APPLICATION OF THE FINITE ELEMENT METHOD IN ANALYZING PLATE INTERACTION SYSTEMS USING COMPOSITE MATERIALS IN THERMAL ENVIRONMENTS

● HONG THAO PHAM - VAN THOAI LE

ABSTRACT:

The use of composite materials is widespread in various civil applications, especially in the construction of large billboards. However, when exposed to pressure loads and temperature variations, these materials can experience sagging, deformation, and a reduction in mechanical properties. This study employs the finite element method to analyze these properties and calculate the sag of composite panels under the influence of mechanical loads and temperature changes. The panel elements are constructed using a four-node isoparametric element, each with five degrees of freedom. The numerical results are compared for different scenarios and show that for composite panels with symmetrical structures under uniformly distributed loads, the deformation components of the symmetrical layers are identical. In panels with any structure, the sag on the average plane of the panel in the z-direction is the largest, while the sag in the x and y directions and the rotation angle around the x and y axes are minimal. Additionally, the study finds that as temperature increases, sag decreases, while an increase in the applied load leads to increased sag. Moreover, an increase in the number of layers in the plate decreases the deflection in the z-direction, and a higher fiber product factor results in greater deflection. This analysis is applied to a practical problem involving a billboard measuring 2440 mm x 4880 mm at three locations: the corner, middle edge, and center of the plate. The results are compared with previous studies to validate the reliability of the algorithm and program.

Keywords: composite material, composite panel, pressure load, temperature variations, finite element method, deflection.

1. Introduction

Composite materials are created by combining two or more different materials to produce new materials with unique properties. A material mixture consists of two phases: a continuous phase and an intermittent phase. The intermittent phase, known as the core, bears force, while the continuous phase, known as the background, provides support, protection, and transmission of the center of gravity [1]. Due to their numerous advantages, composite materials are extensively utilized in various industries such as aviation, aerospace, shipbuilding, automobiles, mechanical engineering, and civil construction, as well as in everyday use.

When composite panels are subjected to loads in varying temperature environments, they tend to sag, deform, and their mechanical properties reduce [2,3]. To address this issue, the location of cracks and the mechanical behavior of layered composite panel details and structures need to be accurately determined in order to predict the structure's current working capacity and find solutions to prevent potential damage. Among the new methods being researched and applied today, the Finite Element Method (FEM) [4-7] is proving to be effective in solving these problems. This article utilizes the finite element method and applies plate theory based on Mindlin's first-order displacement model to analyze the displacement of composite plates under changing thermal environments, applied loads, and sheet material parameters, as well as varying number of layers.

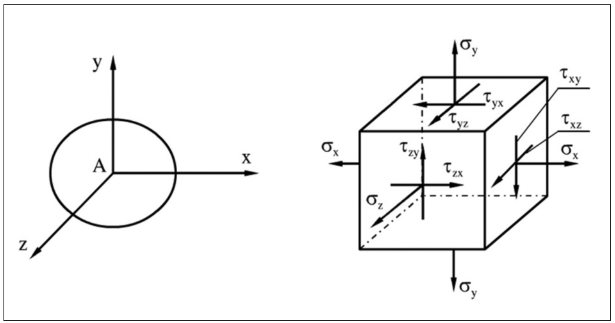
2. Theoretical basis

a. Elastic theory

In general, stress and strain on objects consist of six components (see figure 1).

For stress, $\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx}$ corresponds to normal stress in the x, y, z directions and subsequent stress in the z, x, y directions. For deformation, $\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}$ corresponds to normal tensile deformation in the x, y, z directions and shear deformation in the z, x, y directions [5].

Figure 1: Application components and variables



** Stress - strain - temperature relationship*

For isotropic and elastic materials

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} 1/E & -\nu/E & 0 \\ -\nu/E & 1/E & 0 \\ 0 & 0 & 1/G \end{bmatrix} . \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} + \begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} \quad (1)$$

Write in matrix form

$$\{\varepsilon\} = [E]^{-1}\{\sigma\} + \{\varepsilon_0\} \quad (2)$$

In Which $\{\varepsilon\}$ is the initial deformation vector; [E] is the elastic coefficient matrix (or behavior matrix); E is the elastic modulus; ν is Poisson's coefficient; G is the slip modulus.

With: $$G = \frac{E}{2(1+\nu)} \quad (3)$$

We can express the stress components in terms of strain by solving equation (1).

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} \quad (4)$$

or $\{\sigma\} = [E]\{\varepsilon\} + \{\sigma_0\}$ (5)

$\{\sigma_0\} = -[E]\{\varepsilon_0\}$ represents the initial stress

The initial strain is determined $\{\varepsilon_0\}$ by the temperature.

$$\begin{Bmatrix} \varepsilon_{x0} \\ \varepsilon_{y0} \\ \gamma_{xy0} \end{Bmatrix} = \begin{Bmatrix} \alpha\Delta T \\ \alpha\Delta T \\ 0 \end{Bmatrix} \quad (6)$$

In which, α is the thermal expansion coefficient and ΔT temperature change

b. Plate theory

A plate is a structure that is bounded by two parallel planes and separated by a distance h, which is known as the plate thickness. When a thin plate is subjected to bending due to forces acting perpendicular to the plate plane, the Oxyz coordinate system is selected such that the Oxy plane aligns with the middle surface of the plate, and the z-axis is perpendicular to the plate plane (see Figure 2)

