International Journal of Enhanced Research in Management & Computer Applications
ISSN: 2319-7471, Vol. 13, Issue 4, April-2024, Impact Factor: 8.285
Presented at "ICRETETM-2024", Organized by GSMCOE, Pune, on 22nd - 23rd April 2024

Predictive Analysis and Optimization of Composite Marine T-Joints: A Finite Element Approach

Prof. Sachin Yadav[1], Mr.Vijay kumar Bondar[2], Mr.Vedant Sitafale[3], Ms. Siddhi Ugale [4], Ms. Rutuja Nalawade [5], Mr. Sushant Sonawale [6]

Department of Mechanical Engineering, GSMCOE Pune, India

ABSTRACT

Composite materials are usually Fibre Reinforced Plastic (FRP), these are mostly used in marine application due to their properties. It has properties of lighter weight, corrosion resistance and design flexibility. T-joint has important role in structural integrity of marine vessels, to result the accurate prediction of their damage critically. The aim of this project is to predict the damage criticality of composite marine T- joint using Finite Element Analysis. By meshing and simulation of boundary conditions we can analyze the stress distribution and deformation by applying various loads .To study deflection and stresses, different materials for skin, core and filler are considered. The project not only describe the T- joint but also can be refer to improve the reliability and performance in marine engineering.

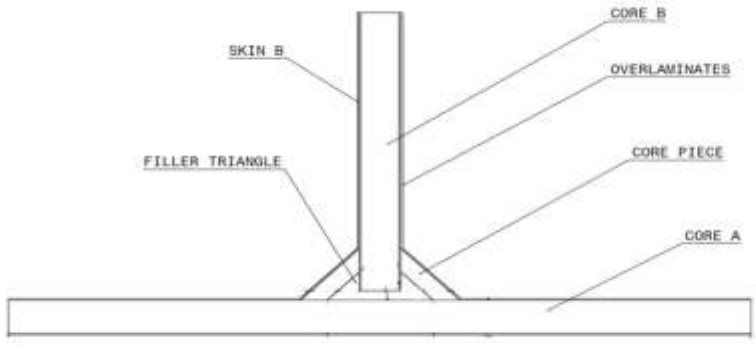
Key Words: T- Joint, Composite Materials, Testing, Loads, Analysis.

INTRODUCTION

Composite materials, such as FRP, offer distinct advantages in marine and military applications, including lighter weight, corrosion resistance, and radar stealth capabilities. This project focuses on predicting the damage criticality of composite marine T-joints to ensure structural reliability. Finite Element Analysis (FEA) is utilized to accurately simulate stress distribution and investigate critical stresses, albeit requiring specialized knowledge and precise modelling. The study emphasizes optimizing T joint design by analyzing various materials for skin, core, and filler.

Through comprehensive FEA simulations, the project aims to enhance the structural integrity and performance of composite marine structures, contributing to cost savings and improved reliability. The complex geometry of T-joints necessitates careful consideration of meshing strategies and boundary conditions to yield accurate results. Ultimately, this research addresses a critical need for predictive methodologies in composite marine engineering, facilitating safer and more efficient maritime operations.

For a hull structure with a number of compartments, a typical joint, known as a T-Joint is used to join the hull and bulkhead sections. It consists of composite over laminates over a shaped fillet constructed by stacking up layers of laminates through hand-lay-up process. Filler made from chopped fiber reinforced resin is used to form the fillet. The function of the T-Joint is to transfer flexural, tensile and shear loads between the hull and bulkhead and to maintain watertight integrity between compartments separated by the bulkhead . There are two types of T-Joints depending on the over laminates shape. They are triangular and circular T-Joints. Unlike triangular T-Joints, much research has been done for circular T-Joints. The over laminates and the resin filler in the fillet are the load transmission path between the hull and bulkhead. Hence, the strength of the joints depends on the strength of both parts stated that there were two load conditions experienced by T-Joints. The first condition was the compression at the interface between hull and the over laminates due to hull pressure. The second one was the tension at the same interface due to heavy machinery's weight. In addition, over laminate can be the main source of delaminating due to the variable quality of the interfaces and the presence of defects. The hull and bulkhead are the primary structures in maintaining the ship stiffness under various loadings.



