Int J Mater Form (2010) Vol. 3 Suppl 1:687–690
DOI 10.1007/s12289-010-0863-4

USING MODAL ANALYSIS TO INVESTIGATE THE VALIDITY OF FINITE ELEMENT MODELS FOR SIMULATING THE THERMOSTAMPING OF WOVEN-FABRIC REINFORCED COMPOSITES

J. Sherwood*, K. Fetfatsidis, D. Winchester, D. Jauffrès, P. Avitabile, J. Chen

University of Massachusetts Lowell - Dept. of Mechanical Engineering - USA

ABSTRACT: Thermostamping/thermoforming of woven-fabric reinforced composites can produce high-volume low-cost composite parts. Finite element modelling of the forming of such composites allows tailoring the manufacturing process to yield quality parts in the minimal time required and for prediction of the resulting mechanical behavior. Visual methods can be used to compare the final fibre directions from simulation results to that of a physically stamped part for single- and two-ply parts. However, such a visual comparison on three or more plies cannot be done without destructive evaluation of the formed part. Modal analysis is a viable option for doing nondestructive comparison of a physically formed part to that of its corresponding finite element modelling result. In this paper, modal analysis is used to compare the vibration modes between a physically stamped part to the same part "formed" using a finite element simulation. The study explores such a comparison for a two-layer plain-weave flat sheet of glass-fibre fabric with orthogonal yarns and a polypropylene thermoplastic matrix.

KEYWORDS: Thermostamping, thermoforming, woven-fabric, composites, modal analysis, finite element

1 INTRODUCTION

The aerospace and automotive industries have increased their use of composite materials due to their high stiffness-to-weight and strength-to-weight ratios, and their corrosion resistance. Also, with passenger safety being a concern, composites have the advantage over metals of being tailor-made to absorb energy and deform in a specific manner upon impact. Processes such as Vacuum Assisted Resin Transfer Moulding (VARTM) [1] and thermostamping of woven-fabric reinforced composites [2, 3] are capable of producing lightweight quality parts relatively fast with reasonably low processing costs.

Depending on the manufacturing process and the parameters used during the manufacturing of a composite part using woven-fabric reinforcements, the yarns will shear and slide relative to one another and the amount of yarn compaction, or flattening, can vary over the surface of the part. In addition, the fibre volume fraction can also be affected, as the liquid matrix material may be "squeezed" such that it can migrate across the part nonuniformly. The final orientation of the yarns, yarn thickness, and the distribution of the matrix within the part after forming affect the structural performance of a composite part. Defects resulting from the manufacturing process such as wrinkling and tearing can compromise structural integrity.

A modelling technique such as the finite element method can simulate the manufacturing process and predict the quality and performance of composite parts. Models have been developed using the commercially available software ABAQUS/Explicit and LS-DYNA to simulate the behaviour of woven-fabric composites during the thermostamping process (Fig. 1) [4]. A natural extension of these models is the option to quantify the structural behaviour of the part after it has been formed. This paper compares the vibration modes and frequencies from an ABAQUS finite element model to those obtained by experimentally studying a Twintex® consolidated glass/polypropylene flat sheet as an initial study before extending the proposed methodology to three-dimensional parts.

2 MODEL DESCRIPTION

Fabric models are generated using a mesh of 1-D (beam) and 2-D (shell) elements (Fig. 2). The 1-D elements account for the tensile contribution and orientation of the principal load paths of the yarns as they rotate to the fabric material behaviour. The 2-D elements account only for the shearing behaviour of the fabric and have no tensile stiffness. Appropriate nonlinear constitutive equations are associated with the 1-D and 2-D elements via user-defined material subroutines to capture the mechanical behavior of the fabric [5].

* Corresponding author: Department of Mechanical Engineering, University of Massachusetts, Lowell
One University Ave., Lowell, MA 01854, USA, phone: (978) 934-3313, fax: (978) 934-3048, James_Sherwood@uml.edu

