



Mohamed Khider University of Biskra
Faculty of Science and Technology
Mechanical Engineering.

Field: Sciences and Techniques

Sector: Mechanical Engineering Specialty: Mechanical Construction

Ref.:

Presented and supported by: **BECHA HOUSSEMEDDINE.**

The date: 17/09/2020.

Title

Numerical modelling of vibration plate composite

Jury:

Dr. Arfaoui Benarfaoui	MCB	University of Biskra	President
Dr. Bilhi Guerira	MCB	University of Biskra	Supervisor
Dr. Dejmoui Mohamdi	MCB	University of Biskra	Examiner

Academic year: 2019 – 2020.

acknowledgments

acknowledgments

Praise is to Allah, by whose grace good deeds are accomplished and may God's prayers and peace is upon His Prophet.

The writer would like to thank everyone who contributed his effort, thought and time in producing this memorandum in the required form. This concerns colleagues and professors at the Department of Mechanical Engineering at *Muhammad Khaider University*.

And to present a special letter of thanks to *Dr. Belhi Guerira*, who supervised, followed up and paid attention to our research.

And I express my thanks and gratitude to a friend / *Rachid Zebila*, who contributed to installing the program in which we found difficulties, and for providing us with his time.

Also, I would like to thank all the distinguished staff of professors who provided their energies for the University and the students.

Finally, I ask God Almighty to accept this modest work, which I hope that the benefit of the reader

dedication dedication

Success is the morning light, always what you see with the singing of the birds that sing in the light of the sunrise, I have persevered for; I liked the study and the university and left after success.

My words have not ended yet, but I will end them with thanks before the farewell. I ask God Almighty to reward you as much as you labored them for our sake and to always bring us together in goodness, and we will be with you in paradise of eternity, O God, Amen.

list of materials

list of materials

General introduction:.....1

Chapter 1: Bibliographic research on laminates in composite materials.

I. General information on composite materials:.....3

1. Introduction:.....3

2. General definition of compound materials:3

3. Basic definitions:4

4. Components of a composite material:5

5. Classification of composite materials:.....5

6. Control of composite structures (concepts) Simple methods:.....5

7. Properties that can be improved through the formation of composite materials, including:.....6

II. ARCHITECTURE OF COMPOSITE MATERIALS:7

1. Introduction:.....7

2. Laminates:.....7

3. Slices based on unidirectional leads:7

4. Positive and negative angles:.....9

5. Symmetrical laminates:.....10

6. Sequences:.....11

7. Architecture of composite materials:.....12

III. Different calculations for the homogeneity of composite materials:.....12

1. Homogenization for the calculation of modules:12

2. Simplified homogenization - Models with "Terminals":	13
3. Mass fractions:	13
4. Relations between volume and mass fractions:	14
5. Hypotheses:.....	15
6. The mechanical characteristics of a ply:.....	16
7. Longitudinal Young's modulus:.....	16
8. Transverse Young's modulus:.....	18
9. Poisson coefficient <i>VLT</i> :.....	20
<i>I. General information of Vibration Theory:</i>	22
1. Introduction:.....	22
2. General definition of Vibration Theory:.....	22
3. Types of vibrations:	23
4. Resonance:	23
5. Vibration Theory:	23
6. Vibration representation:	24
<i>II. Definitions of displacement, speed and acceleration:</i>	25
1. Displacement:	25
2. Speed:.....	26
3. The speed is the derivative of the displacement function:	27
4. Acceleration:	28
5. Choice between displacement, speed and acceleration:	29
6. Natural frequencies for different boundary conditions:	29
7. Link coordinates:	30

III. General information of finite element method:	30
1. Introduction:	30
2. Definition of the finite element method:	31
3. Basic Concepts of the Finite Element Method:	31
4. A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS:	32
5. Preprocessing:	32
6. Types of elements:	33
7. Automatic mesh generation:	33
8. Theory of plate analysis:	34
9. Post processing:	35
IV. Basic laws of the finite element method:	35
a) Three-dimensional elements:	35
b) Shapes of some classical reference elements:	36
c) One-dimensional reference elements:	36
d) Two-dimensional reference elements:	36
e) Three-dimensional reference elements:	37
f) Approximation based on a reference element	38
V. Formulation of finite element equations:	41
1. Galerkin method:	41
2. Variational formulation:	44
3. Solution:	48
VI. Ansys program:	49
1. INTRODUCTION:	49

2. Simulation software: ANSYS	49
3. FINITE ELEMENT MODEL OF THE OPTICAL SYSTEM	50
4. ANSYS	51
5. Methods available for transient dynamic analysis in ANSYS:	51
<i>I. INTRODUCTION:</i>	55
<i>II. MODAL ANALYSIS:</i>	56
1. Modal analysis of a Plate without holes:	56
a) Units of analysis:	56
b) Geometry:	56
c) Mesh:	57
d) Solution:.....	58
e) Harmonic Response	59
f) Epoxy glass	62
<i>III. Typical analysis of a central hole plate</i>	63
<i>IV. A typical plate with two holes analysis</i>	67
g) Geometry	67
h) Solution:.....	67
<i>V. A typical analysis of a three-hole plate</i>	70
Mesh.....	71
<i>Bibliographie</i>	77

list of tables

list of tables

TABLE 1: Unit System.....	56
TABLE 2: Geometric characteristic.....	56
TABLE 3: Plate properties.....	56
TABLE 4: Mesh Properties.....	57
TABLE 5: Natural Frequencies using ANSYS.....	58
TABLE 6: Modal Solution Information.....	59
TABLE 7: Model Solution Results.....	59
TABLE 8: Analysis Harmonic Response Model.....	59
TABLE 9: Model Harmonic Response Analysis Settings.....	59
TABLE 10: Model Harmonic Response Loads.....	60
TABLE 11: Model Harmonic Response Solution Result Charts.....	60
TABLE 12: epoxy glass Orthotropic Elasticity.....	62
TABLE 13: Model Geometry Parts.....	64
TABLE 14: Natural Frequencies using ANSYS.....	64
TABLE 15: Model Solution Results.....	64
TABLE 16: Model Harmonic Response Solution Result Charts.....	65
TABLE 17: Model Geometry Parts.....	68
TABLE 18: Natural Frequencies using ANSYS.....	68
TABLE 19: Model Solution Results.....	68
TABLE 20: Model Harmonic Response Solution Result Charts.....	69
TABLE 21: Model Geometry Parts.....	71
TABLE 22: Model Mesh.....	71
TABLE 23: Natural Frequencies using ANSYS.....	72
TABLE 24: Model Harmonic Response Solution Result Charts.....	73

list of figures

list of figures

Chapter 1: Bibliographic research on laminates in composite materials.

Figure (1-1): Composite material with fiber and matrix (5).	4
Figure (1-2): Constitution of a laminate (7)	8
Figure (1-3): Designation of a laminate (7).	8
Figure (1- 4): Sign convention for the designation of laminates (7)	9
Figure (1- 5): Derivation of the rule of mixtures and inverse rule of mixtures (9).	15
Figure (1-6): Simplified diagram of a longitudinal traction (7)	16
Figure (1-7): Layout diagram of a unidirectional composite (7).	18
Figure (1-8): Orientations in Composite Layers (8).	20

Chapter 2: Vibration Theory of Flat Plates by Finite Element Method.

figure (2-1): Harmonic function (12).	23
figure (2-2): Frequency representation of a harmonic signal (12).	26
figure (2-3): Temporal representation of the vibration speed(12).	27
figure (2-4): Frequency representation of the vibration speed(12)	27
figure (2-5): Temporal representation of acceleration(12).	28
figure (6-2): Frequency representation of acceleration(12).	28
figure (7-2): Element type (17).	33

Chapter 3: Vibration analysis for flat sheets.

Figure 1: The basic dimensions of the model.	57
Figure 2: A 3D model of the body.	57
Figure 3: The FE model of the model.	57
Figure 4: Model Harmonic Response.	60
Figure 5: Model Harmonic Response Solution Frequency Response.	61
Figure 6: Model Harmonic Response Solution Frequency Response 2.	61

Figure 7: Model Harmonic Response Solution Frequency Response 3.	62
Figure 8: Model Harmonic Response Solution Frequency Response 4.	62
Figure 9: 3D model of a plate with a central hole.....	63
Figure10: FE model of a central hole plate.	63
Figure11: Natural Frequencies using ANSYS.	64
Figure 12: Model Harmonic Response Solution.....	65
Figure 13: Model Harmonic Response Solution Frequency Response.	66
Figure14: Model Harmonic Response Solution Frequency Response 2.	66
Figure 15: Model Harmonic Response Solution Frequency Response 3.	66
Figure16: Model Harmonic Response Solution Frequency Response 4	66
Figure17: A 3D model of a two-hole plate.	67
Figure18: FE model for plate.	67
Figure19: Natural Frequencies using ANSYS.	68
figure 20: Model Harmonic Response Solution.....	69
Figure 21: 3D model of the three-hole plate	70
Figure 22: FE model for plate.	70
Figure 23: Natural Frequencies using ANSYS.	72
Figure 24: Model Harmonic Response Solution.....	73
Figure 25: Model Harmonic Response Solution Frequency Response.	74
figure26: Model Harmonic Response Solution Frequency Response 2.	74
Figure 27: Model Harmonic Response Solution Frequency Response 3.	74
Figure 28: Model Harmonic Response Solution Frequency Response 4.	74

GENERAL
INTRODUCTION

General Introduction

General Introduction

Most machines of mass and flexibility are subject to natural vibrations. That is why, since mechanical structures have mass and flexibility, they are subject to vibration. This is what must be taken into consideration when designing machines and structures.

In practical life, when designing a car or plane, the vibrations must be isolated as much as possible so that they do not reach the passenger and also to protect the car or plane from collapse. By staying the system of this vehicle on the one hand knowledge of mechanical vibrations.

The study included these three chapters. The first chapter provides general information about composite materials: introduction and definition of composite materials, composite material components, classification of composite materials, control of composite structures (concepts) and properties that can be improved through the formation of composite materials and also contains engineering of composite materials, as well as calculations different homogeneity of composite materials.

The second chapter includes the theory of vibration for flat plates with the finite element method, including general information on the theory of vibration, definitions of displacement, velocity and acceleration, and important information about the finite element method and the ANSYS program.

Chapter Three continuous vibration analysis of flat panels, typical analysis of a hole less plate, followed by a modular analysis of a center hole, followed by a modular analysis of a two-hole plate, then a typical analysis of a three-hole plate.

CHAPTER 1:
BIBLIOGRAPHIC
RESEARCH ON
LAMINATES IN
COMPOSITE MATERIALS.

Chapter 1: Bibliographic research on laminates in composite materials.

I. General information on composite materials:

1. Introduction:

Compound materials are composed of a mixture of two or more different types of materials, with the aim of developing and improving the properties of two substances into a compound of good effectiveness. Through its composition, the composite materials are heterogeneous and anisotropic (1).

Currently, industries that use composite materials include:

- the automotive industry (for the manufacture, for example, of hoods, oil sumps, steering columns, kinematic chain, springs, laminated panels, etc.)
- the sports and leisure industry (for the manufacture, for example, of boards and sailing boats, kayaks, skis, bicycle frames, golf clubs, tennis rackets, surfboards, etc.).
- The aerospace industry (for the manufacture of solar panels for satellites, antennas, vehicles for launching satellites ...) and other sectors (2).

2. General definition of compound materials:

A composite material consists of the assembly of two materials of different natures, complementing each other and making it possible to obtain a material whose overall performance is greater than that of the components taken separately. Also, they are materials with high mechanical performance, which can be customized according to the designer's desire (1).

At the level of the optical scale to form a third beneficial substance, the essential thing is visual examination, where the components can be distinguished by the Just eye (3).

Composite materials consist of a matrix and fibres; the role of reinforcement is to ensure the mechanical pressure resistance function. The matrix, in turn, ensures consistency between the reinforcements to distribute mechanical stresses (4).

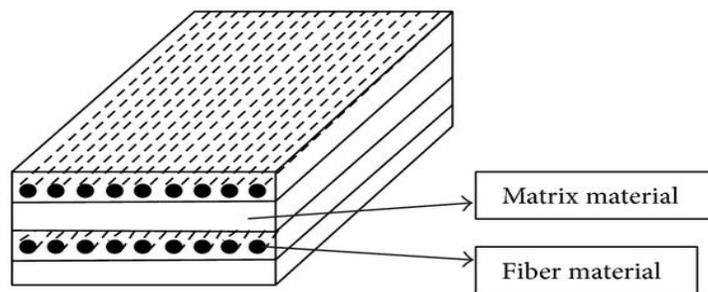


Figure (1-1): Composite material with fiber and matrix (5).

3. Basic definitions:

- ✓ **Homogeneous:** same properties everywhere in the material.
- ✓ **Heterogeneous:** in 2 different points, different properties.
- ✓ **Isotropic:** same properties in all directions.
- ✓ **Transverse isotropic:** there is an axis of symmetry. Symmetry about a line.
- ✓ **Orthotropic:** properties symmetrical about two orthogonal planes.
- ✓ **Anisotropic:** the properties are different according to the different directions (6).

4. **Components of a composite material:**

a) **Reinforcement:** it provides mechanical strength (resistance to traction and rigidity). Often filamentary in nature (organic or Inorganic fibers).

b) **The matrix:** binds the reinforcing fibers, distributes the forces (resistance to compression or to bending), and provides chemical protection (2).

5. **Classification of composite materials:**

There are four famous types of composites:

- Fibrous composite material consisting of fibers within the matrix.
- Layered lamellar composite materials consisting of layers of different materials.
- Granular composite materials consisting of granules within the matrix.
- Assemblies involving all three modes mentioned above or some of them (3).

6. **Control of composite structures (concepts) Simple methods:**

a) **Visual examination:**

This means the first observation is the simplest of all means of analysis. However, it sometimes gives accurate information about the affected areas.

b) Tap test:

This test is frequently used in aeronautics to determine damaged or defect areas (delamination). This test consists of lightly tapping the structure in several places to detect areas of variation in tone and sound on hollow areas. This simple test can be used directly by the technicians and responsible for maintenance and gives relatively reliable results for certain types of damage (ex: interface peeling) (6).

7. Properties that can be improved through the formation of composite materials, including:

- Strength.
- Rigidity.
- Corrosion resistance.
- Vibration resistance.
- The weight.
- Fatigue age.
- Etc (3).

II. ARCHITECTURE OF COMPOSITE MATERIALS:

1. Introduction:

All composites component design processes are shown:

- By surface: plates, shells,
- By stratification of successive layers.

This concept justifies the importance which will be given thereafter to the study of composite materials considered in the form of plates or shells, made up of different layers (or not) . Structures can be designed as a set of plates, and its study conclusion from the study of the plates.

The purpose of this paragraph is to identify the general architecture of composite materials (7) .

2. Laminates:

The laminates are made (Figure I.2) of successive layers (sometimes called folds) Reinforcements (threads, ceilings, mats, fabrics, etc.). We look at different types of laminates (7) .

3. Slices based on unidirectional leads:

Laminates based on unidirectional threads or fabrics constitute a basic type of laminate which theory can be chosen any other type of plate. These laminates are made up (Figure I-3) of layers of unidirectional threads or fabrics, the direction of which is offset in each layer (7) .

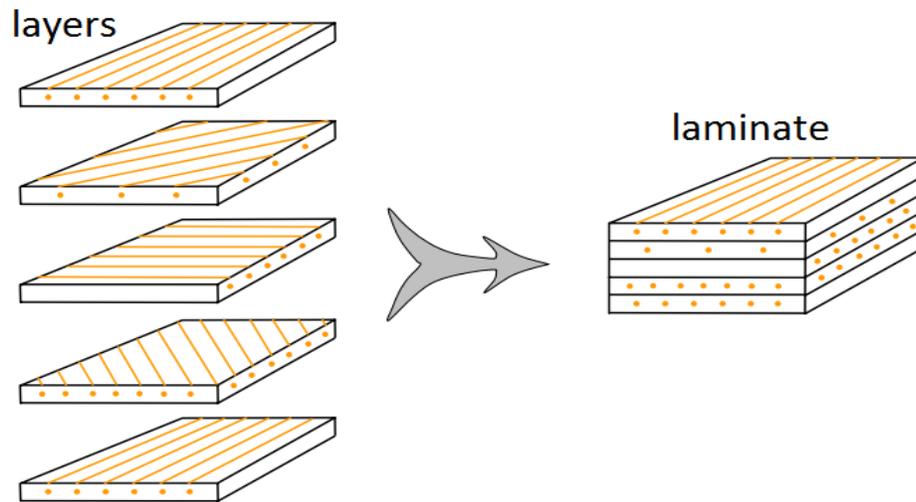


Figure (1-2): Constitution of a laminate (7) .

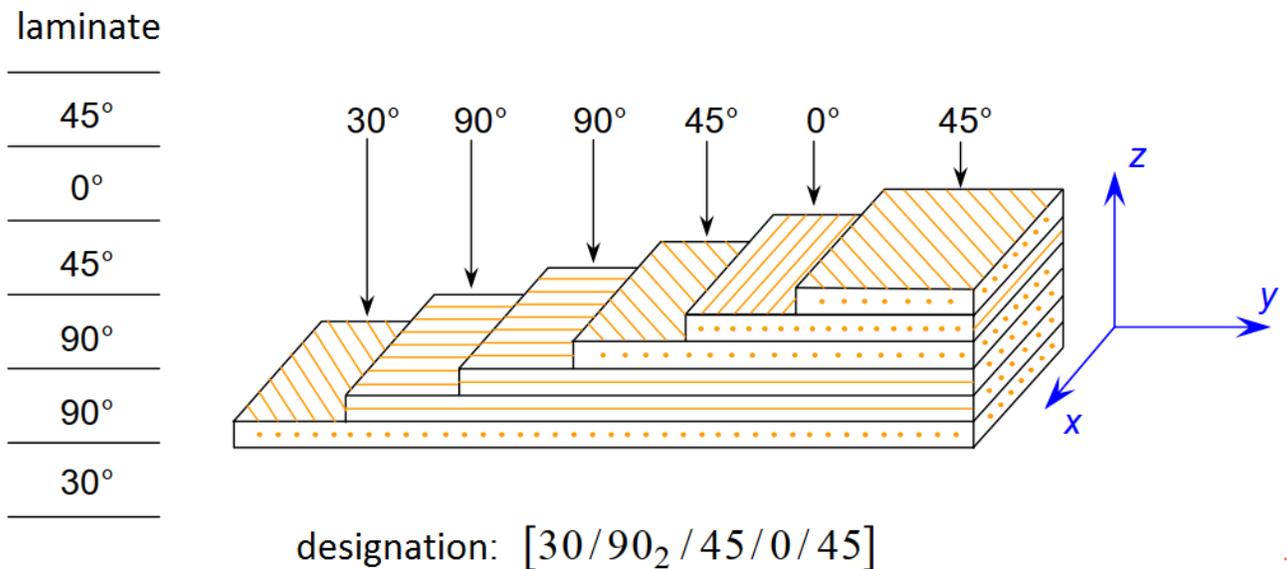


Figure (1-3): Designation of a laminate (7).