LITERATURE REVIEW

1. T-joints in naval ships transfer flexural, tensile, and shear loads between the hull and bulkhead. Research has examined t-joints under various loading conditions and conducted tests on composite t-joints. FEA using software like Ansys has been employed, with results validated against experiments.

2. Composite materials in t-joints for naval ships have been studied, focusing on tensile load effects and different reinforcing methods. Numerical models using FEA were developed to analyze material and joining parameters' influence on t-joint strength.

3. Aluminium sandwich construction of t-joints for lightweight transportation systems has been explored. The study investigated strength characteristics through theoretical and experimental analysis, examining bending deformation, buckling, and crushing strength.

4. Theoretical and experimental effects near intersections of sandwich panel cores were studied, focusing on bending stresses induced in faces. The accuracy of theoretical analyses was verified experimentally for sandwich beams in bending.

5. Adhesively bonded t-joints have been studied, considering the advantages over mechanical joints. Ultimate strength calculations and experimental results on new designs were discussed, emphasizing the impact of fillet geometry and core material.

6. Structural integrity and damage tolerance of composite t-joints in ships were investigated using FEA results highlighted the effects of geometry and dis-bonds on strain distribution, with implications for structural performance.

7. Structural adhesives as an alternative joining method for load-bearing structures were discussed, focusing on epoxy adhesive properties, limitations, and experimental techniques for steel grillage panels.

8. A multifunctional panel concept incorporating shape memory alloy face sheet elements for reversible shape change was presented. Performance aspects such as thermal power, actuation frequency, and load bearing capacity were analyzed.

9. The structural performance of hybrid composite-to-metal bolted joints under flexure loading was studied, considering the effect of bolt type, doublers plate geometry, and foam inserts.

10. The potential of corrugated skin in sandwich panels to carry shear loads was explored, emphasizing improved shear carrying capability and weight savings. Finite element analysis and experimental results were discussed.

11. Parametric variation and optimization using genetic algorithms for structural steel/composite connections in marine applications were proposed. Genetic algorithms were shown to efficiently search the design space for optimal joint configurations.

12. Applications of fiber-reinforced polymer composites to naval ships and submarines were discussed, highlighting benefits such as cost reduction and improved structural and operational performance.

Solid Modelling of composite T Joint:

The finite element method analysis of the composite T joint. There are various solid modelling method and the process for and creating suitable and clean geometry for this analysis .

- CSG – they involves the union of primitive objects
- BREP
- FBM
- CATIA

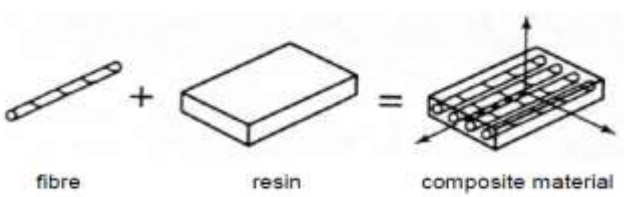
The 3D model are created using CATIA V5 and the tools and port design commands by following these process. A comprehensive solid model of composite T joint is ready for analysis

Material selection:

Composite material are made up of composition of material. Composites are hybrid materials made of a polymer resin reinforced by fibres, combining the high mechanical and physical performance of the fibres and the appearance, bonding and physical properties of polymers .Composite materials are naturally or simple occurring material .These are made from two or more constituents materials significantly different physical or chemical properties. They are consist of a polymer resin matrix reinforced by fibre. Combine various mechanical properties of the fibres.

The most common resin system is used in composite production. They are used in any different kinds of properties:

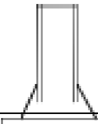
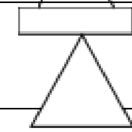
- Mechanical properties
- Micro cracking resistance
- Fatigue resistance
- Resistance to degradation inside water



Epoxy Resin is favoured for its high adhesive strength and mechanical properties. Polyster and vinylester resins are also used but is generally have lower adhesive properties compared to epoxy. Composite materials have balanced of strength weight and durability and these are used to create them ideal for use or application in aerospace, automotive, marine and construction industries.