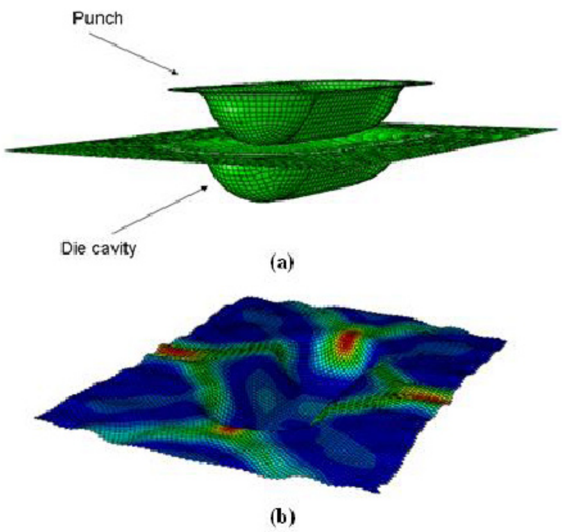

Figure 1: Finite element simulation using ABAQUS/Explicit of the (a) forming process and (b) resulting fabric shear contours

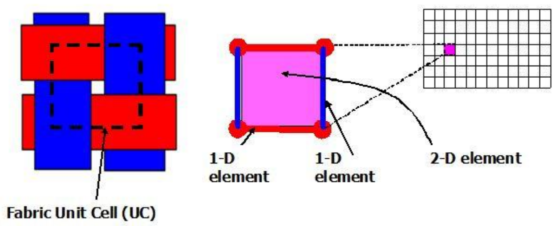

Figure 2: Principle of the discrete mesoscopic modelling using a combination of 1-D and 2-D elements.

The properties of the beam and shell elements evolve as the dry fabric is heated in an oven to melt the polypropylene, is subsequently stretched and sheared in the forming of the part and then is cooled to give the consolidated composite part. For example, the cross-section of the beam elements will change due to yarn compaction, thereby changing the moment of inertia and the resulting bending stiffness. The material properties must be correctly captured by the material model and the geometries of the yarns (e.g., thickness of the shell elements, cross-sectional area and area moment of inertia of the beam elements) must be properly described in the finite element model to have the mechanical behaviour of the final part from the simulation correlate to that of the actual part. Boundary conditions must also be known and appropriately defined in the model.

2.1 CONSOLIDATED FABRIC PROPERTIES

The proposed methodology was explored in this research using a flat plate made from Twintex® plain-weave glass/polypropylene woven-fabric. In the finite element model of the plate, shell elements capture the contribution of the polypropylene (PP) matrix and beam elements describe the contribution of the fibreglass yarns to the consolidated plate. Appropriate values are assigned to the beam and shell elements to define their respective elastic modulus. The flexural stiffness, $\boldsymbol{E_M I_M}$, of the shell elements is defined by knowing the unit cell width, $\boldsymbol{w}$, the shell element thickness, $\boldsymbol{t}$, and Poisson's ratio, $\boldsymbol{\nu}$, such that,

$$I_M = \frac{wt^3}{12(1-\nu^2)} \tag{1}$$

Element material and geometry parameters are summarized in Table 1.

***Table 1.** Element material and geometry parameters*

Parameter	Value
$\boldsymbol{E_Y}$ - Beam Elastic Modulus [MPa]	73,000
$\boldsymbol{E_M}$ - Shell Elastic Modulus [MPa]	1,200
$\boldsymbol{\nu}$ - Poisson's ratio (PP)	0.3
$\boldsymbol{w}$ - Unit cell width (plain-weave) [mm]	4.92
$\boldsymbol{t}$ - Shell thickness [mm]	0.64

The section, $\boldsymbol{I_Y}$, of the beam elements, is given by

$$I_Y = \frac{bh^3}{12} \tag{2}$$

where $\boldsymbol{b}$ is the width and $\boldsymbol{h}$ is the height, or thickness of the beam. Three-point bend tests were conducted on two-yarn wide solidified samples to characterize the consolidated material. For a simply supported beam with a concentrated load applied at the centre, the bending rigidity, $\boldsymbol{B}$, is defined by

$$B = EI = \frac{\left(\frac{L^3}{48}\right)\cdot\left(\frac{F}{\delta}\right)}{2} \tag{3}$$

where $\boldsymbol{L}$ is the length between the two supports, F/δ is the slope of the force-deflection curve obtained from the three-point bend tests, and the number 2 accounts for the two-yarn width of the sample. The bending rigidity of a fabric sample is the sum of the bending rigidity of the yarns, $\boldsymbol{B_Y}$, and of the matrix, $\boldsymbol{B_M}$, such that

$$B = B_Y + B_M = E_Y I_Y + E_M I_M \tag{4}$$

which can be rearranged to solve for the moment of inertia of the beam elements,

$$I_Y = \frac{B - E_M I_M}{E_Y} \tag{5}$$

Assuming a rectangular cross-section and knowing the cross-sectional area of a single yarn (0.846 mm^2), the beam section dimensions $\boldsymbol{b}$ (1.028 mm) and $\boldsymbol{h}$ (0.823 mm) can be found using Eq. 2.

