Gillespie, David I. and Hamilton, Andrew W. and Kruckenberg, Teresa and Neilson, Brian and Atkinson, Robert C. and Andonovic, Ivan and Tachtatzis, Christos (2019) Geometrical thermal analysis as a form of Finite Element Analysis enhancement. In: 17th International Conference on Manufacturing Research ICMR2019, 2019-09-10 - 2019-09-12, Queens University. (In Press) ,

Geometrical thermal analysis as a form of Finite Element Analysis enhancement

David I. Gillespie[a,b,1], Andrew W. Hamilton[b], Teresa Kruckenberg[a], Brian Neilson[a], Robert C. Atkinson[b], Ivan Andonovic[b], Christos Tachtatzis[b]

[a;] *Collins Aerospace, 1 Dow Avenue, Prestwick International Aerospace Park, Ayrshire KA9 2SA UK.*

[b] *Department of Electronic and Electrical Engineering, University of Strathclyde, Royal College Building 204 George Street, Glasgow, G1 1XW, UK*

Abstract. Maintenance Repair and Overhaul (MRO) of composite aerospace components requires rigorous stress and strain analysis, including mechanical and thermal, as part of the repair process. Finite Element Analysis (FEA) as a standard and robust method of determining the transfer of transient thermal results of materials is well established for remanufacture. However, the theoretical material properties upon which FEA models are based, often do not correspond to real life composite aerospace parts, due to manufacturing variations and the addition of previous repairs to the component modifying the fibre orientation and local fibre volume fraction. Such deviations from FEA models can result in inadequate repairs and in extreme instances even cause thermal damage to components. A geometrical analysis method, incorporating existing industry standard heating elements and single point thermal capture sensors, could be used to quickly verify transient thermal results and provide useful data in order to correct the FEA model.

Keywords. Re-manufacture, Thermography, Composite, Aerospace, Quality Assurance, Maintenance Repair Overhaul.

1. Introduction

The preparation of composite repairs within the aerospace industry is a rigorous and technically demanding task which relies heavily on engineering data in regards to stress and strain analysis. With the advancement of digital technologies it is now possible to calculate the interactions of complex geometries with external and internal forces [1, 2] quickly. This has resulted in an improvement to the efficiency of which repairs can now be designed. However in composite repairs it is still possible for unexpected stresses and strains to develop within a component during the repair process which can result in an unsuccessful repair or the creation of damage within an area previously considered undamaged. When considering material properties there is evidence of low levels of repeatability between composite components when compared to Finite Element Analysis [3, 4, 5, 6]. It has been suggested that these variations can be explained due to minor variations between the fabrication set up which, unlike other material fabrications (e.g. CNC milling of metals), are difficult to achieve high

[1] Corresponding Author. david.gillespie@strath.ac.uk

repeatability. The result of this can impact the fibre orientation, cure ply thickness and fibre volume fraction of the composites.

2. Simulation of thermal spread within a composite laminate

In order to validate a Finite Element model the creation of data which can be validated must first be achieved. The creation of geometry representative of two Unidirectional material fibre laminates with a density of 1625 kg m^{-1} and a single biaxial woven material fibre laminate panel of density 1589 kg m^{-1}, were conducted within the ANSYS 17.1 mechanical workbench. These were then used to generate FE models of composite laminate structures within the ANSYS 17.1 Composite PrePost environment with the material properties representative of the Aerospace material (Table 1) used in the validation section. The two unidirectional laminate panels and a biaxial laminate panel were simulated with the ply lay ups:

- Panel 1 0/0/0/0 Unidirectional
- Panel 2 0/90/90/0 Unidirectional
- Panel 3 0/90/90/0 Biaxial (woven)

Table 1. Material thermal properties for the aerospace unidirectional and biaxial composite plies.

