AIJREAS VOLUME 1, ISSUE 10 (2016, OCT) (ISSN-2455-6300) ONLINE
ANVESHANA'S INTERNATIONAL JOURNAL OF RESEARCH IN ENGINEERING AND APPLIED SCIENCES

DESIGN AND ANALYSIS OF DYNAMIC LOAD CELL FOR FATIGUE TESTING MACHINE

K. MOUNIKA
14R31D0402, M.Tech
Nova College Of Engineering And Technology, Jafferguda, Hayatnagar, Hyderabad

Dr. D.RAMESH
Principal, Nova College Of Engineering And Technology, Jafferguda, Hayatnagar, Hyderabad

ABSTRACT:

In this paper fatigue testing facilities are first classified in accordance with a number of features which include purpose, type of loading, method of load application and transmittal as well as control system. Owing to the significant role played by the loading system in defining the design features, scope and limitations of the testing machine, such systems are herein studied at some length. Typical examples of test rigs built for uniaxial and multi axial loading are presented. A calibration method of wide applicability for uni-axial material testing machines is proposed and experimentally verified. The result of the calibration is an estimate of the linear deviation between the forces indicated by the load cell of the machine, and that experienced by the tested specimen. The analysis relies upon parameter estimation of a measured frequency response function, with a dynamic model derived from a low-frequency vibration analysis of the mechanical machine structure. Details of higher order, possibly created by a flexible machine base, are also investigated. The accuracy is then considerably enhanced whenever the spectrum of the exciting force has significant amplitude at a weakly damped resonance of the base. Since the suggested method indicates the achieved model accuracy, it is easy to detect the cases when the approach is not applicable. The dynamic model can also be used for optimizing machine performance. Optimal machine damping is shown to equal 1= ffiffiffi 2 p , similar to the maximum flat design of second order Butterworth filters. The measured machine was found to have a damping of only 3%.

INTRODUCTION

A Custom build test rig design and fabrication arises, as the functional requirements of engineering components made of composites needs to be evaluated based on their performance requirements and have to be analyzed through specific testing procedure. Whenever a body undergoes tension, compression, bending, torsion or all then that type of failure is called as a Flexural Fatigue Failure. This is generally seen in the cyclic process of the engineering components like connecting rod which undergoes tension and compression for every cycle and so on.

Recently the usage of composite materials has been identified as an attractive alternative for various engineering applications. Since these materials are emerging as feasible substitutes to the conventional materials, in view of the deriving specific advantages like weight reduction with minimum investment etc., the components made by these materials need to be evaluated based on their performance requirements which has to be analyzed through a standard test procedure. Thus the design and analysis of a test rig is an important task.

As these material failures are not completely understood till date, a lot of research work is under the progress. As specific industry standards are not established as of now in the areas such as fatigue and micro failure mechanisms to deal with, the custom build test rig have been designed to meet the immediate requirements of industrial applications.

The present work is aimed at establishing a standard test procedure for analyzing and understanding the flexural fatigue failure behaviour of composite laminates. Composite materials' failure behavior is very complex from the conventional isotropic materials due to the influence of matrix interfacial relations and the polymer matrix fracture behavior.

Considering the above mentioned factors, the test-rig is designed to continuously monitor the health of the laminate throughout the test. The capability of the test-rig critically depends on the dynamic load sensing transducer and data logging system. The FEA software played a key role in the design and fabrication process of the load transducer. This work provides an approach to the design and fabrication methodology of dynamic load transducer and the data logging system development.

The present work establishes the critical approach to design an accurate load measuring device called a "Load Cell" to be used in fatigue test rig, which is capable of measuring dynamic loads by virtue of generating analog signals proportional to the load applied on the body of the load cell. Since this design requires a careful estimation of body strains that have to be transmitted through the strain gauges affixed to the body, the proposed load cell design is utilized in flexural fatigue test rig which is meant to carryout flexural fatigue analysis of laminated components.

Since the sensitivity of the load cell should be good enough to measure the reduction in stiffness of the flexing composite laminates cycle by cycle, it is very essential to have a precise measurement of reaction load offered by composite laminate specimen for which the selection of load cell body material and strain gauge characteristics must be carefully identified.