The designation of these laminates is generally carried out according to the following steps:

1. Each layer is designated by a number indicating the value in degrees of the angle made by the direction of the fibers with the reference x-axis.
2. The successive layers are separated if their angles are different.

3. Successive layers of the same orientation are designated by a numerical index.
4. The layers are named successively, going from one face to the other. Square brackets (or parentheses) indicate the beginning and the end of the code (7).

The designation depends on the axis system chosen. An example is given in Figure I-3.

4. Positive and negative angles:

When layers are oriented at angles equal to absolute values, but with opposite signs, the + or – signs are used. The convention for positive or negative angles depends on the system of axes chosen: an inversion may appear according to the choice made (Figure I-4) (7).

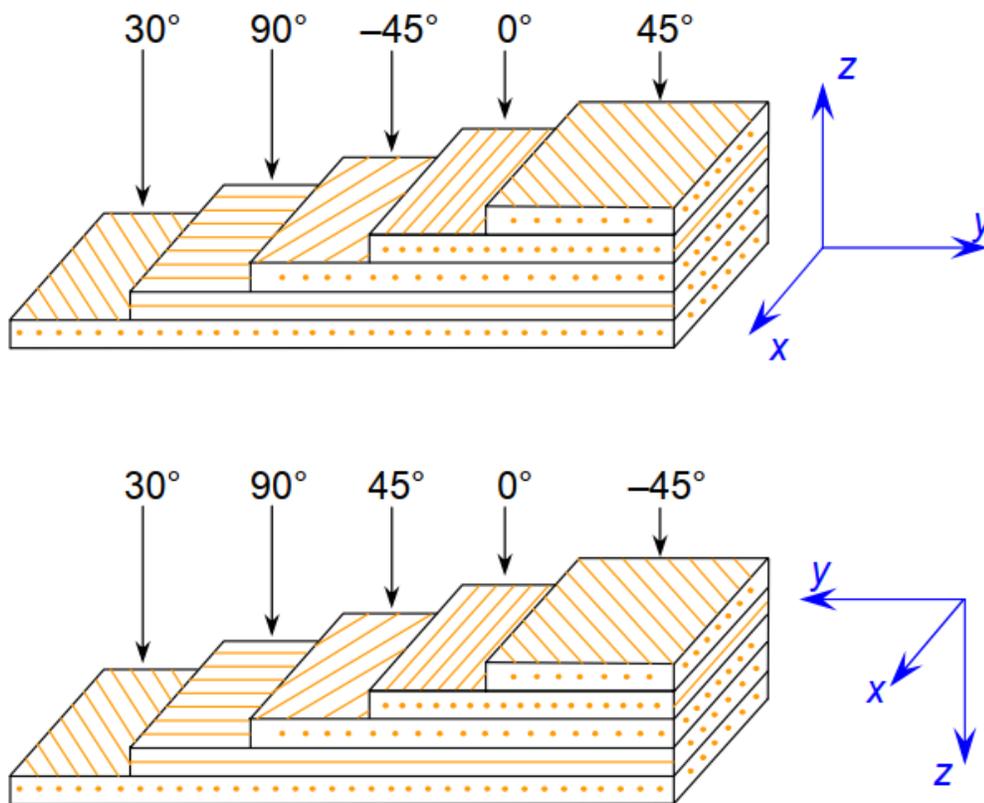


Figure (1- 4): Sign convention for the designation of laminates (7).

5. Symmetrical laminates:

A laminate is symmetrical if its mean plane is plane of symmetry. Its then requires designation half of the successive layers.

If the laminate has an even number of layers, the designation begins on one side and ends at the plane of symmetry. An index S indicates that the laminate is symmetrical. For example:

$$\begin{array}{c}
 90^0 \\
 \hline
 45^0 \\
 \hline
 45^0 \\
 \hline
 0^0 \\
 \hline
 0^0 \\
 \hline
 45^0 \\
 \hline
 45^0 \\
 \hline
 90^0
 \end{array}
 \qquad
 [90/45_2/0]_s$$

If the laminate has an odd number of layers, the designation is similar to the previous one, with the distinction of the central layer. For example:

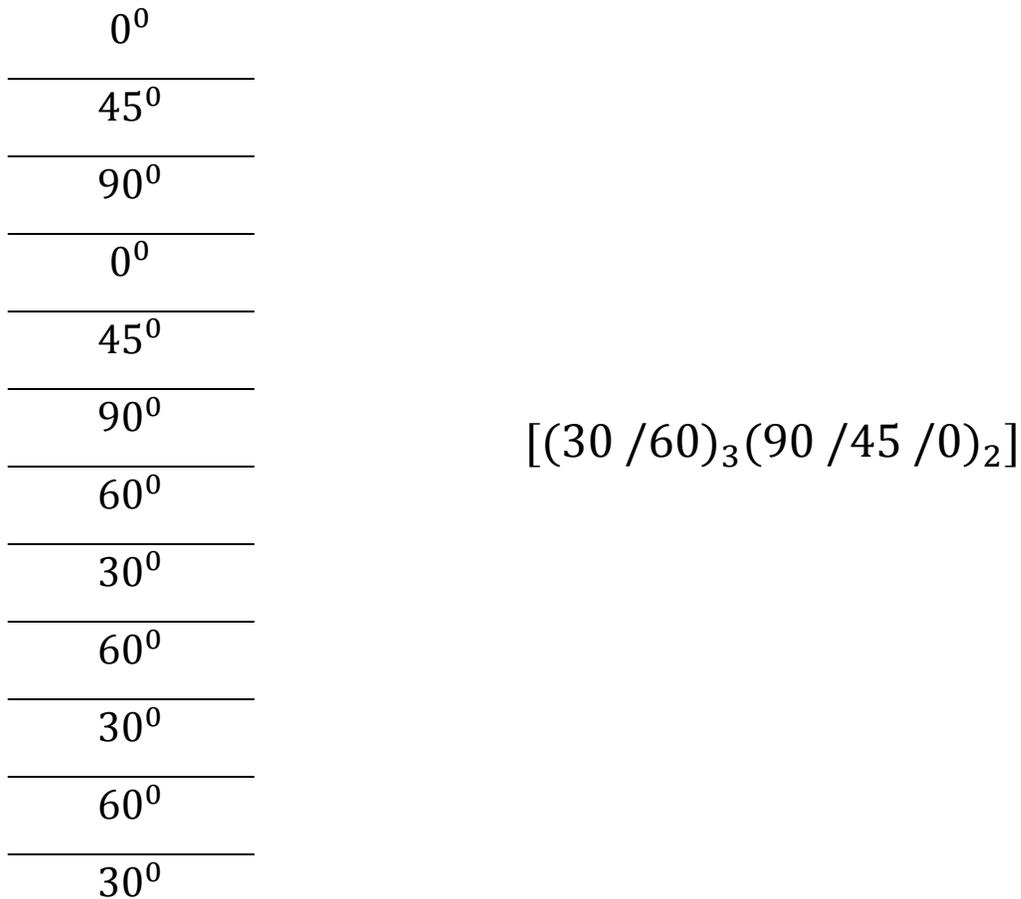
$$\begin{array}{c}
 90^0 \\
 \hline
 45^0 \\
 \hline
 45^0 \\
 \hline
 0^0 \\
 \hline
 45^0 \\
 \hline
 45^0 \\
 \hline
 90^0
 \end{array}
 \qquad
 [90/45_2/\bar{0}]_s$$

6. Sequences:

The repetition of sequences can be indicated by a subscript indicating the number of times a sequence is successively repeated, for example:

$$\begin{array}{c} 0^0 \\ \hline 45^0 \\ \hline 90^0 \\ \hline 0^0 \\ \hline 45^0 \\ \hline 90^0 \\ \hline 90^0 \\ \hline 45^0 \\ \hline 0^0 \\ \hline 90^0 \\ \hline 45^0 \\ \hline 0^0 \end{array} \quad [0/45/90]_{2s}$$

7. Architecture of composite materials:



The alternation $[0/90]_n$ is often referred to as cross laminate (7).

III. Different calculations for the homogeneity of composite materials:

1. Homogenization for the calculation of modules:

The first step in the calculation is to determine the mechanical properties of the material according to the properties of its components. There are different models for homogeneity, including:

2. Simplified homogenization - Models with "Terminals":

Or a composite material with orthotropic mark (l, t), consisting of fibers embedded in a polymer matrix. Consider an elementary cell with volume fraction $V = 1$ consisting of fibers and matrix with:

$$V_m: \text{Volume fraction of matrix.} \quad V_m = \frac{v_m}{v_c} \quad (1.1)$$

$$V_f: \text{Volume fraction of fiber.} \quad V_f = \frac{v_f}{v_c} \quad (1.2)$$

$$\text{With:} \quad V = V_f + V_m = 1 \quad (1.3)$$

$$\text{Because:} \quad v_c = v_f + v_m \quad (1.4)$$

3. Mass fractions:

The mass fractions are defined in the same way from the respective masses M_c , M_f , M_m of composite material, of fibers, of matrix. The mass fractions or mass fractions of fibers and of matrix are written respectively:

$$M_f = \frac{M_f}{M_c} \quad (1.5)$$

$$M_m = \frac{M_m}{M_c} \quad (1.6)$$

$$\text{With:} \quad M_m = 1 - M_f \quad (1.7)$$

4. Relations between volume and mass fractions:

The relationships between the volume and mass fractions involve the densities ρ_c, ρ_f, ρ_m respective of the composite material, of the fibers, of the matrix. The masses and volumes are linked by the relationships:

$$M_c = \rho_c v_c ; \quad M_f = \rho_f v_f ; \quad M_m = \rho_m v_m ; \quad (1.8)$$

The total mass of the composite material is:

$$M_c = M_f + M_m , \quad (1.9)$$

or

$$\rho_c v_c = \rho_f v_f + \rho_m v_m , \quad (1.10)$$

The density of the composite material is therefore written as a function of the following density fractions:

$$\rho_c = \rho_f V_f + \rho_m (1 - V_f) , \quad (1.11)$$

Similarly, starting from the total volume of the composite:

$$v_c = v_f + v_m , \quad (1.12)$$

We are getting:

$$\frac{M_c}{\rho_c} = \frac{M_f}{\rho_f} + \frac{M_m}{\rho_m} , \quad (1.13)$$

Hence the expression of the density as a function of the mass fractions (7):

$$\rho_c = \frac{1}{\frac{M_f}{\rho_f} + \frac{M_m}{\rho_m}} \quad , \quad (1.14)$$

5. Hypotheses:

The local level, we have the following hypotheses:

- **Fibers:** isotropic fragile linear elastic behavior of coefficients E_f and V_f .
- **Matrix:** non-linear, isotropic elastic behavior of coefficients E_m and V_m .

The goal is to define the relationships between $E_L, E_T, E_f, E_m, V_m,$ and V_f .

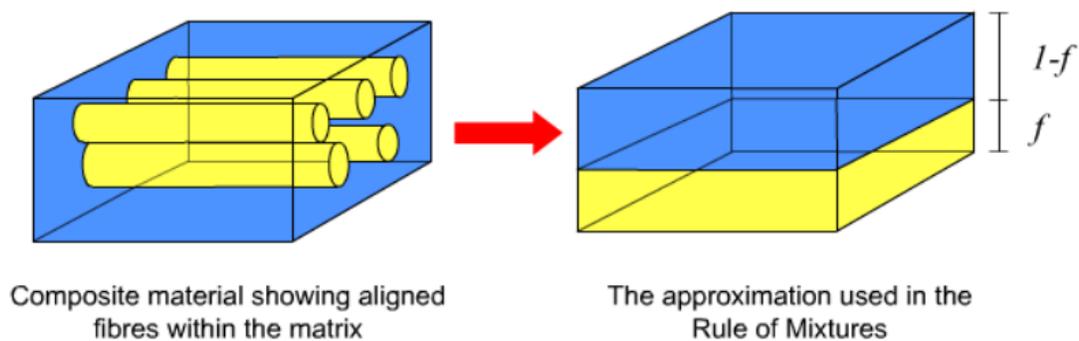


Figure (1- 5): Derivation of the rule of mixtures and inverse rule of mixtures (9).

- We work in linear elasticity.
- We assume that the fiber / matrix bond is perfect.
- Locally, we have: $\sigma_f = E_f \cdot \varepsilon_f$ and $\sigma_m = E_m \cdot \varepsilon_m$ (6).

6. The mechanical characteristics of a ply:

Simplified and practical expressions of the modules can be obtained by making a very simplified approach to the mechanical behavior of the elementary cell of the unidirectional composite material (1).

- **Young's coefficient:** This is the coefficient of elongation or inclination of the strength / strain of the material.

- **Poisson's ratio:** This is the negative ratio of the lateral contraction trigger to the axial strain caused by axial elongation stress.

- **Shear modulus:** the ratio of shear stress to shear strain at some point (3).

7. Longitudinal Young's modulus:

The longitudinal Young's modulus is determined in a longitudinal tensile test. The simplifying hypothesis is to suppose a uniform deformation in the fiber and in the matrix:

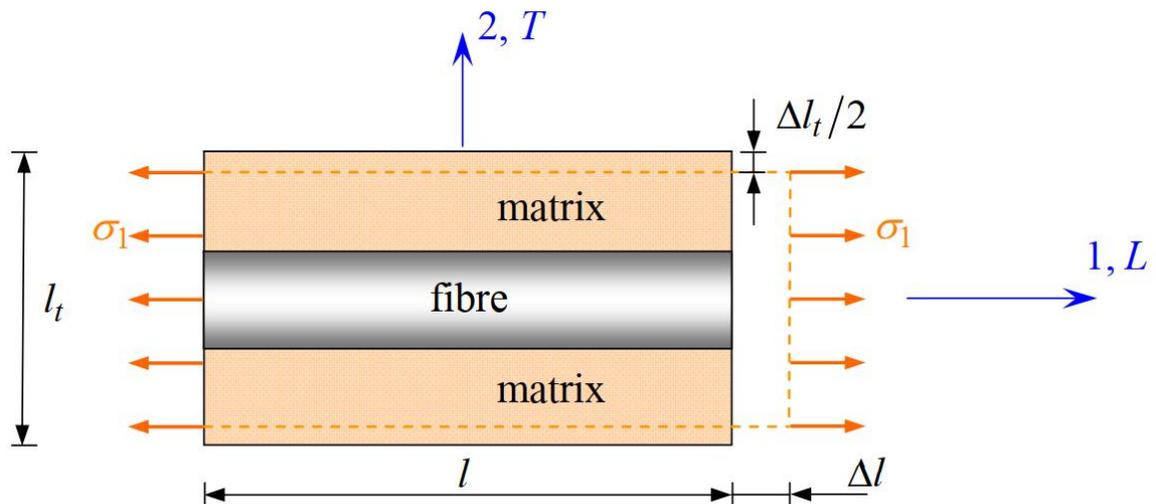


Figure (1-6): Simplified diagram of a longitudinal traction (7).

Uniform and identical strain in the fiber and the matrix.

$$\varepsilon l = \frac{\Delta l}{l} , \quad (1.15)$$

Where l is the cell length considered. The identity of the deformations in the fiber and in the source requires:

$$\varepsilon_f = \varepsilon_m = \varepsilon_l , \quad (1.16)$$

Since the fibers and the matrix have a flexible behavior, the stresses are expressed in the fibers and the matrix as shown:

$$\sigma_f = E_f \cdot \varepsilon_l , \quad (1.17)$$

$$\sigma_m = E_m \cdot \varepsilon_l , \quad (1.18)$$

The total load applied is:

$$f_1 = \sigma_f \cdot S_f + \sigma_m \cdot S_m , \quad (1.19)$$

Where S_f and S_m are the areas of the cross sections of the fiber and the matrix respectively.

If S is the cross-sectional area of the mean cell, the mean stress $\sigma_1 = \frac{f_1}{S}$

Is written: $\sigma_1 = \sigma_f V_f + \sigma_m (1 - V_f)$ (1.20)

This average stress is linked to the deformation of the cell by the longitudinal Young's modulus: $\sigma_1 = E_1 \cdot \varepsilon_1$ (1.21)

The combination of relations (17) and (18) to (21) leads to the expression of the longitudinal Young's modulus: $E_l = E_f V_f + E_m (1 - V_f)$ (1.22)

This expression is known as the law of mixtures for the Young's modulus in the direction of the fibers. The variation of the modulus is linear between the values E_m modulus of the matrix and E_f modulus of the fibers, when the volume fraction V_f of fibers varies from 0 to 1 (1).

8. Transverse Young's modulus:

The transverse Young's modulus is determined in a transverse tensile test where the composite is loaded in the direction normal to the fibers.

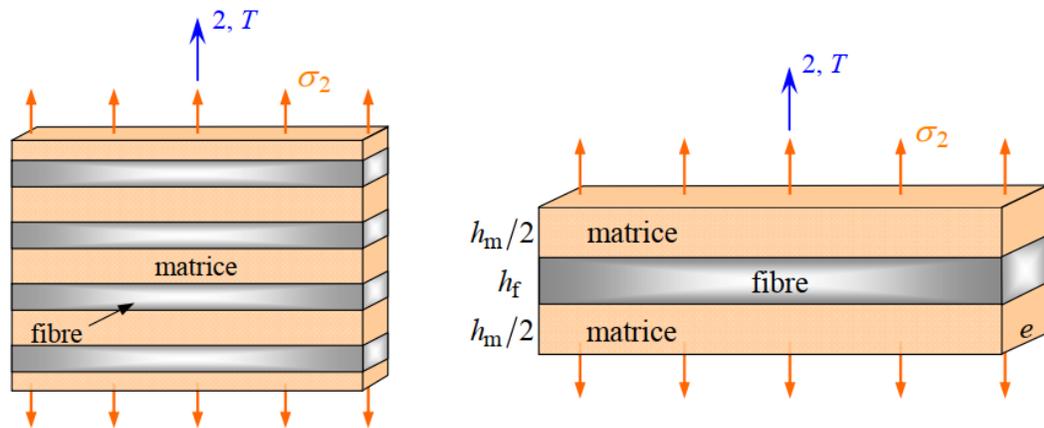


Figure (1-7): Layout diagram of a unidirectional composite (7).