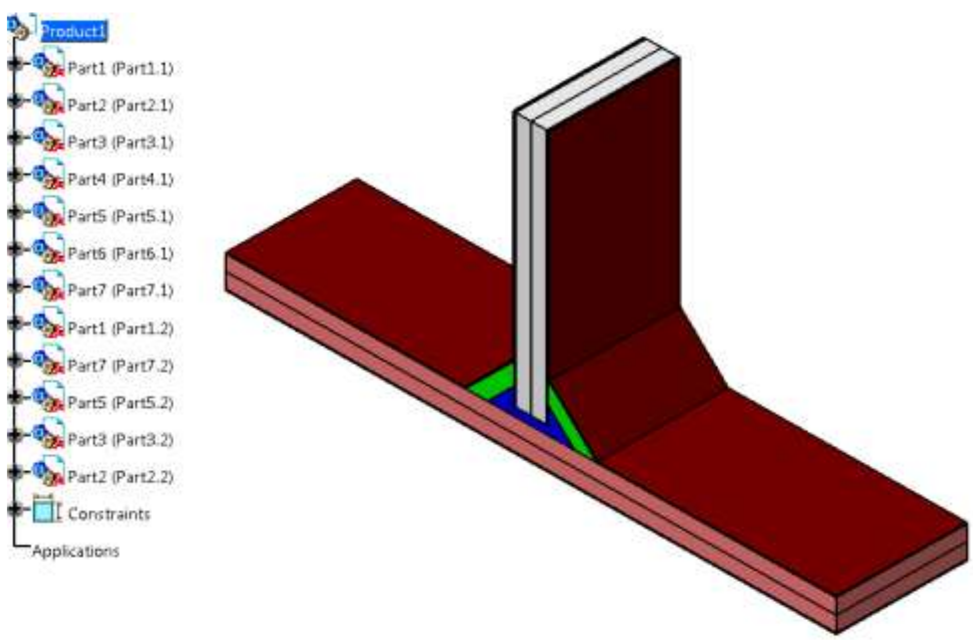
Design of Composite T joint:

The design of a composite T-joint involves careful consideration of structural requirements, material selection, weight, appearance, and manufacturability. A conceptual T-joint design typically includes core materials, fiberglass skins, filler materials, and over laminates. The goal is to create a joint that provides sufficient strength and rigidity while minimizing weight and complexity. Various factors, such as the angle of joint, loading conditions, and environmental factors, influence the design process. 3D modelling software, such as CATIA V5, is often used to visualize and analyze the proposed design. The design is iterated upon to optimize performance and meet design requirements. Special attention is paid to the selection of materials to ensure compatibility, durability, and cost-effectiveness. The final design is evaluated based on factors like structural integrity, dimensional accuracy, ease of manufacturing, and aesthetics. Prototyping and testing may be conducted to validate the design before full-scale production

Sr.No.	Part Name	Materials	Dimensions	Value (mm)	Quantity	Shape
1	T-Joint	See below	Overall width	150	-	
			Overall length	700		
			Overall height	400		
2	Triangular Fillets	PVC Foam	Base length	90	2	
			Height	40.70		
			Angle of base	60^0		
3	Skin laminates	Glass Fiber	Thickness	2	4	
4	Core A and B	PVC Foam	Thickness	43	2	
5	Filler	Epoxy Resin 520	Thickness around the fillet	3	-	Liquid

Manufacturing T Joint:

Manufacturing process of a composite T-joint involves several steps to fabricate the various components and assemble them into the final structure. Core materials, such as PVC foam sheets, are cut to size using saws or CNC machines. Surface preparation is essential to ensure proper bonding between materials, often involving sanding or chemical treatments. Hull and bulkhead plates are assembled by applying epoxy resin and fiberglass skins, which are then cured to form a strong bond. Core pieces and filler triangles are fabricated separately and inserted into the assembly to provide additional support and reinforcement. Careful attention is paid to dimensional accuracy and alignment to ensure a precise fit. Over laminates are applied to the joint area to provide additional strength and protection. The entire assembly is then subjected quality control checks to verify structure integrity and adherence to design specification. The manufacturing process may vary depending on the specific materials and techniques used, but the goal is always to produce high quality reliable T joint suitable for application.



Experimental Stress Analysis:

We are determining the static tensile load and displacement of a T-joint with the help of stress analysis. We have studied the strength and failure mode experimentally and by finite element analysis. Load Vs Displacement curves plotted with the help of servo hydraulic testing machine. After building the structure, it's good practice to verify the FEM using experimental stress analysis. We are using experimental stress analysis also FE analysis. In this case, the goal is to identify the crack an existing t-joint in order to resolve problems.