Also before proceeding further, the S–N curve of the material is thoroughly studied to identify Max. Stress levels to be considered in the design. From the S- N curve the endurance limit is found to be 140mpa. While in the design consideration, the maximum stress levels to be borne by the load cell material is further reduced and is considered 50% well below the endurance limit in view of keeping the load cell life far beyond the experiment test deviation and to prevent the failure of the load cell itself during flexural fatigue experimentation.

The Present load cell is a pillar type load cell from the previous experiences the square cross section of a load cell exhibits variable strains across the gauge bonding areas i.e., from centre of the surface to the edge of the load cell body. Hence it has been decided that the circular cross section is better than square cross section. This consideration played a vital role in exhausting the bonding aspects which prevents peeling off effect. The present available test rig load cell is designed to test glass epoxy laminates, since the glass epoxy laminates are of very low stiffness.

LITERATURE SURVEY

The literature survey pertaining to dynamic load cell and fatigue failure of composite materials under static and dynamic loading conditions is presented in this chapter. The survey also includes various experimental methods supported by simulation and modelling of the failure behaviour of polymer matrix composite materials.

Dr. D. V. Ravi Shankar et al. 2014.[3], The stiffness degradation curves established from the data generated by the test rig clearly exhibits that the stiffness reduction rate is very high during first few fatigue cycles. Then the specimen attains pivoting state where in the top and bottom layers of the specimen were damaged and then due to continuous redistribution of bending stresses further damage to the laminate in the subsequent layers is prevented. The curves fitted to the experimental data are having the exponential decay nature. The test result yielded for $\pm 45^0$ is best orientation of tacking for flexural fatigue for critical applications. Since the data is plotted between bending load in Newton versus number of fatigue cycles in the constant amplitude flexural fatigue test, the curve can also be treated as stiffness versus number of cycles.

Michael Gary Wyzgoski et al. 2008. [4], in this it suggest that Fatigue lifetime predictions for both materials in many cases are indistinguishable from the actual measured data. By contrast, the lifetime predictions of the earlier model could deviate by more than an order of magnitude from the measured data. No additional adjustable parameters are required if one uses the initial breaking strength of the material as part of the new model calculations.

Jae-Mean Koo, et al. 2013. [7], by conducting the tensile and fatigue tests on woven CFRP after low-velocity impact, the residual strength and fatigue life were evaluated. The residual strength decreases as the impact energy increases when the energy is larger than 2.6 J. The fatigue life of composite materials decreases due to the impact damage, and the scattering range of fatigue testing data after impact is larger than that with no impact damage. In order to evaluate the influence of the residual strength prediction model on the fatigue life, the proposed equation for fatigue life after impact and the result of substituting Caprino's model were compared to the fatigue test results.

S.N. Kukureka, et al, 2003. [9], discuss the fatigue life of GRP rods used as strength members in optical fibre cables and fatigue damage The fatigue failure in these materials is proposed by examining the relationship between residual strength and residual modulus. The level of fatigue damage was quantified using percentage modulus degradation. Fatigue failure in bending is considered to occur when the modulus loss reaches 1.5% and a 10% modulus loss is used to define the failure point in tensile fatigue.

The relationship between stress and life was determined for a stress ratio of 0.6. A discussion was presented of the differences in fatigue behaviour under two loading regimes: flexural fatigue and tension-tension fatigue.

Xiaoling Liao, et al. 2008. [10], To understand the effect of tensile fatigue loads on the flexural behaviour of 3D C/C composites and the flexural load-displacement curves of original and fatigued specimens were examined. It revealed that the interfacial strength was degraded after fatigue loads from the observation of the cross-section and the fracture surface of the C/C composites. It is suggested that the weakened interface and reduced residual thermal stresses by fatigue loads play important roles in enhancing the property of C/C composites.

Davi S. De vasconcellos et al, 2014. [18], This study deals with the tensile-tensile fatigue behaviour of a woven fabric hemp/epoxy composite for two different stacking sequences ($0^0/90^0$h and ±45°h), adding up an extensive analysis of fatigue damage mechanisms by combining different techniques. All these results have permitted a complete description of damage mechanisms developing in these hemp/epoxy composites. It has also been demonstrated that the three stages damage scenario for fatigue tests, which is characteristic of carbon/epoxy and glass/ epoxy composites, also describes the evolution of damage mechanisms for the woven hemp/epoxy composite.