The height of the layers should simply check:

$$V_f = \frac{h_f}{h_f + h_m} \quad \text{end} \quad 1 - V_f = \frac{h_m}{h_f + h_m} \quad (1.23)$$

The load F_2 imposed in the transverse direction is transmitted entirely in the fiber and in the matrix, imposing equal constraints:

$$\sigma_m = \sigma_f = \sigma_2 \quad (1.24)$$

It follows that the respective deformations of the fiber and the matrix in the transverse direction are written:

$$\varepsilon_f = \frac{\sigma_2}{E_f} \quad ; \quad \varepsilon_m = \frac{\sigma_2}{E_m} \quad (1.25)$$

The cross of an elementary cell elongation resulting cumulative elongation in the fiber and the matrix is:

$$\Delta l_2 = \varepsilon_f h_f + \varepsilon_m h_m \quad (1.26)$$

And the transverse strain is:

$$\varepsilon_2 = \frac{\Delta l_2}{h_f + h_m} = \varepsilon_f \frac{h_f}{h_f + h_m} + \varepsilon_m \frac{h_m}{h_f + h_m} \quad (1.27)$$

So be it:

$$\varepsilon_2 = \varepsilon_f V_f + \varepsilon_m (1 - V_f) \quad (1.28)$$

This deformation is linked to the stress imposed on the cell, by the transverse module:

$$\sigma_2 = E_T \varepsilon_2 \quad (1.29)$$

The combination of expressions (29) to (25) leads to the expression of the transverse module:

$$\frac{1}{E_T} = \frac{V_f}{E_f} + \frac{1 - V_f}{E_m} \quad (1.30)$$

This expression is known as the inverse law of mixtures (1).

9. Poisson coefficient V_{LT} :

The Poisson coefficient represents the contraction in the transverse direction to when a ply is subjected to tensile loading in the longitudinal direction

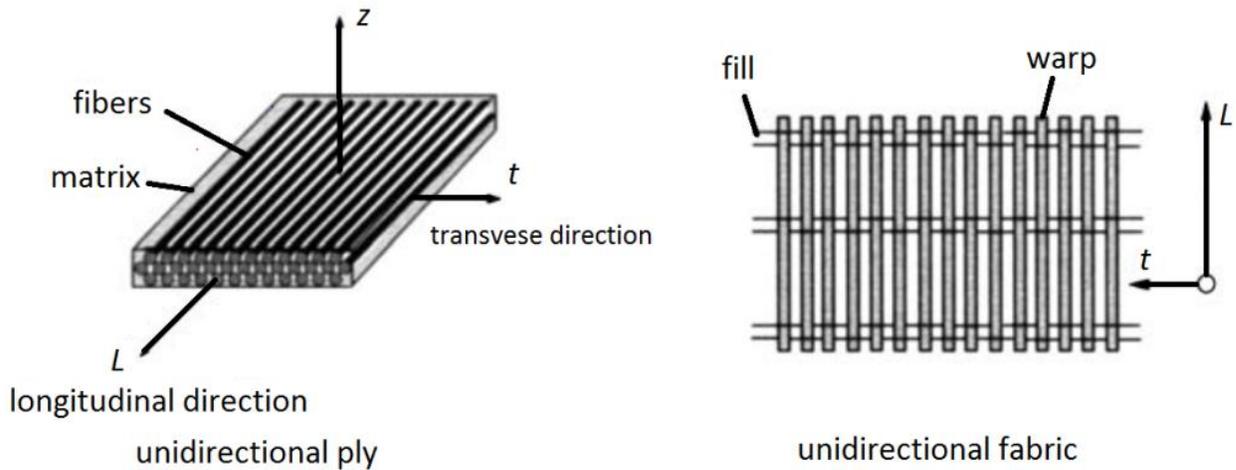


Figure (1-8): Orientations in Composite Layers (8).

Where the phrase is given as:

$$V_{LT} = v_f V_f + V_m (1 + V_f) \quad (1.31)$$

This expression is the law of mixtures for the longitudinal Poisson's ratio (8).

CHAPTER 2: VIBRATION THEORY OF FLAT PLATES BY FINITE ELEMENT METHOD

Chapter 2: Vibration Theory of Flat Plates by Finite Element Method.

I. General information of Vibration Theory:

1. Introduction:

Knowledge of the natural frequencies of a rectangular plate with holes is essential. In engineering applications, a plate is often required to accommodate fixtures or accessories; therefore a plate with holes is a common structure. Hence the knowledge of free vibration analysis of plates with openings at different positions and under different boundary conditions is important (2).

A variety of plates has been extensively used as important structural elements in many industrial fields (10).

With the advent of digital computers, discrete problems can generally be solved readily even if the number of elements is very large (11).

The study of vibrations relates to the oscillatory movements of bodies following the application of the forces imposed on them. It can be said, beyond any doubt, that anybody with mass and elasticity is susceptible to vibration (12).

2. General definition of Vibration Theory:

Vibration is the movement of the body swing around the position of balance .All bodies with mass and elasticity have a natural readiness to vibrate. So most of the machines and engineering constructions are exposed to vibration. This is what should be taken into account in the design of machinery and construction. There is two types of vibrations: free and forced vibration (13).

3. Types of vibrations:

Free vibration: Free vibration occurs when the body swings under the influence of the forces inherent in the device itself, in the absence of external forces. And the natural vibration is one of the dynamic properties of the device.

Forced vibration: Vibration that occurs under the influence of external forces is called forced vibration. Under the excitation forces alternately (13).

4. Resonance:

The state of resonance occurs when the forced vibrations are equal to the normal pulse, and this leads to the appearance of large fluctuations. And that the capacity in this case is only the amount of damping in the system, and to avoid reaching this stage, it is necessary to know the natural vibration of the system in advance (14).

5. Vibration Theory:

A body vibrates when it is animated by an oscillatory movement when it is in a position of equilibrium (figure 1.1). The simplest form of oscillatory movement is the sinusoidal form characterized by an amplitude, a frequency and a phase (12).

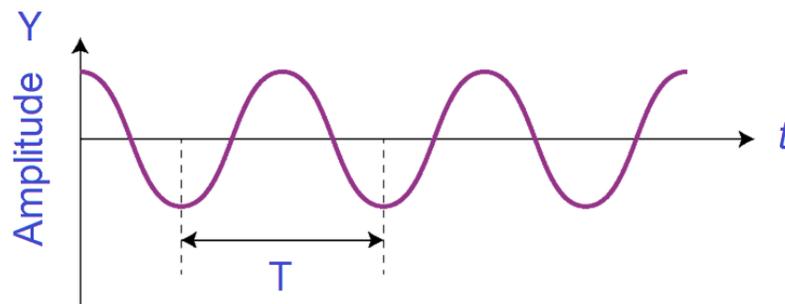


figure (2-1): Harmonic function (12).

A harmonic movement is defined by a sinusoidal function of the type:

$$y(t) = Y \sin(\omega t + \theta) \quad (2 - 1)$$

$y(t)$: denotes the simple harmonic function,

Y : represents the amplitude. It is the maximum value of the harmonic function.

t : is the time variable (in seconds);

θ : The phase angle at the origin of the function (rad);

ω : The pulsation (rad / s),

The prevailing relationship between heartbeat, frequency and period is defined as follows:

$$T = \frac{2\pi}{\omega} = \frac{1}{f} = \frac{60}{N} \quad (2 - 2)$$

T : The period (in seconds);

f : denotes the frequency which represents the number of periods per time unit (in Hertz (Hz) or in revolutions / sec or in cycles / s).

N : represents the number of revolutions / minutes (12).

6. Vibration representation:

Let two vibrations be represented by the following equations:

$$X_1(t) = a_1 \sin(\omega t) \quad (2 - 3)$$

$$X_2(t) = a_2 \sin(\omega t + \theta) \quad (2 - 4)$$

We note that if their period is the same, it is not the same for their maximum displacement which is not reached at the same time. One is offset by an angle θ relative to the other. The angle θ is called the phase angle.

The phase represents the period of time during which this quantity advanced or delayed, compared to the origin of the time variable.

The phase shift is the difference of the respective phases of two periodic movements of the same frequency, which, in the case of harmonic movements, results in the difference of the phase angles calculated from the same origin.

By periodic vibration is meant a quantity which is reproduced identically and at regular intervals with regard to a variable on which it depends (time, space, etc.)

The harmonic movement can be generalized by a periodic movement if there is repetition of the movement after a given period of time T . So we can write:

$$y(t) = y(t + T) \quad (2 - 5)$$

T Denotes the period (time it takes to go around), and is defined as the inverse of the frequency (equation 2.2) (12).

II. Definitions of displacement, speed and acceleration:

1. Displacement:

Any displacement is a vector quantity which defines the change in position of a given body or point in relation to a reference system.

The latter consists of a system of axes relating to the rest position or an equilibrium position.

As a general rule, any displacement can be represented by a vector-rotation or a vector-translation.

A displacement is qualified as “relative displacement” if it is calculated with respects to a reference system other than the basic reference system. To calculate the relative displacement between two points, it suffices to make the vector difference between the displacements of these two points.

Let us consider a displacement function of harmonic type:

$$y(t) = Y \sin(\omega T + \theta) \quad (2 - 6)$$

The amplitude of the signal is y and its frequency is $f = \omega / 2\pi$. The temporal form of this function is illustrated in figure 2.1. In the frequency domain, the same signal is more simply represented by figure 2.2 (12).

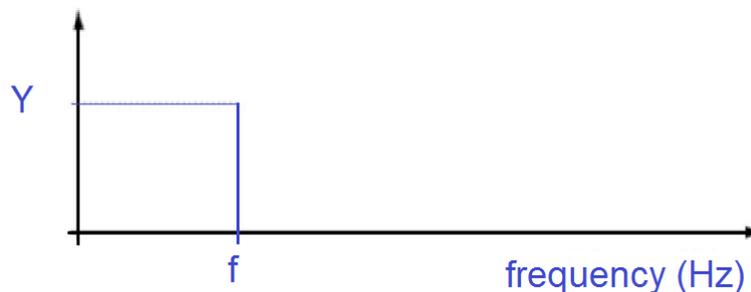


figure (2-2): Frequency representation of a harmonic signal (12).

2. Speed:

The speed represents the derivative of the displacement compared to time and is defined as being the limit of $\Delta x / \Delta t$ when Δt tends towards 0.

A speed is called a "relative speed" if it is calculated in a reference system other than the basic reference system. The relative speed between two points is obtained by calculating the vector difference between the speeds of two given points.

3. The speed is the derivative of the displacement function:

$$\dot{y}(t) = \omega Y \cos(\omega t + \theta) = \omega Y \sin\left(\omega t + \theta + \frac{\pi}{2}\right) \quad (2 - 7)$$

Consequently, we can say that the vibratory speed is a harmonic signal phase shifted by 90 degrees compared to displacement with the same frequency, but whose amplitude V is equal to ωY (Cf. figures 2.3 and 2.4).

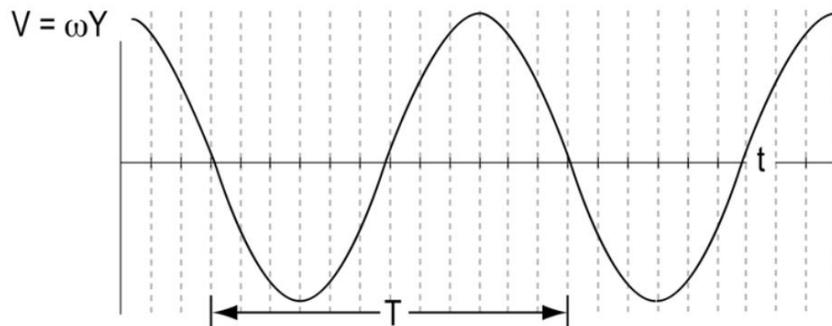


figure (2-3): Temporal representation of the vibration speed(12).



figure (2-4): Frequency representation of the vibration speed(12).

4. Acceleration:

Acceleration is a vector that represents the derivative of speed with respect to time.

The acceleration is for a harmonic signal:

$$\ddot{y}(t) = -\omega^2 Y \sin(\omega t + \theta) \quad (2 - 7)$$

Acceleration is therefore a harmonic signal, phase shifted by 90 degrees relative to speed and 180 degrees relative to displacement, whose amplitude A is equal to $\omega^2 Y$ and whose pulsation is ω (Cf. figures 2.5 and 2.6).

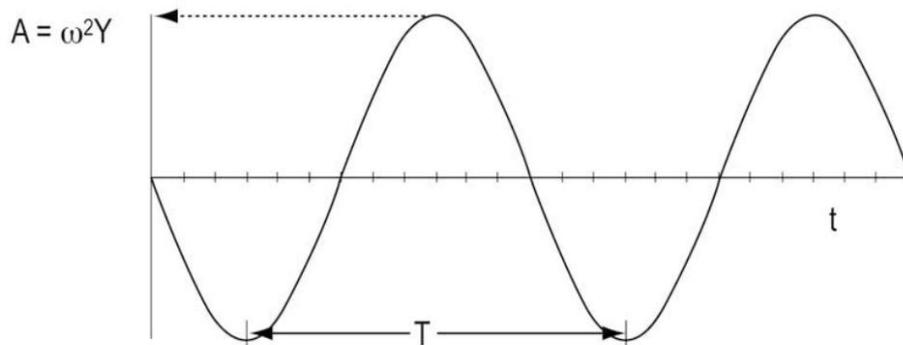


figure (2-5): Temporal representation of acceleration(12).



figure (6-2): Fréquence representation of acceleration(12).

5. Choice between displacement, speed and acceleration:

We note that there is a very simple relationship between the amplitudes of displacement Y , speed V , acceleration A and pulsation ω . Also we can write that:

$$A = \omega V = \omega^2 Y \quad (2 - 8)$$

Therefore, we only need to know two of these parameters to calculate the other two (12).

6. Natural frequencies for different boundary conditions:

To find the fundamental frequencies of the plate, the stiffness matrix $[k]$ and the mass matrix $[M]$ are needed. For linear problems the local stiffness matrix $[k]$ is independent of element displacement $\{X\}$ and the following relation is valid for all instants of time.

$$\{\ddot{X}\} = -\omega^2 \{X\} \quad (2 - 9)$$

Therefore, the global matrix equation can be written as:

$$([K] - \omega_m^2 [M])\{X\} = \{0\} \quad (2 - 10)$$

We assume that the plate is simply supported at two edges while the boundary conditions at the other two edges, are a combination of boundary conditions. These combinations of supports are SSSS, SSSC, SCSC, SFSC, SFSS and SFSE, where S stands for simply support, C for clamped, and F for free boundary conditions. The first letter describes the boundary condition at $x = 0$, the second letter is the boundary condition at $y = 0$ and third and fourth letters describe the boundary conditions at $x = a$ and $y = b$ respectively. Primary boundary conditions that are used for displacement based finite element analysis are as follow (14).

7. Link coordinates:

It can be said in general that the movement of devices with two degrees of freedom are related to each other. Where the coordinates are appear in all the equations of movement and in general can be written two equations of movement thus:

$$m_{11}\ddot{x}_1 + m_{12}\ddot{x}_2 + k_{11}x_1 + k_{12}x_2 = 0 \quad (2 - 11)$$

$$m_{21}\ddot{x}_1 + m_{22}\ddot{x}_2 + k_{21}x_1 + k_{22}x_2 = 0 \quad (2 - 12)$$

Or in the form of a matrix, such as (14):

$$\begin{bmatrix} m_{11} & m_{12} \\ m_{21} & m_{22} \end{bmatrix} \begin{pmatrix} \ddot{x}_1 \\ \ddot{x}_2 \end{pmatrix} + \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \end{pmatrix} \quad (2 - 13)$$

III. General information of finite element method:

1. Introduction:

The Finite Elements Method or short FEM is a comfortable way for engineers to carry out complex calculations in mechanics and other physical fields. It enables the engineers to make accurate estimations of stress, strain, frequencies and much more. Not only analyses of single parts are possible, but also of subassembly- blues and complete assemblies. It is even used to simulate crash tests of cars. The Finite Elements Analysis (FEA) is nowadays an invaluable tool in the product development. It is already applied in the state of concept generation and extensively used for the further development of parts. This does not save only a lot of time, but also a lot of money, since the budget for tests and prototypes can be reduced to a minimum. The Finite Element Method

is in cooperation with CAD (Computer Aided Design) the most powerful procedure to rationalize and optimize the work of an engineer qualitatively. But it is not advisable to have a blind trust into FEM. The basics of mechanics and the basics of the FEM- procedure must be understood by the calculation engineer. Otherwise a calculated structure could end up in a disaster in real life, which often results in deaths. The calculation engineer is liable and has to be aware of his responsibility (15).

2. Definition of the finite element method:

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), (16). It is a numerical method mainly used to solve partial differential equations. And as for the engineering field, the finite element method is used in the analysis of engineering constructions, such as: mechanical vibrations, material durability ... etc (15). Boundary value problems are also sometimes called field problems (16).

3. Basic Concepts of the Finite Element Method:

This method has basic steps for solving differential equations. Sometimes, the scope of the solution is divided into sections called elements, which may be equal or not equal. These elements are related to each other and each component writes an approximate solution to the differential equation, from that the equations of the elements are obtained in the form of a matrix called the matrix equation of the element, by grouping the equations of the elements taking into account that the end of the element is the beginning of the other element we get the total matrix equation, and know the cases Numerical for the range of the

solution and its substitution in cosmic matrix equations we get a solvable equation (15) (16).

4. A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS:

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, or some other problem. These steps are embodied in commercial finite element software packages. The steps are described as follows (16).

5. Preprocessing:

The pre-processing step is, quite generally, described as defining the model and includes

Define the geometric domain of the problem.

Define the element type(s) to be used.

Define the material properties of the elements.

Define the geometric properties of the elements (length, area, and the like).

Define the element connectivity's (mesh the model).

Define the physical constraints (boundary conditions).

Define the loadings.

The pre-processing (model definition) step is critical. In no case is there a better example of the computer-related axiom. A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem (16).