Let's consider a plate structure (see Figure 2) that is being subjected to a surface load *p(x, y)* acting perpendicular to the middle plane. The distribution of *p(x, y)* is arbitrary at this point, and no specific dimensions or boundary conditions are provided. Now, let's cut out an infinitesimal section element with dimensions dx and dy from the plate and analyze the local equilibrium conditions. At the negative sectional edges, we have the applied moment flows M^0_{xx}, M^0_{yy}, M^0_{xy}, and transverse shear force flows Q_x, Q_y. At the positive sections, we have the respective infinitesimal increments of these quantities [9]

$$M^0_{xx} + \frac{\partial M^0_{xx}}{\partial x}\, dx,\quad M^0_{yy} + \frac{\partial M^0_{yy}}{\partial y}\, dy,$$

$$M^0_{xy} + \frac{\partial M^0_{xy}}{\partial x}\, dx,\quad M^0_{xy} + \frac{\partial M^0_{xy}}{\partial y}\, dy,$$

$$Q_x + \frac{\partial Q_x}{\partial x}\, dx \text{ and } Q_y + \frac{\partial Q_y}{\partial y}\, dy,$$

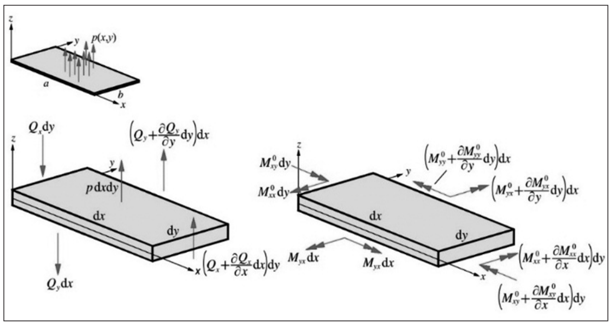
are applied. The sum of transverse forces is obtained from this

$$\left(Q_x + \frac{\partial Q_x}{\partial x}\, dx\right) dy + \left(Q_y + \frac{\partial Q_y}{\partial y}\, dy\right) dx$$
$$- Q_x\, dy - Q_y\, dx + p dxdy = 0 \qquad (7)$$

or after divison by da = dxdy:

$$\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} + p = 0$$

Figure 2: Local equilibrium at an infinitesimal section element



In the same way, the shear force components Q_x, Q_y are determined.

$$\frac{\partial M^0_{xy}}{\partial x} + \frac{\partial M^0_{xy}}{\partial y} = Q_x\ ;$$
$$\frac{\partial M^0_{xy}}{\partial x} + \frac{\partial M^0_{yy}}{\partial y} = Q_y \qquad (8)$$

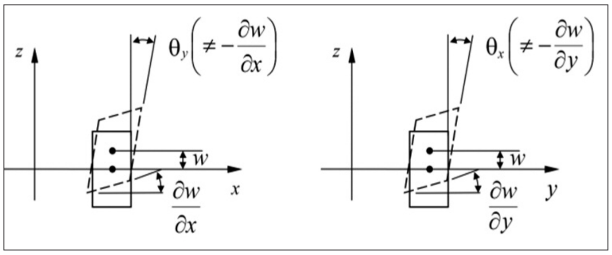
c. Reissner – Mindlin plate theory

If the plate thickness is not small, the Reissner-Mindlin plate theory is applied. This theory calculates the angular deflection of the cross section [9]:

$$\gamma_{xz} \neq 0, \gamma_{yz} \neq 0 \qquad (9)$$

This implies that the lines that are at right angles to the middle plane stay straight when the material deforms, but they are no longer at right angles to the middle plane. The total rotation angle of the section is made up of two parts: the first part is due to the bending of the plate when the lines are still at right angles to the middle plane; the second part is due to the average shear deformation (see figure 3)

Figure 3: Rotational angle of normals and shear deformation of cross-section



The average shear strains, denoted as γ_{xz} , γ_{yz}, for the corresponding cross-section xy, are calculated.

$$\gamma_{xz} = \frac{\partial w}{\partial x} - \theta y\ ;\quad \gamma_{yz} = \frac{\partial w}{\partial y} - \theta x \qquad (10)$$

In the thin plate problem, the shear strains approach zero and then:

$\theta_x \to \partial w/\partial y$ and $\theta_y \to \partial w/\partial x$.

The relationship between shear stress (τ_{xz} , τ_{yz}) and shear strain (γ) according to Hooke's Law is as follows.

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \frac{E}{2(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \qquad (11)$$

The average shear strain (γ_{xz} , γ_{yz}) is assumed to be constant across the thickness of the plate. As a result, the combined effect of these shear stresses on the curved surface of the section produces the shear force (Q_x, Q_y), which is associated with the shear strain in the following manner:

$$\begin{Bmatrix} Q_x \\ Q_y \end{Bmatrix} = \frac{Eh\alpha}{2(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \qquad (12)$$

or $\{Q\} = h\alpha\, [D_c]\, \{\gamma\}$ (13)

In there:

$$[D_c] = \frac{E}{2(1+\nu)} \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

× is the matrix that represents the material's response to shear. (14)

α represents the slip correction factor (shear correction factor).

d. 4-node quadrilateral element

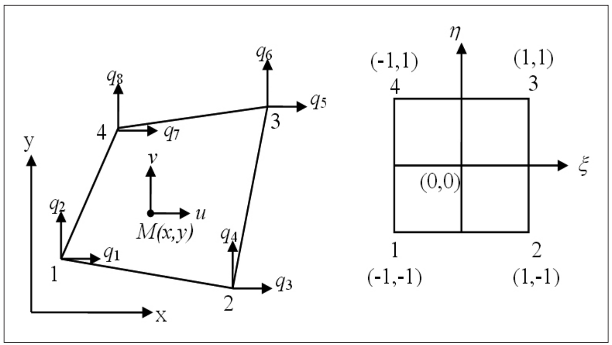
Take a general quadrilateral element shown in Figure 4. The element has four nodes [3,4] numbered 1, 2, 3, and 4 in a counterclockwise direction. Each node has 2 degrees of freedom, and their coordinates are represented by (xi, yi). The nodal displacement vector of the element is denoted by:

$$q = \{ q_1 \ q_2 \ ... \ q_8 \}^T \qquad (15)$$

The displacement of any point M(x, y) in the element is represented by the variable u.

$$u = [u(x, y), v(x, y)]^T \qquad (16)$$

Figure 4: 4-node quadrilateral element



3. Applied problem

The composite panel has the dimensions shown in Figure 5: panel length (L) = 0.8m, panel width (W) = 0.8m; each layer thickness (h) is 0.005m. It has a symmetrical configuration and is subjected to an evenly distributed load (q) on the entire panel surface with q = 875000 N/m^2. The panel is clamped on all four sides.