Davi S. De vasconcellos et al, 2014. [18], This study deals with the tensile-tensile fatigue behaviour of a woven fabric hemp/epoxy composite for two different stacking sequences ($0^0/90^0$h and ±45°h), adding up an extensive analysis of fatigue damage mechanisms by combining different techniques. All these results have permitted a complete description of damage mechanisms developing in these hemp/epoxy composites. It has also been demonstrated that the three stages damage scenario for fatigue tests, which is characteristic of carbon/epoxy and glass/ epoxy composites, also describes the evolution of damage mechanisms for the woven hemp/epoxy composite:

M. P. Cavatorta et al, 2007. [32], The fatigue and post-fatigue behavior of carbon-glass biaxial fabric reinforced epoxy composites manufactured by RTM and HL has been experimentally evaluated. Tensile and flexural static tests show slightly higher stiffness and strength for RTM specimens when tested off-axis owing to the better matrix properties and significantly higher stiffness and strength for HL specimens when tested on-axis owing to the achieved higher fiber volume fraction. No significant loss in stiffness during cycling nor in residual properties was observed when the maximum fatigue stress was below 30% of the average ultimate flexural strength, regardless of the manufacturing technology and sample fiber orientation.

S. Belouettar et al, 2009. [36], Fatigue tests in four-point bending were performed on two different sandwich configurations; one with an aluminium cores and one with aramide fibres cores. The fatigue test results were presented in standard S-N diagrams. It was also found from this experimental program that the stiffness might not be a good monitoring measure for the "health" of a specimen. When the stiffness starts to decrease during the last part of the fatigue life tests there was already considerable damage present in the core material. Damage initiated in the zone of high shear stresses over the entire length of the zone and in the middle of the specimen.

DESIGN AND FABRICATION OF LOAD CELL

The selection of the suitable load transducer is followed by the following steps:

a. Material selection.

b. Proposing various geometric models.

c. FEM implementation to the proposed designs.

d. Selection criteria for the load cell body design

(a) Material Selection:

The material selection is primarily based on the elastic property that is young's modulus. This should be capable of providing sufficient elastic strain for a given load application range.

As per the present project is focusing on developing load cells with various load measuring ranges such as application range of 0 - 1000N, 0 -1500N, 0 -2000N, 0-2500N, 0 – 3000N , 0 – 3500N, 0 – 4000 N, the material selected for this application is an aluminium alloy of Young's modulus 70 GPa. The same has been cross checked by a tensile test with IS specifications. This test is very essential for FEA implementation to the present problem. The S-N curves of the aluminum alloy were considered during the design of the load cell body.

Composition of Aluminium 6061 alloy:

Component	Amount (wt %)
Magnesium	0.8-1.2
Silicon	0.4-0.8
Iron	0.7
Copper	0.15-0.40
Zinc	0.25
Titanium	0.15
Manganese	0.15
Chromium	0.04-0.35
Aluminium	Balance

Physical Properties:

Density: 2.7 g/cm^3

Melting Point: Approx 580°C

Modulus of Elasticity: 70-80 GPa

Poissons Ratio: 0.33

The typical S – N curve of a given metallic material subjected to fatigue appears to be as shown in the figure.

(b) Proposing Various Geometric Models for Various Sensitivities

From the consensus cylindrical shape body is considered for experimentation. To arrive at the required specifications of the load cell body FE analysis is carried out on designs considered, and the present flexural fatigue equipment design's functional requirements such as load application range dynamic behavior of the test rig. The FE analysis is carried out on the two designs with 20 mm. diameter and 18mm diameter to see the deformation behavior. The Geometric model of the load cell is as shown in the fallowing figure.

(c) FEM Implementation to the Proposed Designs

The final load cell design is arrived from extensive analysis of the proposed load cell body structures. The transducer body is strained within the load range of 0-1000 N and sufficiently strong enough to withstand from the axial tension compression fatigue throughout the experiment. The material selected for the fabrication of load cell body is Aluminum Alloy, where this body produces a linear strain in the applied load range.

The Finite Element Analysis (FEM) is carried out to check the deformation behavior. From the analysis it is clearly understood that the gauge length area is uniformly strained which is the important aspect to be considered for the design of the load cell body. The analysis is carried out by applying load ranges from 100 N to 1000 N. The details are explained in the next chapter.

