A MODELING METHOD OF CURING DEFORMATION FOR CFRP COMPOSITE STIFFENED PANEL

WANG Yang[1], GAO Jubin[1] BO Ma[1] LIU Chuanjun[1]

[1] Beijing Key Laboratory of Civil Aircraft Structures and Composite Materials, Beijing Aeronautical Science & Technology Research Institute of COMAC, Future Science and Technology Park, wangyang6@comac.cc

Keywords: CFRP, Curing Deformation, Stiffened Panel, Modelling Method

ABSTRACT

Based on the finite element analysis a simulation process is developed to predict the deformation of CFRP composite panel after cure. The Classic Laminate Theory is employed to assign the composite mechanical properties, and 2D shell element is used to simulate both the skin and stiffener of the panel. A hard contact behaviour is introduced between the rigid mode and stiffened panel, and thermal stress effect in the autoclave is addressed with uniform thermal environmental. The influence of contact boundary conditions on the maximum curing deflection is studied and the mode is validated by comparing with experimental result. The thermal expansion property of composite structure is highly related to the ply stacking sequence. The experimental result shows the curing deformation is reduced by half when the ply sequence is optimized. This modelling method can capture the trend of optimizing and predict the magnitude of curing deformation. The maximum difference between numerical and experimental result is less than 1mm. This modelling procedure is straight forward, and compared with the multi-physics coupling method it requires less memory resources. Curing deformation of two panel with different stacking sequence are successfully predicted with is method. The result shows that the modelling method described in this paper can effectively analysis the curing deformation of composite panel with relatively simple steps. Additionally, numerical simulation results from this method can support the ply optimization on composite panel curing deformation.

1 INTRODUCTION

Composite materials are used increasingly in the aviation industry due to their high specific strength and design freedom. The anisotropic nature of composite brought advantages in the structure optimization. Unidirectional CFRP ply is used to build the laminate of the composite structure, and the percentage of ply with different orientations is optimized for weight saving. Therefore the coefficient of thermal expansion (CTE) in the fibre direction is different than it in the normal direction. During the curing process in the autoclave, mismatch of CTE produces the interlaminate residual stress and tend to deform the structure. Several researches on numerical analysis of curing residual stress have been performed [1,2] . White and Hahn's [3,4] work gives a model that links the curing dynamic with the residual stress to predict the curing deformation. Bogetti and Gillespie [5.6] studied the stress difference caused by the matrix, typically for the thick laminate in different curing temperatures.

Using numerical analysis to simulate the curing deformation of the composite can help the tooling mould design and optimize the production process. There are studies that has been focused on the influence of mould on the curing deformation [7,8]. They provide methods of componentise design on the mould to reduce the risk in the bulk production.

In this paper, the numerical method is targeted on an available production problem during the manufacturing. As the expansion coefficients of the warp and weft are different, classical laminate theory is employed to generate the CTE. Interaction between mould and CFRP laminate are created in the numerical mode and boundary conditions are altered in the simulation steps. The trend of laminate optimization for the curing deformation is studied with the simulation result.

2 SIMUATION OF CURING DEFROMATION

2.1 Structure properties and manufacturing process

The structure of the stiffened panel consist with three "T" shaped stiffeners and a rectangle skin that made out of CFRP (Figure 1). Ply sequence in the skin and stiffener are both symmetry and balance. The skin thickness and the basic dimensions are shown in the Table 1.

The stiffened panel is manufactured with co-bonding technique, where the skin is cured first, and then the wet stringers are co-bounded to the skins. Therefore this modelling method targets at the internal thermal stress on the panel during the curing process. The nonlinear behaviour of boundary conditions in the production process is simulated through mulit-step analysis. When the panel experiences the second autoclave cycle, the thermal mismatch between the metal mode and composite panel introduces internal stress on the skin. Those stresses are attended to deform the panel after the connection between the panel and mode is removed.

