CHAPTER – 4 - EXPERIMENTAL METHODOLOGIES

4.1 PREPARATION OF SPECIMENS:
Materials Used:
1. Bi-woven Glass (E, S), Graphite, Carbon, Kevlar-49 fibers
2. Epoxy Resin
3. Hardener
4. Wax Polish
5. Polyvinyl Alcohol (PVA),
6. Mylar Film sheets
7. Chalk Powder

Fig – 4.1 - Epoxy resin and hardener

Fig – 4.2 - Mylar film and glass fiber
Fig 4.3 - Wax polish is applied on table on which acts as a release film.

Fig 4.4 - Poly vinyl Alcohol (PVA) is mixed with water and is applied on the table.
Fig – 4.5 - Powdered form of PVA can be mixed with water to form gel and then can be applied on the table.

Fig – 4.6 - Epoxy-resin and poly-amine hardener are mixed in the ratio 100:10
Fig 4.7 - After 9 layers are done Mylar film is spread on it to get a smooth surface finish.

Fig 4.8 – Laying of fiber cloth
Fig – 4.9 – Graphite cloth being layered
Fig- 4.10 - Resin being applied
Fig – 4.11 – Laminate being covered by Mylar Film & Vacuum being applied
Investigations on Dynamic Properties of Polymer Based Composites

**Fig 4.12** – Vacuum bagging technique

**Fig 4.13** – Laminate after curing

**Fig 4.14** – Temperature controlled Furnace
4.2 - Design of Vibration Test Fixture:
A fixture is a device for locating, holding and supporting a work piece during a manufacturing operation or while conducting an experiment. It should provide the required constraint to the member. The design of the fixtures that were to be used in this analysis was extremely important, as the dynamic behavior of any structure/member depends on the boundary conditions, apart from several other factors.

Vibration test fixtures are required to allow mounting of the test specimen to the fixture as well as to allow for testing in all the three orthogonal directions. The design of vibration test fixtures is critical to avoid errors in equipment test response due to any resonances of the impact hammer, table and the fixture itself. Ideally the laboratory mounting should replicate the physical conditions observed in service such as the stiffness, mass and the consequent resonant responses of the actual service installation.

The key issues in the design of vibration test fixture are as follows:

- Allow for ease of mounting of the test item on the vibration machine
- Allow for vibration testing in each of the three orthogonal directions with minimum cross talk. i.e. motion in the two orthogonal directions not being tested.
- Ensure the absence of fixture resonances within the specified test frequency range by tailoring the dynamic response of the fixture and the table
- Weight and force limitations of vibration machine
- Distribution of vibration energy uniformly throughout the test item.

From the Literature survey carried out, it was found that most of the experimental procedures were carried out using:

1. Hanging the specimen by Elastic cords in a specially designed Frame
2. Using Sponge pads

Keeping this in mind, we designed a special low cost fixture on which all the three boundary conditions could be carried out along with different thickness of the specimen.

The fixtures were designed and fabricated shown in fig 4.17 below keeping in mind the stiffness of the laminated panel as well as its dimensions. The fixture were fabricated to accommodate three boundary condition namely, cantilever (c-f-f-f) and two sides fixed (c-f-c-f), all sides fixed (c-c-c-c).

**4.3 - PROCEDURE:** To conduct a model test, a test fixture was designed. The test fixture is made up of mild steel having the dimensions of (150*150*20 mm) square plate. We prepared two brackets to mount above the test fixture. The dimensions are (150*20*20mm). In the both plate we made a drill hole, the each hole distance is 37.5mm and the hole diameter is 6mm and we carried a threading operation in the each hole, for matching to each hole we procured aligned bolt. The diameter of the aligned bolt is 6mm and bolt head is 8mm. For correct fitting of the specimen we prepared a supporting plate with a dimension (150*20*2mm). The whole weight of a test fixture is around 7kgs.

![Fig - 4.17 - View of Fixture](image)
Thus a suitable fixture to hold the Specimens under all the 3 boundary Conditions were designed and fabricated. A number of trials were conducted to see whether the specimens were held properly without damaging them and ultimately it was found to be a good fixture capable of holding the specimens properly.
4.4 - MODELLING

Various fibers such as Glass, Carbon, Graphite, Kevlar were chosen to model using Finite Element tool ANSYS. The size of the laminate was fixed at 150x150mm with thickness of 2mm and 4mm and 3 boundary conditions namely Cantilever, both Ends Fixed & All sides fixed were simulated. A number of trials were made in the selection of element for meshing of the fibers and analyzed. Finally, SHELL99 appears to yield good results and was chosen to mesh the laminates with different orientation. Stacking sequence of Fibers (0° and 90°) was done. Initially, coarse mesh was chosen and results were found to be inappropriate, hence model was prepared with fine mesh which improved the results drastically thereby confirming to the convergence criteria.