Panel	Temperature (K)	Thermal conductance X Direction (W m^{-1} K^{-1})	Thermal Conductance Y Direction (W m^{-1} K^{-1})	Thermal Conductance Z Direction (W m^{-1} K^{-1})	Specific Heat (J kg^{-1} K^{-1})
Panel 1 / Panel 2	218.15	9.1037	0.74422	0.74422	678.26
	295.93	10.99	0.91729	0.91729	904.35
	394.26	13.084	1.1077	1.1077	1184.9
Panel 3	218.15	9.3287	9.3287	0.6923	678.26
	295.93	11.198	11.198	0.9346	904.35
	394.26	12.981	12.981	1.0384	1184.9

The resultant FE models produced two panels that displayed a 0.508 mm constant uniformity thickness and 300mm by 300mm in the XY directions. Panel 3 produced with uniform thickness of 0.7366 mm and 300mm by 300mm in the XY directions. The Transient Thermal toolbox system was used in order to provide a temperature against time across the FE model under simulation in controllable time segments. A heater mat with dimensions 150mm by 150mm was simulated. In order to ensure that the transient thermal simulation and the experimental validation are both reacting to the same thermal input (in the form of a heater mat) the real world heater mats thermal profiles were captured using a two minute step heating profile broken into two parts, the ramp and the dwell (Table 2).

Table 2. Step heating profile

Stage	Time Step(secs)	Start Temperature (K)	End Temperature (K)	Ramp rate (K/s)
Ramp	0-60	298.15	353.15	0.917
Dwell	60-120	353.15	353.15	0.00

The ramp and dwell cycle were conducted thirty times in order to capture the temperature output across the heater mat. The temperature data was captured using six RTDs (Resistance Temperature Detector) split into three groups; group one contained a single RTD that was used to record ambient air temperature and provide this as an input to the simulation. Group two consisted of four RTDs that each recorded the temperature output of the heater mat "cells". The final group consisted of one RTD, centrally located upon the heater mat to provide feedback to the control system to control the ramp and dwell temperatures. Figure 1 shows in orange the setpoint profile used by the control system (as in Table 2) and in blue, the mean temperature response of the experimentally captured data for the thirty repetitions. It was noted during the experimental capture, that there was an area of 15mm by 25mm at the connection point of the control wiring to the heater mat which did not contain heating elements. This geometry was used within the simulation to represent the heat surface of the mats used in the experimental section. The mean ramp/dwell profile was then used to set the temperature profile of the heater mat cells within the ANSYS 17.1 simulation.

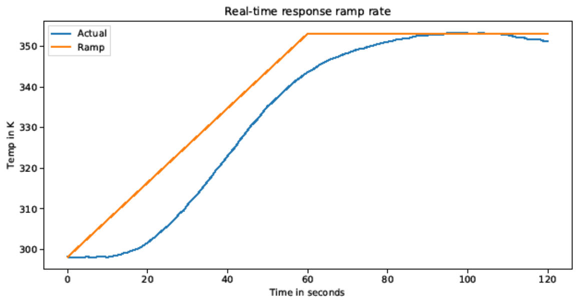


Figure 1 Real-time response to ramp rate of heater mat.

The simulations were repeated for five heat zones location as shown in Figure 2, using the same real time response obtained in the previous step. The expectation is that due to the symmetry of the fibre orientations the thermal spread profile obtained for locations (1,3) and (2,4) for the 0/0/0/0 will be mirrored, while the same symmetry will hold for locations (1,4) and (2,3) for the 0/90/90/0 orientation. For the centre injection point (location 5) a balanced thermal spread would be expected with the 0/90/90/0 lay-up due to the balanced nature of the layup in the X/Y directions. The centre injection point of the 0/0/0/0 lay-up will be expected to show a further spread along the fibre direction.

In order to ensure that the comparison between simulation and experimentally captured results are valid, 64 points capture points were created (Figure 2, blue 'X') evenly spaced 37.5 mm in X and Y directions, with the initial capture point being 18.75mm in both the X and Y directions from the absolute bottom left corner of the panel (0,0). Within the simulation, temperature probes were placed at these capture points on the front and back of the panel on the laminate panels and RTDs placed at same locations for experimental validation.