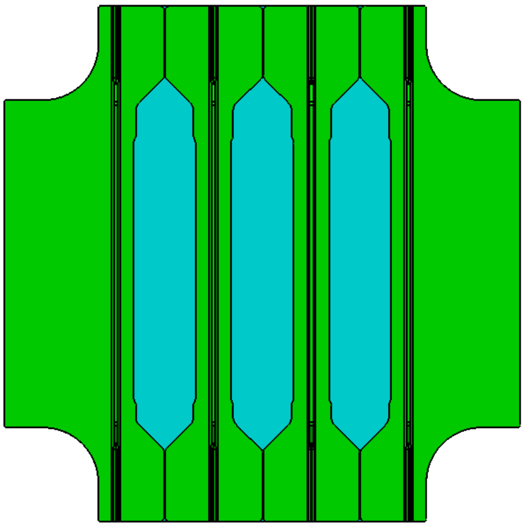


Figure 1: CFRP Stiffened panel made with autoclave

Terms	Unit	Magnitude
Length	mm	743
Width	mm	743
Thickness	mm	6.876
E_1	GPa	150
E_2	GPa	10
v	-	0.3

Table 1: Properties of the Stiffened panel

2.1 Laminate properties

Based on the classical laminate theory a thermal elasticity function for the composite structure can be formed as such

$$\begin{Bmatrix} N^T \\ M \end{Bmatrix} = \begin{Bmatrix} A \ B \\ B \ D \end{Bmatrix} \begin{Bmatrix} \varepsilon^0 \\ \kappa \end{Bmatrix}$$

Where N^T represents the thermal load caused by the different thermal expansion, A B D are the sub matrices of laminate stiffness. Matrix A B and D define the membrane coupling and bending stiffness of the laminate which formed as

$$A_{ij} = \sum_{k=1}^{n} Q_{ij}(z_k - z_{k-1})$$

$$B_{ij} = \sum_{k=1}^{n} \frac{Q_{ij}}{2}(z_k^2 - z_{k-1}^2)$$

$$D_{ij} = \sum_{k=1}^{n} \frac{Q_{ij}}{3}(z_k^3 - z_{k-1}^3)$$

For symmetry and balance plate, the coupling matrix B and terms D_{16} and D_{26} in the bending stiffness matrix are zero. The curing deformation of the stiffened panel is bending upward, namely the thermal stress is mainly act as bending force. It is clear that if the stacking sequence of the laminate is altered then the B and D matrix change. Also the CTE in warp and weft direction increases with respect to the percentage of ply in the corresponding direction. Therefore under the same manufacturing method, the curing deformation of plate can be improved by optimize the stacking sequence.

2.2 Thermal field and Heat transfer

Composite panel is curried in the autoclave and during the process neither the temperature nor pressure is unstable. Therefore the heat transfer over the composite structure can be a simulated through the by

$$\frac{\partial}{\partial x}\left[k_x \frac{\partial T}{\partial x}\right] + \frac{\partial}{\partial y}\left[k_y \frac{\partial T}{\partial y}\right] + \frac{\partial}{\partial z}\left[k_z \frac{\partial T}{\partial z}\right] = \rho C_T \frac{\partial T}{\partial t}$$

The right hand side of the equation is the energy from the environment that k_x k_y and k_z are thermal-conductivity coefficients on three directions. Relatively, the left and side is the temperature of difference in material, where ρ is the material density and C_T is the specific heat.

2.3 Residual stress after cure

Residual stress build up in the curing process normally consist with internal thermal expansion and matrix shrink. In this case the skin of panel is cured already, the matrix shrink can only appear on the stiffener. Also the stiffness of the cover does not change during the curing cycle and viscoelasticity model can be reduce to