6. Types of elements:

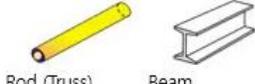
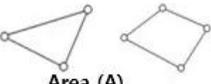
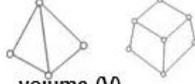
Type	Actual Models	Finite Element Expressions (Geometric Properties Defined by Nodes)	Additional Requirements(Actual Volume Calculation)
1D	 Rod (Truss) Beam	 Length (L)	Area (A, cross-sectional shape) $\rightarrow V = L \times A$
2D	 Shell, Plane Stress, Plane Strain, <u>Axisymmetric</u> , etc	 Area (A)	Thickness (t) $\rightarrow V = A \times t$
3D	 Solid	 volume (V)	None (volume calculation possible)

figure (7-2): Element type (17).

The previous described elements are automatically fitted together to one big so-called mesh, what extends to the entire part. But the automatically generated mesh is not always perfect. In some analysis types (17).

7. Automatic mesh generation:

Mesh generation has always been a time-consuming and error-prone process. This is especially true in the practical science and engineering computations, where meshes have to be generated for three-dimensional geometries of various levels of complexity. The attempt to create a fully automatic mesh generator, which is a particular mesh generation algorithm that is capable of generating valid finite element meshes over arbitrary domains and needs only the information of the specified geometric boundary of the domain and the required distribution of the element size, started from the work of Zienkiewicz and Phillips¹ in the early 1970s. Since then many methodologies

have been proposed and different algorithms have been devised in the development of automatic mesh generators (11).

Before proceeding further on mesh generation schemes it is necessary to specify the kind of mesh we desire. Here we should give the following information:

1. The type of element and the number of nodes required on each;
2. The size of the desired element, here the minimum size of each element generally is specified;
3. Specification of regions of different material types or characteristics to be attached to a given element
4. In some cases, the so-called stretch ratio if we wish to present elements which are elongated in some preferential direction. This is often needed for problems in fluid mechanics in regions where boundary layers and shocks are encountered (11).

8. Theory of plate analysis:

Displacement dependency to time and place is usually used. Generally displacement of different points across the page is as follows:

Displacement in direction $1(x) U_1(x_1, x_2, x_3, t)$.

Displacement in direction $2(y) U_2(x_1, x_2, x_3, t)$.

Displacement in direction $3(z) U_3(x_1, x_2, x_3, t)$ (18).

9. Post processing:

Analysis and evaluation of the solution results is referred to as post processing. Post processor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

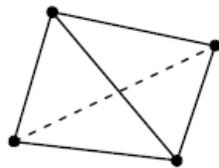
While solution data can be manipulated many ways, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable (16).

IV. Basic laws of the finite element method:

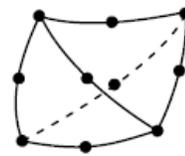
a) Three-dimensional elements:

These are tetrahedrons, hexahedrons or prisms, the faces of which are polynomial surfaces of the first, second or third degree.

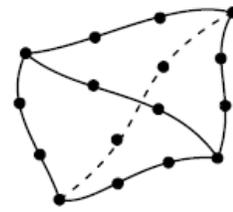
Tetrahedral elements:



Linear (4)

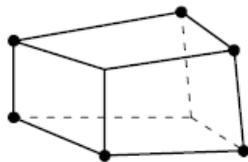


Quadratic (10)

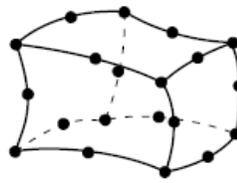


Cubic (16)

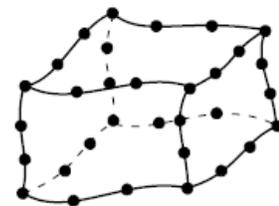
Hexahedral elements:



Linear (8)

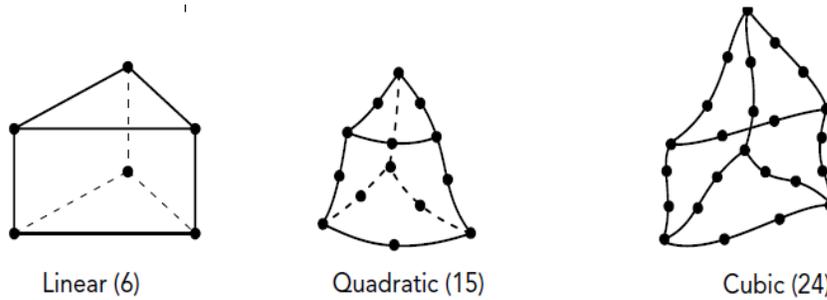


Quadratic (20)



Cubic (32)

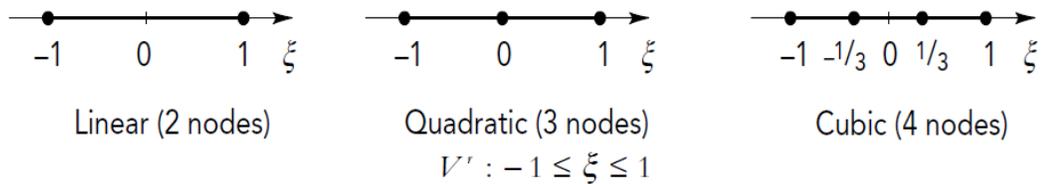
Prismatic elements:



b) Shapes of some classical reference elements:

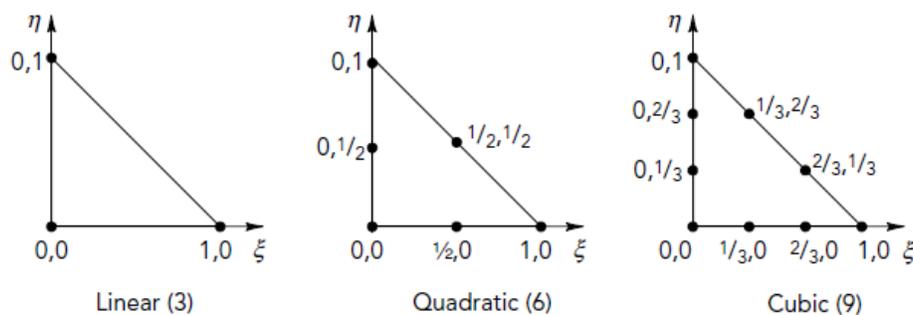
We now illustrate the shape and analytical definition of the reference elements corresponding to the classical elements covered in section 1.2.3.

c) One-dimensional reference elements:

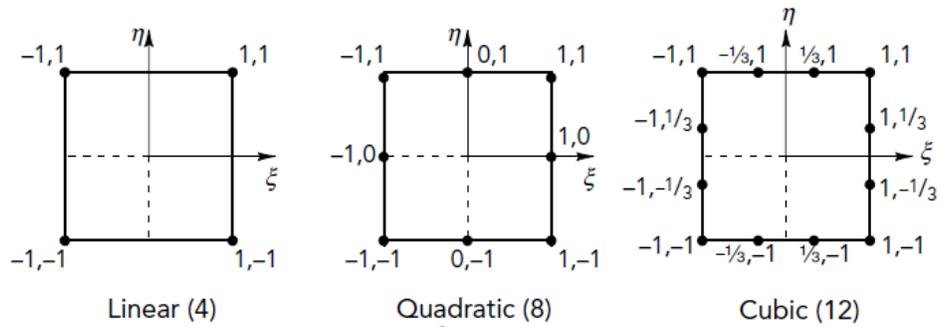


d) Two-dimensional reference elements:

Triangular elements:

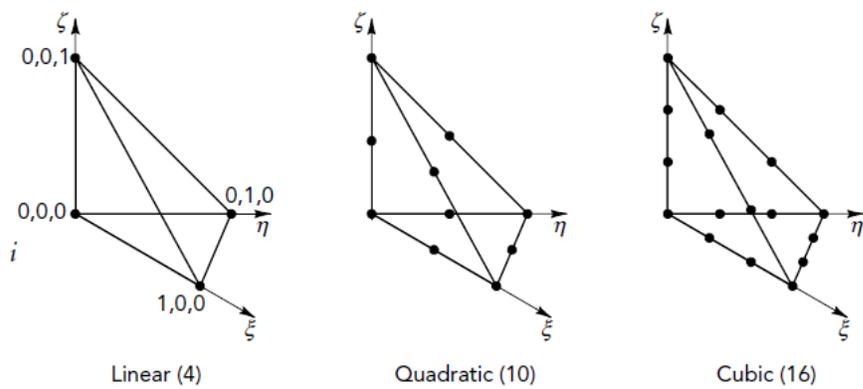


Square elements:

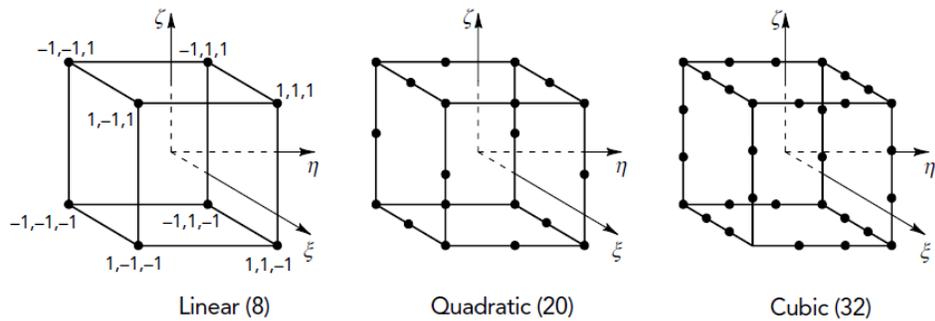


e) Three-dimensional reference elements:

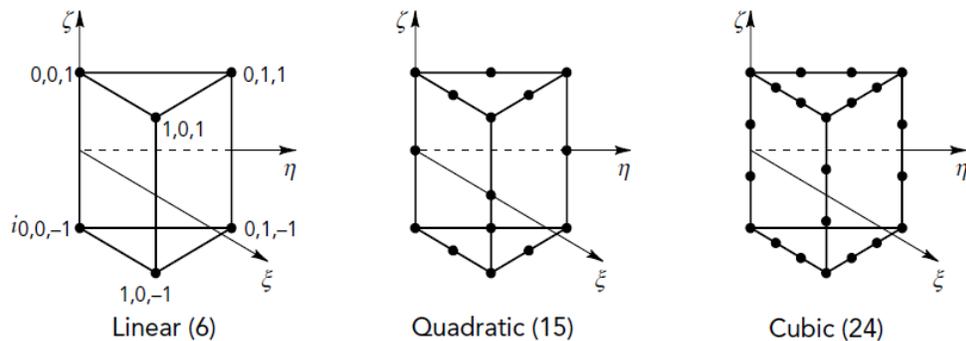
Tetrahedral elements:



Cubic elements:



Prismatic elements:



f) Approximation based on a reference element

a) Expression of the approximate function $u(x)$:

Let us choose a set of n interpolation nodes of coordinates x_i in the domain V , which may or may not coincide with the geometrical nodes. For each

Element V^e let us use a nodal approximation of type (1.5) for the exact function $u_{ex}(x)$:

$$u_{ex}(X) \approx u(x) = \langle N_1 N_2 \dots N_{n^e}(X) \rangle \begin{Bmatrix} u_1 \\ \vdots \\ u_{n^e} \end{Bmatrix} = \langle N(X) \rangle \{u_n\} \quad (2-14)$$

Where x belongs to V^e ; $N_1 N_2 \dots N_{n^e}(X)$ are the values of u_{ex} at the interpolation nodes n^e of the element, i.e. nodal variables; and $N(x)$ are the interpolation functions on the real element.

b) PROPERTIES OF APPROXIMATE FUNCTION $u(x)$

1. Interpolating property of the nodal approximation

The approximate function $u(x)$ takes the value u_i at nodes with coordinates x_j :

$$u(X_i) = u_i = \langle N_1(X_i) N_2(X_i) \dots N_{n^e}(X_i) \rangle \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ \cdot \\ u_{n^e} \end{Bmatrix} \quad (2 - 15)$$

Similarly, using the approximation on the reference element:

$$u(\xi_i) = u_i = \langle N_1(\xi_i) N_2(\xi_i) \dots N_{n^e}(\xi_i) \rangle \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ \cdot \\ u_{n^e} \end{Bmatrix} \quad (2 - 16)$$

2. Continuity over the element:

If the approximate function $u(x)$ is required to be continuous over the element together with all its derivatives up to the order s , functions $N_i(x)$ with continuous derivatives up to the order s , must be used.

3. Inter-element continuity:

If the approximate function $u(x)$ and its derivatives up to the order s are required to be continuous on a common boundary with another element, then $u(x)$ and its derivatives up to the order s must depend solely on the nodal variables associated with the nodes on this boundary.

Let us consider the continuity of $u(x)$ over a boundary (continuity C^0):

$$u(X) = \langle N_1(X) N_2(X) \dots N_{n^e}(X) \rangle \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ \cdot \\ u_{n^e} \end{Bmatrix} \quad (2 - 17)$$

Jacobian matrix of a four-noded quadrilateral element Using

$$[J] = \frac{1}{4} \begin{bmatrix} -(1-\eta) & (1-\eta) & (1+\eta) & -(1+\eta) \\ -(1-\xi) & -(1+\xi) & (1-\xi) & (1-\xi) \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ x_3 & y_3 \\ x_4 & y_4 \end{bmatrix} \quad (2-18)$$

$$\det(J) = A_0 + A_1\xi + A_2\eta \quad (2-19)$$

$$A_0 = \frac{1}{8} [(y_4 - y_2)(x_3 - x_1) - (y_3 - y_1)(x_4 - x_2)]$$

$$A_1 = \frac{1}{8} [(y_3 - y_4)(x_2 - x_1) - (y_2 - y_1)(x_3 - x_4)] \quad (2-20)$$

$$A_2 = \frac{1}{8} [(y_4 - y_1)(x_3 - x_2) - (y_3 - y_2)(x_4 - x_1)]$$

$$[K_e]\{q_e\} = \{F_e\} \quad (2-21)$$

Where $[K_e]$ is the stiffness matrix of the element. $\{q_e\}$ the vector of displacements and $\{F_e\}$ the vector of the equivalent nodal forces:

$$\{F_e\} = \{F_e^V\} + \{F_e^S\} + \{F_e^O\} \quad (2-22)$$

$[K_e] = \int_{V_e} [B]^T [C][B] dV$ is the stiffness matrix of the element.

$\{F_e^V\} = \int_{V_e} [N]^T [F^v] dV$ set the vector of volume forces

$$\{F_e^S\} = \int_{S_e} [N]^T [F^S] ds \quad \text{set the vector of surface forces}$$

$$\{F_e^O\} = \int_{V_e} [B]^T [\sigma^O] dV \quad \text{set the vector of initial constraints}$$

4. Assembly of elements:

This expression promises to reduce the elementary quantities in the structural frame

$$[K_e]_{global} = [T_e]^t [K_e]_{local} [T_e]$$

$$\{F_c^u\}_{global} = [T_e]^t \{F_e^u\}_{local} \quad (2 - 23)$$

$$\{F_c^S\}_{global} = [T_e]^t \{F_e^u\}_{local}$$

$$\{F_c^O\}_{global} = [T_e]^t \{F_e^u\}_{local}$$

V. Formulation of finite element equations:

Several approaches can be used to transform the physical formulation of the problem to its finite element discrete analogue. If the physical formulation of the problem is known as a differential equation then the most popular method of its finite element formulation is the Galerkin method. If the physical problem can be formulated as minimization of a functional then variational formulation of the finite element equations is usually used.

1. Galerkin method

Let us use simple one-dimensional example for the explanation of finite element formulation using the Galerkin method. Suppose that we need to solve numerically the following differential equation:

$$a \frac{d^2 u}{dx^2} + b = 0; \quad 0 \leq x \leq 2L \quad (2 - 24)$$

With boundary conditions

$$u|_{x=0} = 0$$

$$a \left. \frac{du}{dx} \right|_{x=2L} = R \quad (2 - 25)$$

First, consider a finite element presented on the right of Figure. The element has two nodes and approximation of the function $u(x)$ can be done as follows:

$$u = N_1 u_1 + N_2 u_2 = [N] \{u\}$$

$$[N] = [N_1 \quad N_2] \quad (2 - 26)$$

$$\{u\} = \{u_1 \quad u_2\}$$

Where N_i are the so called shape functions

$$N_1 = 1 - \frac{x - x_1}{x_2 - x_1}$$

$$N_2 = \frac{x - x_1}{x_2 - x_1} \quad (2 - 27)$$

Which are used for interpolation of $u(x)$ using its nodal values. Nodal values u_1 and u_2 are unknowns which should be determined from the discrete global equation system.

After substituting u expressed through its nodal values and shape functions, in the differential equation, it has the following approximate form:

$$a \frac{d^2}{dx^2} [N]\{u\} + b = \psi \quad (2 - 28)$$

Where ψ is a nonzero residual because of approximate representation of a function inside a finite element. The Galerkin method provides residual minimization by multiplying terms of the above equation by shape functions, integrating over the element and equating to zero:

$$\int_{x_1}^{x_2} [N]^T a \frac{d^2}{dx^2} [N]\{u\} dx + \int_{x_1}^{x_2} [N]^T b dx = 0 \quad (2 - 29)$$

Use of integration by parts leads to the following discrete form of the differential equation for the finite element:

$$\int_{x_1}^{x_2} \left[\frac{dN}{dx} \right]^T a \left[\frac{dN}{dx} \right] dx \{u\} - \int_{x_1}^{x_2} [N]^T b dx - \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} a \frac{du}{dx} \Big|_{x=x_2} + \begin{Bmatrix} 1 \\ 0 \end{Bmatrix} a \frac{du}{dx} \Big|_{x=x_1} = 0 \quad (2 - 30)$$

Usually such relation for a finite element is presented as:

$$[k]\{u\} = \{f\}$$

$$[k] = \int_{x_1}^{x_2} \left[\frac{dN}{dx} \right]^T a \left[\frac{dN}{dx} \right] dx \quad (2 - 31)$$

$$\{f\} = \int_{x_1}^{x_2} [N]^T b dx + \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} a \frac{du}{dx} \Big|_{x=x_2} - \begin{Bmatrix} 1 \\ 0 \end{Bmatrix} a \frac{du}{dx} \Big|_{x=x_1}$$

In solid mechanics $[k]$ is called stiffness matrix and $\{f\}$ is called load vector. In the considered simple case for two finite elements of length L stiffness matrices and the load vectors can be easily calculated:

$$[k_1] = [k_2] = \frac{a}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$\{f_1\} = \frac{bL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}; \quad \{f_2\} = \frac{bL}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} + \begin{Bmatrix} 0 \\ R \end{Bmatrix}$$

The above relations provide finite element equations for the two separate finite elements. A global equation system for the domain with 2 elements and 3 nodes can be obtained by an assembly of element equations. In our simple case it is clear that elements interact with each other at the node with global number 2. The assembled global equation system is:

$$\frac{a}{L} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{bL}{2} \begin{Bmatrix} 1 \\ 2 \\ 1 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ R \end{Bmatrix} \quad (2-31)$$

After application of the boundary condition $u(x=0) = 0$ the final appearance of the global equation system is:

$$\frac{a}{L} \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{bL}{2} \begin{Bmatrix} 0 \\ 2 \\ 1 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ R \end{Bmatrix} \quad (2-32)$$

2. Variational formulation

The differential equation

$$a \frac{d^2 u}{dx^2} + b = 0; \quad 0 \leq x \leq 2L$$

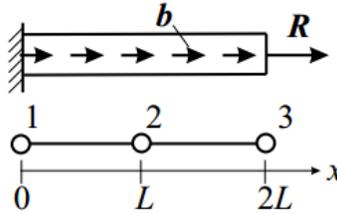
$$u|_{x=0} = 0 \quad (2-33)$$

$$a \left. \frac{du}{dx} \right|_{x=2L} = R$$

With $a = EA$ has the following physical meaning in solid mechanics. It describes tension of the one dimensional bar with cross-sectional area A made of material with the

elasticity modulus E and subjected to a distributed load b and a concentrated load R at its right end as .Such problem can be formulated in terms of minimizing the potential energy functional Π :

$$\Pi = \int_L \frac{1}{2} a \left(\frac{du}{dx} \right)^2 dx - \int_L b u dx - R u|_{x=2L} \quad (2-34)$$



Tension of the one dimensional bar subjected to a distributed load and a concentrated load.