Consider the force acting in one direction on the entire panel, with the position of the force being at the center of the panel. The mechanical properties of the layers are as follows: elastic modulus E1 = 150 GPa; E2 = 9.65 GPa; E3 = E2; G12 = 4.14 GPa; G13 = 4.14 GPa; G23 = 3.45 GPa; Poisson's ratio ν = 0.3; material density ρ = 1600 kg/m^3.

Average Linear Expansion Coefficient α $(^oC)^{-1}$ in direction 1 (OX): $\alpha_1 = 0.3 \times 10^{-6}$ $(^oC)^{-1}$

Average Linear Expansion Coefficient $\alpha(^oC)^{-1}$ in direction 2 (OY): $\alpha_2 = 0.5 \times 10^{-6}$ $(^oC)^{-1}$

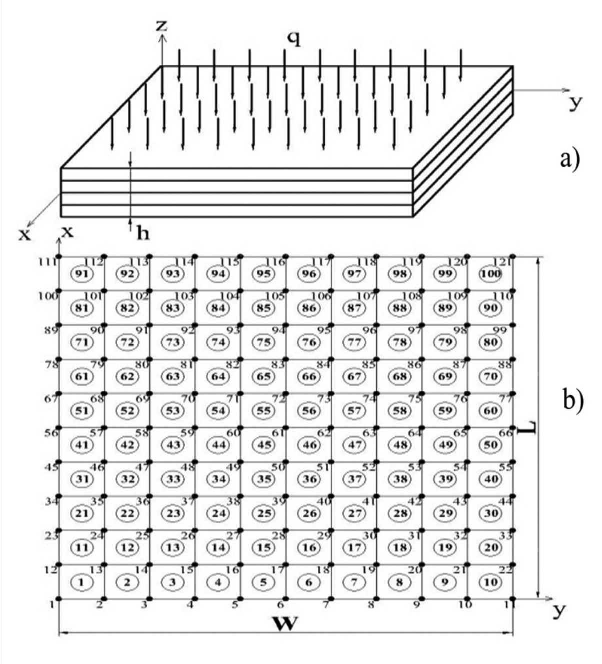
The calculation is performed for the case W/L=1 with the fiber arrangement of (90 degrees/0 degrees/0 degrees/90 degrees). Vf = 0.5; Vm = 0.5.

In our analysis, we use a 2 x 2 Gaussian point numerical integration for the bending stiffness matrix and a 1 Gaussian point numerical integration for the shear stiffness matrix. The boundary conditions are as follows:

- At the nodes located on the boundary edges parallel to the x-axis: u, w, and θx are all zero.
- At the nodes located on the boundary edges parallel to the y-axis: v, w, and θy are all zero.

Based on the problem model, the author calculates the nodal displacement of the element in the plate.

Figure 5: *a) Schematic diagram of the flexural composite plate; b) 10 x 10 element mesh on the composite plate*



3.1. Case 1: Calculate the deflection of the composite plate in three different scenarios, then compare the results with the article [10].

- Scenario 1: The plate is under uniform load with no temperature change (ΔT = 0°C).
- Scenario 2: The plate is under uniform load and subjected to a temperature change of 100 degrees C (ΔT = 100°C).
- Scenario 3: The plate is under a uniform load with no temperature change (ΔT = 0°C) and when subjected to a temperature change of 100°C.

Hypothesis 1: The plate is evenly loaded when there is no heat.

Deflection

The maximum deflection is w = 0.45 x10^-3 m

Compare the results of the thesis with the results in the paper [10] with the same data (same model).

The results in the reference paper w = 0.00045 m

Figure 6: Element mesh of composite panel

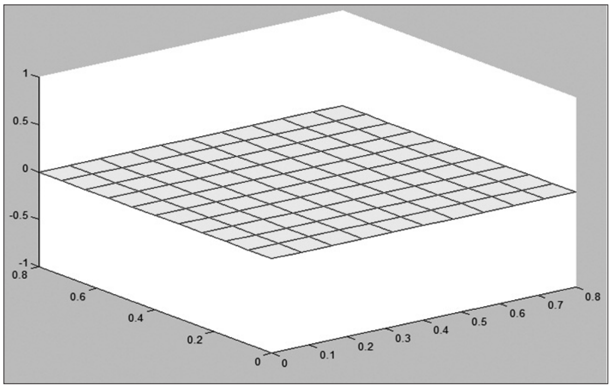
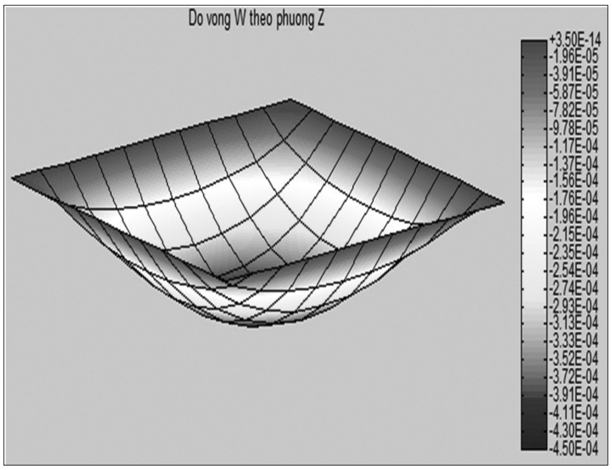


