PROGRESSIVE FORMING OF THERMOPLASTIC LAMINATES

John R. Smith and Uday K. Vaidya

Department of Materials Science and Engineering
The University of Alabama at Birmingham (UAB)
Birmingham, AL 35294 USA

ABSTRACT

Thermoplastic composite laminates can be post-manufactured by progressively thermoforming them to generate contoured parts from prior flat panels. This process is attractive for expanding the potential usage of composite materials in next generation transportation, infrastructure, marine, and military sectors for part replacement and structural applications. Thermoforming has proven to be an efficient means to creating parts of complex geometries. Accurately predicting material properties and temperatures prior to forming is of utmost importance to minimize waste and reduce cost for mass production applications. This paper presents a finite element modeling approach to establish the manufacturing parameters for locally formed thermoplastic composite plates.

INTRODUCTION

Thermoplastic composites are attractive because of their recyclability, indefinite shelf-life, and superior fracture toughness. Progressive thermoforming of thermoplastic composite laminates is a post-manufacturing process used to create parts of three dimensional shapes from a planar sheet of composite material, similar to that of sheet metal forming. The essential ingredients to shaping a flat laminate include heat, a forming force, and membrane tensile forces (typically achieved with a clamping assembly). Forming induces a wide array of mechanical and rheological changes in the sheet. It is crucial to fully understand these changes so that part characterization and mechanical predictions can be made and verified. Because of significant fiber tow reorientation associated with forming, it is not uncommon to witness buckling, wrinkling, interply slip, intraply shear, thickness changes, and delaminations.

Depending upon the initial laminate, (whether it is a sequence of unidirectional plies, or a sequence of woven plies, or discontinuous fiber sheets) there is significant tow/fiber reorientation associated with forming. It is the ability of the fiber to reorient and shear, in-plane, that allows the preform to drape across tooling surfaces and alter its shape. It is not uncommon to witness buckling, wrinkling, interply slip, intraply shear, thickness changes, and delaminations because of this reorientation, see Figure 1. Fiber lock is experienced when the tows are unable to rotate any further, giving rise to many of these defects. Figure 2 shows that increased torque (forming force) is necessary to create deep drawn impressions due to these rotations. This phenomenon results in an area reduction and is described by Trellis shearing.
The extent, ease, and ability of forming are mostly driven by matrix rheology and fiber/weave characteristics. There are several interrelated factors that affect thermoforming of composite laminates. The thermoforming process couples temperature with displacement, incorporating heat transfer, viscosity, flow, non-orthogonal material properties, and large deformations. In reinforced composites, high fiber volume fraction offers a significant amount of resistance to shearing/deformation needed to form shape(s), rendering forming difficult.

There are several studies that have reported composite forming and its associated challenges. Labeas et al reported that too much heat was being conducted away by non-heated tooling and that the panel would not remain within forming temperatures (1). Nowacki et al agreed that heated tooling aids in achieving quality parts (2). Muzzy et al found that heated tooling permitted an increase in matrix viscosity during forming (3). The present research has been conducted with the use of only localized heating. That is, the tooling is the sole source of heat generation. External heating is not studied, even though it is known that local heating results in wrinkling, delaminations, and buckling because interply slip is only applicable to the heated region (4).

Contact heating provides three modes of heat transfer: conduction between platens and panel, convection within the surface irregularity air gaps of the platen and panel, and isotropic/anisotropic conduction within the panel (5). Concerning interfacial conductance between the tooling and composite panels, the thermal energy remains on the contact surfaces of the panel and increases their local temperatures. This is due to the resistive nature of thermoplastic matrices. Even heat through conductive fibers is inhibited in effectiveness due to the temperature being firstly transferred through the matrix material.