(d) Selection Criteria for Load Cell Body Design:

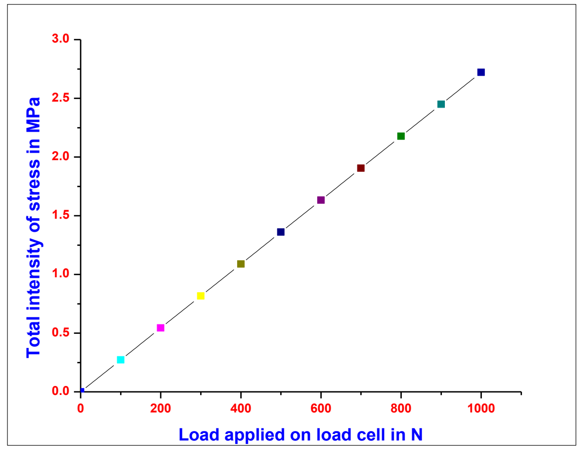
The Finite Element Analysis performed on load cells provided data consisting of load and the respective strain on the gauge length. Table 3.1 furnishes the total stresses and strains induced in the body for the given load. In the present experiment the maximum load applied is 100 N to 1000 N. The geometric load cell is meticulously designed by considering the fatigue characteristics of the load cell body.

The design criteria adopted is the maximum experimentation load that induces a stress level in the gauge portion of the load cell or well below the endurance limit of the load cell body material.

Before arriving at geometric size and shape, FEM is implemented to the load cell design to cross check the stress and strain levels. An analysis is carried out on various geometric sizes and finally the load cell design is obtained to cater the needs of present experimentation functional requirements. The results from FEM analysis indicates that the stresses and strains of the load cell material are below its endurance limit.

The load cell body with pillar type is showing a linear response with respect to load with uniformly distributed load. Hence the pillar type load cell is chosen for fabrication in view of achieving required sensitivity to measure the load accurately.

There by the load cell can with stand up to 1000 N load and works for infinite number of cycles. Table 3.1 clearly shows the load cell body response for a given load from 100 N to 1000 N. Further the load cell ends of the body is transformed into tapered shape to make it compatible to the test rig components and supporting links.



MACHINING OF THE LOAD CELL

The chosen aluminum block is loaded onto a lathe machine available. The piece is perfectly centered. Once the piece is centered in a small 8 mm hole is drilled at both sides of the piece to easily identify the center. Whenever this piece is loaded onto any other machine it does not require centering (saves time and effort). The block is firmly held in a three jaw chuck (three jaw chuck is preferred because only in a three jaw chuck a perfect center can be obtained) and is properly supported. The lathe is run at medium speed and low feed is preferable to attain good surface finish to avoid cleaning of surface at a later stage.

As per the selected model of the load cell, initially the block is rounded to 37 mm diameter rod of aluminum by ordinary HSS tool and at 37 mm the tool is to be changed and a carbide tool is to be loaded to get a good finish.100 mm from one edge of the rod a marking is made. Metal is removed till a diameter of 20 mm is attained from the marking to 40 mm towards other edge of the piece.

Once 20mm diameter is attained the machine is stopped. The tool post is rotated though an angle of 6.5 degrees and machining is done to attain taper of 10 mm(i.e,50 mm length of the piece is tapered from 25 mm increasing to 35 mm as per drawing). Tapering operation is performed on both edges of the aluminum rod.

A 14mm drill is loaded onto the lathe.14 mm hole is drilled into the block on both sides. A 14mm tap is used to make threads on inside of the drilled hole.(also 14mm taps one of right hand and other of left hand threading can be used on the piece if the load cell is to be used as a turn buckle).The whole piece while loaded on the lathe is given an emery paper finish to round the edges.

PREPARATION OF LOAD CELL

- The load cell is cleaned with the toluene ($C_6H_5CH_3$) having a molecular weight of 92.13 kg.
- Marking is done to mark the center (for differentiating top from bottom)
- Center of the rib where strain gauges are to be fixed is marked using a vernier height gauge.
- Another marking at a distance that is one third the center distance is marked, (here on a later stage the strain gauge is placed).
- Angle markings are made accordingly at 0, 90,180,270.
- These markings are completed and the load cell is given an emery finish to give a good grip to the adhesive. The mid rib is marked into four sectors each at a distance of the rib by four.[38.13/4=value].
- This value marking is set on the vernier and same marking is made into sectors all along the piece.