Figure 7: Composite panels sag under non-thermal loads



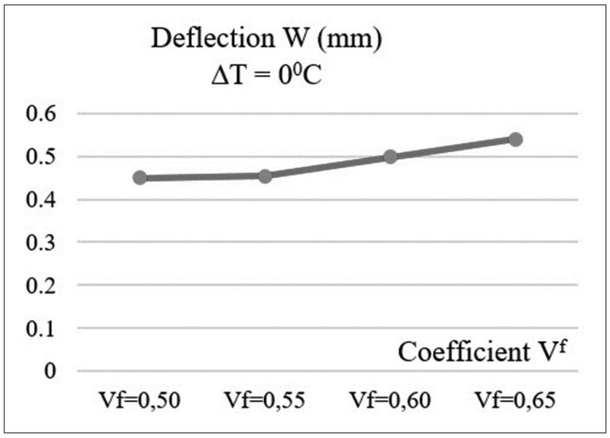
The results of the thesis w = 0.00045 m

Then the author maintained the applied load while altering the material parameter V^f and comparing the thesis results with those of the article [10].

Table 1: Deflection with ΔT = 0°C and different material parameters

V^f	$\Delta T = 0^oC$	
	Deflection (mm)	
	ref	Thesis
$V^f = 0{,}50$	0.450	0.450
$V^f = 0{,}55$	0.453	0.455
$V^f = 0{,}60$	0.498	0.499
$V^f = 0{,}65$	0.534	0.540

Graph 1: Relationship between coefficient Vf and deflection W with ΔT = 0°C

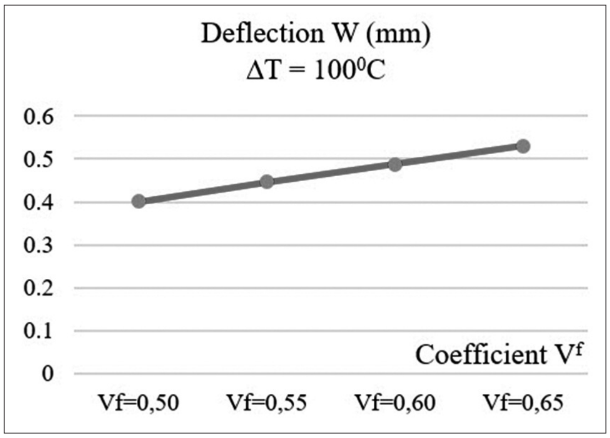


Hypothesis 2: A uniform load-bearing plate is subjected to a temperature change (ΔT) of 100^oC.

Table 2: Deflection with ΔT=100°C and different material parameters

V^f	$\Delta T = 100^oC$	
	Deflection (mm)	
	ref	Thesis
$V^f = 0{,}50$	0.400	0.401
$V^f = 0{,}55$	0.443	0.447
$V^f = 0{,}60$	0.485	0.488
$V^f = 0{,}65$	0.528	0.531

Graph 2: Relationship between coefficient Vf and deflection W with ΔT = 100°C

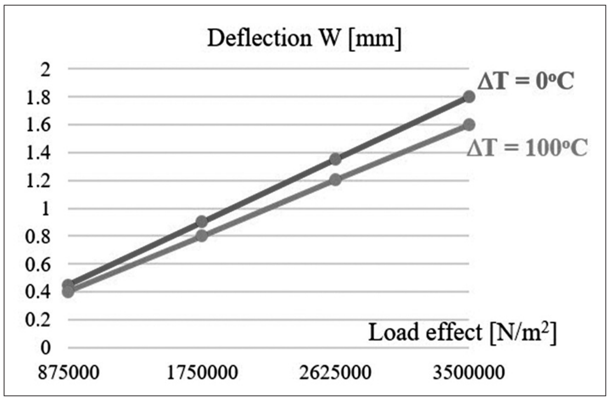


Hypothesis 3: The load plate changes when there is no heat ($\Delta T = 0^oC$) and when subjected to a temperature change of $\Delta T = 100^oC$.

Table 3: Deflection values with different applied loads.

Load effect N/m^2	$\Delta T = 0^oC$		$\Delta T = 100^oC$.	
	Deflection (mm)		Deflection (mm)	
	ref	Thesis	ref	Thesis
P = 875000	0.450	0.450	0.398	0.401
P x 2	0.896	0.900	0.800	0.801
P x 3	1.348	1.350	1.203	1.204
P x 4	1.798	1.800	1.597	1.600

Gaph 3: Effect of applied load on deflection W



3.2. Case 2: The author changes the temperature in some cases ΔT = [30^oC, 50^oC, 100^oC, 200^oC, 300^oC] *(Table 4, Graph 4)*

3.3. Case 3: The author increases the number of layers n to (4, 8, 12, 16, 32) respectively. The test temperature is $\Delta T=300^oC$. The fiber arrangement is [90 0 0 90]s. *(Table 5, Graph 5)*

4. Application problem

Composite panels, particularly advertising panels, are widely used across various fields. The figure 8 shows the boundary conditions of 4-sided clamps (bones). The author conducted a survey in Ho Chi Minh City, considering the acting force to be the wind force with an average speed of 6 m/s, equivalent to 220 N/m^2. The analysis temperature is under normal conditions, ranging from 19^oC to 40^oC, with the maximum surveyed temperature being 40^oC. External influences such as humidity are disregarded by the author.