3 EXPERIMENTS

While the analytical models are capable of simulating complex geometries, a simple flat plate was used as an example of the proposed correlation methodology between the finite element model and an actual composite part. Two layers of a Twintex® plain-weave glass/PP woven fabric were heat-pressed together and allowed to cool and solidify to fabricate a 304.8-mm x 304.8-mm square plate. Both layers of fabric were laid in similar $0^{o}/90^{o}$ orientations. The consolidated flat fabric sheet, ~1.0-mm thick, was clamped inside a square "rigid" steel frame with several bolts around the perimeter to simulate a fixed boundary condition (Fig. 3). The steel frame was thick enough (19 mm) to ensure that the dynamics of the flat sheet would not be influenced by the dynamics of the steel frame. The flat sheet and the steel frame were bolted to a wall where a shaker was used to excite the clamped composite sheet. A random burst excitation was applied for 75% of each time block, such that enough time was given for the signal to be damped and prevent leakage.

Figure 3: *Solid composite plate clamped in steel fixture*

3.1 3-D SCANNING LASER VIBROMETER

A Polytec 3-D scanning laser vibrometer (Fig. 4) was used as a noncontacting method of obtaining frequencies and mode shapes of the clamped plate. A mesh of several sampling points was selected on the video display of the flat sheet. Three independent lasers were co-aligned to a sampling point but at different interrogation angles to determine the complete 3-D velocity vector at that point. After a complete scan of all preselected sampling points, frequency response functions and mode shapes were viewed and analyzed. In addition to the flat sheet, sampling points were included on the steel frame and on the wall to verify that the fixed boundary condition was satisfied.

Figure 4: *Polytec PSV-400-3-D Scanning Laser Vibrometer sensor heads and data management system*

3.2 THREE-POINT BEND TESTS

After the laser vibrometer testing, the flat composite sheet was cut into several beam samples, and three-point bend tests were performed on the cut samples to investigate the bending rigidity of the plate at the edge and the middle sections. Samples were extracted at each of the edge and centre locations in the 0 and 90^{o} orientations to investigate any warp and weft dependence. A Mark-10 mechanical testing machine (Fig. 5) was used to perform the three-point bend tests. The span between the two supports was 76.4 mm. The width and thickness of the cut samples were (31.6 ± 1 mm) and (1.0 ± 0.1 mm), respectively. Force and displacement values were recorded, and the bending rigidity of each sample was calculated.

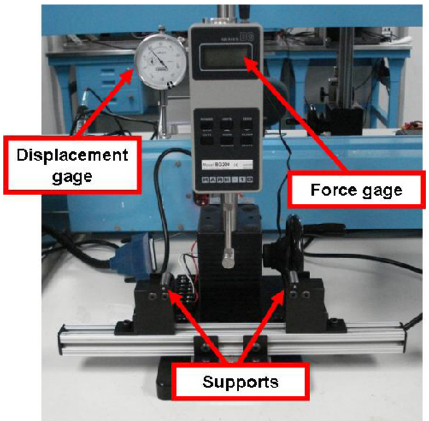

Figure 5: *Mark-10 mechanical testing machine*

4 RESULTS AND DISCUSSION

Modal results from the 3-D scanning laser vibrometer were compared to modal results obtained by the ABAQUS finite element model. The flexural stiffnesses of the beams cut from the plate were summarized to explain any difference between the model and the experimental frequencies.

4.1 MODAL ANALYSIS

The first three mode shapes correlated very well between model and experiment (Fig. 6). Note that due to the geometry of the square plate, the finite element model predicted repeated frequencies for modes 2 and 3, but the experiments showed an 8.1% difference between frequencies 2 and 3 (Table 2) for the actual plate. The finite element model assumed a uniform thickness and uniform bending stiffness across the plate.

Measurements of the formed plate showed that the plate was thicker on the perimeter in comparison to the centre of the plate. This difference between centre and edge thicknesses is reasonable because the force on the forming platen was applied in the centre and thereby the edges of the top platen could "bend up". This slight bowing of the forming platen, while small, does explain how there could be a thickness gradient across the formed plate. Thus, the differences between the model and experimental frequencies are partially due to variations in thickness of the part as a result of fabrication and the model assuming a uniform thickness for the plate.

