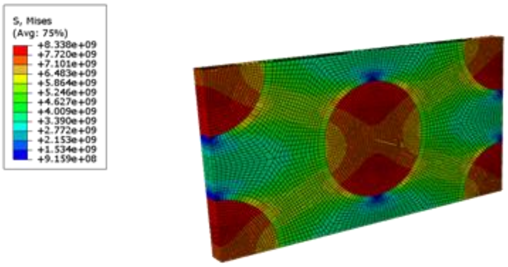


Figure 9. Von Mises stress contours in an RVE

5. Hyperelastic material parameters calibration using Isight

In the forming processes, i.e. injection molding, thermoforming, etc. composites are generally described using highly nonlinear material behavior laws (hyperelastic, viscoelastic, etc.). The material constants embedded within these laws can usually be determined using different optimization techniques in order to minimize the difference between the experimental data and the theoretical response.

However, the implementation of optimization algorithms such as Levenberg-Marquardt (Levenberg, 1944; Marquardt, 1963), or Powell's iterative algorithm (Fletcher, 1987) proved to be a tedious task.

Abaqus/CAE allows the evaluation of hyperelastic and viscoelastic material behaviors by automatically creating response curves using selected strain energy potentials. However the user has no control over the optimization algorithms.

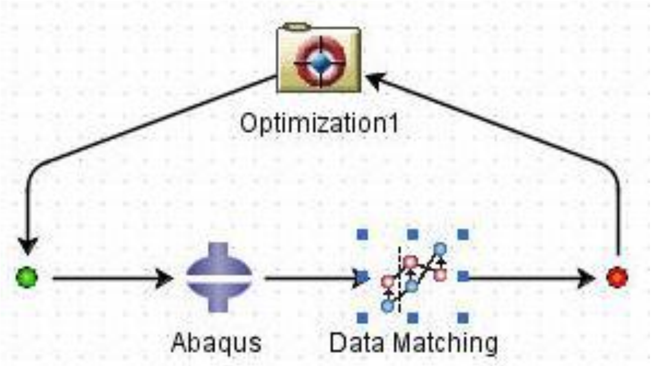


Figure 10. Experimental Data versus

Isight offers a great alternative in terms of material models calibration with a very straight forward approach. A large library of optimization algorithms is available within the software. The use of Isight shows great efficiency in identification problems, even with highly nonlinear hyperelastic material models. In the example shown in Figure 10, Isight was coupled with Abaqus in a loop in order to minimize the error between the experimental data and 3rd order Ogden hyperelastic material model for a rubber sample in uniaxial tension.

The calibration results are shown in Figure 11. The Isight results were compared with Abaqus identification results to show the order of accuracy.

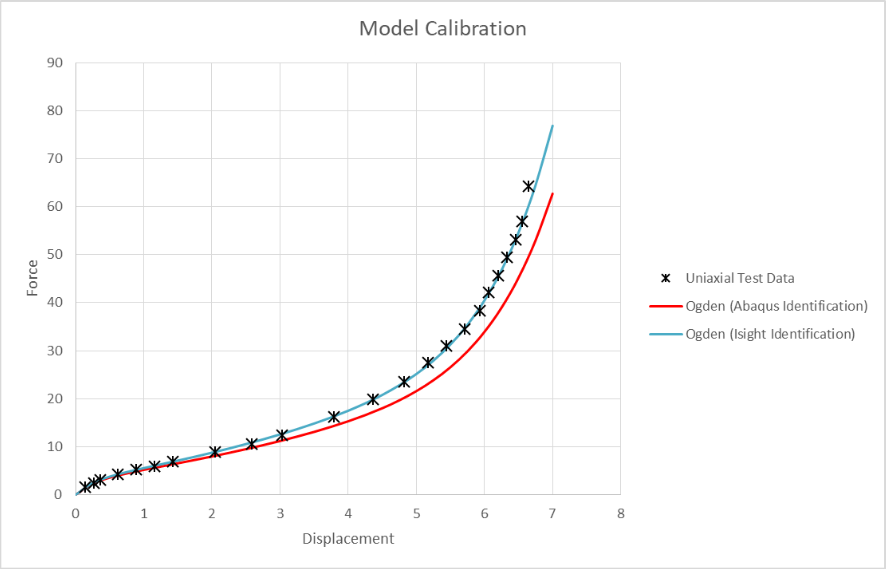
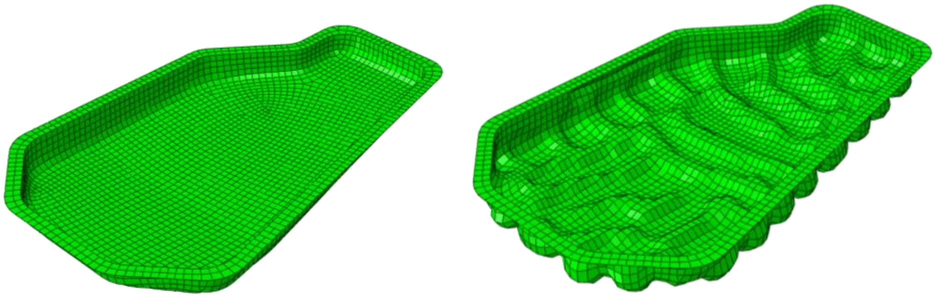