The overall objective of the proposed research is to locally form thermoplastic composite panels with the combined application of heat and pressure. In the present work, forming concentrated impressions, bead-forms, localized dimples, and/or other shapes from starting sheet form are being investigated. The end goal of this work is to be able to successfully create composite parts with locally heated and thermoformed features, in multiple locations. This research is valuable to the transportation, military, and marine industries. This paper is limited to initial manufacturing parameters that are being established, and the physics of the process based on lab-scale experiments and finite element analysis (FEA).
FINITE ELEMENT MODEL

A simplified computer aided drafting (CAD) model of the manufacturing cell for local forming of a thermoplastic composite panel was developed. The cell comprises a female die, a male die (not shown in the model), the clamp and the thermoplastic composite sheet that is being locally formed. The model is configured for forming hemispherical dome shaped features in the present case. Features such as threaded mounts, rounded corners, and through-holes were eliminated in favor of making the model more orthogonal for meshing advantages and simplification. Pro/Engineer Wildfire 4.0 was used for the CAD work. A half-model was utilized to reduce computational costs. Altair HyperWorks Version 9.0 HyperMesh was used to create the mesh and assigned specific elements necessary for analysis. An 8-noded thermal brick element SOLID70 was chosen for all materials (6,7). Contact and target elements were assigned to all interference locations. The target element chosen for the study was TARGE170, a 4-noded shell element while the contact element is CONTA173, also a 4-noded shell element. A positive aspect to contact and target mesh generation is that the nodes do not have to coincide; this means their respective meshes can be different.

![Figure 3. Representative Mesh Model of Manufacturing Cell and Thermoplastic Composite Panel (Male die not shown in model)](image)

EXPERIMENTAL

Two unidirectional composite test panels were fabricated with thermocouples embedded within. Three through-thickness thermocouples were placed in three selected planar locations, resulting in nine thermocouples per test panel, shown in Figure 4a. The test panels were made of 60 wt % S2-glass fibers/thermoplastic polyurethane (TPU) matrix, and 58 wt% carbon fibers/polyacrylic acid (PAA) matrix. Both matrices are high temperature polymers and were used to ensure that melting would not occur during various tests.
Four distinct planar panel positions, in relation to the tooling, were established to acquire temperatures with the generated test panels, as shown in Figure 4b. The locations were chosen from the preliminary FEA results. General trends and hot spots due to local heating are generated no matter which composite constituents are chosen. To best ensure that the model was working correctly, the panel results from these locations were compared to the results from FEA. Since the thermal model is a half representation, the two left most locations can be mirrored, when in reality those would be measurements taken on the right hand side of the center line.

When a composite panel is placed on top of the female die, a cavity of entrapped air is formed. While it is not air-tight, the amount of heat transfer through this pocket is considerable. It was not possible to model convection from the curved surface of the die to the flat bottom surface of the panel; hence the air pocket was modeled as a separate material in which contact elements were assigned between the surface of the die to the air pocket and between the panel’s bottom surface and the air pocket.

Transport properties for unidirectional composite panels were determined from methods of Agarwal et al (8). These are valid for heat conduction, permeation, electrical conduction, and transport for electrical and magnetic fields. For a unidirectional composite panel, longitudinal conduction and its effective specific heat can be determined from the rule of mixtures.

\[ K_4 \text{ or } C_p = V_f K_f + V_m K_m \]

In this case subscript “f” denotes fiber and “m” denotes matrix. “K” is the denoted conduction value, while “V” is the volume fraction. When determining specific heat, “C_p”, the rule of mixture values for conduction needs to be replaced with each constituent’s respective specific heat constant. Since the composite panel is unidirectional, it can be treated as transversely isotropic. Hence, conduction in the 2 and 3 directions are the same. Conduction for inclusions, or fibers, can be estimated based upon material properties and cross section geometry.
In this case, “a” and “b” are associated to both the width and height of the fiber or inclusion. For a round fiber, the ratio a/b can be taken as 1. Hence: $\xi = 1$. A scaling factor is considered.

\[ \eta = \left( \frac{K_2}{K_3} \right) - 1 \]