In the static tensile test of the T-joint, the sample is mounted to the load cell at the top of the T-panel. Bottom plate (hull) of T-joint has fixed on movable platform of hydraulic testing machine with the help of C-clamps as shown above. The fixture attached with vertical plate (bulkhead) was clamped with mechanical arrangement to upper jaw of machine. Ensured that the T-joint at machine's center to avoid offset loading. The base of the test rig is 750mm long heavy section steel beam.

The test sample is loaded through pairs of square steel tube. Steel road are mounted in spherical bearings (SKF GX 17 F) to allow for free rotation of the test sample at the ends. The top of the T-panel is connected to the testing machine through two 10mm thick steel plates of 320 x 50 mm (H X W). The steel plates are bolted to the test sample by six M16 x 100 bolts. The top end of the steel plates is connected to the load cell through a spherical bearing (SKF GX 45 F)

The load is recorded by certified 100KN load cell. The displacement is both recorded as the movement of piston (cross-head) on the tensile machine, and by calibrated linear variable differential transformer device. The tests are also recorded on video and still-photos with the camera in fixed position during the test, showing side view of the T-joint. A voltmeter within the picture frame shows the load cell voltage, for identifying the load level at each individual picture. The tensile test carried out on a 250KN hydraulic testing machine (KIC-2-XXXX-C), at a constant cross-head speed of 5mm/min.

We filled necessary information in computer attached to hydraulic testing machine. This machine operated and controlled by software and result is displayed on the computer screen. We kept initially preload zero and displacement also brought to zero.

- **TRIAL NO 1.**

For first trial the limiting load has kept 3 kN, but it was free for further loading. Load has gradually increased on T-joint with speed of 5 mm/min. Load Vs Displacement curve displayed on computer screen shows progress of gradual loading. At 5938.8 N load, the upper bolt has failed i.e. fixture has failed in trial no.1 but T-joint is not damaged. That means it might take more load.

- **TRIAL NO. 2:**

The failed bolt in trial no.1 is replaced by high tension bolt of class 8.8. we used special bolt for heavy duty application. The size of bolt is same as that of previous one i.e. M10 X 150. Again same conditions were applied as in case of trial no.1 and results were recorded. At 13965.0 N load, the high tension bolt is failed i.e. failure has occurred again in fixture but there were no harm to T-joint.

- **TRIAL NO. 3:**

For trial no. 3 we replaced whole fixture by new one. New fixture contained MS plates with thickness more than previous one. We increased Drilled size from 12mm to 16mm. Distance between plates as well as upper surface of vertical plate to jaw is reduced. High tension bolts of size M15 X 150 used in fixing plates as well as to clamp with upper jaw. In this case, T-joint has failed at 19815.6 N.

Value	Unit	Sample-T1	Sample-T2	Sample-T3
Failure load	KN	19.8	21.5	16.4
Failure load per unit length	KN/mm	9	9.5	14
Tensile strength	N/mm^2	1.981	1.985	2.61

The result of 3-test of T-joint on hydraulic testing machine. We are getting different result for each joint as shown in table.