Using representation of $\{u\}$ with shape functions (1.3) – (1.4) we can write the value of potential energy for the second finite element as:

$$\Pi_e = \int_{x_1}^{x_2} \frac{1}{2} a \{u\}^T \left[\frac{dN}{dx} \right]^T \left[\frac{dN}{dx} \right] \{u\} dx - \int_{x_1}^{x_2} \{u\}^T [N]^T b dx - \{u\}^T \begin{Bmatrix} 0 \\ R \end{Bmatrix} \quad (2-35)$$

The condition for the minimum of Π is:

$$\delta \Pi = \frac{\delta \Pi}{\delta u_1} \delta u_1 + \dots + \frac{\delta \Pi}{\delta u_n} \delta u_n = 0 \quad (2-36)$$

Which is equivalent to:

$$\frac{\delta \Pi}{\delta u_n} = 0, \quad i = 1 \dots n \quad (2-37)$$

It is easy to check that differentiation of Π in respect to u_i gives the finite element equilibrium equation which is coincide with equation obtained by the Galerkin method:

$$\int_{x_1}^{x_2} \left[\frac{dN}{dx} \right]^T EA \left[\frac{dN}{dx} \right] dx \{u\} - \int_{x_1}^{x_2} [N]^T b dx - \begin{Bmatrix} 0 \\ R \end{Bmatrix} = 0 \quad (2 - 38)$$

Assembly of the global equation system

The aim of assembly is to form the global equation system

$$[K]\{Q\} = \{F\}$$

using element equations

$$[k_i]\{q_i\} = \{f_i\}$$

In order to derive an assembly algorithm let us present the total potential energy for the body as a sum of element potential energies π :

$$\Pi = \sum \pi_i = \sum \frac{1}{2} \{q_i\}^T [K_i] \{q_i\} - \sum \{q_i\}^T \{f_i\} + \sum E_i^0 \quad (2 - 39)$$

Where E_i^0 is the fraction of potential energy related to free thermal expansion:

$$E_i^0 = \int_{V_i} \frac{1}{2} \{\varepsilon^t\} [E] \{\varepsilon^t\} dV \quad (2 - 40)$$

Let us introduce the following vectors and a matrix where element vectors and matrices are simply placed:

$$\{Q_d\} = \{q_1\} \{q_2\}$$

$$\{F_d\} = \{ \{f_1\} \{f_2\} \dots \}$$

$$[K_d] = \begin{bmatrix} [k_1] & 0 & 0 \\ 0 & [k_2] & 0 \\ 0 & 0 & \dots \end{bmatrix} \quad (2 - 41)$$

It is evident that it is easy to find matrix $[A]$ such that

$$\{Q_d\} = [A]\{Q\}$$

$$\{F_d\} = [A]\{F\}$$

The total potential energy for the body can be rewritten in the following form:

$$\begin{aligned} \Pi &= \frac{1}{2} \{Q_d\}^T [K_d] \{Q_d\} - \{Q_d\}^T \{F_d\} + \sum E_i^0 \\ &= \frac{1}{2} \{Q\}^T [A]^T [K_d] [A] \{Q\} - \{Q\}^T [A]^T \{F_d\} + \sum E_i^0 \end{aligned} \quad (2 - 42)$$

Using the condition of minimum of the total potential energy

$$\left\{ \frac{\delta \Pi}{\delta Q} \right\} = 0$$

We arrive at the following global equation system:

$$[A]^T [K_d] [A] \{Q\} - [A]^T \{F_d\} = 0 \quad (2 - 43)$$

The last equation shows that algorithms of assembly the global stiffness matrix and the global loadn vector are:

$$[K] = [A]^T [K_d] [A]$$

$$\{F\} = [A]^T \{F_d\}$$

Here $[A]$ is the matrix providing transformation from global to local enumeration. Fraction of nonzero (unit) entries in the matrix $[A]$ is very small. Because of this the matrix $[A]$ is never used explicitly in actual computer codes.

Element properties

Element matrices and vectors are calculated as follows:

Stiffness matrix

$$[K] = \int_V [B]^T [E] [B] dV \quad (2 - 44)$$

Force vector (volume and surface loads)

$$\{p\} = \int_V [N]^T \{p^V\} dV + \int_S [N]^T \{p^S\} dS \quad (2 - 45)$$

thermal vector (fictitious forces to simulate thermal expansion)

$$\{h\} = \int_V [B]^T [E] \{\varepsilon^t\} dV \quad (2 - 46)$$

3. Solution:

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow (16).

VI. Ansys program

1. INTRODUCTION:

The development of finite element methods for the solution of practical engineering problems began with the advent of the digital computer. And it was only through the use of the digital computer that this process could be rendered effective and given general applicability (19).

2. Simulation software: ANSYS

The Finite Element Method is a numerical tool for analyzing structures and continua as well as solving problems of engineering and mathematical physics. These problems often concern stress analysis, heat transfer, fluid flow, vibrations, electric and magnetic fields and several other areas which are used in the design of buildings, engines, electric motors, cars, etc. FEM is a method for converting a partial differential equation into systems of algebraic equations, which are well suited for being solved with computers (20).

ANSYS is a software using FEM to solve differential equations. The main simulation technology includes Structural Mechanics, Systems and Multiphysics, Electromagnetics, Fluid Dynamics, Explicit Dynamics and Hydrodynamics. The workflow technology involves ANSYS Workbench Platform, High-Performance Computing, Geometry Interfaces and Simulation Process and Data Management (20).

ANSYS is a general purpose FEA tool which is commercially available and can be used for wide range of engineering application. Before we start using ANSYS for FEA modeling and simulation, there are certain set of questions

which need to be answered based on observation and engineering judgment. Questions may be like what is the objective of analysis? How to model entire physical system? How much details should be incorporated in system? How refine mesh should be in entire system or part of the whole system? To answer such questions computational expense must be compared to the level of accuracy of the results that needed. After that ANSYS can be employed to work in an efficient way after considering the following:

- Type of problem
- Time dependence
- Nonlinearity
- Modeling simplification

From observation and engineering judgment, analysis type has to be decided. In this study the analysis type is structural; to be specific out of different other structural problem focus in this study is on Static and Dynamic analysis. Non-linear material and geometrical properties such as plasticity, contact, and tensile strength are available (21).

3. FINITE ELEMENT MODEL OF THE OPTICAL SYSTEM

For finding dynamic characteristics of an optical system, finite element method can be used. In this thesis, ANSYS is used for solving the finite element model of the optical system. ANSYS has two versions, one is ANSYS Classic and another is ANSYS Workbench. Both have pre and post processing modules. Moreover, both can be connected directly to CAD programs, such as Pro/ENGINEER, in other words; designed optical system can be read by ANSYS from Pro/ENGINEER and also if something is changed in CAD model, finite element model can easily be updated (22).

4. ANSYS

ANSYS is a software publisher specializing in digital simulation. The company is headquartered in Canonsburg, Pennsylvania, in the United States. Its major products are software that implements the finite element method, in order to solve previously discretized models. The company has numerous subsidiaries¹ around the world, notably in Europe and Asia (23).

As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination, is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text (24)

5. Methods available for transient dynamic analysis in ANSYS:

Three methods are available to do a transient dynamic analysis in ANSYS:

(I) Full method that uses the full system matrices with no matrix reduction. It is the most general of the three methods with the following advantages,

- It is easy to use because there is no need to worry about choosing master degree of freedoms or mode shapes;
- It allows all types of nonlinearities to be included;
- It uses full matrices with no matrix approximation involved;
- It calculates all displacements and stresses in a single pass;

- It works with all types of loads (i.e., nodal forces, imposed displacements and pressure and temperature load);

The main disadvantage of the method is that it is the most expensive method than either of the other methods and must be avoided if we are not interested to consider nonlinearities in the simulation.

(II) Reduced method which condenses the problem size by using master degrees of freedom and reduced matrices. After finding the displacements at the master degrees of freedom, the program expands the solution to the original full degree of freedom set. The advantage of the reduced method is that it is faster and less expensive than the full method. However, the method has several disadvantages,

- The time step must remain constant throughout the solution.
- The only nonlinearity allowed is simple node-to-node contact (gap condition)
- All loads must be applied at master degrees of freedom. Element loads such as temperature and pressure cannot be applied

(III) Mode superposition that uses the mode shapes (eigenvectors) from a modal analysis to calculate the structure's response. The method has the following advantages,

- It is faster and less expensive than the reduced or full method for many problems.
- It accepts considering damping ratio as a function of mode number. The main disadvantages of the mode-superposition method are,

- The time step must remain constant throughout the analysis and automatic time stepping is not allowed.
- The only nonlinearity allowed is simple node-to-node contact (gap condition)
- It does not accept imposed nonzero displacements (25).

Aucune source spécifiée dans le document actif.

CHAPTER 3:
VIBRATION ANALYSIS
FOR FLAT SHEETS.

Chapter 3: Vibration analysis for flat sheets.

Abstract - it is an analytical study of free vibration of plates composed of composite materials containing an intermediate aperture using finite elements

We will perform the calculations using ANSYS software,

Through this study, we compare the results obtained with the results found in the references or in galactic experiments in this field

If the results match, then we do a numerical analysis of the vibrational behavior of the plates with or without holes, where a study will be conducted to find out the effect of factors on the subjective frequencies.

Key words - Free Vibration, Composite Materials, FEM, ANSYS Software,

I. INTRODUCTION:

Vibration is the body's swinging movement around its equilibrium position, and objects that have mass and elasticity have a natural predisposition to vibrate. Therefore, most engineering machinery and facilities are subject to vibration. This is what must be taken into consideration when designing and building machinery. There are two types of vibrations: free and forced vibrations (13).

It is therefore essential to study these natural frequencies and find ways to avoid resonance.

Resonance occurs when the forced vibrations equal the normal pulse, and this leads to the emergence of large fluctuations. And that the capacitance in this case is only the amount of damping in the system, in order to avoid reaching this stage, it is necessary to know in advance the natural vibration of the system (14).

II. MODAL ANALYSIS:

1. Modal analysis of a Plate without holes:

a composite beams of dimensions $550\text{mm} \times 50\text{mm} \times 10\text{mm}$ of epoxy glass fibres having thickness 0.5 mm is modeled using ANSYS. the material properties for woven glass fabric are considered as:

$$E_{11} = 250\text{GPA}, E_{22} = 250\text{GPA},$$

$$E_{33} = 0.6E_{11}$$

and modulus of rigidity $G = 4\text{GPA}$.

The plate is fixed at one end.

a) Units of analysis:

TABLE 1: Unit System.

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s

b) Geometry:

TABLE 2: Geometric characteristic.

Object Name	Geometry
State	Fully Defined
Definition	
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	550, mm
Length Y	50, mm
Length Z	10, mm

TABLE 3: Plate properties.

Object Name	Solid
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	epoxy glass
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Properties	
Volume	2,75e+005 mm ³
Mass	0,6325 kg

Centroid X	-9,031e-015 mm
Centroid Y	2,8222e-016 mm
Centroid Z	5, mm
Moment of Inertia Ip1	137,04 kg·mm ²
Moment of Inertia Ip2	15950 kg·mm ²
Moment of Inertia Ip3	16076 kg·mm ²
Statistics	
Nodes	2228
Elements	275
Mesh Metric	None

	Controlled
Statistics	
Nodes	2228
Elements	275
Mesh Metric	None

Figure 1: Dimensional determination of the studied model Figure 2: The 3D model shows the studied body. Figure 3 shows the FE model.

c) Mesh:

TABLE 4: Mesh Properties.

<i>Object Name</i>	<i>Mesh</i>
State	Solved
Sizing	
Relevance Center	Coarse
Element Size	10,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	10,0 mm
Inflation	
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
Patch Conforming Options	
Triangle Surface Mesher	Program

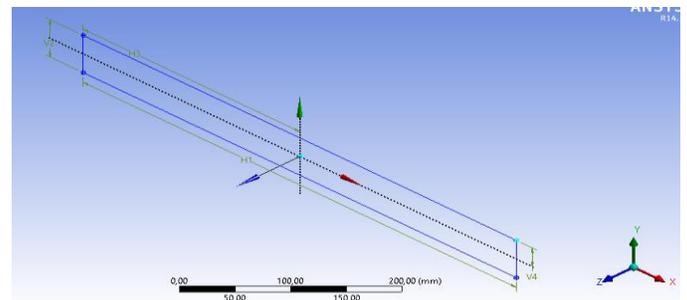


Figure 1: The basic dimensions of the model.

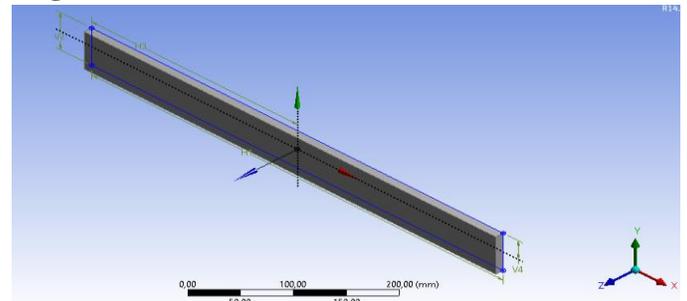


Figure 2: A 3D model of the body.

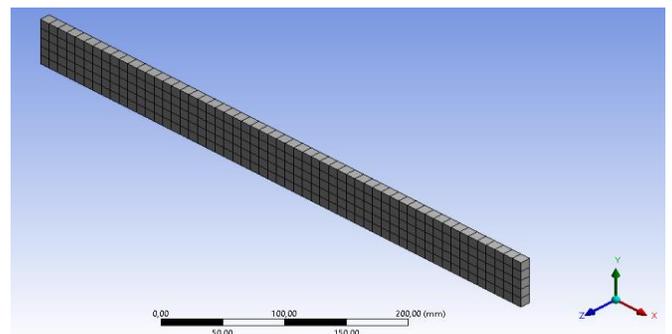
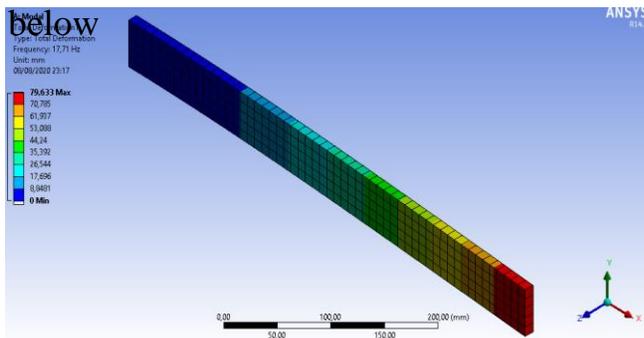


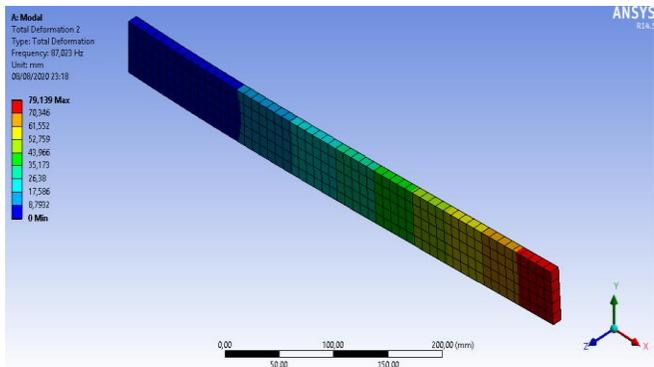
Figure 3: The FE model of the model.

d)Solution:

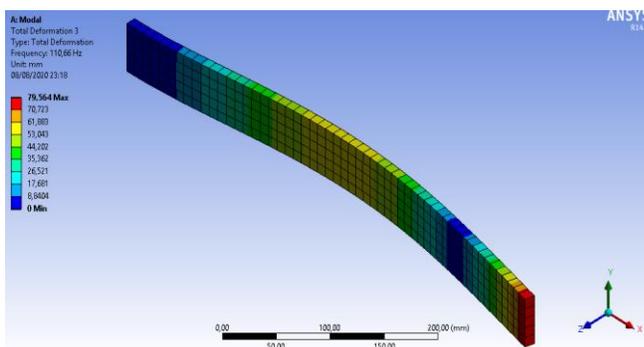
The 1st 4 modes of vibration were found using ANSYS, the results of the modal analysis are tabulated in Table 5



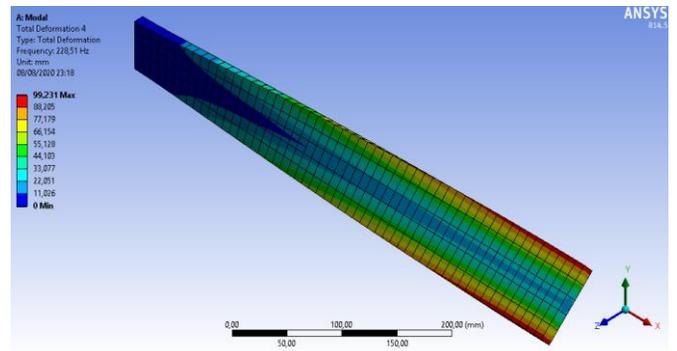
Mode 1



Mode 2



Mode 3



Mode 4

The following bar chart indicates the frequency at each calculated mode.