$$\sigma_{ij} = D_{ij}\alpha_{ij}\Delta T$$

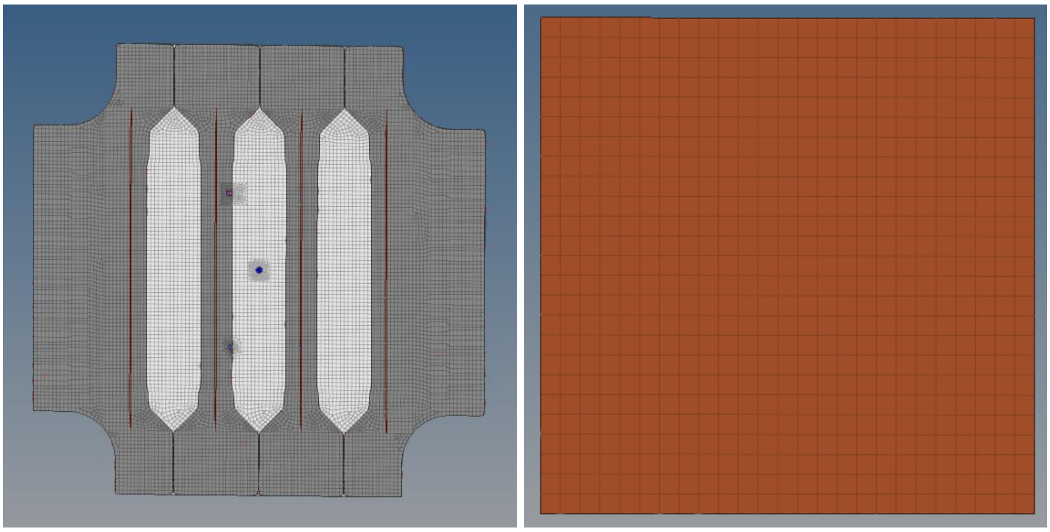
Where D_{ij} are the membrane stiffness terms and α_{ij} are the CTE difference between the mould and composite panel. Residual stress from thermal expansions can be directly acquired from the temperature difference.

2.3 Mould effectives

Previous research shows the thermal expansion of metal mould and composite are acting in the same direction. The shrinking magnitude of metal mould larger than the composite which tend to stop the panel from shrink. However the composite skin is tied up on the mould in this case. During the simulation the edge between panel and mould is consider to be coupled.

3 NUMERIAL MODE

This simulation is targeted at predicting the cured deformation after mould unloading. The numerical mode for the stiffened panel consist with 37209 four nodes shell element which shows in the Figure 2a. Same type of element is used on the steel mould, however the mesh density is much smaller than the stiffened panel. It has 625 shell element and the element property is rigid to save the computational resources (Figure 2b).



(a) (b)

Figure 2: Numerical mode

Residual stress in the composite panel due to the thermal expansion is address with the THERMAL module in the ABAQUSE. The CTE properties of the composite panel and temperatures in different stages are shown in the Table 2, where the T_{inital} represents the maximum temperature in the autoclave and T_{end} is the temperature at the end of curing cycle. The CTE_{ij} in the table represent the expansion coefficients of stiffened panel in longitudinal and horizontal direction.

Dimension	Unit	Magnitude
T_{inital}	℃	180
T_{end}	℃	20
CTE_1	m/℃	3.2e-06
CTE_2	m/℃	3.1e-05

Table 2: Thermal Properties & Load

3.1 Simulation steps

When the stiffened panel is finished with curing, the residual stress is gathered in the skin of panel. Then the structure is separated from the steel mould with tools, during this process the residual stress start to release and deformed the panel as the Figure 3 shows

Thus, the simulation content with two steps: First step the thermal load is acting in the system and created a uniform temperature decreases on the composite panel. Second step, the composite panel is a free body where only the geometry middle point fixed, thermal stress simulated in the first step is input to the second step as the load. The composite panel deformed under this internal stresses, and the displacement is the curing deformation.