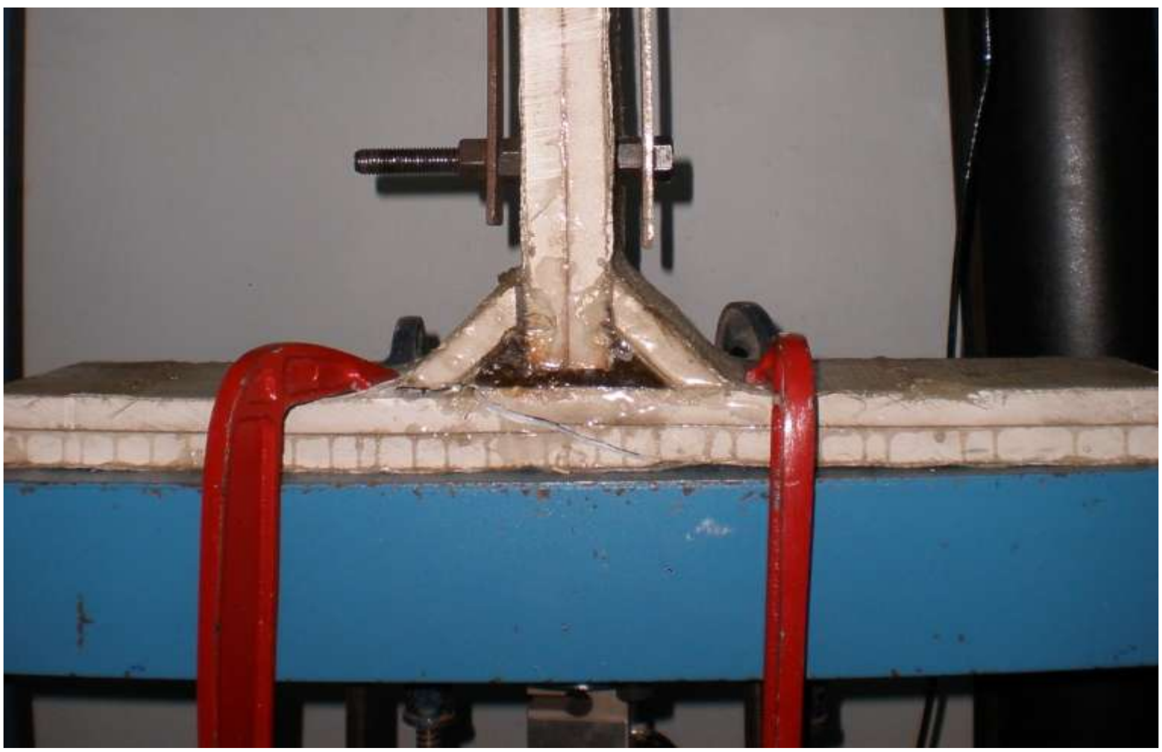


Fig. A failed T-joint with crack to the bottom plate

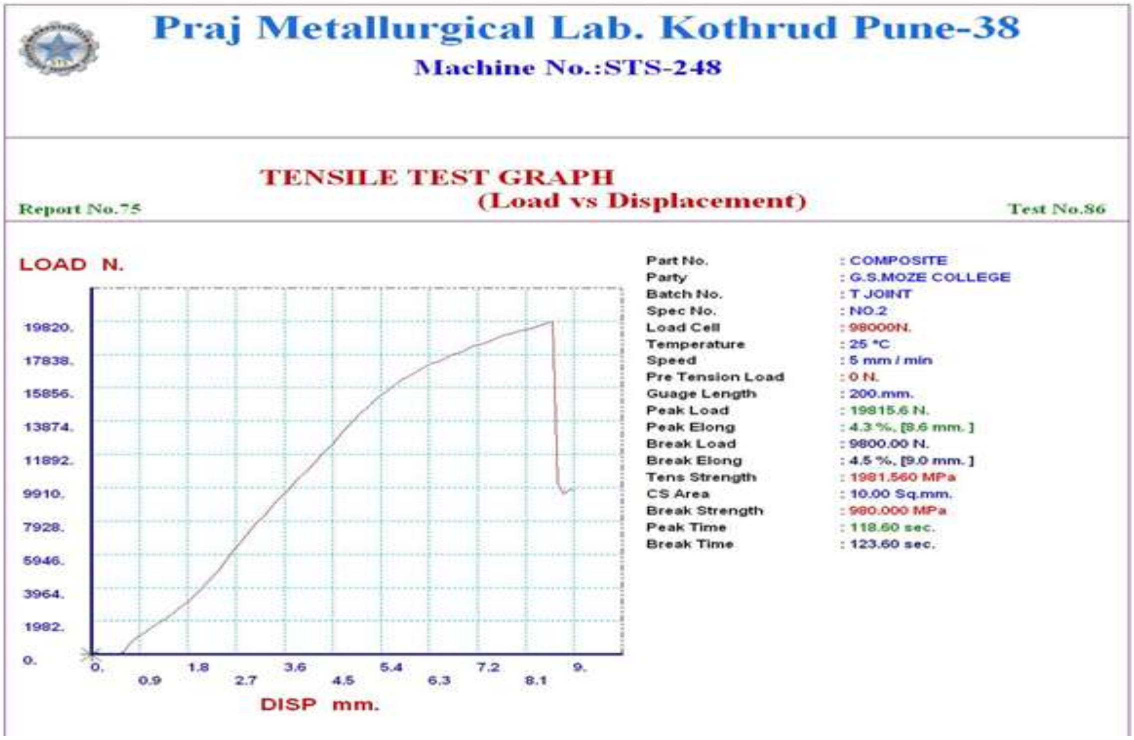


Fig. Load vs. displacement curve for T-joint with full specification.