The panel parameters are as follows:

- Panel length (L): 4.880m
- Panel width (W): 2.440m

Table 4: Deflection value with temperature change

Temperature ΔT	Deflection W [mm]
30^oC	0.0354
50^oC	0.0293
100^oC	0.0231
200^oC	0.0092
300^oC	0.0062

Graph 4: Effect of temperature on deflection W

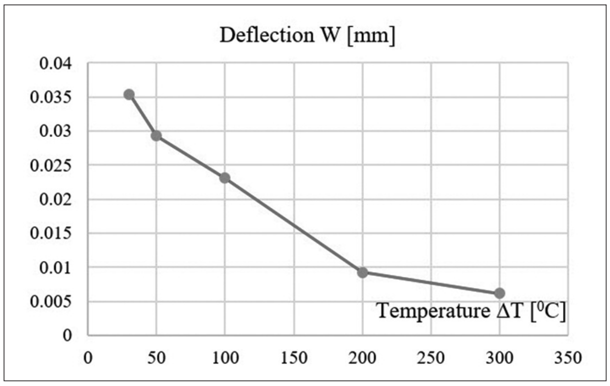
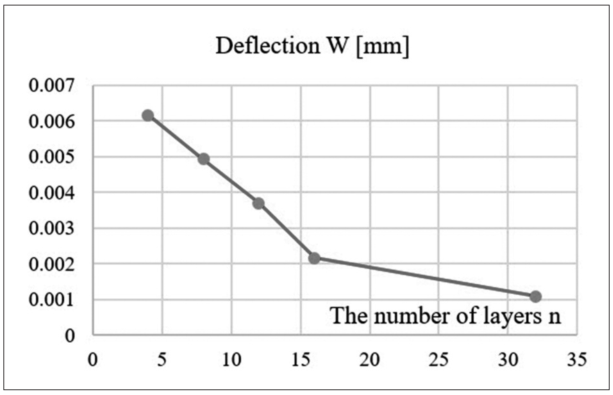


Table 5: Deflection values with varying number of layers

The number of layers n	Deflection W [mm]
4	0.00616
8	0.00493
12	0.0037
16	0.00216
32	0.00109

Graph 5: Effect of temperature on deflection W.



- Each layer thickness (h_k): 0.006m
- Total panel thickness (h): 0.048m
- The panel has 8 layers, with a symmetrical configuration, and the load is evenly distributed over the entire panel surface.

The mechanical properties of the layers are as follows:

- Elastic modulus (E1): 150 GPa
- Elastic modulus (E2): 9.65 GPa
- Elastic modulus (E3): E2
- Shear modulus (G12): 4.14 GPa
- Shear modulus (G13): G12
- Shear modulus (G23): 3.45 GPa
- Poisson's ratio (v): 0.3

Average Linear Expansion Coefficient $\alpha(^oC)^{-1}$ in direction 1 (OX): $\alpha_1 = 0.3 \times 10^{-6}\ (^oC)^{-1}$

Average Linear Expansion Coefficient $\alpha(^oC)^{-1}$ in direction 2 (OY): $\alpha_2 = 0.5 \times 10^{-6}\ (^oC)^{-1}$

The calculation is performed on the case of the fiber arrangement being (900/00/00/900). $V^f = 0.5$; Vm = 0.5. The author analyzes the displacement of the plate at 3 positions: Position 1 at the corner of the plate. Position 2 at the middle edge of the plate. Position 3 is the center of the plate

Figure 8: Application of composite panels in practice

4.1. Data analysis

When analyzing the problem data, the following information was gathered:

- Number of elements in the x direction: 10
- Number of elements in the y direction: 10
- Total number of elements = 100
- Number of nodes on 1 element = 4
- Total number of nodes in the whole system = (100+1)^2 = 121
- Number of degrees of freedom (BTDs) on 1 node = 5
- Number of BTDs on 1 element = 4x5 = 20
- Total number of BTDs in the whole system = 5x121 = 605

The author then analyzed the displacement of the plate at 3 specific locations. Each location cell has the following dimensions: length = 0.488m, width = 0.244m.

Figure 9: Schematic diagram of 8-layer composite plate subjected to bending

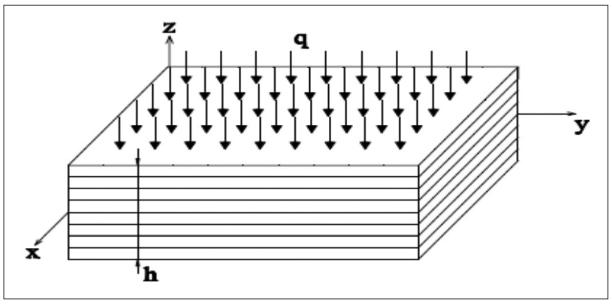
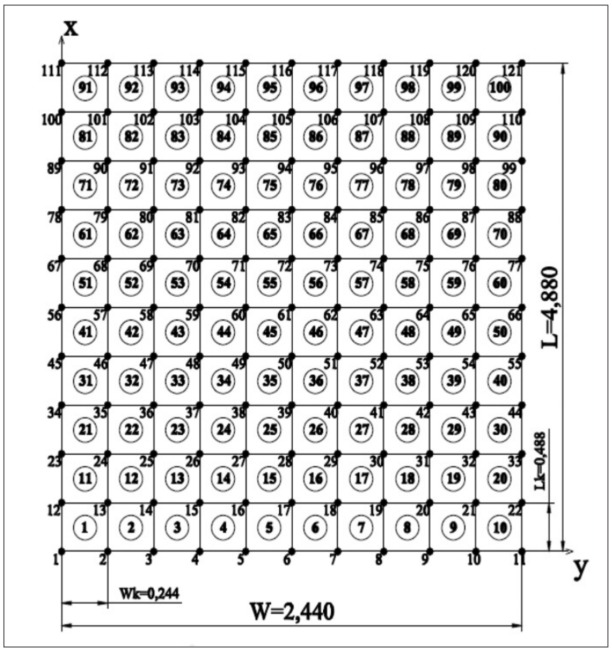


Figure 10: 10x10 element mesh on 8-layer composite plate



4.2. Calculation results in MATLAB

Mesh model with 100 elements:

❖ *Position 1 at the corner of the plate*

- Deflection
- The maximum deflection is w = 4.99x10-4m.