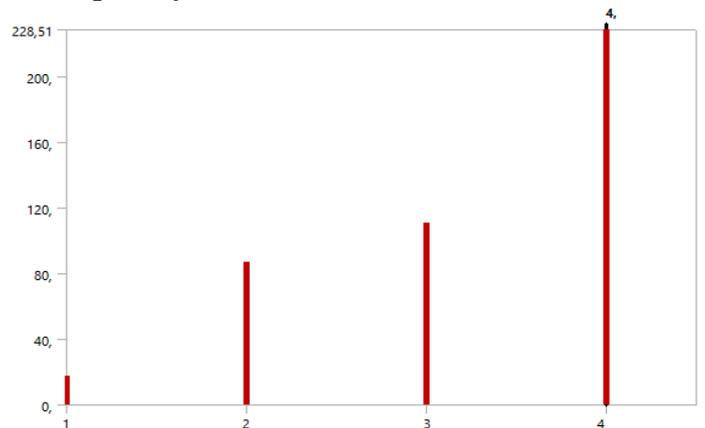


TABLE 5: Natural Frequencies using ANSYS.

<i>Mode</i>	<i>Frequency [Hz]</i>
1,	17,71
2,	87,023
3,	110,66
4,	228,51

TABLE 6: Modal Solution Information

Object Name	Solution Information
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2,5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type

TABLE 7: Model Solution Results.

Object Name	Total Deformation 1	Total Deformation 2	Total Deformation 3	Total Deformation 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Total Deformation			
Mode	1,	2,	3,	4,
Identifier				
Suppressed	No			
Results				
Minimum	0, mm			
Maximum	79,633	79,139	79,56	99,23

m	mm	mm	4 mm	1 mm
Information				
Frequency	17,71 Hz	87,023 Hz	110,6 Hz	228,5 Hz

e) Harmonic Response

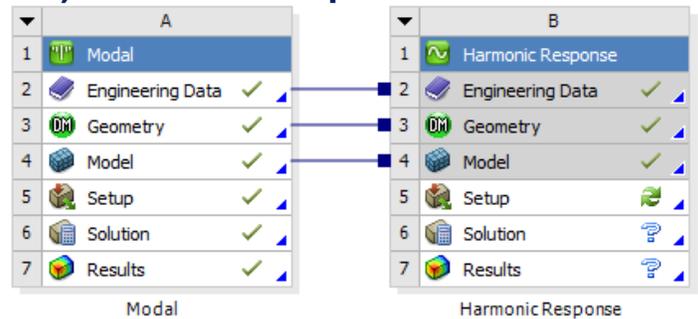


TABLE 8: Analysis Harmonic Response Model.

Object Name	Harmonic Response
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Harmonic Response
Solver Target	Mechanical APDL
Options	
Environment Temperature	22, °C
Generate Input Only	No

TABLE 9: Model Harmonic Response Analysis Settings.

Object Name	Analysis Settings
State	Fully Defined
Options	
Range Minimum	15, Hz
Range Maximum	250, Hz
Solution Intervals	100
Solution Method	Mode

	Superposition
Modal Frequency Range	Program Controlled
Store Results At All Frequencies	Yes
Output Controls	
Stress	Yes
Strain	Yes
Nodal Forces	Yes
Calculate Reactions	Yes
General Miscellaneous	No
Damping Controls	
Constant Damping Ratio	0,
Stiffness Coefficient Define By	Direct Input
Stiffness Coefficient	0,
Mass Coefficient	0,

X Component		0, N
Y Component		0, N
Z Component		150, N
Phase Angle		0, °

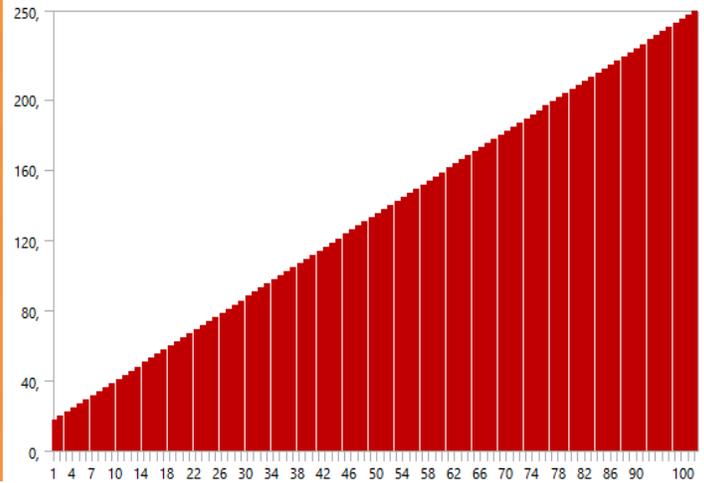


Figure 4: Model Harmonic Response.

TABLE 10: Model Harmonic Response Loads.

Object Name	Fixed Support	Force
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	1 Edge
Definition		
Type	Fixed Support	Force
Suppressed	No	
Define By		Components
Coordinate System		Global Coordinate System

TABLE 11: Model Harmonic Response Solution Result Charts.

Object Name	Frequency Response	Frequency Response 2	Frequency Response 3	Frequency Response 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	6 Faces			
Spatial Resolution	Use Average			
Definition				

Type	Normal Stress	Normal Elastic Strain	Directional Deformation	Directional Acceleration
Orientation	Z Axis			X Axis
Suppressed	No			
Options				
Frequency Range	Use Parent			
Minimum Frequency	15, Hz			
Maximum Frequency	250, Hz			
Display	Bode			
Results				
Maximum Amplitude	2,962e-010 MPa	8,3161e-013 mm/m	747,45 mm	1,2658e-004 mm/s ²
Frequency	203, Hz	22,05 Hz	17,35 Hz	111,35 Hz
Phase Angle	180, °		0, °	180, °
Real	-2,962e-010 MPa	-8,3161e-013 mm/m	747,45 mm	-1,2658e-004 mm/s ²

Imaginary	0, MPa	0, mm/m	0, mm	0, mm/s ²
-----------	--------	---------	-------	----------------------

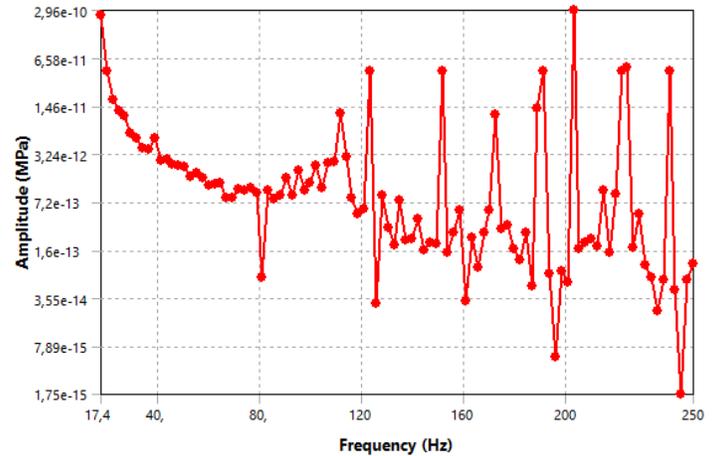


Figure 5: Model Harmonic Response Solution Frequency Response.

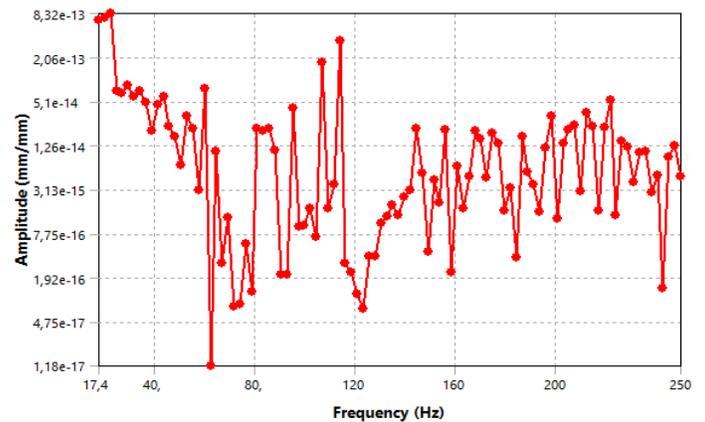


Figure 6: Model Harmonic Response Solution Frequency Response 2.

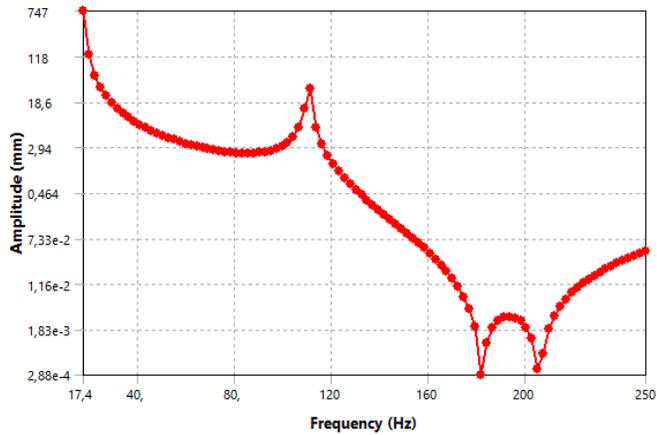


Figure 7: Model Harmonic Response
Solution Frequency Response 3.

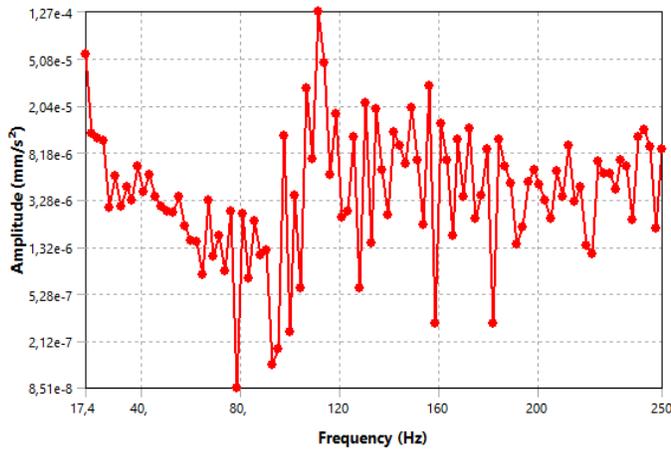


Figure 8: Model Harmonic Response
Solution Frequency Response 4.

f) Epoxy glass

TABLE 12: epoxy glass Orthotropic Elasticity.

Young's Modulus X direction MPa	Young's Modulus Y direction MPa	Young's Modulus Z direction MPa
25000	25000	15000
Poisson's Ratio XY	Poisson's Ratio YZ	Poisson's Ratio XZ
0,38	0,38	0,25
Shear Modulus XY MPa	Shear Modulus YZ MPa	Shear Modulus XZ MPa
4000,	4000,	4000,

III. Typical analysis of a central hole plate

Figure 1: The 3D model shows the studied body. Figure 2 shows the FE model.

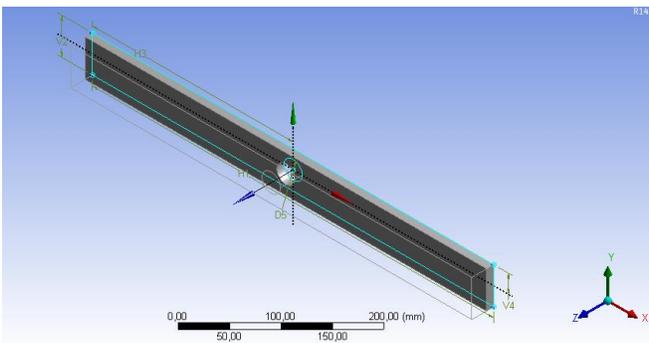


Figure 9: 3D model of a plate with a central hole.

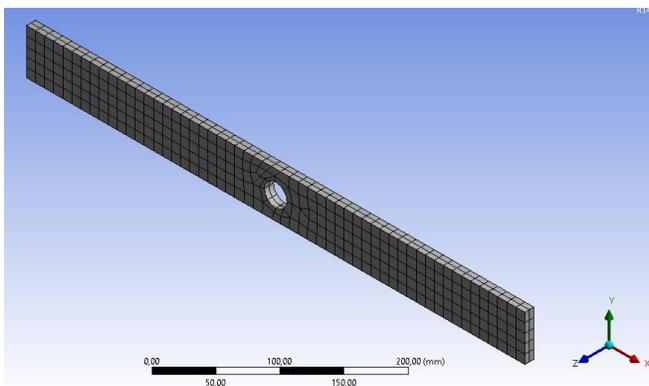
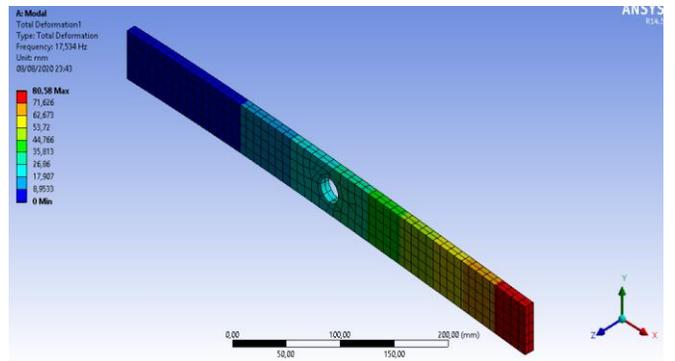
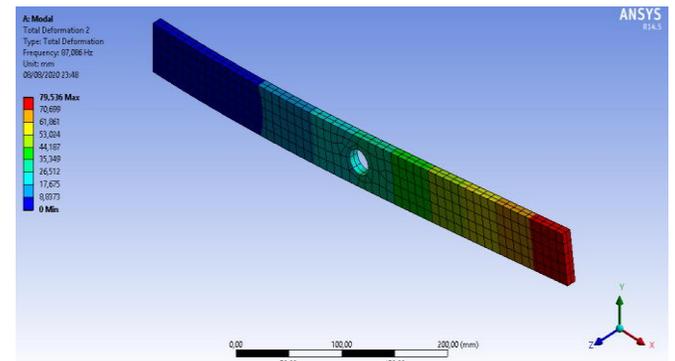


Figure 10: FE model of a central hole plate.

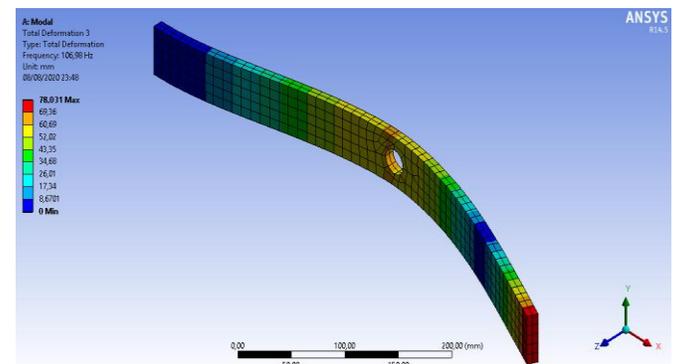
The 1st 4 modes of vibration were found using ANSYS, the results of the modal analysis are tabulated in Table 15 below



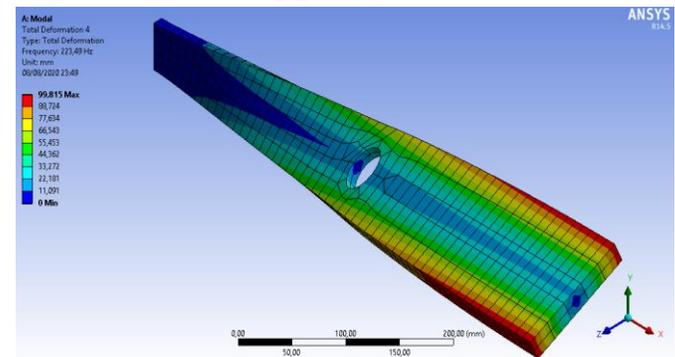
Mode 1



Mode 2



Mode 3



Mode 4

TABLE 13: Model Geometry Parts.

Object Name	Solid
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	epoxy glass
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	550, mm
Length Y	50, mm
Length Z	10, mm
Properties	
Volume	2,7009e+005 mm ³
Mass	0,62121 kg
Centroid X	-2,1908e-015 mm
Centroid Y	0, mm
Centroid Z	5, mm
Moment of Inertia Ip1	136,51 kg·mm ²
Moment of Inertia Ip2	15949 kg·mm ²
Moment of Inertia Ip3	16075 kg·mm ²
Statistics	
Nodes	3551
Elements	554

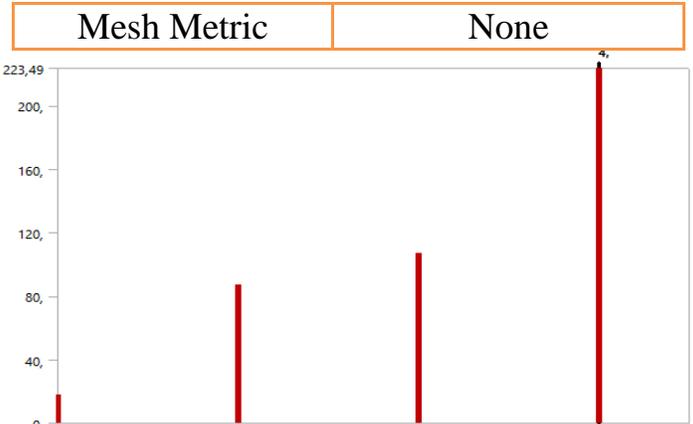


Figure11: Natural Frequencies using ANSYS.

TABLE 14: Natural Frequencies using ANSYS.

Mode	Frequency [Hz]
1,	17,534
2,	87,086
3,	106,98
4,	223,49

TABLE 15: Model Solution Results.