ANALYSIS of T-joint:

The process methodology adopted in the static analysis of T-joint is described as follows. The T-joint is modeled in design software CATIA V5 which is compatible with the simulation software ANSYS 12.

The CAD model is imported into the simulation software. The first step is preparing a proper process plan for the analysis of the T-joint. This process plan involves building the CAD model, determining the boundary conditions, study the material properties and loading pattern. The material properties and orientation used for each part in various computations are given in table together with description of each part. The elastic constants E_X, E_Y, G_{xy} and V_{xy} in the local coordinate system are given together with tensile strength X and Y in the local X-direction and Y-direction respectively. The material are assumed to be linear elastic and orthotropic.

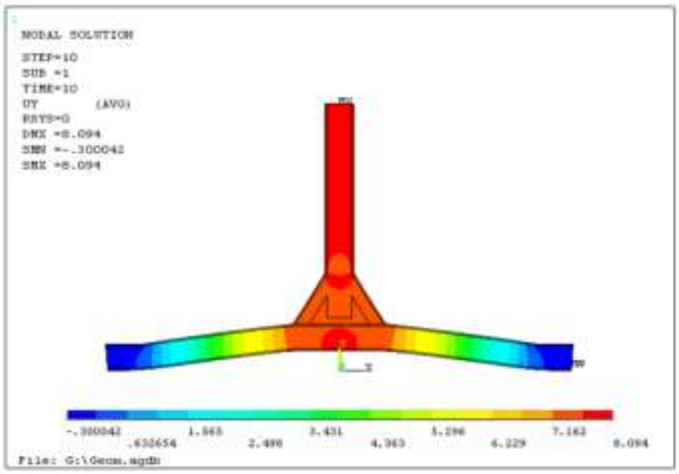
The element (PLANE82) used in the model is a plane 8 node orthotropic element with a 2 x 2 Gauss integration scheme. We used automatic mesh. The automatic mesh initially consists of approximately 10000 elements. The element side length varies from 1mm in the upper skin of panel A and the filler, to 3 mm in the core of panel A and B.

Table:- Material properties for each part in various configurations.

Part No.	Part description	Material description	E_X (MPa)	E_Y (MPa)	G_{xy} (MPa)	S (MPa)	V_{xy}
1	A skin bottom	Glass fiber	26,100	11,500	4,400	31.4	0.14
2	A skin top	Glass fiber	26,100	11,500	4,400	31.4	0.14
3	B skin	Glass fiber	26,100	11,500	4,400	31.4	0.14
4	A Core	PVC Foam	104	104	40	1.4	0.3
5	B core	PVC Foam	104	104	40	1.4	0.3
6	Filler	Epoxy Resin 520	500	500	170	8.7	0.47

By the finite element analysis, it is possible to understand the failure mechanisms associated with the complex structure of the T-joint by indentifying the critical points and improving them. Figure shows the distribution of the stresses σ_y (in the direction of the Y-axis) for a displacement 8mm. The maximum value is located at the corner where there is a stress concentration.

The effects of changes in the T-Joint geometry (overlaminate angle and hull thickness) were studied by comparing the strain distribution in critical regions of the joint. We are reported their observations of the failure mode for the T-joint under static tensile load. We found that the crack started either from the top corner of the interface between the bulkhead and the over laminate or at the bottom corner of the interface between the hull and the overlaminate, as shown in Figure. The exact starting position could not be determined, as even the high speed video camera used to observe the failure has too slow to capture the crack initiation.