Models

The word “model” has the traditional meaning of a scaled copy or representation of an object. We use here the term in a more modern sense, which has become increasingly common since the advent of computers:

A model is a symbolic device built to simulate and predict aspects of behavior of a system.

To predict everything, in all physical scales, we must deal with the actual system. A model abstracts aspects of interest to the modeler. The qualifier symbolic means that a model represents a system in terms of the symbols and language of another discipline. For example, engineering systems may be (and are) modeled with the symbols of mathematics and/or computer sciences.

Mathematical Models

Mathematical modeling, or idealization, is a process by which an engineer or scientist passes from the actual physical system under study, to a mathematical model of the system, where the term model is understood in the sense of. The process is called idealization because the mathematical model is necessarily an abstraction of the physical reality — notes the phrase aspects of behavior. The analytical or numerical results produced by the mathematical model are physically re-interpreted only for those aspects.

Element Nodes

Each element possesses a set of distinguishing points called nodal points or nodes for short. Nodes serve a dual purpose: definition of element geometry, and home for degrees of freedom. When a distinction is necessary we call the former geometric
nodes and the latter connection nodes. For most elements studied here, geometric and connector nodes coalesce.

Nodes are usually located at the corners or end points of elements. In the so-called refined or higher-order elements nodes are also placed on sides or faces, as well as possibly the interior of the element.

**FIG. 4.21 - SHELL99 LINEAR LAYERED STRUCTURAL SHELL**

4.5 - Element Description

SHELL99 may be used for layered applications of a structural shell model. SHELL99 allows up to 250 layers. If more than 250 layers are required, a user-input constitutive matrix is available. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The input summary and the capabilities of the element SHELL99 shown in Fig – 4.21 is provided below:

**Table – 9 – Input summary for modal testing**

<table>
<thead>
<tr>
<th>Element Name</th>
<th>SHELL 99</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>I, J, K, L, M, N, O, P</td>
</tr>
<tr>
<td>Degrees of Freedom</td>
<td>UX, UY, UZ, ROTX, ROTY, ROTZ</td>
</tr>
<tr>
<td>Real Constants</td>
<td>If KEYOPT(2) = 0, supply the following 12+(3*NL) constants: NL, LSYM, LP1, LP2, EFS, ADMSUA, (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), MAT, THETA, TK for layer 1, MAT, THETA, TK for layer 2, etc. up to layer NL</td>
</tr>
<tr>
<td></td>
<td>If KEYOPT(2) = 1, Supply the following 12+(6*NL) constants: NL, LSYM, LP1, LP2, EFS, ADMSUA, (Blank), (Blank), (Blank), (Blank), (Blank), (Blank), MAT, THETA, TK(I), TK(J), TK(K), TK(L) for layer 1, etc. up to layer NL</td>
</tr>
<tr>
<td></td>
<td>If KEYOPT(2) = 2, supply the following 79 constants: A(21), B(21), D(21), MT(6), BT(6), AVDENS, THICK, EFS, ADMSUA</td>
</tr>
<tr>
<td></td>
<td>If KEYOPT(2) = 3, supply the following 127 constants: A(21), B(21), D(21), E(21), F(21), MT(6), BT(6), QT(6), AVDENS, THICK, EFS,</td>
</tr>
</tbody>
</table>
ADMSUA

If KEYOPT(2) = 0 or 1, supply the following 13*NM properties where NM is the number of materials (maximum is NL):
EX, EY, EZ, ALPX, ALPY, ALPZ, (PRXY, PRYZ, PRXZ or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, for each of the NM materials.

If KEYOPT(2) = 2 or 3, supply none of the above.

Supply DAMP and REFT only once for the element (use MAT command to assign material property set). See the discussion in Section 4.99.1 for more details.

Pressures:
face 1 (I-J-K-L) (bottom, in +Z direction),
face 2 (I-J-K-L) (top, in -Z direction),
face 3 (J-I), face 4 (K-J), face 5 (L-K), face 6 (I-L)

Temperatures:
T1, T2, T3, T4, T5, T6, T7, T8 if KEYOPT(2) = 0 or 1, or none if KEYOPT(2) = 2 or 3

Plasticity, Creep, Swelling, Stress stiffening, Large deflection, Large strain, Birth and death, Adaptive descent.