Figure 11: Element mesh of composite panel at panel corner

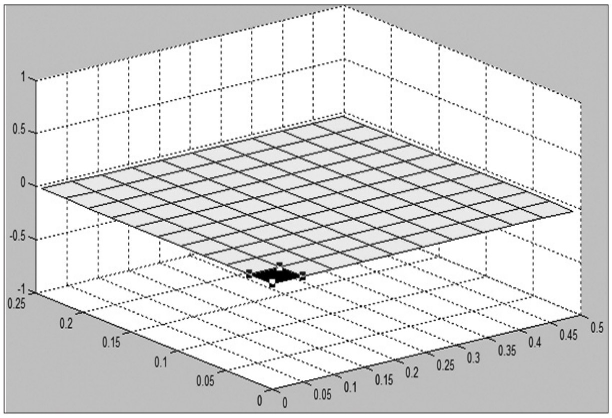


Figure 12: Composite panel sagging at the corner of the panel

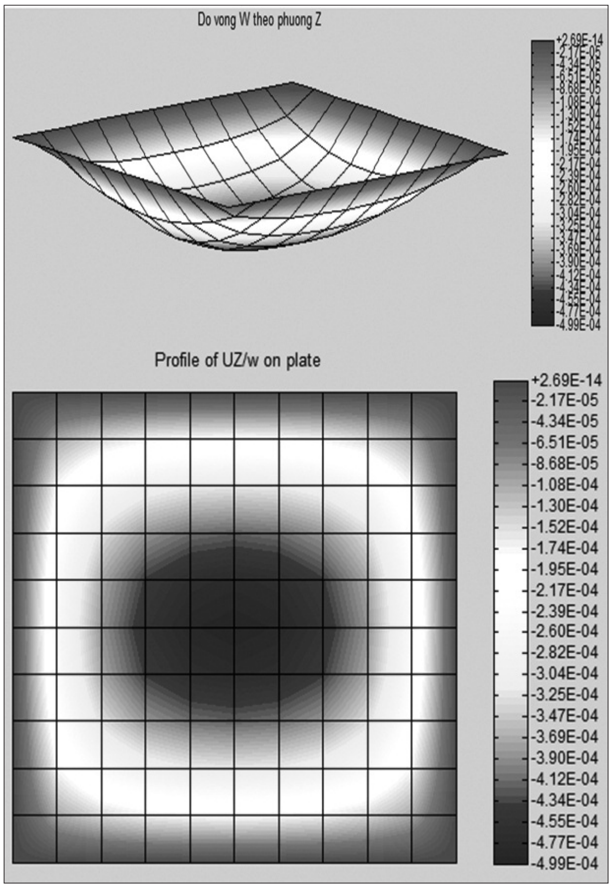


Figure 13: Element mesh of composite panel at the middle boundary of the panel.

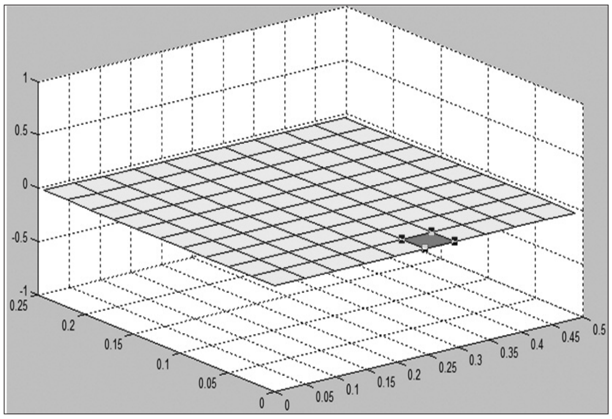
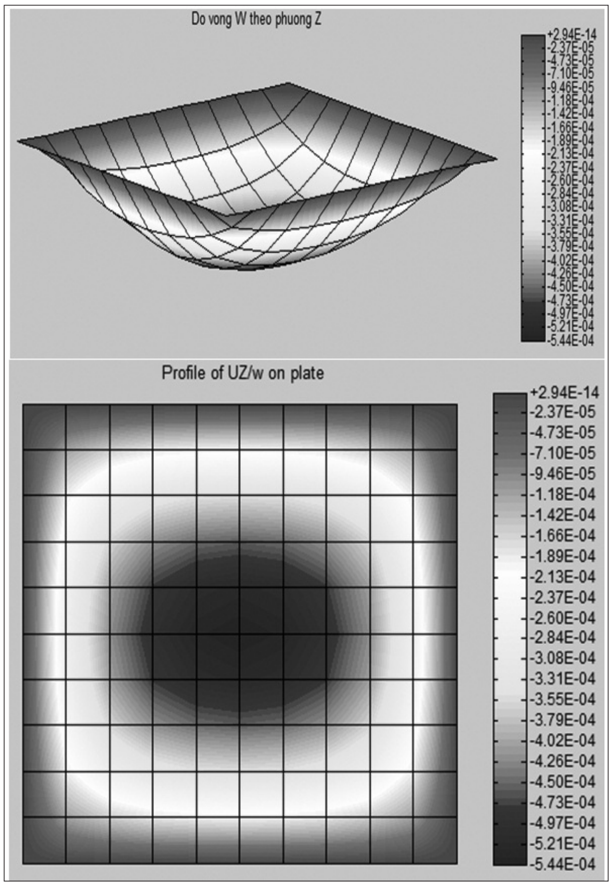


Figure 14: Composite panel with deflection at the middle edge of the panel



❖ *Position 2 at the middle edge of the plate*

- Deflection

- The maximum deflection is w = 5.44x10-4m.

