Finite Element Analysis of Composite Laminate by ANSYS Software

Bathala Surendra 1, Vinod D. 2*
1,2 Department of Mechanical Engineering, Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, Tamil Nadu, India
Email: *vinodhd.sse@saveetha.com

Abstract. The Glass Fiber Reinforced Polymer (GFRP) is used for several industrial purposes where it is required to exhibit good Flexural behavior. The flexural strength of composite is very essential property to make it as a good structural component. In the present work Glass Fiber/Epoxy resin composite laminate of six layer is modelled with the help of catia software. The Finite Element Analysis for Flexural Strength was also studied with the help of FEA software ANSYS to validate the tensile, compression and impact results with that of FEA software.

Key words: Epoxy, E-glass fiber, composite, Puncture resistance, impact strength, stiffness, impact velocity, laminates, reinforcement, and deformation.

1. Introduction
Fibre-reinforced composite structures allow the design improvements, materials and processes sufficient space and receptivity. Their balance of force with weight, rigidity, performance, and availability of simple raw materials make them the obvious choice for a vast array of commercial and industrial applications like electronics, furniture, the electrical, oil, etc [1]. The major areas of institutional implementation are aircraft, space, automotive and sports products, naval and infrastructure [2].

Flexural strength is very important whenever the composite material is subjected to the compression loading. There are many methods which can be used to measure flexural strength, but this also comes with great deal of confusion as which method will be best to use [3]. The test should be simple giving the pure flexural properties which require no special devices and can be used on presently available machines [4]. The three-point test method has been widely used to characterize the flexural strength of the fiber reinforced composites. In this method a beam of certain dimension is placed under simply Conditions assisted shall be added at the sample centre, and line load shall be applied [5]. Their fault is not nur due to undue tension but is often caused by buckling of many industrial applications like columns, beams or plates.

2. Related Work
The case study considers only rectangle thin plates [6]. When a flat plate has low pressure loads on the plane, it stays flat and is balanced. But the stability structure of a plate transitions to a non-flat structure, with the volume of the in-plane compressive load increases [7]. A composite material consists of two or more components which provide considerable weight savings in systems because of the high weight to density and excellent stiffness to weight ratio. The compression strain in which the
plate becomes brittle is considered a "critical buckling strain." In addition, the mechanical characteristics of a fibrous fiber can vary by orientation of fibers as needed [8]. The fabrics are the main load-bearing components of these materials, and the high modulus matrix and the endothelial cell of the fiber matrix provides the requisite durability and protects themselves from the atmosphere [9]. New programs and polymers are being produced more rapidly, because of the need for materials with uncommon combinations of properties which traditional monolithic products could not comply with [10]. Indeed, because of its diverse existence, composite materials will in all ways follow these criteria. Composite products are formed according to their component components, their delivery and their relationship, and an uncommon mix of physical properties can therefore be achieved [11].

Due to its exceptional rigidity and specific strength, laminated composites are more commonly used in mechanical and aerospace applications. The beams strengthened by fibers are commonly used in form of comparatively fine plate and, thus, researchers have been heavily considering the load-bearing capacity of the composite plate against humping under different loading and border conditions [12]. Polymers attract greater interest from architects, scientists and architects due to their outstanding stiffness and weight properties [13]. The concrete laminate boards are normally exposed to compressive loading during operation, which can trigger buckling when overwhelmed [14]. Their hunching action is therefore essential in ensuring that these systems are stable and reliable. In order to fix the issue of humpback of laminated composite sheets, experiments became central in the challenging conceptual and statistical method for laminated structural comparability [15]. This thesis concerns the analysis of loading symmetrical and through the plate in the clamped, non-clamped, free boundary state. The findings on buckling loads are evaluated by cutting distance, longitudinal / thickness ratio, ply direction and width / length ratio [16].

3. Proposed Methodology

3.1. CATIA

CATIA is one CAD / CAM / CAE kit among the world's leading. It not only blends 3D parametric characteristics with 2D tools as a powerful platform for modelling, but tackles all architecture by manufacturing.

- CATIA - Computer Aided Dimensional Interactive Application.
- CATIA is a recently designed family of CAD / CAM / CAE software solutions, which is produced by Desalt Systems, France.
- With numerous desks, CATIA executes the simple design task; some working benches in this kit are accessible.
- Surface design and wireframe workbench
- Architecture of generative workbench type.
- DMU cinematics
- Development
- Construction of mould

3.2. ANSYS Evaluation

ANSYS has been established by ANSYS Inc. – the US for its full FEA numerical simulation kit. In nearly any scientific field is used by technicians globally.