Figure 11. Experimental Data versus 3rd order Ogden hyperelastic material model for rubber

6. Bead optimization of a composite thin structure using Tosca structure

Thin composite structures have generally low stiffness, which affects significantly their performance under different environmental and structural conditions. Introduction of bead patterns increases the stiffness and eigenfrequency of these structures. Bead optimization offers the optimal bead layouts for improved static and dynamic properties of composite shell structures.

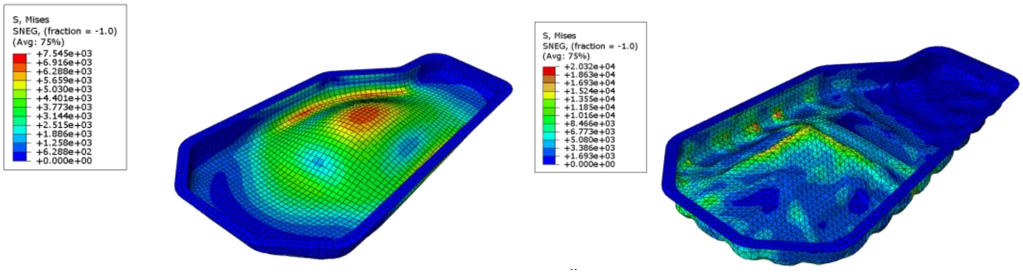
Tosca structure offers great capabilities in terms of nonparametric optimization, including topology, shape, size, and bead optimization including a wide range of nonlinearities. Taking into consideration different manufacturing restrictions, such as geometric restrictions, symmetries, demolding control, etc.



(a) **(b)**

Figure 12. Introduction of bead patterns in an oil pan, (a): the original oil pan design, (b): the optimized oil pan design

In the example shown in Figure 12, a bead optimization has been performed using Tosca structure in order to increase the eigenfrequency value of the first mode in an oil pan. Using Abaqus/CAE, modal frequency analysis was performed in order to extract the first mode eigenfrequency of the first design, this value was 342.90 Hz. The composite material properties were generated using the FE-RVE plug-in (Figure 9). The introduction in beads in the design increased this eigenfrequency to 638.87 Hz. Comparative results are illustrated in Figure 13.



(a) **(b)**

Figure 13. Results of natural frequency analysis of the oil pan, (a): the original oil pan design, (b): the optimized oil pan design

7. Fatigue analysis of a composite structure using Fe-safe

Fatigue of composite materials presents a tremendous challenge when one considers the number and the variety of parameters that can possibly affect the governing mechanisms (Muc, Barski, Chwał, Romanowicz, & Stawiarski, 2018). The nature of loads in field generally englobes complicated loading scenarios involving residual stresses, cyclic fatigue stresses, thermal stresses, etc. The consideration of the loading scenarios is necessary for a more accurate prediction of the parts behavior in real life.