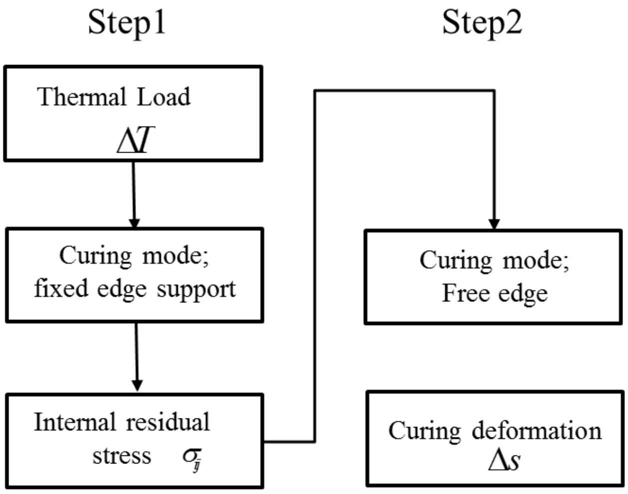


Figure3 Simulation process.

3.2 Boundary conditions

The boundary condition of the numerical mode is altered as there are two steps in the simulation. In the first step, the composite panel is tie to the mould and simple supported boundary condition is addressed around the four sides of the panel. Then the tie is replaced by a hard contact interface in the second step which allow the composite panel to deform. Composite panel is symmetrized about the geometry centre point, so it assumed to be fixed at the centre.

4 RESULT VALIDATION

The cure deformation of the composite panel is measured with feeler. Figure 4 indicates the magnitude of curing deformation in along the panel edge. It can be realized that the panel has been bended towards the center. Since the bending stiffness along the stiffness direction is much greater than it on the perpendicular direction, the displacement on web of stiffener is larger. Maximum deflection is reached 7.8mm.

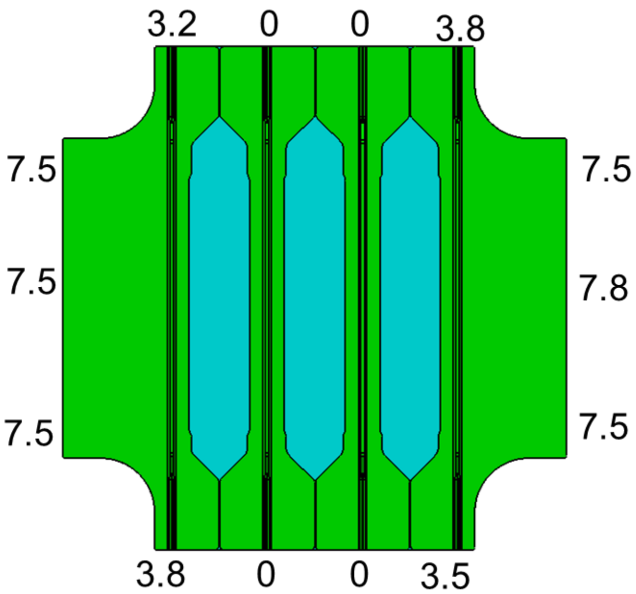


Figure 4: Curing deformation

The numerical simulation is performed to analysis the internal stress and curing deformation of the composite panel. Experimental result that shows in the Figure 4 is used to validate the simulation result. Then a sensitivity study is performed on the stacking sequence.

4.1 Validation

Step 1: The numerical result is generated by the ABAUSE, the residual strain generate by the thermal environment in the autoclave is shown in the Figure 5. Composite panel edges are constrained during this process, therefore the residual stress is build up at the center of the skin. The internal strain is symmetrically distributed about the skin center and superimposed over different directions. Therefore the highest deflection is located in the middle of the panel.