Object Name	Total Deformation 1	Total Deformation 2	Total Deformation 3	Total Deformation 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Total Deformation			
Mode	1,	2,	3,	4,
Identifier				

Suppressed	No			
Results				
Minimum	0, mm			
Maximum	80,58 mm	79,536 mm	78,031 mm	99,815 mm
Information				
Frequency	17,534 Hz	87,086 Hz	106,98 Hz	223,49 Hz

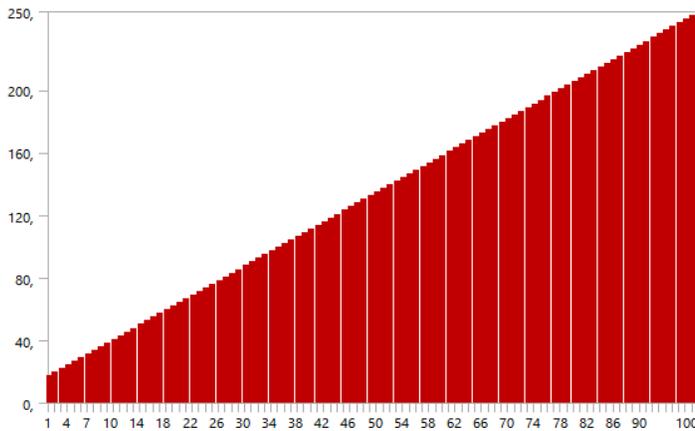


Figure 12: Model Harmonic Response Solution.

TABLE 16: Model Harmonic Response Solution Result Charts.

Object Name	Frequency Response	Frequency Response 2	Frequency Response 3	Frequency Response 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	7 Faces			
Spatial Resolution	Use Average			

Definition				
Type	Normal Stress	Normal Elastic Strain	Directional Deformation	Directional Acceleration
Orientation	Z Axis			
Suppressed		No		
Options				
Frequency Range	Use Parent			
Minimum Frequency	15, Hz			
Maximum Frequency	250, Hz			
Display	Bode			
Results				
Maximum Amplitude	6,429 1e-010 MPa	6,8232 e-013 mm/m	148 2,2 mm	3,387 2e+007 mm/s ²
Frequency	31,45 Hz	17,35 Hz		106,65 Hz
Phase Angle	180, °	0, °		
Real	- 6,429 1e-010 MPa	6,823 2e-013 mm/mm	1482,2 mm	3,38 72e+007 mm/s ²
Imaginary	0, MPa	0, mm/mm	0, mm	0, mm/s ²

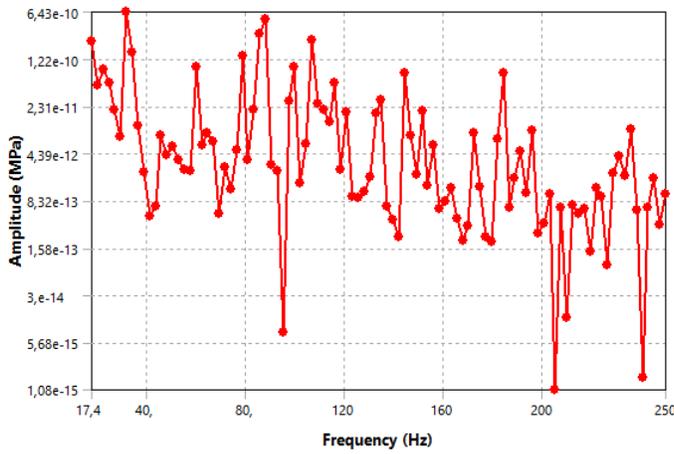


Figure 13: Model Harmonic Response
Solution Frequency Response.

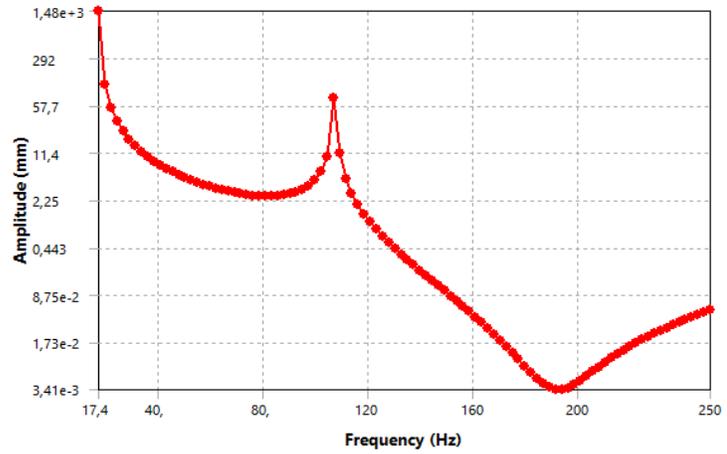


Figure 15: Model Harmonic Response
Solution Frequency Response 3.

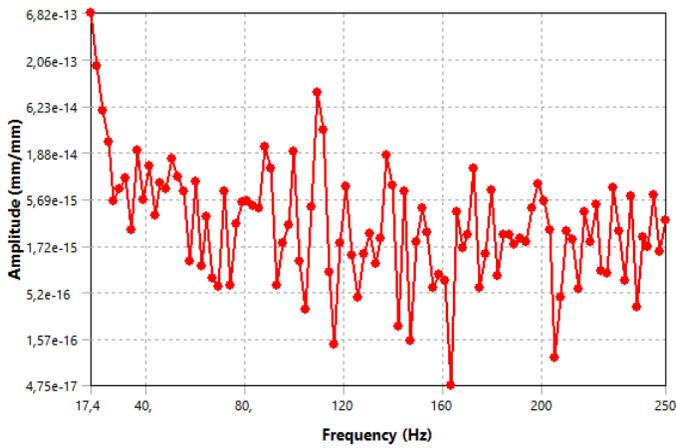


Figure 14: Model Harmonic Response
Solution Frequency Response 2.

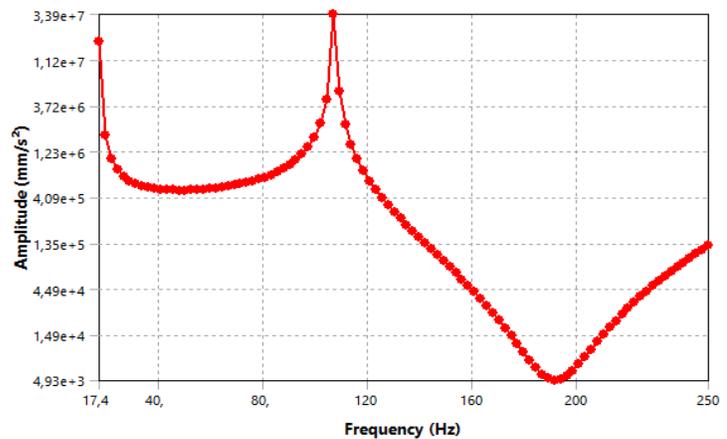


Figure 16: Model Harmonic Response
Solution Frequency Response 4

IV. A typical plate with two holes analysis

Figure 1: The 3D model shows the body under study. Figure 2 illustrates the FE model.

g) Geometry

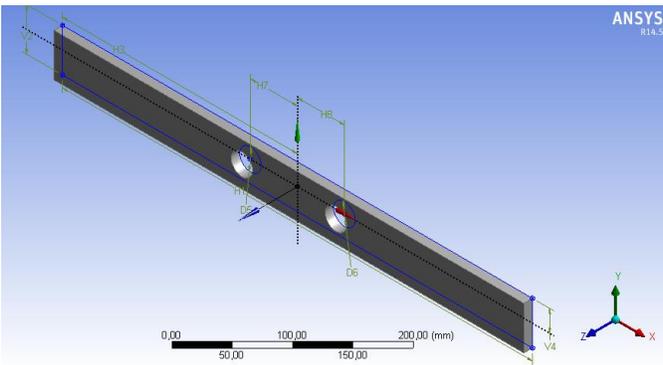


Figure17: A 3D model of a two-hole plate.

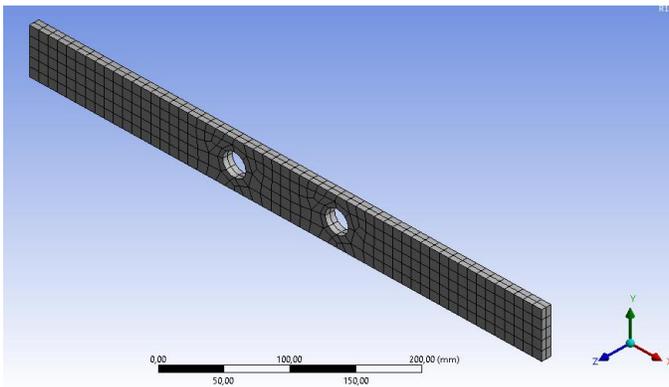
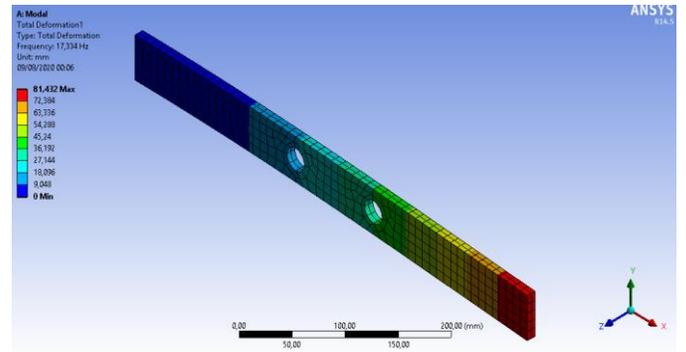


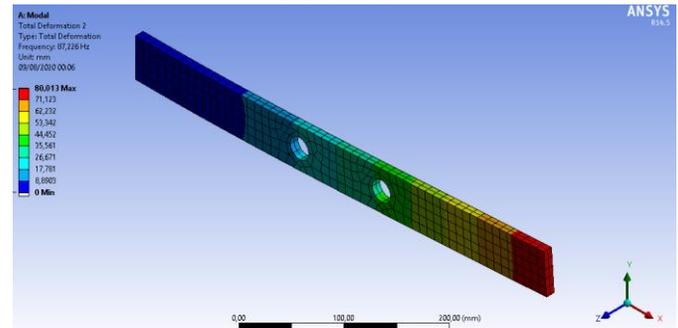
Figure18: FE model for plate.

h) Solution:

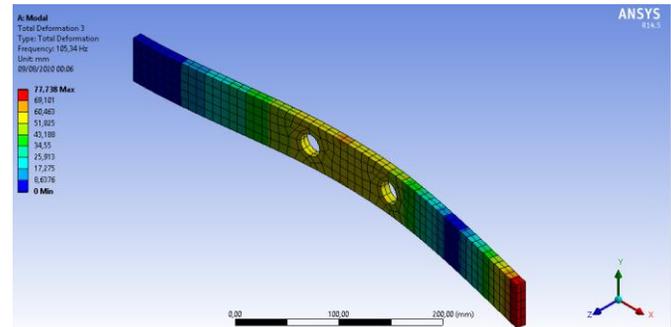
The 1st 4 modes of vibration were found using ANSYS, the results of the modal analysis are tabulated in Table 19 below.



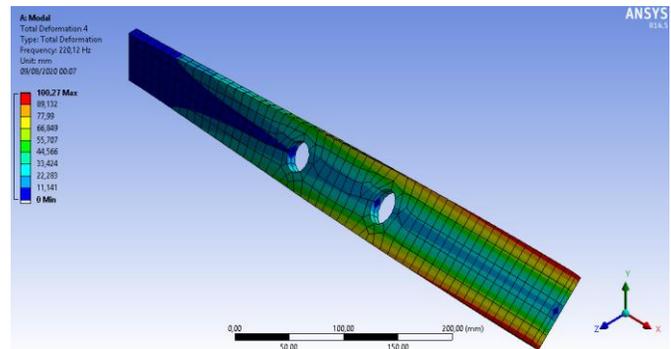
Mode 1



Mode 2



Mode 3



Mode 4

TABLE 17: Model Geometry Parts.

Object Name	Solid
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	epoxy glass
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	550, mm
Length Y	50, mm
Length Z	10, mm
Properties	
Volume	2,6518e+005 mm ³
Mass	0,60992 kg
Centroid X	-8,6869e-015 mm
Centroid Y	1,6002e-016 mm
Centroid Z	5, mm
Moment of Inertia Ip1	135,98 kg·mm ²
Moment of Inertia Ip2	15881 kg·mm ²
Moment of Inertia Ip3	16006 kg·mm ²
Statistics	
Nodes	3586
Elements	554
Mesh Metric	None

The following bar chart indicates the frequency at each calculated mode.

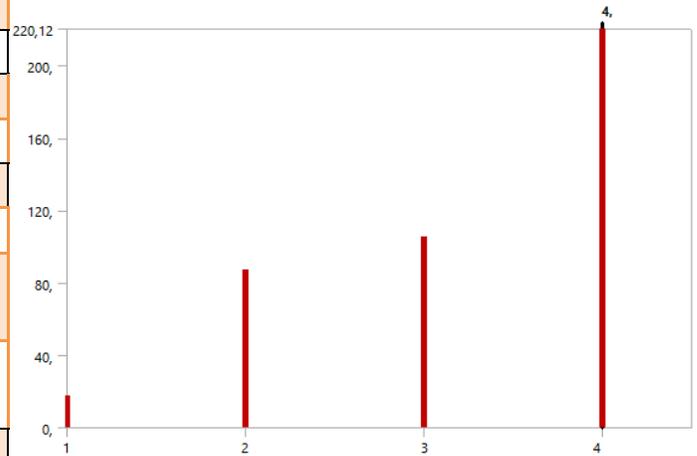


Figure19: Natural Frequencies using ANSYS.

TABLE 18: Natural Frequencies using ANSYS.

Mode	Frequency [Hz]
1,	17,334
2,	87,226
3,	105,34
4,	220,12

TABLE 19: Model Solution Results.

Object Name	Total Deformation 1	Total Deformation 2	Total Deformation 3	Total Deformation 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Total Deformation			
Mode	1,	2,	3,	4,

Identifier				
Suppressed	No			
Results				
Minimum	0, mm			
Maximum	81,432 mm	80,013 mm	77,738 mm	100,27 mm
Information				
Frequency	17,334 Hz	87,226 Hz	105,34 Hz	220,12 Hz

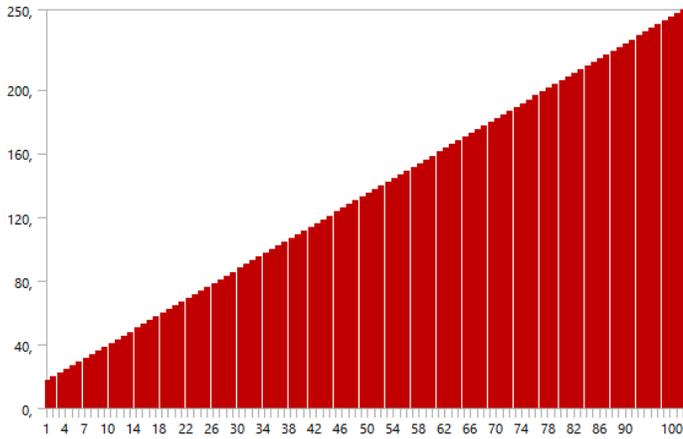


figure 20: Model Harmonic Response Solution.

TABLE 20: Model Harmonic Response Solution Result Charts.

Object Name	Frequency Response 1	Frequency Response 2	Frequency Response 3	Frequency Response 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	8 Faces			
Spatial	Use Average			

Resolution				
Definition				
Type	Normal Stress	Normal Elastic Strain	Directional Deformation	Directional Acceleration
Orientation	Z Axis			
Suppressed	No			
Options				
Frequency Range	Use Parent			
Minimum Frequency	15, Hz			
Maximum Frequency	250, Hz			
Display	Bode			
Results				
Maximum Amplitude	1,0125e-008 MPa	4,2789e-011 mm/m	17360 mm	2,0631e+008 mm/s ²
Frequency	17,35 Hz			
Phase Angle	180, °			0, °
Real	-1,0125e-008 MPa	-4,2789e-011 mm/m	-17360 mm	2,0631e+008 mm/s ²
Imaginary	0, MPa	0, mm/m	0, mm	0, mm/s ²

V. A typical analysis of a three-hole plate

Figure 1: The 3D model shows the body being studied. Figure 2 illustrates the FE model.

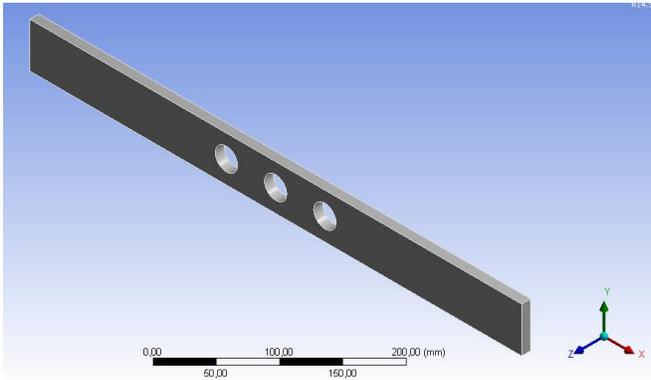


Figure 21: 3D model of the three-hole plate

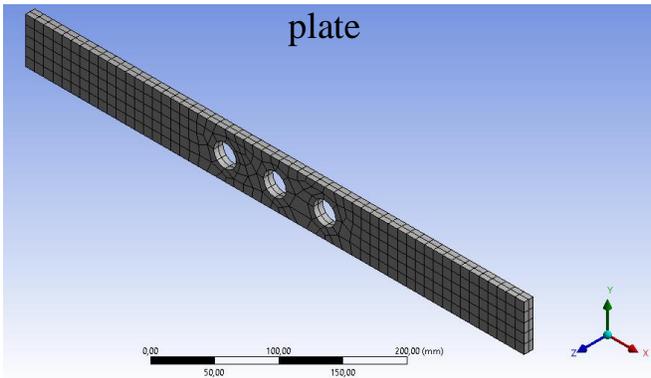


Figure 22: FE model for plate.

The 1st 4 modes of vibration were found using ANSYS, the results of the modal analysis are tabulated in Table 5.