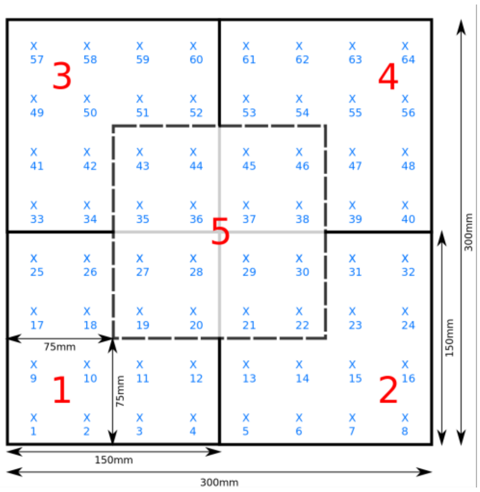


Figure 2 Experimental and simulation set up of laminate panels showing 150mm by 150mm heater mat placement zones 1-5 (Red) and probe locations (Blue X) and probe number (blue) on 300mm by 300mm laminate panel.

3. Probe analysis

The RTD data was captured from the three panels in an experimental set up that mimicked the simulation transient thermal analysis. The heater mat and RTDs were temporarily bonded to the panels via a thermally conductive silicon grease and thermally resistant tape. The unidirectional 0/0/0/0 laminate tests provided results that supported the transient thermal spread direction and shape, however produced lower temperatures outside of the heater mat area than in the transient thermal simulation (Figure 3).

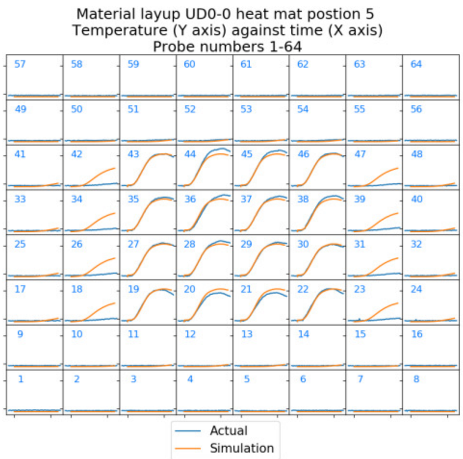
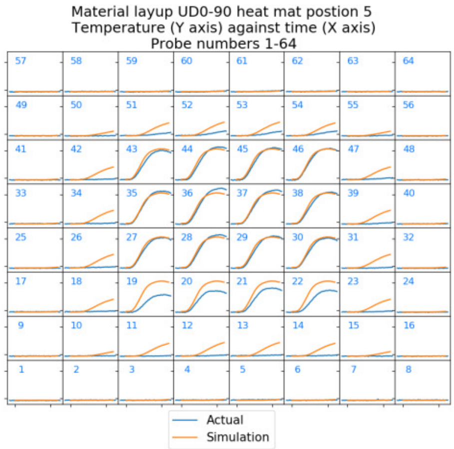
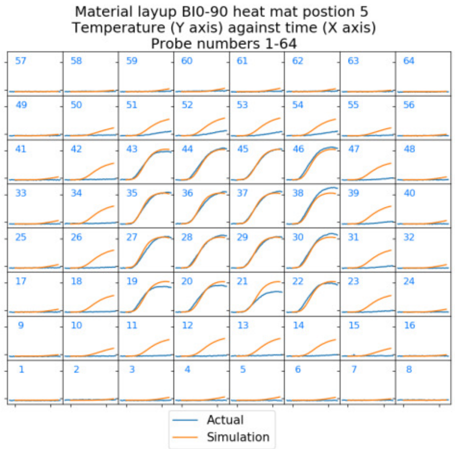


Figure 3 Heat map results of unidirectional material laminate panel with ply orientations 0/0/0/0 in heater mat position 5; probes arranged as shown in **Figure 2**

In the case of the unidirectional 0/90/90/0 (**Error! Reference source not found.** (a)) in which the transient thermal simulation produced a balanced thermal spread in both the X and Y axis it can be seen that the experimental results differ in that they concentrate in the Y axis, most notably at the top of the panel (locations 50-55). The imbalance between the top (locations 50-55) and bottom (locations 10-15) heat spread can be explained due to the heater mat shape incorporating power and control leads at the base, where the thermal spread temperatures are lower. Even so, this shape and difference in temperature had been modelled within the simulation. With regards to the concentration of thermal spread in the Y axis, along the 90 degree fibre orientation. The hypothesis is that the two ply layers meeting in the same orientation act to re-enforce the thermal transmission along the fibres as less energy is lost in the boundary between these plies than when two plies of dissimilar orientation meet. This concentration along the Y axis is repeated within the biaxial 0/90/90/0 sample (**Error! Reference source not found.** (b)).