MODELLING AND ANALYSIS OF DYNAMIC LOAD SENSOR

INTRODUCTION TO SOLID-WORKS

Solid Works is mechanical design automation software that takes advantage of the Familiar Microsoft Windows graphical user interface. This easy-to-learn tool makes it Possible for you to quickly sketch out ideas, experiment with features and dimensions, and Produce models and detailed drawings.

Drawing or building a model in solid works starts with a 2 Dimensional sketch and sometimes 3Dimensional sketches are also available for power users. The 2D sketch or 3D sketch comprise of geometry such as points lines, arcs, conics and spines. To define the size and location of the geometry dimensions are added to the sketch. And to define attributes such as tangency, parallelism, concentricity and perpendicularity relations are being used. For controlled dimension in the sketch they can be controlled independently or by relationships to other parameters inside or outside of the sketch.

Drawing can be finally created either from parts or assemblies and they can be generated automatically form the solid model. We can easily add notes, dimensions and tolerances to the drawing whenever required. The drawing module includes most paper sizes and standards such as, ANSI, ISO, DIN, GOST, JIS, BSI, and SAC.

5.2 DIFFERENT MODULES IN SOLID WORKS

- ✓ Sketcher
- ✓ Part Modeling
- ✓ Surfacing
- ✓ Sheet Metal
- ✓ Drafting
- ✓ Simulation

✓ Manufacturing

5.3 COMPUTER AIDED ENGINEERING (CAE)

Computer-aided engineering (CAE) is the broad usage of computer software to aid in engineering analysis tasks. It includes Finite Element Analysis (FEA), Computational Fluid Dynamics (CFD), Multi-body dynamics (MBD), and optimization.

CAE areas covered include:
In general, there are three phases in any computer-aided engineering task:

- Pre-processing – defining the model and environmental factors to be applied to it. (typically a finite element model, but facet, voxel and thin sheet methods are also used)
- Analysis solver (usually performed on high powered computers)
- Post-processing of results (using visualization tools)

5.3.1 Finite Element Analysis (FEA)

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called "Finite Elements". The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called "Nodes" or "Nodal Points". Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called "shape functions". This will represent the displacement within the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an aircraft. In the recent years, FEA has been universally used to solve structural engineering problems. The departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to rely on this technique to stay competitive.

COSMOS WORKS

COSMOS is an application which works on design analysis and fully integrated with Solid Works. Finite Element Method (FEM) is used for this software to simulate the design conditions as per the requirement. Finite element method needs a larger equation as a solution. For finding out the optimum solutions this software is used and it is user friendly software. This software helps to reduce the time frame and cost as it uses computer for analysis, which in turn avoids the field tests.

COSMOS uses the FEM and this FEM is a numerical technique for designing. FEM will works by splitting the model in to tiny pieces of simple shape which are called as elements. This replaces the complexity of the problem in to simple problems and solves simultaneously. COSMOS Works provides one screen solution for stress, frequency, buckling, Drop test, thermal, and optimization analyses.

A product development cycle typically includes the following steps:

1. Build your model in the Solid Works CAD system.
2. Prototype the design.
3. Test the prototype in the field.
4. Evaluate the results of the field tests.
5. Modify the design based on the field test results.

This process continues until a satisfactory solution is reached. Analysis can help you accomplish the following tasks:

➢ Reduce cost by simulating the testing of your model on the computer instead of expensive field tests.

➢ By using this process reduces the data exchange errors.

➢ Reduce time to market by reducing the number of product development cycles.

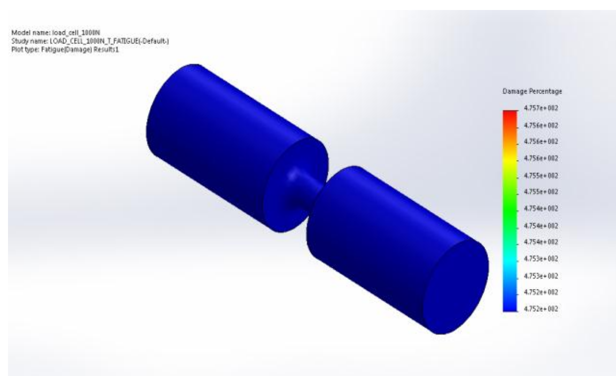
➢ Improve products by quickly testing many concepts and scenarios before making a final decision, giving you more time to think of new designs.