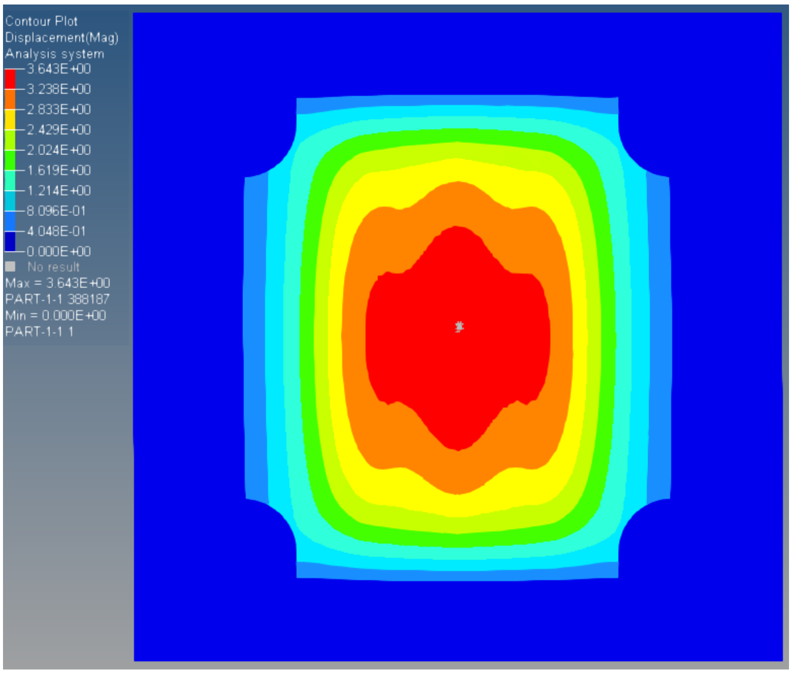


Figure 5: Residual stress distribute

Step 2: Simulation is restated based on the Step1, the residual stress from the Step1 is the load input. Boundary conditions on the mode is changed, constrains along the edges are removed and replaced by a signal point constrain at the middle. During this step the composite panel is separated

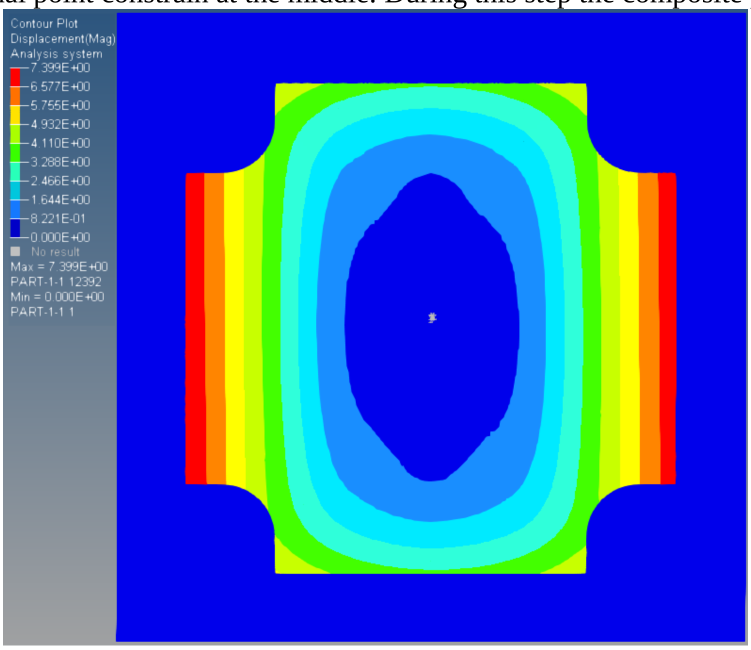


Figure 6: Simulated curing deformation result

from steel mould. Curing deformation of the panel from the simulation shows in the Figure 6. Result indicate after connection between panel and mould is removed, residual stress will release from the edge of panel. The maximum deflection appears at the edges parallel to the stiffeners. Curing deformation trend of composite panel is captured by the simulation. The maximum deflection is 7.2mm compare with the experimental data the difference is 4%. Thus the numerical model successfully recurring the curing deformation of the composite panel.

4.2 Effect of laminate sequence

In the section 2 the relation between the CTE and laminate stacking sequence indicates that by altering the orientation of ply CTE and bending stiffness can be changed. So to study it effects on the curing deformation, the skin laminate stacking sequence is optimized for increasing the bending stiffness along horizontal direction. One 0° ply at the symmetry face is change to 90°, this change would also balance the CTE in two different directions and also maintain the symmetry and balance sequence.