❖ *Position 3 is the center of the plate*

- Deflection

- The maximum deflection is w = 9.07x10-4m.

4.3. Summary of analysis results

(Figure 17, Table 6, Graph 6)

Figure 15: Element mesh of composite plate at the center of the plate

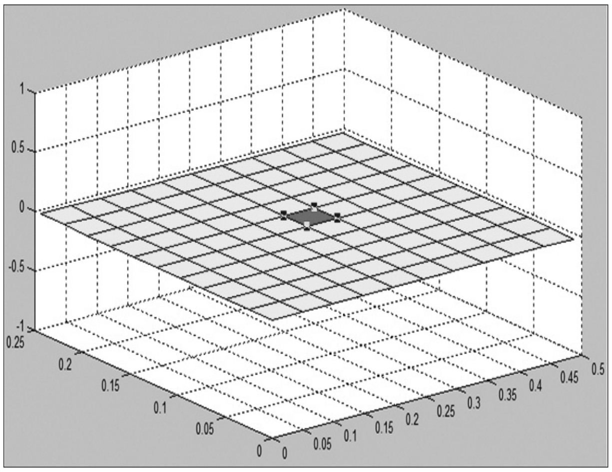


Figure 16: Composite panel sagging at the center of the panel

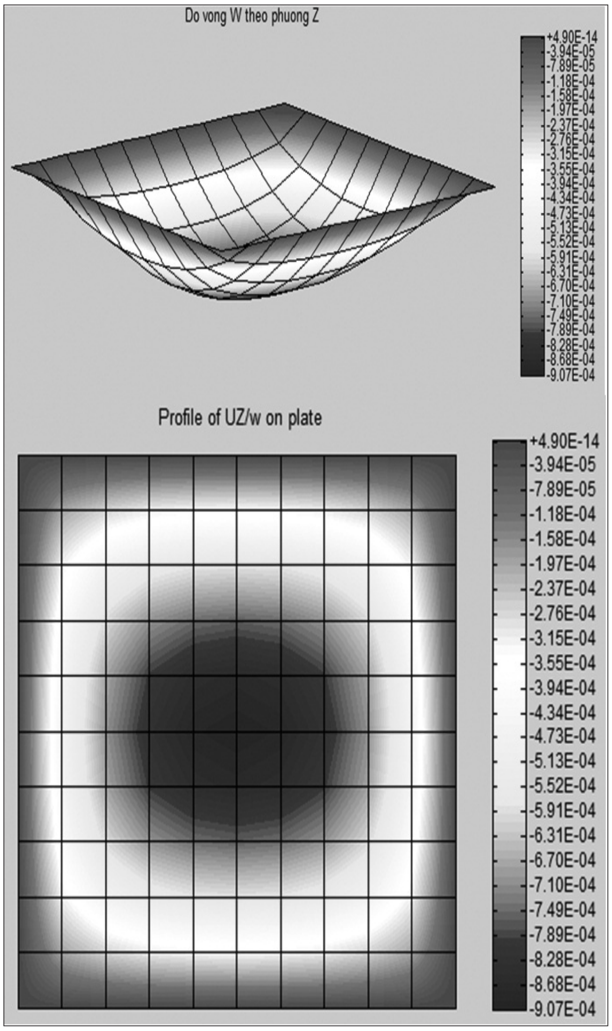


Figure 17: Element mesh of composite plate at 3 positions of the plate

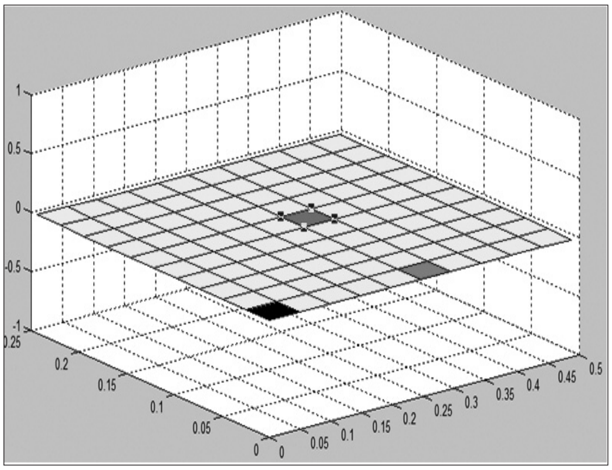
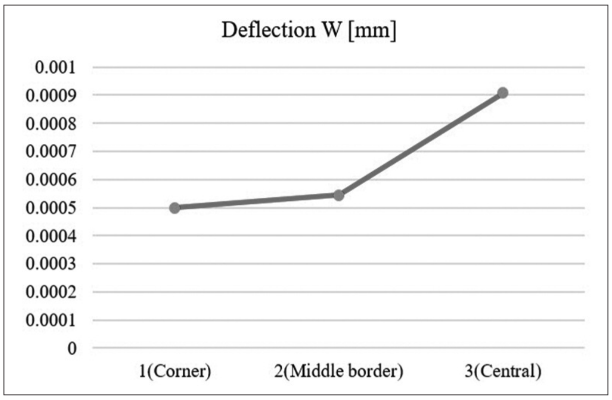