ANALYSIS STEPS

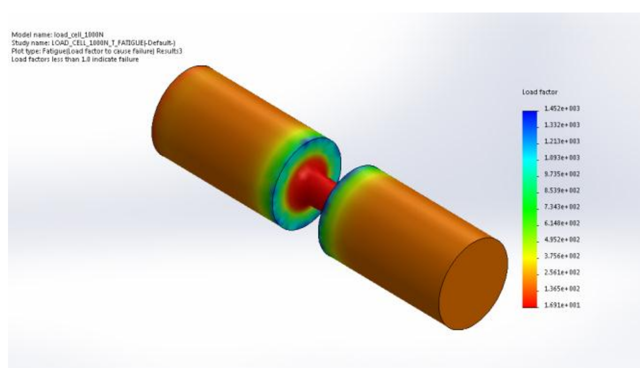
Complete a study by performing the following steps:

- Create a study defining its analysis type and options.
- If needed, define parameters of your study. Parameters could be a model dimension, a material property, a force value, or any other entity that you want to investigate its impact on the design.
- Define material properties. This step is not required in COSMOS Works if material properties were defined in Solid Works.
- Specify restraints. For example, in structural studies you define how the model is supported.
- Specify the loads.
- Mesh the model where COSMOS Works divides the model into many small pieces called elements.
- Link the parameters to the appropriate study inputs.
- Define as many design scenarios as you want (up to 100 design scenarios).
- Run the study or selected design scenarios.
- View and list the results

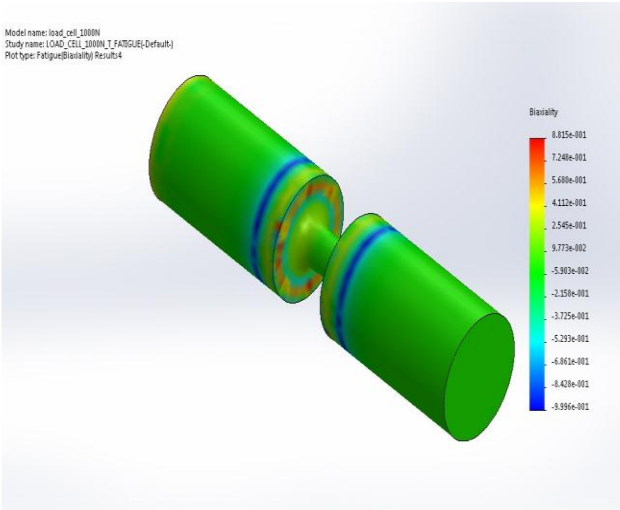
Similarly same procedure is applied for the remaining loads like 3000N, 4000N and the results are analyzed with the help of Cosmos Works.



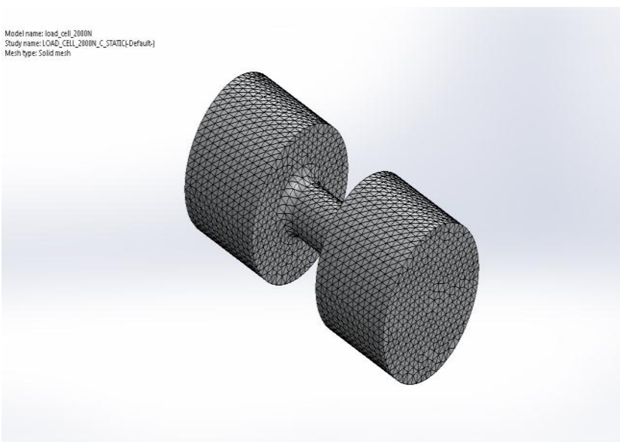
Static analysis of Load Cell-1000N_T_Fatigue_(Damage Plot)



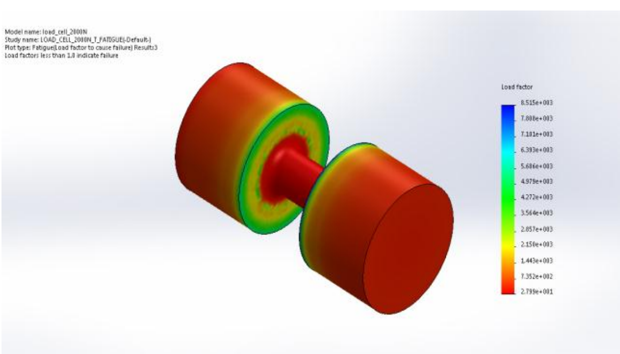
Static analysis of Load Cell-1000N_T_Fatigue_(Load Factor)