Simulation result is generate with the validated model with the new stacking sequence. Figure 7 shows the result of predicated curing deformation where the maximum deflection has been reduced to 3.2mm. Compare it with the original 7.8mm, curing deformation has been reduced by 58%.

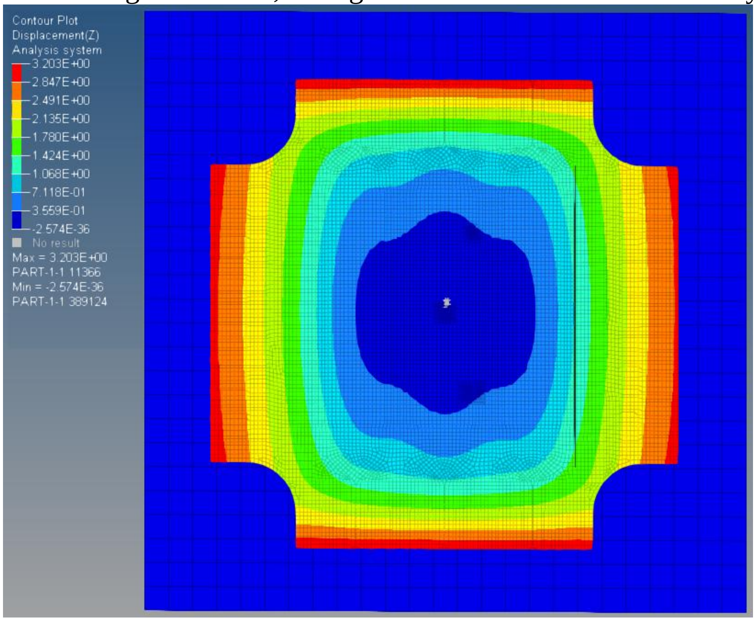


Figure 7 Curing deformation after laminate optimization

5 COULUTION

Composite panel has different CTE in warp and weft direction which would cause the unbalanced displacement in the autoclave. For co-bonding manufacturing technic, the curing deformation happens after the skin is released from the mould. In the modelling process thermal load is simulated in the step1, then in step2 the curing deformation predicated. Numerical result on maximum curing deformation and trend of deflection shows a good agreement with the experimental result. Finally in the sensitivity study, the curing deformation has been reduced more than half by altering the stacking sequence.

REFERENCES

[1] A.Johnston, R. Vaziri, A.Pourartip. A Plane Strain Model for Process-induced Deformation of Laminated Composite Structures. Journal of Composite Materials. 2001, 35(16):1435-1469

[2] Lange J, Toll S, Manson J , Hult A Residual Stress Build-up in Thermoset Films Cured above Their Ultimate Glass Transition Temperature. Polymer. 1995, 36(16):3135-3141

[3] S. R. White and H. T. Hahn. Process Modeling of Composite Materials: Residual Stress Development during Cure. Part I : Model Formulation. Journal of Composite materials 1992. 26(16): 2402-2422

[4] S. R. White and H. T. Hahn. Process Modeling of Composite Materials: Residual Stress Development during Cure. Part II : Experimental Validation. Journal of composite materials. 1992. 26(16):2423-2454

[5] T. A. Bogetti and J. W. Gillespie, Jr. Two-Dimensional Cure Simulation of Thick Thermosetting Composite. Journal of Composite Materials. 1991,25(3):239-273

[6] T. A. Bogetti and J. W. Gillespie and R. I. MeCullouqh. Iinfluence of Processing on the Development of Residual Stresses in Thick Section Thermoset Composites International Journal of Materials & Product Technology. 1994, 9(1-3):170-182

[7] G.Twigg, A. Poursartip and G. Fernlund. Tool-part Interaction in Composite Processing. Part I: Experimental Investigation and analytical Model. Composites Part A. 2004. 35:121-133

[8] G.Twigg, A. Poursartip and G. Fernlund. Tool-part Interaction in Composite Processing. Part II: Numerical Modeling. Composite Part A.2004, 35 135-141