Structural
Thermal facilities
Fluids
Electromagnetic Medium & High Frequency.

3.3. Procedure

It consists of 3 main steps:
Pre-processor
Solver
Post processor

3.4. Structural Analysis
The far more popular use of the maximum balance methodology is actually structural analysis. The term "structural" encompasses not just civil structures like roads or structures, but also marine, aviation, Mechanical structures including ship hulls, plane boards and machinery, and metal equipment, such as gears, machine equipment and materials.

3.5. Elements Used In Structural Analyses
Most ANSYS component forms vary from basic spars and radii to complicated structured shells and widespread straining solids. Any of these components may be used for building structures. Table 1 shows the structural element research analysis.

**Table 1: Structural Research Elements**

| Category     | Element Name(s)                          |
|--------------|------------------------------------------|
| Spars        | LINK1, LINK8, LINK10, LINK180           |
| Beams        | BEAM3, BEAM4, BEAM23, BEAM24, BEAM44, BEAM54, BEAM188, BEAM189 |
| Pipes        | PIPE16, PIPE17, PIPE18, PIPE20, PIPE59, PIPE60 |
| 2-D Solids   | PLANE2, PLANE25, PLANE42, HYPER56, HYPER74, PLANE82, PLANE83, HYPER84, VISCO88, VISCO106, VISCO108, PLANE145, PLANE146, PLANE182, PLANE183 |
| 3-D Solids   | SOLID45, SOLID46, HYPER58, SOLID64, SOLID65, HYPER86, VISCO89, SOLID92, SOLID95, VISCO107, SOLID147, SOLID148, HYPER158, SOLID185, SOLID186, SOLID187, SOLID191 |
| Shells       | SHELL28, SHELL41, SHELL43, SHELL51, SHELL61, SHELL63, SHELL91, SHELL93, SHELL99, SHELL150, SHELL181 |
| Interface    | INTER192, INTER193, INTER194, INTER195 |
| Contact      | CONTAC12, CONTAC52, TARGE169, TARGE170, CONTAT171, CONTAT172, CONTAT173, CONTAT174, CONTAT175 |
| Coupled-Field| SOLIDS, PLANE13, FLUID29, FLUID30, FLUID38, SOLID62, FLUID79, FLUID80, FLUID81, SOLID98, FLUID129, INFINI10, INFINI11, FLUID116, FLUID130 |
| Specialty    | COMBIN7, LINK11, COMBIN14, MASS21, MATRIX27, COMBIN37, COMBIN39, COMBIN40, MATRIX50, SURF153, SURF154 |
| Explicit Dynamics | LINK160, BEAM161, PLANE162, SHELL163, SOLID164, COMBI165, MASS166, LINK167, SOLID168 |

3.6. Material Model Interface
If we use the GUI, the material we simulate must be specified using an interactive design with the material model. This application contains a hierarchy of content category tree structures to help us select the suitable model to evaluate it.

3.7. Load Types
In a static analysis, all the following load types apply. ROTX, ROTY, ROTZ and UX (Reds) DISPLACEMENTS. The load types are shown in Table 2.
Table 2: Load Types

| Displacement (UX, UY, UZ, ROTX, ROTY, ROTZ) | Constraints DOF Constraints in the ANSYS Basic Analysis Guide |
|---------------------------------------------|---------------------------------------------------------------|
| Force, Moment (FX, FY, FZ, MX, MY, MZ)     | Forces (concentrated loads) in the ANSYS Basic Analysis Guide |
| Pressures (PRES)                            | Surface loads in the ANSYS Basic Analysis Guide               |
| Temperature (TEMP), Fluence (FLUE)          | Body loads Body Loads in the ANSYS Basic Analysis Guide       |
| Gravity, Spinning, and so on               | Inertia loads Inertia Loads in the ANSYS Basic Analysis Guide |

3.8 Modeling Composites
Composites are somewhat simpler to model than an isothermal substance like iron and steel. The characteristics and orientations of the various layers must be described with careful attention, since each layer has various orthotropic material characteristics. The following facets of constructing a composite model will be focused in this section:

- Pick the proper type of element
- Describes the layered setup
- Failure requirements listed
- Simulation and post-processing standards being met

3.9 Element Type Used
3.9.1. SOLID45 Description
SOLID45 is being used for 3D modelling of concrete slabs as shown in Figure 1. And this part is described by eight nodes with 3 degrees of freedom for each module: x, y & z module-direction transformations. Plasticity, slowness, swell, tension rigging, large deflection, & high stress are part of the portion.