KEYOPT(2) 0 - Constant thickness layer input (250 layers maximum)

4.6 - ANSYS

ANSYS FEA software provides tools tailored to meet the demands of diverse users in nearly every Engineering industry. While delivering effective inter-operability with other ANSYS products, ANSYS provides a range of solutions which enables companies to see how their designs will react to the physical demands of the real world, thus allowing them to make crucial changes prior to production. The entire range of ANSYS products includes from disciplines like Mechanical, Automotives, Aerospace, Machine Tools, etc. ANSYS is a general purpose finite element modeling package for numerically solving wide variety of mechanical problems. The analysis capabilities of ANSYS include the ability to solve:

1. Static and Dynamic Structural Analysis
2. Steady State and Transient Heat Transfer problems
3. Modal frequency and Buckling Eigen value problems
4. Static or time varying magnetic analysis
5. Various types of field and coupled-field applications

The program contains many special features which allow non-linearities or secondary effects to be included in the solutions such as plasticity, large strain, hyper elasticity, creep, swelling, large deflections, stress-stiffening, temperature dependency, material anisotropy and radiation. As ANSYS has been developed, other special capabilities
such as sub-structuring, sub-modeling, random vibration, kineto-statics, kineto-dynamics, free convection fluid analysis, acoustics, magnetic, piezoelectric, coupled field analysis and design optimization have been added to the program.

Program Review:
The ANSYS element library contains more than 136 various types of elements for static, dynamics, and heat transfer and depending on the application. This variety of elements allows the ANSYS program to analyze two and three dimensional frame structure, piping systems, two-dimensional plane and axi-symmetrical solids, three dimensional solids, flat plates, and non-linear problems which also includes contact problems. The input data for an ANSYS analysis are prepared using a pre-processor. The pre-processor (PREP7) contains powerful solid modeling and mesh generations capabilities and is also used to define all analysis data (geometric properties, natural properties, constraints, loads etc). With the benefit of data base definition and manipulation of analysis data, parametric input, user files, macros, and extensive online documentation are also available, providing more tools and flexibility for the analyst to define the problem. Extensive graphic capability is also available throughout the ANSYS program, including isometric, perspective section edge, and hidden line displays of three dimensional structures, X-Y graphics of input quantities and results and contains of solution results.

Stages Involved:
1. **Pre-processing** – It is the initial part of analysis. It involves preparation of input data like the geometry information (length of the domain, boundary conditions etc), the data of the problem (co-efficient in the differential equations, source terms etc), FEM mesh generation (number of elements, element length, connectivity, etc). The major steps in preprocessing are given below:
   - Assembling the model
   - Defining the constraints or fixity conditions
   - Define element type and material/geometric properties
   - Mesh lines/ area/ volumes are required
2. **Processor / Solver** – Assigning Loads, Constraints, and solving the required analysis. Here we specify the loads (point or pressure), constraints (translational and rotational) and finally solve the resulting the set of equations.
3. **Post-processor** – Further viewing the results are carried out in this phase. Here the list of nodal displacements, element forces, stresses & strains can be done. In addition, stress contours can also be plotted.

![Diagram of Node indicating Degrees of Freedom](image)

**Fig – 4.22 - Details of Node indicating Degrees of Freedom**

### 4.7 - THE ANALYSIS PROCEDURES

The objective of finite element analysis is to accurately represent the behavior of the physical structure being analyzed. The degree of success in achieving this objective depends largely on the modeling techniques and assumptions employed in the analysis. The general purpose finite element analysis program ANSYS, was used for the analysis. This program offers a family of layered elements to analyze structural models. The element formulation is based on the standard iso-parametric element similar to that given by Ahmad et al. The element used denoted by Shell99, in the analysis has four corner nodes and four middle side nodes with six degrees freedom at each node, viz., translations (u, v, w) in nodal x, y and z directions and rotations (θx, θy, θz) about the x, y and z axes. These are connected by quadric shape functions which describe both the Original shape and displacements of the element. A 3-D quadratic failure criterion Tsai-Wu which includes interaction among the stress or strain components, used with Shell99 element, has been applied as the failure criterion to predict the ultimate failure strength of the laminated composite plates under static and dynamic distributed loads.