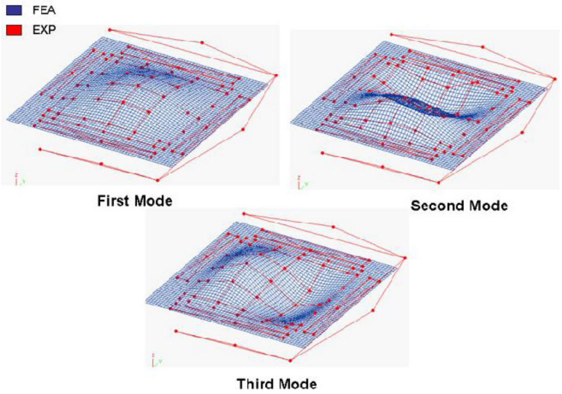

Figure 6: *Modal analysis of flat composite plate: Comparison of first three experimental and analytical mode shapes*

Table 2. *Comparison of first three frequencies between model and experiment*

Frequency	EXP (Hz)	FEA (Hz)	% Diff.
1	117.44	110.00	6.34 %
2	213.99	226.26	5.73%
3	232.88	226.26	2.84%

4.2 THREE-POINT BEND TESTS

Warp and weft-orientation beam samples were cut from the middle and edge locations of the flat sheet and subjected to three-point bend tests. The measured flexural rigidities are summarized in Table 3.

Table 3. *Bending rigidity of composite plate cut samples*

Direction	Edges - EI ($N\text{-}mm^2$)	Middle – EI ($N\text{-}mm^2$)	% Diff.
Warp	25,428	26,496	4.03 %
Weft	25,750	22,224	13.69%
% Diff.	1.25%	16.12%	

At the edge of the plate, the bending rigidity varied by only 1.25% between warp and weft directions. However, a 16.12% difference was measured in the flexural rigidity between warp and weft directions in the middle of the plate. Also, the weft direction at the edge of the plate was 13.69% stiffer than the weft direction in the middle of the plate. The variation in the flexural stiffness at different locations in the plate and in different directions is probably the more significant source of the frequency differences between the finite element model and the experiment.

Future work will explore methods for including a more complete description of the warp and weft yarn behaviour in the forming simulation and the potential pressure variation experienced by the part from the tool resulting in a thickness variation across the final part. Nevertheless, the model and experiment frequencies correlated within 10%, so the proposed methodology of using modal analysis to compare the stiffness of the formed part to that found using the simulation does show promise and can aid in refining the geometry and material parameters used in the simulation.

5 CONCLUSIONS

A modal analysis was performed on a flat, consolidated sheet of a plain-weave woven fabric composite. Experimental mode shapes and frequencies were obtained using a 3-D scanning laser vibrometer and were compared to mode shapes and frequencies obtained from an ABAQUS finite element model. The first three mode shapes correlated very well and the frequencies correlated within 10%. The differences between the model and experimental frequencies were due to the fabrication of the part, which caused some thickness and stiffness variability in the actual part.

ACKNOWLEDGEMENT

The authors would like to acknowledge The National Science Foundation for its support of this research through Grant No. DMI-0522923.

REFERENCES

[1] Aoki Y., Sugimoto S., Hirano, Y., Nagao, Y.: Non-Destructive Inspection Technologies for VARTM Composite Structures. *SAMPE Journal*, 46(1):22-27, 2010.

[2] Gorczyca, J., Sherwood, J. and Chen, J.: "Development of a Friction Model for Use in the Thermostamping of Commingled Glass-Polypropylene Woven Fabrics," *Composites Part A*, Vol. 38, p. 393-406, 2007.

[3] Wakeman M., Cain T.A., Rudd C.D., Brooks R., Long A.C.: Compression moulding of glass and polypropylene composites for optimised macro and micro-mechanical properties. 1 Commingled glass and polypropylene. *Compos Sci Technol* 58:1879–1898, 1998.

[4] Jauffrès, D., Morris, C.D., Sherwood, J.A., and Chen, J.: Simulation of the Thermostamping of Woven Composites: Mesoscopic Modelling Using Explicit FEA Codes. *International Journal of Material Forming*, 2(1):173-176, 2009.

[5] Jauffrès, D., Sherwood, J.A., Morris, C.D. and Chen, J.: Discrete mesoscopic modelling for the simulation of woven-fabric reinforcement forming. International Journal of Material Forming (in press, available online on www.springerlink.com), 2009.