Deflection Plot of composite t-joint

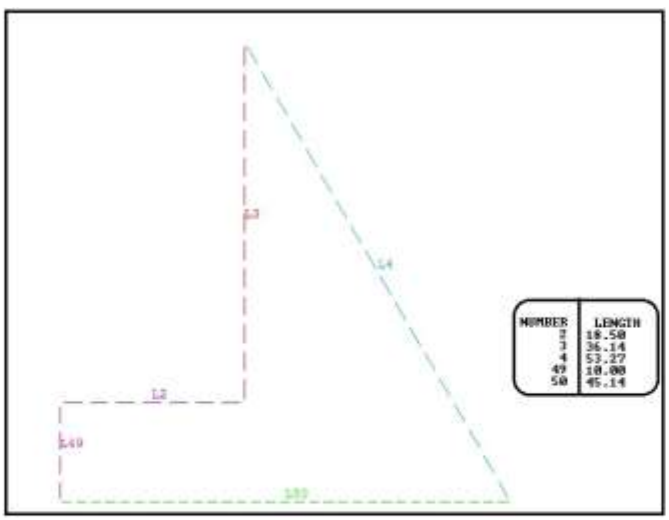
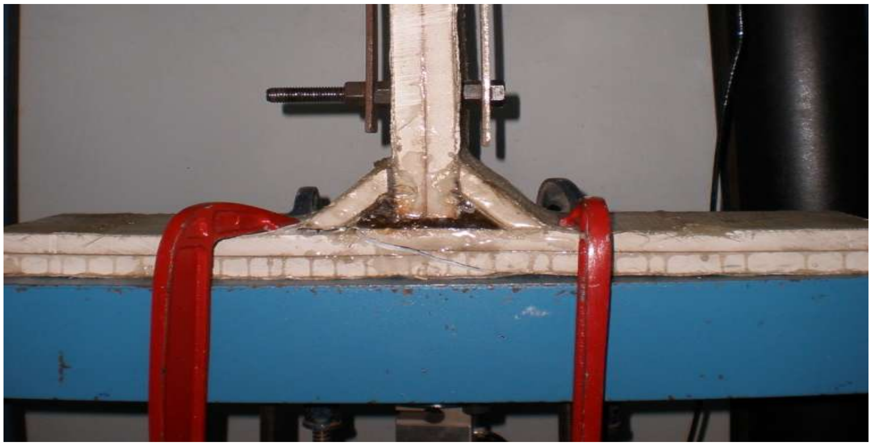


Fig. length and line number of triangle region



The T-Joint loaded as described previously and clamped at the reaction points. The maximum strain in the overlaminate would be expected to occur in the sloping region adjacent to the filler which is subjected to bending and axial loads, both of which are functions of the overlaminate angle and hull thickness. Three regions in the overlaminate were selected for comparison of the strain distribution. These were the top, mid and bottom sections as described. The top and bottom corner sections correspond to the regions where failure initiation would be expected, while the mid-section corresponds with the region where the maximum overlaminate strain would be expected.

RESULTS AND DISCUSSION

The crack initiated from the interface between the over laminate and bottom skin through core A at about 19.8 KN after 8.4 mm displacement. The load versus displacement curve is shown in figure.

Comparison of Results:

LOAD (N)	ANSYS RESULTS IN (mm)	EXPERIMENTAL RESULTS IN(mm)
1982	0.809423	1.2
3964	1.619	1.9
5946	2.428	2.6
7828	3.238	3.1
9910	4.047	3.6
11892	4.857	4.3
13874	5.666	5.1
15856	6.475	5.6
17838	7.285	6.7
19820	8.094	8.4

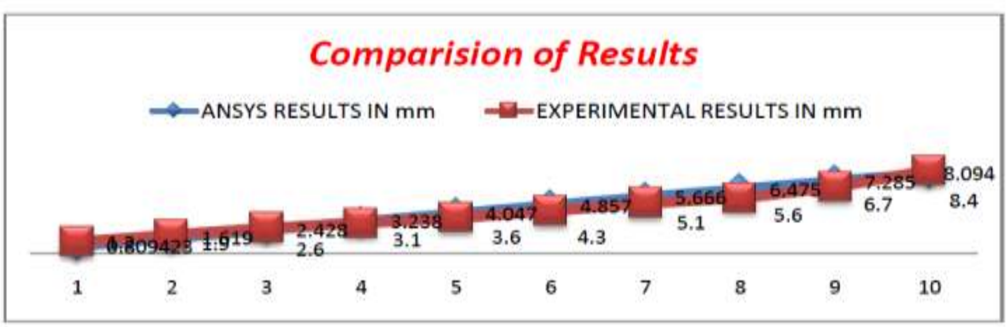


FIG. Comparison of Results graph.