### 4.8 - SHELL99 Assumptions and Restrictions

- Zero area elements are not allowed. This occurs most often whenever the elements are not numbered properly.
- Zero thickness layers are allowed only if a zero thickness is defined at all corners. Tapering down to zero is not allowed.
- If KEYOPT (11) = 0, all nodes are assumed to be at the midthickness of the element.
All inertial effects are assumed to be in the nodal plane, i.e., unbalanced laminate construction and offsets have no effect on the mass properties of the element.

No slippage is assumed between the element layers. Shear deflections are included in the element, however, normal to the center plane before deformation are assumed to remain straight after deformation.

This element may produce inaccurate results under thermal loads for non-flat domains.

The applied transverse thermal gradient is assumed to be linear through the element and over the element surface.

The stress varies linearly through the thickness of each layer.

Interlaminar transverse shear stresses are based on the assumption that no shear is carried at the top and bottom surfaces of an element. Further, these interlaminar shear stresses are only computed at the centroid and are not valid along the element boundaries. If accurate edge interlaminar shear stresses are required, shell-to-solid sub modeling should be used.

The element matrices are reformed every iteration unless option 1 of the \texttt{KUSE} command is active. Only the lumped mass matrix is available. The mass matrix is assumed to act at the nodal plane.

The large deflection option for SHELL99 is not as convergent as it is for SHELL91 (the nonlinear layered shell element). SHELL91 may be the preferred element type when constructing models that include large deflection.

**SHELL99 - Linear Layered Structural Shell Element**

- SHELL99 is an 8-node, 3-D shell element with six degrees of freedom at each node. It is designed to model thin to moderately thick plate and shell structures with a side-to-thickness ratio of roughly 10 or greater. For structures with smaller ratios, we may consider using SOLID46. The SHELL99 element allows a total of 250 uniform-thickness layers. Alternately, the element allows 125 layers with thicknesses that may vary bi-linearly over the area of the layer. If more than 250 layers are required, we can input our own material matrix. The element also allows failure criterion calculations. It also has an option to offset the nodes to the top or bottom surface.

**SHELL91- Nonlinear Layered Structural Shell Element**

- SHELL91 is similar to SHELL99 except that it allows only up to 100 layers and does not allow us to input a material property matrix.

  However, SHELL91 supports plasticity, large-strain behavior and a special
sandwich option, whereas SHELL99 does not. SHELL91 is also more robust for large deflection behavior.

The procedure for a modal analysis consists of four main steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Expand the modes.
4. Review the results.
4.9 - EXPERIMENTAL SET-UP

The following equipment shown in fig.4.24 below was used to perform an impact test:

1. Impact Hammer with a load cell attached to its head to measure the input force.
2. An Accelerometer to measure the response acceleration at a fixed point and direction.
3. A 4 Channel FFT Analyzer to compute FRF’s.
4. Post-processing Modal software [LMS Test Lab developed LMS International, Belgium & ME SCOPE developed by VIBRANT TECHNOLOGIES USA] for identifying modal parameters and displaying the mode shapes in animation.

![Fig – 4.24 – FFT Analyzer Set-up](image-url)
Investigations on Dynamic Properties of Polymer Based Composites

<table>
<thead>
<tr>
<th>➢ Accelerometer specifications</th>
<th>➢ Hammer specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Sensitivity: 10.02 mV/gm</td>
<td>• Sensitivity: 2.33 mV/N</td>
</tr>
<tr>
<td>• Sealing: epoxy</td>
<td>• Resonant freq: &gt; 20 kHz</td>
</tr>
<tr>
<td>• Sensing element: Quartz</td>
<td>• Sensing element: quartz</td>
</tr>
<tr>
<td></td>
<td>• Sealing: epoxy</td>
</tr>
</tbody>
</table>

Fig – 4.25 – FFT Analyzer with Impact Hammer

Fig – 4.26 – Impact Hammer used

Fig- 4.27— Experimental Modal Test Set-up with FFT Analyzer

Fig- 4.28- Impact hammer being applied on nodes
4.10 - Specimen Preparation

On the whole 30 composite specimen models were fabricated and tested. The fibers chosen were bi-woven E-Glass, S-Glass, Carbon, Graphite & Kevlar fiber. The laminate was cut to the required size and bonded to the graphite fiber cloth by using an adhesive made from a mixture of LY556 resin [LY556 resin is a bifunctional epoxy resin i.e., diglycidyl ether of bisphenol-A (DGEBA) supplied by Ciba Co]. Epoxy is chosen primarily because it happens to be the most commonly used polymer and because of its insulating nature (low value of thermal conductivity, about 0.363 W/m-K) & HY 951 hardener [Araldite HY 951 (triethylene tetramine – TETA) from Ciba Company] which is a liquid of low viscosity of an aliphatic amine basis, in proportions of 100:10 by weight. The surfaces were thoroughly cleaned in order to ensure that they were free from oil, dirt, etc., before bonding at room temperature and pressure. The models were allowed to cure for about 24 hours. Thicknesses of the Specimens were maintained at 2 mm & 4 mm throughout the experiments for both the Grades of fibers. Details of the composite models fabricated are as shown in Fig 4.29 below.

