AN FE IMPLEMENTATION OF COUPLED RESIN FLOW AND PREFORM DEFORMATION LINKED TO RESIDUAL STRESS AND CURE-INDUCED DISTORTION ANALYSIS IN ABAQUS

Mohammad Rouhi [1], Maciej Wysocki[1]

[1] RISE Research Institute of Sweden, Department of Polymers, Fibers & Composites, Unit Composite Technology, Argongatan 30, SE-431 22 Mölndal, Sweden
Email: mohammad.rouhi@ri.se
Email: Maciej.wysocki@ri.se

Keywords: Process simulation, VARTM, Cure simulation, Residual stresses, ABAQUS

Abstract

The developed methodologies for process simulation of composite manufacturing, in general, are used to understand the simultaneous physical phenomena to optimize the part quality by predicting the defects arising during the process. One of the manufacturing technologies that are extensively popular for both small and large composite structures is Vacuum Assisted Resin Transfer Molding (VARTM). The objective here is to simulate the resin infusion process using ABAQUS, as an intermediary step in the manufacturing chain, cf. Figure 1. The significant challenges for modeling the resin infusion step are i) the migrating free surface due to resin infiltration into the fibrous preform in different scales, ii) the highly deformable preform and its shape due to the interaction between external pressure loading, and the intrinsic fluid pressure, and iii) the prediction of the permeability of the deformable preform.

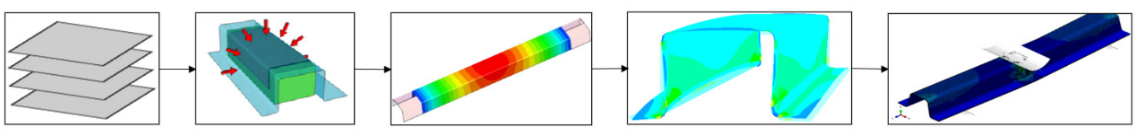

Figure 1: Simulation chain for a composite part manufactured by VARTM.

The theory of porous media for process simulation of resin infusion is established by Larsson et al.[1]. The formulation can predict the flow and the free surface at the flow front coupled to the preform deformation. The compressible two-phase porous media formulation includes an additional liquid mass balance relationship compared to the standard compressible porous media formulation to directly monitor the free surface problem in terms of the compressible continuum formulation.
As shown in Figure 1, the VARTM process chain starts from draping (preforming), resin infusion under vacuum, curing analysis, and finally de-molding; before any structural analysis is performed to predict damage and failure. In this paper, the step of resin infusion into highly deformable preform is simulated in 3D using a user-defined element (UEL) subroutine in the commercial finite element software package ABAQUS. It provides the possibility to predict local characteristics which are then linked to the developed curing and de-molding simulations using ABAQUS where path dependent cure model considers thermorheologically

[1] R. Larsson, M. S. Rouhi, and M. Wysocki, Free surface flow and preform deformation in composites manufacturing based on porous media theory," European Journal of Mechanics A/Solids, vol. 31, no. 1, pp. 1-12, 2012

complex materials and the dependency of the glassy and rubbery stiffnesses on temperature[2]. It is beneficial since both process steps are done in one platform and omit the need to buy and learn different simulation tools and the trouble of transferring and mapping data between them.

To evaluate the method properly, a complicated part geometry is selected. A curved Z–profile is used to incorporate male and female radii and double curvature. The part is approximately 500mm long, 100mm high and 175mm wide, and has a nominal theoretical thickness of 3.54 mm. The lower flange varies in length along the part, see Figure 2.

The simulations are performed on resin infusion followed by cure simulation for predicting the residual stresses and distortion. The connection between the two, on the simplest level, is the prediction of the local volume fractions which are used the calculate the initial composite properties for cure analysis.

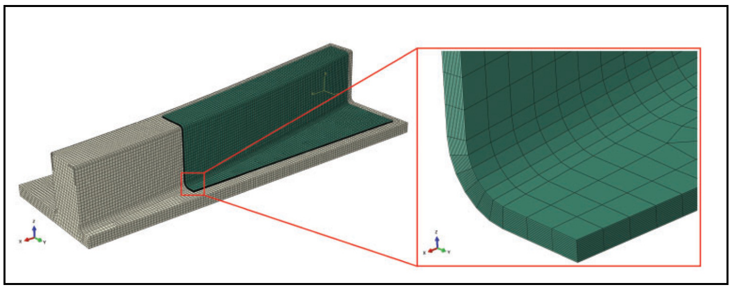

Figure 2: Geometry of the Z-profile and tool (left), and a close-up of the laminate thickness (right).

Figure 3 left shows the predicted element-wise volume fraction from the infusion simulation. Figure 3 right shows the deformation pattern from the path-dependent cure analysis, using the data predicted by resin infusion analysis. A simple python script reads the results from step one and calculates the inputs to the second step.

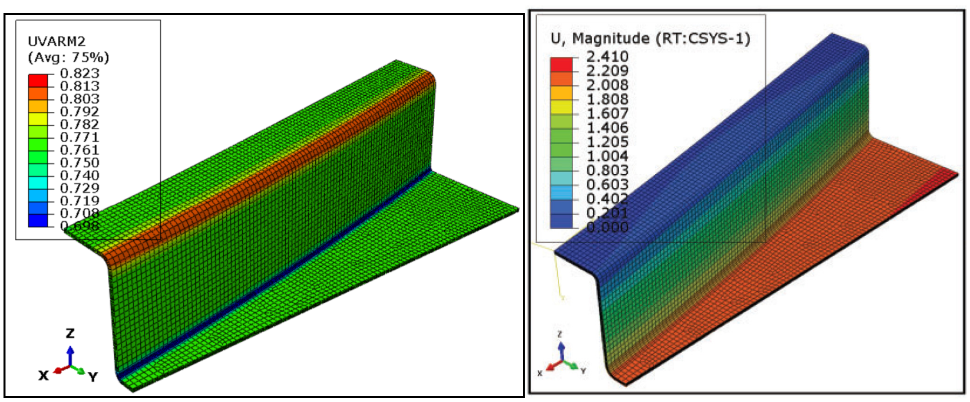

Figure 3: left is the results from resin infusion in terms of volume fractions and the right is the results from cure simulation in terms of displacement.

The results are indicative and further studies are being performed. Experimental validation is planned for both steps. A comparison of the model size and simulation time for each step is performed. Considering that all the steps are performed in ABAQUS using developed subroutines and mapping data directly from one step to another provides a level of simplicity with increasing computational efficiency and accuracy.

[2] C. Cameron, S. Saseendran, F. Stig, and M. S. Rouhi, A rapid method for residual cure stress analysis for optimization of cure induced distortion effects, DOI: 10.1177/00219983211024341