Fe-safe is a multi-axial strain based fatigue software that offers a wide range of capabilities for durability assessment and failure prevention. As a powerful and comprehensive tool, Fe-safe allows to model and simulate fatigue of metals, rubber, composites, etc.

The fatigue life generated by the mode 1 stresses shown in Figure 13 has been studied using Fe-safe.

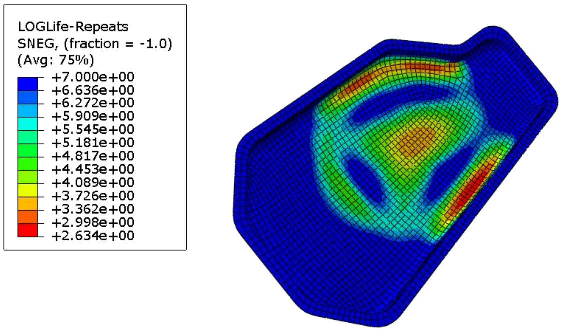


Figure 14. Fatigue assessment of an oil pan using mode 1 stresses

The amplitude of the fatigue stresses applied to the oil pan was 10% of the mode 1 stress of the first design (Figure 12 (a)). Fatigue contours illustrated in Figure 14 allow the analysis of the durability of the first design of the oil pan. The fatigue life also highlighted in the worst elements.

Conclusion

The main objective of this study was to provide a general assessment of the capabilities of Abaqus to model composite materials. The focus was on performing a thermoforming simulation on long fiber composites.

Other tools from SIMULIA, i.e. fe-safe, Isight, and Tosca were introduced with a focus on their capabilities to model the behavior of composite materials.

References

Abbassi, F., Elfaleh, I., Mistou, S., Zghal, A., Fazzini, M., & Djilali, T. (2011). Experimental and numerical investigations of a thermoplastic composite (carbon/PPS) thermoforming. *Structural Control and Health Monitoring*, *18*(7), 769–780.

Akkerman, R., & Haanappel, S. P. (2015). Thermoplastic composites manufacturing by thermoforming. In *Advances in Composites Manufacturing and Process Design* (pp. 111–129). Elsevier.

Fletcher, R. (Roger). (1987). *Practical methods of optimization*. Wiley.

Gong, Y., Peng, X., Yao, Y., & Guo, Z. (2016). An anisotropic hyperelastic constitutive model for thermoplastic woven composite prepregs. *Composites Science and Technology*, *128*, 17–24.

Ji, H., McLendon, W., Hurtado, J., Oancea, V., & Bi, J. (2017). *Multi-scale Material Modeling with the Mean-Field Homogenization Method*.

Levenberg, K. (1944). A method for the solution of certain non-linear problems in least squares. *Quarterly of Applied Mathematics*, *2*(2), 164–168.

Margossian, A., Bel, S., & Hinterhoelzl, R. (2016). On the characterisation of transverse tensile properties of molten unidirectional thermoplastic composite tapes for thermoforming simulations. *Composites Part A: Applied Science and Manufacturing*, *88*, 48–58.

Marquardt, D. W. (1963). An Algorithm for Least-Squares Estimation of Nonlinear Parameters. *Journal of the Society for Industrial and Applied Mathematics*, *11*(2), 431–441.

Michel, J. C., Moulinec, H., & Suquet, P. (1999). Effective properties of composite materials with periodic microstructure: a computational approach. *Computer Methods in Applied Mechanics and Engineering*, *172*(1–4), 109–143.

Muc, A., Barski, M., Chwał, M., Romanowicz, P., & Stawiarski, A. (2018). Fatigue damage growth monitoring for composite structures with holes. *Composite Structures*, *189*, 117–126.

Wu, L., Noels, L., Adam, L., & Doghri, I. (2012). A multiscale mean-field homogenization method for fiber-reinforced composites with gradient-enhanced damage models. *Computer Methods in Applied Mechanics and Engineering, Volume 233-236, 164-179.*