![Vacuum Bag Technique](image_url)

**Fig- 4.29-** – Vacuum Bag Technique

The following equipment shown in figure below was use to perform an impact test,

a) An **impact hammer** with a load cell attached to its head to measure the input force.

b) An **accelerometer** to measure the response acceleration at a fixed point & direction.

c) A 4 channel **FFT analyzer** to compute FRFs.

d) **Post-processing modal software** for identifying modal parameters and displaying the mode shapes in animation.
For FRF, at each singular point the modal hammer was struck three times and the average value of the response was displayed on the screen of the display unit. At the time of striking with modal hammer to the point son the specimen, precaution were taken for making the stroke to be perpendicular to the surface of the plates. Then by moving the cursor to the peaks of the FRF the frequencies are measured.

Typical properties of the hardener HY 951 are shown in Table

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Molecular Weight</td>
<td>146.24</td>
<td></td>
</tr>
<tr>
<td>Viscosity (Hoepller) at 25°C</td>
<td>450</td>
<td>Cp</td>
</tr>
<tr>
<td>Specific gravity at 25°C</td>
<td>0.973</td>
<td>g/cc</td>
</tr>
<tr>
<td>Flash point</td>
<td>129</td>
<td>°C</td>
</tr>
<tr>
<td>Vapor pressure at 20 °C</td>
<td>&lt; 0.01</td>
<td>mmHg</td>
</tr>
<tr>
<td>Color</td>
<td>Clear, pale yellow or yellow liquid</td>
<td></td>
</tr>
<tr>
<td>Boiling point</td>
<td>284-287</td>
<td>°C</td>
</tr>
</tbody>
</table>

(HY 951, Material Safety Data Sheet, Ciba co., version. 12, 2/June/2005.)

4.11 - LMS TEST LAB

The post processing software used for analysis was LMS Test Lab. LMS Test Lab is a complete, integrated solution for tested-based engineering, combining high speed multiple-channel data acquisition with a suite of integrated testing, analysis and report-generation tools. LMS Test is a designed to make testing more efficient and more convenient for each and every user.

It is the idea for testing departments that need to be future-focused: offering the right balance between ease of use and functional flexibility. LMS Test. Lab significant increases a test facility’s productivity, delivering more reliable results, even when the availability of prototypes is dramatically reduced.
Investigations on Dynamic Properties of Polymer Based Composites

Features

- Dedicated workflow-based user interface for impact parameter settings and measurements.
- Expert and operator mode with password protection.
- Real-time and parallel measurement of modal test functions (windowed / unwindowed time data, APS, FRF, coherence).
- Trigger setting wizard for hammer hit.
- Versatile graphical displays for measurement control.
- Configurable test template & report.

Benefits

- Fast and reliable execution of impact test.
- Reduce time for the measurement and learning curve.
- Immediate validation of modal test quality.
- Immediate feedback on measurement status.
- Customize without programming.

4.12 Pump Selection for Vacuum Bag Process

The size and shape of the mould and type and quantity of the material being laminated will determine the minimum pump requirements. If we are laminating flat panels consisting of a few layers of glass, flat veneers or a core material, 5” or 6” Hg (2.5–3 psi) vacuum pressure will provide enough clamping pressure for a good bond between all of the layers. If the area of the panel is limited to a few square feet, a 1 or 2 CFM pump will be adequate to maintain that clamping pressure. As the panel area increases, the CFM requirement increases proportionately. Generally, the best pump for a specific vacuum bagging operation will have the largest air moving capacity for the vacuum/clamping pressure required while operating at a reasonable horsepower.
Investigations on Dynamic Properties of Polymer Based Composites

**Fig. 4.30** - Nodes marked on Specimens

**Figure 4.31** - Specimen being impacted

**Fig. 4.32** - Accelerometer Location

**Fig. 4.33** - Data-Physics FFT Analyzer
Investigations on Dynamic Properties of Polymer Based Composites

Fig - 4.34 - EXPERIMENT BEING CONDUCTED ON GRAPHITE SPECIMEN

Fig - 4.35 - Accelerometer used