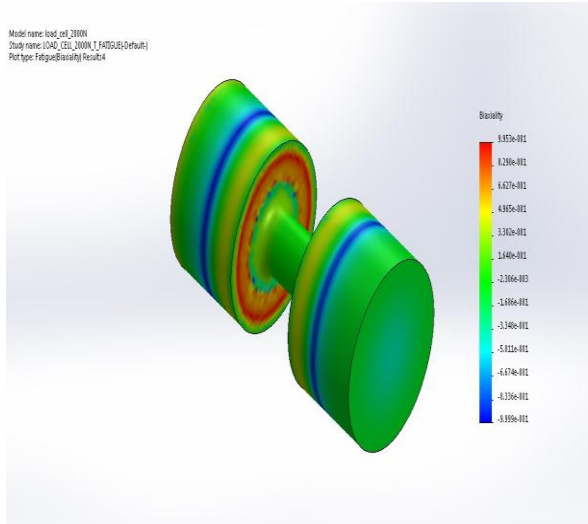
Static analysis of Load Cell-1000N_T_Fatigue_(Indicator Plot)



Meshing of Load Cell-2000N



Static analysis of Load Cell-2000N_T_Fatigue_(Load Factor



Static analysis of Load Cell-2000N_T_Fatigue_(Indicator Plot)

WORKING PRINCIPLE OF THE TEST RIG

The objective of the test rig is to simulate the desired reversed cyclic bending load application on the composite laminate specimen which is fixed vertically as furnished in Fig. 6.5.

The eccentric mechanism is rotated through the worm gear pulley system by 3 HP Induction motor. And obtain the rotating speed as 1.93 RPS. The signal continuously coming from the strain gauge of the load cell is fed to the signal conditioning system to amplify the signal to a readable extent. The electronic circuit of signal conditioning system is capable of amplifying and conditioning the signal precisely.