1) The load increases linearly until a load and a displacement respectively of about 1982N and 0.7mm are reached.
2) The load increases but the slope changes because the base starts to deform. a load and displacement respectively of about 19800N and are 8.4mm are reached. In this case bending of the base element is evident.
3) The load increases but with the lower slope. Moreover, the load shows instability due to shear cracks propagation of the core. Initially micro cracks occur at the bottom skin/core interface of the plating element. Then these cracks propagate in the core showing the typical trend at 60^0, mainly in presence of the notches of the PVC blocks, starting by the more external one shown in figure
4) The load drops drastically due to the simulation failure of the base of the T-joint. The cracks reach the top skin or core interface of the plating element, the sandwich collapse and as a consequence, a crack propagates between the plating top skin and the mat of the over laminate on one side of the T-joint.as shown in figure
5) Finally, the load drops drastically due to full failure of the joint. The failure mechanism of this undamaged T-Joint verifies the experiment done by that the failure has always along the over laminate bond line. However, the initial location where the crack begins to grow may be either at the hull- over laminate interface or the bulkhead-over laminate interface. No matter where the crack initiates, the crack will grow along the filler region to cause fracture along the over laminate bond line. We are suggested that the initial crack growth is due to the manufacturing imperfections, such as poor bonding

According to Finite Element Analysis result, maximum stress has developed on over laminate and bottom skin. Hence it was obviously failed at high stress region i.e. from over laminate.

CONCLUSION

- Failure stress is measured through ANSYS software as well as testing for composite T joint specimens.
- Deflection is measured through ANSYS software as well as Testing.
- The load increases linearly until a load and a displacement respectively of about 19820N and 0.8mm are reached.
- The load increases but the slope changes because the base starts to deform. a load and displacement respectively of about 19820N and are 8.4mm are reached. In this case bending of the base element is evident.
- The load increases in Tension Test but the slope changes because the base starts to deform. a load and displacement respectively of about 32030N and are 43.44mm are reached. In this case bending of the base element is evident.
- The load increases in Compression Test but with the lower slope. Moreover, the load shows instability due to shear cracks propagation of the core. Initially micro cracks occur at the bottom skin/core interface of the plating element. Then these cracks propagate in the core showing the typical trend at 60^0, mainly in presence of the notches of the PVC blocks, starting by the more external one shown
- The T-joint fails in the Tension Test shear in the core of the base panel at a load of 19.82 KN
- The T-joint fails in the compression Test shear in the core of the base panel at a load of 32.030 KN

REFERENCES

[1]. D.W. Zhou and L.A. Louca "Numerical simulation of sandwich T-joints under dynamic loading" Composites Part B 39 (2008) 973–98520.
[2]. G. Di Bella, C.Borsellino "Experimental and numerical study of composite T-joints for marine application" international journal of adhesion & adhesives 30(2010) 347-358.
[3]. Jeom Kee Paik, Anil K.Thayamballi, Gyu Sung Kim "The strength characteristics of aluminum honeycomb sandwich panels" Thin-walled structures 35 (1999) 205-231
[4]. Elena Bozhevolnayaa and Ole T.Thomsen "Local effects across core junctions in sandwich panels" Composites: Part B 34 (2003) 509–517.
[5]. S.M.R. Khalili and A.Ghaznavi "Numerical analysis of adhesively bonded T-joints with structural sandwiches and study of design parameters" International Journal of Adhesion & Adhesives 31 (2011) 347–356.
[6]. Ferry Dharmawan and Rodney S "Geometry and damage effects in a composite marine T-joint" Australia Composite Structures 66 (2004) 181–187.
[7]. S.A. Hashim "Adhesive bonding of thick steel adherents for marine structures" Marine Structures {12 (1999) 405}423.
[8]. Dana M. Elzey "A shape memory-based multifunctional structural actuator panel" International Journal of Solids and Structures 42 (2005) 1943–1955.
[9]. Jean-Paul Kabche "Experimental characterization of hybrid composite-to-metal bolted joints under flexural loading". Composites: Part B 38 (2007) 66–78.

[10]. Matthew Kampner "On using corrugated skins to carry shear in sandwich beams". Composite Structures 85 (2008) 139–148 PA 18015.

[11]. .S.W.Boyd "Optimization of steel–composite connections for structural marine applications". Composites: Part B 39 (2008) 891–906.

[12]. A.P.Mouritz "Review of advanced composite structure for naval ships and submarines" Composite structure 53 (2001) 21.

[13]. Aage Lystrup "Design and test of lightweight sandwich T-joint for Naval ships".Riso national laboratory, DK-4000 Roskilde, Denmark November 2008