4. Analysis Results
Results are shown in Figure 2 to Figure 7. Results of mono compressive stress and multi compressive stress are shown in Figure 2 and Figure 5. Its corresponding impacts are shown in Figure 3 and Figure 6. The tensile stress is shown in Figure 4 and Figure 7.
Figure 2: Monocompressive Stress

Figure 3: Impact
Figure 4: Tensile

Figure 5: Multi compressive Stress
5. Conclusion
The numerical analysis of the composites is significant in reliability and safety of a number of mechanical components made of composites. The tensile, compression and impact analysis was done to find the flexural strength of the glass fibre reinforced laminate with the help of finite element software ANSYS. The three-dimensional Finite element simulation was done to find Flexural strength.
with obtained numerical results with the FEA results. From the results of this study, the composite laminate was sin-researched for tensile, compression and impact strength.

References
[1] Zeng, L. (2021, March). Static finite element analysis of three-closed box thin-wall beam based on Pseudo-elastic SMA hybrid composite material ANSYS. In Journal of Physics: Conference Series (Vol. 1802, No. 2, p. 022095). IOP Publishing.
[2] Amena, Bekan, and BalachandraPattanaik. "High Voltage Gain using Cockcroft Walton Voltage Multiplier (CWVM).” International Journal of MC Square Scientific Research 11.3 (2019): 23-32.
[3] http://www.nhtsa.dot.gov/cars/testing/procedures/TP-581-01.pdf.
[4] Lei, Junfeng, et al. "An Image Rain Removal algorithm based on the depth of field and sparse coding.” 2018 24th International Conference on Pattern Recognition (ICPR). IEEE, 2018.
[5] Shareefa Ahmad Abu Shahada, Suzan Mohammed Hreiji, Saleh Ibrahim Atudu, SherminShamsudheen“Multilayer Neural Network Based Fall Alert System” International Journal of MC Square Scientific Research Vol 11 No 4 2019.
[6] Zonghua Zhang, Shutian Liu, Zhiliang Tang, “Design REFERENCES
[7] Fahad, Autho Abdul Aziz. "Design and implementation of blood bank system using web services in cloud environment." International Journal of MC Square Scientific Research 11.3 (2019): 09-16.
[8] Pandey, Amit, and Gyan Prakash. "Deduplication with Attribute Based Encryption in E-Health Care Systems." International Journal of MC Square Scientific Research 11.4 (2019): 16-24.
[9] Mohamed, K. B., Lijo, L. A., Ramya, R., Lakshmi, P. S., Harini, R., & Ramasamy, N. (2021). Structural and Thermal Analysis of Composite Wind Turbine Blade for Wind Mill Applications. Advances in Design and Thermal Systems: Select Proceedings of ETDMMT 2020, 1.
[10] Imran, M., Khan, R., & Badshah, S. (2021). Experimental, analytical, and finite element vibration analyses of delaminated composite plates. Scientia Iranica. Transaction B, Mechanical Engineering, 28(1), 231-240.
[11] Demirtaş, E., H. Özkan, and M. Nofer. "Extrusion Foaming of High Impact Polystyrene: Effects of Processing Parameters and Materials Composition." Int J Mater Sci Res 1, no. 1 (2018): 9-15.
[12] Thomas, Tina Rachel, Prashant Kumar Singh, and Syed Aulia. "Acceptance and Use of Social Media Banking in Sultanate of Oman."
[13] Jansri, Wilawan. "Incorporating Customers Perceived Value of Luxury Natural Consumption."
[14] Kim, Young Jun. "Application of Materials and Cost Model Decision System using Case-Based Reasoning."
[15] Olagunju, Mukaila, K. M. Owolabi, and A. O. Afolayan. "Determination of a Price Index for Escalation of Building Material Cost in Nigeria."
[16] Singh, V., & Rastogi, V. (2021). Design and Static Analysis of Mono Composite Leaf Spring Made of Various Types of Composite Materials Using Finite Element Method. In IOP Conference Series: Materials Science and Engineering (Vol. 1033, No. 1, p. 012041). IOP Publishing.