(a)



(b)

Figure 4 Heat map results of (a) unidirectional 0/90/90/0 (b) biaxial 0/90/90/0 woven

The suggested explanation for the FE model simulation results is down to the way in which FE software packages simulate composite fibre layers. It is common for the fibres and yarn within composite fabrics to be modelled as solids [7] with increased concentration of material properties along the fibre direction. There is also the modelling of fibre undulation to consider, again the complex micromechanics associated with this can be dealt with by using estimates based on a constant undulation across the material layup [8].

The FE model appears to simulate the 1D transient thermal accurately through the depth of the panel as seen by probes at locations 27-30, 35-38 and 43-46. These probe locations are all directly above the heat source. As previously explained, probes at locations 19-22 are affected by the shape heater mat. The remaining probes rely on the 3D transient thermal heat transfer through the X and Y axis of the panel in order to produce a temperature reading on the RTDs.

The FE software in this instance have resulted in two panels (both 0/90/90/0) which do not reflect the thermal properties in both magnitude and direction as suggested by the FE simulations and the remaining panel (unidirectional 0/0/0/0) where the magnitude of the thermal spread is inaccurate. As such the two minute test using a temperature range well within the safe thermal loads of the composite materials has indicated that the FEA carried out is not an accurate reflection of the material.

4. Conclusion

A comparison of the transient thermal results of an industry standard FEA package with a simple heat transmission simulation it indicates that the results can differ significantly from experimentally captured heat transmission behaviour. The cause for the difference between 0/90/90/0 FEA simulation and the experimentally captured results is owing to a different degree of thermal transmission at the boundaries between plies depending on the similarities in the fibre orientation. Therefore caution must be taken when interpreting FEA results, as these may not reflect the true thermal behaviour of parts.

Acknowledgements

The Authors would like to acknowledge Collins Aerospace, a United Technologies company, for the provision of aerospace quality composite test samples used in the experimental validation of this paper.

References

[1] J. Stelzer, “Consideration and strategies in developing finite element software for desktop computers,” *Engineering Computations,* vol. 1, no. 2, pp. 106-124, 11 2 1984.

[2] J. N. (. N. Reddy and D. K. Gartling, The finite element method in heat transfer and fluid dynamics, CRC Press, 2010, p. 500.

[3] P. Boisse, B. Zouari and A. Gasser, “A mesoscopic approach for the simulation of woven fibre composite forming,” *Composites Science and Technology,* vol. 65, no. 3-4, pp. 429-436, 1 3 2005.

[4] S. V. Lomov, D. S. Ivanov, I. Verpoest, M. Zako, T. Kurashiki, H. Nakai and S. Hirosawa, “Meso-FE modelling of textile composites: Road map, data flow and algorithms,” *Composites Science and Technology,* vol. 67, no. 9, pp. 1870-1891, 1 7 2007.

[5] F. Nosrat-Nezami, T. Gereke, C. Eberdt and C. Cherif, “Characterisation of the shear–tension coupling of carbon-fibre fabric under controlled membrane tensions for precise simulative predictions of industrial preforming processes,” *Composites Part A: Applied Science and Manufacturing,* vol. 67, pp. 131-139, 1 12 2014.

[6] C. E. Jones, A. W. Hamilton, P. J. Norman, A. Cleary, S. J. Galloway, R. Atkinson, G. M. Burt, C. Michie, I. Andonovic and C. Tachtatzis, “A novel methodology for macro-scale, thermal characterisation of carbon fibre reinforced polymer for integrated aircraft electrical power systems,” *IEEE Transactions on Transportation Electrification,* pp. 1-1, 2019.

[7] R. Misra, A. Dixit and H. S. Mali, “Finite Element (FE) Shear Modeling of Woven Fabric Textile Composite,” *Procedia Materials Science,* 2014.

[8] Y. C. Zhang and J. Harding, “A numerical micromechanics analysis of the mechanical properties of a plain weave composite,” *Computers and Structures,* 1990.