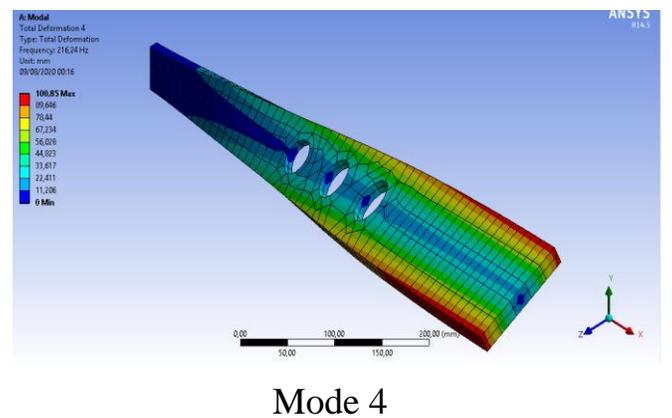
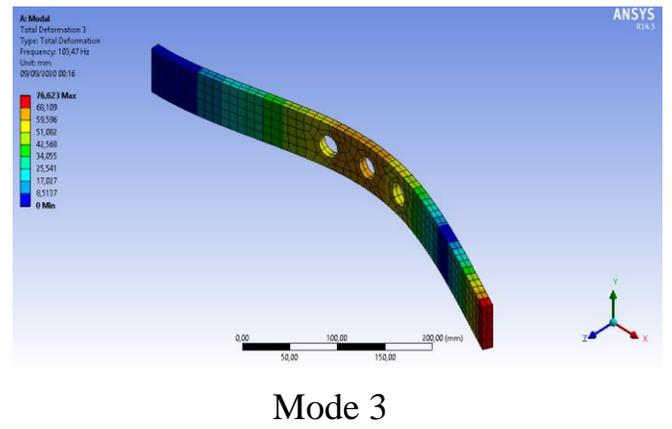
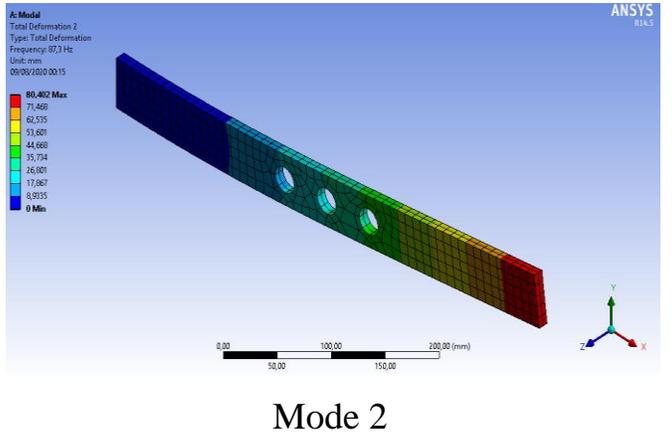
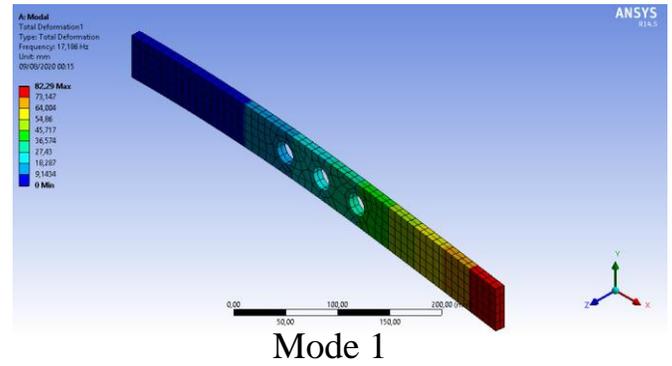


TABLE 21: Model Geometry Parts.

Object Name	<i>Solid</i>
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	epoxy glass
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	550, mm
Length Y	50, mm
Length Z	10, mm
Properties	
Volume	2,6027e+005 mm ³
Mass	0,59863 kg
Centroid X	-7,9556e-015 mm
Centroid Y	5,1236e-017 mm
Centroid Z	5, mm
Moment of Inertia Ip1	135,45 kg·mm ²
Moment of Inertia Ip2	15880 kg·mm ²
Moment of Inertia Ip3	16005 kg·mm ²
Statistics	
Nodes	3594
Elements	552

Mesh Metric

None

Mesh

TABLE 22: Model Mesh.

Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	10,0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	10,0 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Advanced	
Shape Checking	Standard Mechanical
Element Midside	Program

Nodes	Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	3594
Elements	552
Mesh Metric	None

The following bar chart indicates the frequency at each calculated mode.

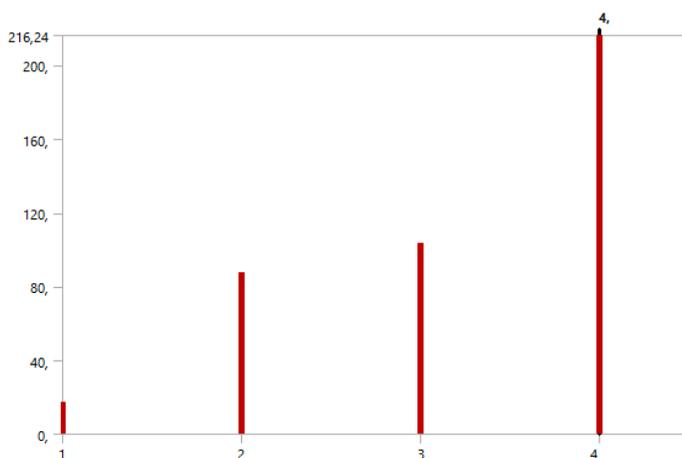


Figure 23: Natural Frequencies using ANSYS.

TABLE 23: Natural Frequencies using ANSYS.

Mode	Frequency [Hz]
1,	17,186
2,	87,3
3,	103,47
4,	216,24

Object Name	<i>Total Deformation1</i>	<i>Total Deformation 2</i>	<i>Total Deformation 3</i>	<i>Total Deformation 4</i>
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	All Bodies			
Definition				
Type	Total Deformation			
Mode	1,	2,	3,	4,
Identifier				
Suppressed	No			
Results				
Minimum	0, mm			
Maximum	82,29 mm	80,402 mm	76,623 mm	100,85 mm
Information				
Frequency	17,186 Hz	87,3 Hz	103,47 Hz	216,24 Hz

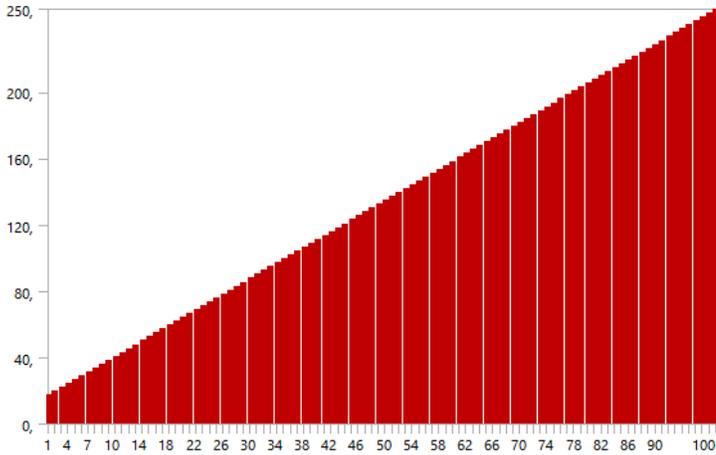


Figure 24: Model Harmonic Response Solution.

TABLE 24: Model Harmonic Response Solution Result Charts.

Object Name	Frequency Response	Frequency Response 2	Frequency Response 3	Frequency Response 4
State	Solved			
Scope				
Scoping Method	Geometry Selection			
Geometry	9 Faces			
Spatial Resolution	Use Average			
Definition				
Type	Normal Stress	Normal Elastic Strain	Directional Deformation	Directional Acceleration
Orientation	Z Axis			
Suppressed	No			

Options				
Frequency Range	Use Parent			
Minimum Frequency	15, Hz			
Maximum Frequency	250, Hz			
Display	Bode			
Results				
Maximum Amplitude	6,8971 e-008 MPa	7,455 e-013 mm/mm	1712,6 mm	2,0353 e+007 mm/s ²
Frequency	17,35 Hz			
Phase Angle	0, °	180, °		0, °
Real	6,8971 e-008 MPa	- 7,455 e-013 mm/mm	- 1712,6 mm	2,0353 e+007 mm/s ²
Imaginary	0, MPa	0, mm/mm	0, mm	0, mm/s ²

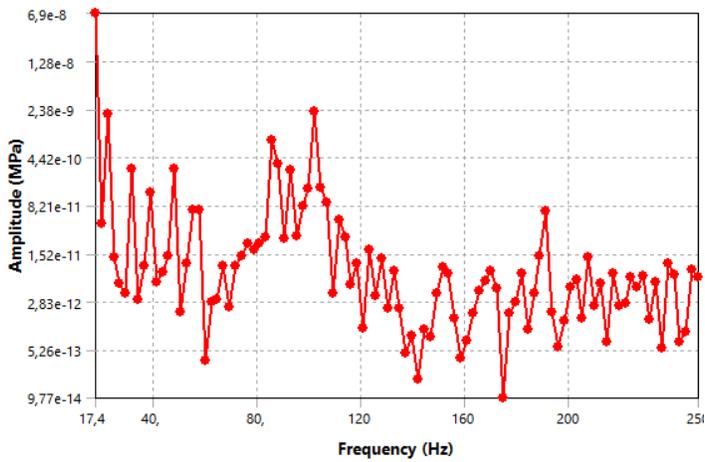


Figure 25: Model Harmonic Response Solution Frequency Response.

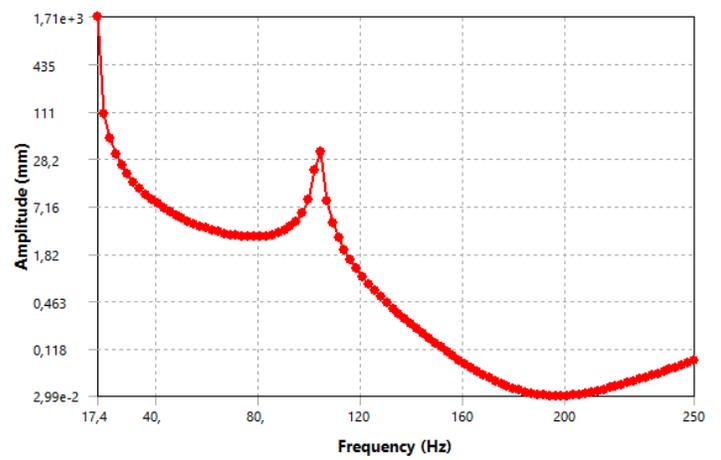


Figure 27: Model Harmonic Response Solution Frequency Response 3.

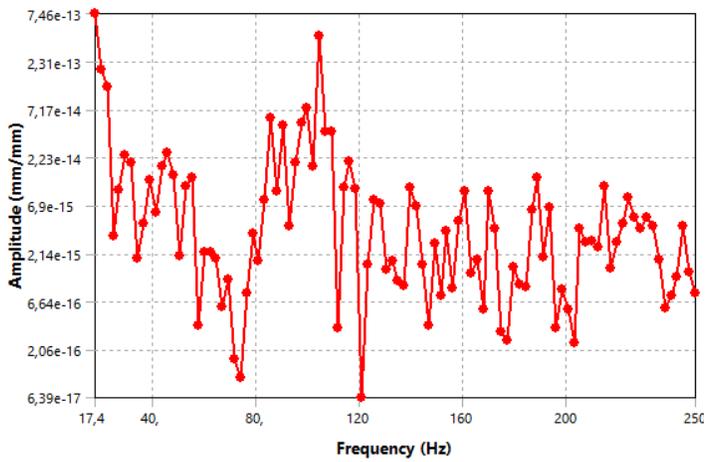


figure26: Model Harmonic Response Solution Frequency Response 2.

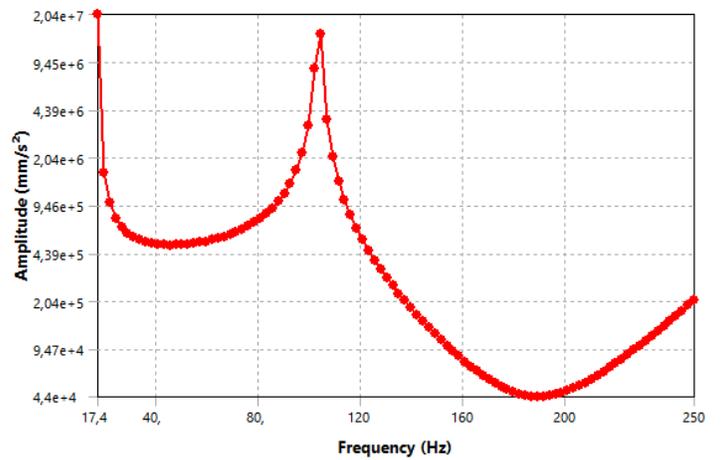


Figure 28: Model Harmonic Response Solution Frequency Response 4.

GENERAL CONCLUSION

The summary of the work is an analytical study of the free vibration of flat panels (with or without holes), which are composed of composite materials; Use of ANSYS program whose business concept is "finite element method".

When conducting this study, we found that the results are identical. Where we give all dimensions of the plate to the ANSYS program and enter the properties of the composite material that makes up the plate using the laws of composite materials. Then we installed one end of the board and determined the type of network we want the element size: 10 mm, the number of nodes: 2228 knots, then calculate the first four natural frequencies.

The first mode frequency: 17.71 Hz, the second mode frequency: 87.023 Hz, the third mode frequency: 110.66 Hz, and the fourth mode frequency: 228.51 Hz. We note that these results are approximately identical to the results in reference (26). Which includes: The first mode: the frequency 17.5 Hz, the second mode: the frequency 91.00 Hz, and the third mode: the frequency 118 Hz. These results are relatively identical.

Then we put a central hole in the plate and again we made two and three holes, then we kept the same dimensional properties and the same composite material properties for the studied body. We note that as the number of holes increases, the frequency ratio decreases for the first position, increases for the second position, decreases for the third position, and also decreases for the fourth position.

Thus we conclude that holes affect the plate frequencies; therefore, we should pay attention to him when studying any plate and it can be calculated with ANSYS program; Knowing that the principle of operation of this program ANSYS is the finite element method, through which we are able to find the closest solution to the exact solution. And

A typical analysis of a three-hole plate Modal analysis of a Plate without holes:

as we mentioned earlier, all devices of mass and flexibility have a natural predisposition to free vibration.

Bibliographie

1. Berthelot, J. M. (1999). *Matériaux composites. Comportement mécanique et analyse des structures.*
2. SAFA, LAYACHI et ABDELOUAHAB, T. A. T. I. *Vibration des plaques stratifiées menus d'ouvertures rectangulaires excentrées.*
3. بسوق عُكاظ - جونز روبرت م - ميكانيك المواد المركبة
4. Laurent Gornet. *Généralités sur les matériaux composites. Engineering school. 2008.*
5. SUBRAMANI, T. et VISHNUPRIYA, S. *Finite Element Analysis of a Natural Fiber (Maize) Composite Beam. International Journal of Modern Engineering Research, vol. 4, no 6, p. 1-7.*
6. Nadia BAHLOULI *Cours Matériaux Composites / DESS Mécanique avancée et Stratégie industrielle.*
7. Berthelot, Jean-Marie. *"Mécanique des matériaux et structures composites." Institut Supérieur des Matériaux et Mécaniques Avancés (2010).*
8. Gay, D. (2014). *Composite materials: design and applications. CRC press.*
9. https://www.doitpoms.ac.uk/tlplib/bones/derivation_mixture_rules.php.
10. <https://asmedigitalcollection.asme.org/appliedmechanicsreviews/articleabstract/40/7/879/42117>
11. Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. (2005). *The finite element method: its basis and fundamentals. Elsevier.*
12. Thomas, M., & Laville, F. (2007). *Simulation des vibrations mécaniques: par Matlab. Simulink et Ansys—Presses de l'Université du Québec—9 Pages.*

13. مذكرة محاضرات في الاهتزازات الميكانيكية السودان
14. Aagaah, M. R., Mahinfalah, M., & Jazar, G. N. (2006). *Natural frequencies of laminated composite plates using third order shear deformation theory. Composite structures, 72(3), 273-279.*
15. (2017). محمد عبد اللطيف صالح ,معاوية إبراهيم أحمد ,ياسين صالح إبراهيم ,اشراف & محمود يس عثمان . اهتزاز الهياكل باستخدام طريقة العناصر المحددة .
16. Hutton, D. V. (2004). *Fundamentals of finite element analysis. McGraw-hill. .*
17. Afkar, A., & Kamari, M. N. (2016). *Analysis of free and forced vibration of FGM rectangular floating plates (in contact with fluid) using the theory of Mindlin. Journal of Materials and Environmental Science, 7(9), 3264-3277.*
17. Grunwald, B. (2018). *Vibration analysis of shaft in SolidWorks and ANSYS.*
18. Afkar, A., & Kamari, M. N. (2016). *Analysis of free and forced vibration of FGM rectangular floating plates (in contact with fluid) using the theory of Mindlin. Journal of Materials and Environmental Science, 7(9), 3264-3277.*
19. Bathe, K. J. (2006). *Finite element procedures. Klaus-Jurgen Bathe.*
20. Ge, X. (2014). *Simulation of vibrations in electrical machines for hybrid-electric vehicles (Doctoral dissertation, MS Thesis, Department of Applied Mechanics, Chalmers University of Technology, Göteborg (Sweden)).*
21. Bharucha, F. M. (2016). *Computational Analysis of IT Rack Under Vibration Load.*
22. Avşar, A. L. (2008). *Dynamic modeling and analysis of vibration effects on performance in optical systems (Master's thesis).*

23. *Amhammed, M., & Hamza, O. (2014). Détermination expérimentale et par simulation numérique des paramètres modaux pour les plaques minces (Doctoral dissertation).*
24. *Bathe, K-J. Finite Element Procedures. Englewood Cliffs, NJ: Prentice-Hall, 1996.*
25. *Liaghat, T., Guibault, F., Allenbach, L., & Nennemann, B. (2014, April). Two-way fluid-structure coupling in vibration and damping analysis of an oscillating hydrofoil. In ASME 2014 International Mechanical Engineering Congress and Exposition. American So.*
26. *Vader, S. S., & Raikar, V. A. (2017). Crack detection in composite cantilever beam by Vibration analysis and Numerical method.*

Summary Summary

All structures that have mass and flexibility have a natural Susceptibility to vibration such as structural structures and machines. This research deals with an analytical study of the vibrations of the flat plate with or without holes, which is made up of composite materials. The calculations were performed by ANSYS 14.5 program whose working principle is the finite element method. The most important finding of the study is that holes change the natural vibrations of flat plates.

المملخص المملخص

جميع الهياكل ذات الكتلة والمرونة لديها استعداد طبيعي للاهتزاز مثل الهياكل والآلات الهيكلية. يتناول هذا البحث دراسة تحليلية لاهتزاز للألواح المسطحة مع أو بدون ثقوب، والتي تتكون من مواد مركبة. تم إجراء الحسابات بواسطة برنامج ANSYS 14.5 الذي يتمثل مبدأ عمله في طريقة العناصر المحدودة. أهم ما توصلت إليه الدراسة هو أن الثقوب تغير الاهتزازات الطبيعية للألواح المسطحة.