Table 6: Deflection values with changing analysis position

Position	Deflection W [mm]
1(Corner)	0.000499
2(Middle border)	0.000544
3(Central)	0.000907

Graph 6: Effect of changing analysis location on deflection W



5. Conclusion

When analyzing a composite plate in a thermal environment, the author used the finite element method and the Matlab programming language to find the displacement of element nodes. It was concluded that for a plate with a symmetrical structure, subjected to evenly distributed forces, the deformation components of the symmetrical layers are equal, so the author only calculates on the average surface of the plate. For a plate with any structure, the deflection on the average surface of the plate in the z direction is the largest, while the deflection in the x and y directions is very small. The rotation angle around x and y is also very small. When the temperature increases, the deflection decreases. Additionally, the deflection increases with an increase in load, but decreases with an increase in the number of layers in the plate. Furthermore, the deflection increases with an increase in the fiber volume coefficient. It is recommended to arrange multi-layer composite plates with the same size and the same mass of material. The conducted analysis also included practical application, specifically analyzing an advertising panel using composite materials to predict the working capacity of the structure and to find solutions to prevent possible damage ■

REFERENCES:

1. Ngo Tri Dung, (2020). Vật liệu Composite và Nanocomposite - Từ Kiến thức đến Ứng dụng Công nghiệp. IntechOpen.

2. M.W. Hyer, (2009). Stress Analysis of Fiber-Reinforced Composite Materials, McGraw-Hill, Virginia Polytechnic Institute and State University, New York.

3. Nguyen Hoai Son, Mai Thanh Phong, Mai Duc Dai, (2011). Ứng dụng phương pháp phần tử hữu hạn trong tính toán kết cấu. National University Publishing House, Ho Chi Minh City.

4. Nguyen Hoai Son et al, (2018). Phương Pháp Tính Ứng Dụng Với Matlab. National University Publishing House, Ho Chi Minh City.

5. Young W. Kwon, Hyochoong Bang, (2017). The Finite Element Method Using MATLAB. CRC Press.

6. Tran Ich Thinh, Ngo Nhu Khoa, (2007). Phương pháp phần tử hữu hạn. Hanoi Publishing House.

7. Nhu Phuong Mai, (2009). Lý thuyết đàn hồi. Education Publishing House.

8. S.C. Tan, (2017). Stress Concentration in Laminated Composite. England, UK: Routledge.

9. Christian Mittelstedt, (2023). Theory of Plates and Shells. Berlin, Germany: Springer.

10. Rajiv Kumar, Amit Sharma, Rajesh Kumar, (2013). Thermal Buckling Analysis of a Laminated Composite Plate Resting on Elastic Foundation Using Finite Element Method Based on Micromechanical Model, International Journal of Engineering, Business and Enterprise Applications (IJEBEA), India.

Received date: August 1, 2024
Reviewed date: August 5, 2024
Accepted date: August 20, 2024

Author information:

1. PHAM HONG THAO[1]

2. LE VAN THOAI[1]

[1]Faculty of Engineering and Technology
Nguyen Tat Thanh University

ÁP DỤNG PHƯƠNG PHÁP PHẦN TỬ HỮU HẠN TRONG PHÂN TÍCH ỨNG XỬ CỦA HỆ TƯƠNG TÁC TẤM SỬ DỤNG VẬT LIỆU COMPOSITE TRONG MÔI TRƯỜNG NHIỆT

● PHẠM HỒNG THAO[1]
● LÊ VĂN THOẠI[1]
[1]Khoa Kỹ thuật - Công nghệ
Trường Đại học Nguyễn Tất Thành

TÓM TẮT:

Vật liệu composite được sử dụng rộng rãi trong nhiều lĩnh vực dân dụng, điển hình là trong các tấm biển quảng cáo cỡ lớn. Khi vật liệu này hoạt động trong môi trường chịu tải trọng áp suất và nhiệt độ thay đổi, nó có xu hướng bị võng, biến dạng và giảm cơ tính. Nghiên cứu này phân tích các đặc tính trên dựa vào phương pháp phần tử hữu hạn để tính toán độ võng của tấm composite dưới tác động của tải trọng cơ học và nhiệt độ. Phần tử đẳng tham số bốn nút, mỗi nút có năm bậc tự do, được sử dụng để xây dựng phần tử tấm. Các kết quả số được khảo sát cho các trường hợp khác nhau và so sánh cho thấy, đối với tấm composite có kết cấu đối xứng chịu lực phân bố đều, các thành phần biến dạng của các lớp đối xứng là như nhau. Với tấm composite có cấu trúc bất kỳ, độ võng trên mặt phẳng trung bình của tấm theo phương z là lớn nhất, trong khi độ võng theo phương x và y rất nhỏ, và góc xoay quanh trục x và y cũng rất nhỏ. Khi nhiệt độ tăng, độ võng giảm, nhưng khi tải trọng tác dụng tăng, độ võng lại tăng. Khi số lớp trong tấm tăng, độ võng theo phương z giảm, và hệ số tích sợi càng lớn thì độ võng càng tăng. Phân tích độ võng này được áp dụng vào bài toán thực tế là tấm biển quảng cáo có kích thước 2440 mm x 4880 mm tại ba vị trí: Vị trí 1 là góc tấm, vị trí 2 là cạnh giữa tấm, và vị trí 3 là tâm tấm. Các kết quả được so sánh với các nghiên cứu trước đó cho thấy độ tin cậy của thuật toán và chương trình.

Từ khóa: vật liệu composite, tấm composite, tải nhiệt, phần tử hữu hạn, độ võng.