The transverse isotropic conduction is determined.

\[ \frac{K_2}{K_3} = \frac{1 + \xi \eta K_2}{1 - \eta K_2} \]

Table 1 provides the thermal values used for each test panel.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>C/PAA</td>
<td>S2/TPU</td>
<td>C/PAA</td>
</tr>
<tr>
<td>K1</td>
<td>0.298</td>
<td>0.046</td>
</tr>
<tr>
<td>K2/K3</td>
<td>0.046</td>
<td>0.028</td>
</tr>
</tbody>
</table>

Comparisons between thermocouple values obtained during testing and FEA predictions for the carbon/PAA panel and the S2-glass/TPU panel are given in Figs. 5 and 6 respectively. 25% heating was used for both the clamp and die while testing the carbon/PAA panel. 100% heating of the clamp and 12% heating of the die was used for the S2-glass/TPU panel. When comparing experimental measurements to FEA predictions, it must be recognized that the top and bottom surface readings of the test panel are taken one ply deep. A large amount of heat is entrapped in the panel surfaces when conductive heating is used. The FEA predictions show this surface temperature. When through-thickness gradients are considered, the surface temperature decreases quickly. The 100% heating of the clamp and 12% heating of the die was observed to create significant hot spots, especially for the surfaces of the panel. The 25% heating of the clamp and die was used to create more uniform panel temperatures. A slight bias is imposed to heat the panel with the clamp, effectively eliminating hot spots due to the increased amount of heat input into the panel by the die. The thermocouple placement within either laminate is not precise; their locations are approximately correct. This fact becomes very crucial for heating regimes which generate hot spots. The results of the test are valid for verifying the general temperature ranges experienced at various locations in the panel.
Figure 5: (a) Thermocouple Readings Showing Temperature Distributions for Heating of 58 wt% Carbon/PAA Test Panel for 10 Minutes with 25% Heater Output for Both Clamp and Die and (b) the Predicted Temperature Profiles for the Lower, Original, and Upper Bounds of Potential Heat Fluxes. All Temperatures are given in [deg F].

Figure 6: (a) Thermocouple Readings Showing Temperature Distributions for Heating of 60 wt% S2-glass/TPU Test Panel for 10 Minutes with 100% Heater Output for the Clamp and 12% for the Die Beginning with a Room Temperature of 89 [deg F] and (b) the Predicted Temperature Profiles for the Lower, Original, and Upper Bounds of Potential Heat Fluxes. All Temperatures are given in [deg F].
SUMMARY

This study has addressed the steps necessary to accurately predict thermoplastic composite panel temperatures using localized contact heating by means of finite element analysis. Model generation was outlined, as well as an explanation to characterizing the manufacturing cell using thermal steady state testing. Thermal contact conductance was used to model separate material interfaces and to introduce temperature drops across their boundaries. Comparing results from test panels with embedded thermocouples to predicted results provides insight into the planar and through-thickness temperature profiles created in composite panels.

The study concludes that heat fluxes must be matched to avoid drastic though-thickness variations. However, planar variations are bound to exist despite close match of the output heat values. The mismatch in contact area from the clamp and die is too great to achieve thermal planar uniformity. Panels made of constituents with high thermal conduction aid in distributing heat more evenly, as well as using the panel platform. The platform in itself acts as a heat sink and allows the heat to be more evenly distributed on the bottom face of the composite panel.

The model should typically use the upper bounds of the heat flux outputs to best match results obtained in the test panels. Variations between the model and test panel exist; however, on average, are within an acceptable margin of error. While the model is not exact in matching the test values, the trends it displays match the trends given from acquisition, yielding a trustworthy tool for temperature predictions for local forming of thermoplastic composite sheets.

REFERENCES


ACKNOWLEGEMENTS

Support from the National Science Foundation EPSCoR program of the Alabama and the Army Research Laboratory, Aberdeen Proving Ground (APG) is gratefully acknowledged.