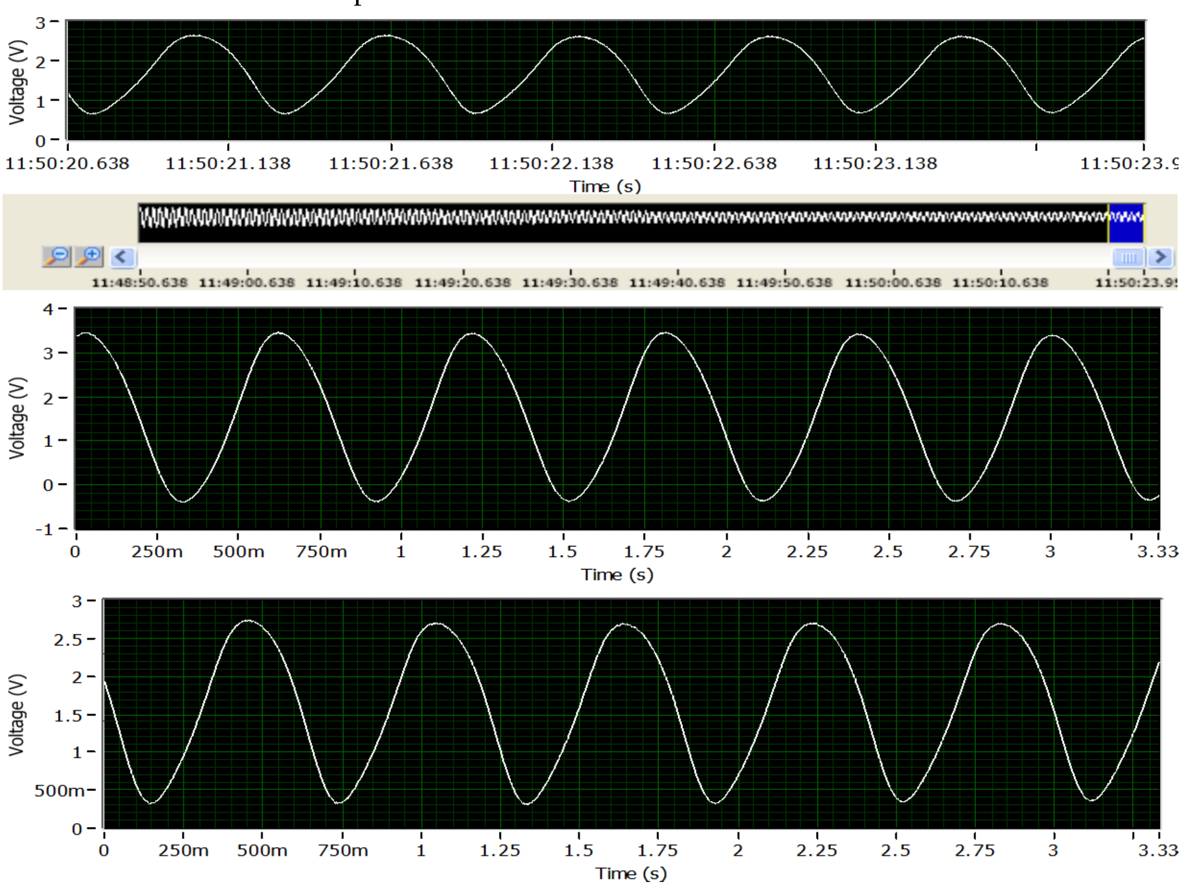
Fig: Actual Flexural fatigue test rig

This analog signal is proportional to the load applied on the composite laminate specimen. The signal is fed to the data acquisition system with 8 channels. The data generated and logged is huge, and the LAB VIEW software continuously logs the data and stores the data in repository. The LAB VIEW software provides a facility to capture the data in the form of snap shots. The data logging system has a capability of sampling frequency of 48 kilo samples per second.

With the frequency of loading cycles at 1.93 RPS, the snap shot may not be plotted in 3.33 seconds as a complete cycle of loading, there by the sampling frequency is reduced to 300 samples per second. The each snap shot data is exported to excel format. The typical sample in excel format is shown in Table.

Table: The sample data in the form of time versus voltage

The sample front end of the LAB VIEW software during experiment and the data represented in this figure has to be further processed, Conversion from time vs. voltage data which has to be converted into load in Newton vs. number of cycles of load application.



The cyclic bending loads are estimated by simulating stresses with an order of 50% of maximum tensile strength. To estimate the bending load, tensile tests are carried out on laminates fabricated as per ASTM D-638 specifications.

The cyclic bending loads to be applied on the specimen are calculated with reference to the beam bending equation.

$$\frac{M}{I}=\frac{F}{Y}=\frac{E}{R} \text{ ----------------------- (1)}$$

The specimen is placed on test-rig in cantilever mode, then the maximum bending moment M=WL (where W is the bending load) is to be applied on the specimen. From the neutral axis (Y) the distance of surface of specimen is equal to half the thickness of the specimen.

Y=d/2 ------------------(2)

Moment of Inertia of the specimen $I=\pi d^4/64$ ---------------- (3) and

The induced bending stresses of the specimen are F = (Ultimate Tensile strength of the specimen)/2

From this the bending load for each specimen is calculated.

The specimen of 150 mm long 30 mm width and 5 mm thick is evaluated subjected to flexural fatigue test. The fluxing is the flexural load applied on the specimen, which is considered from the tensile test results conducted on the specimens and prepared from the same laminates that are used for the fatigue test. The maximum tensile strength becomes the basis for the bending load to be applied on the specimen. The bending load which is to be applied on the specimen is arrived from the calculations derived from bending equation, so that that, the stresses due to the bending are equivalent to 50% of the stresses of maximum tensile strength of the laminate.

By rotating the eccentric mechanism with the help of electric motor, the cyclic bending is applied on the specimen at the frequency of 1.93 RPS.

The bending load is sensed by a sensor (load cell). The signal generated from the load cell is amplified to an extent of a fraction of milli-volts to volts, proportional to the load. The iso-elastic type of sensor strain gauges are used which in turn respond instantaneously to the load applied on the load cell. The voltage so generated is fed to the NI 6009 data logger through signal conditioning system. It is capable of sampling 1000 samples per second.

With the capacity of the signal conditioning system the data is logged and fed to the computer repository with the help of LAB VIEW software. The large quantity of data which is generated from the test coupons were not handled when the limitations of windows based excel software is not in a position to handle more than 64,000 numbers of data points. In such situation, snapshots were collected at regular time intervals. Each snapshot consists of 1000 data points for a period of 3.33 seconds.

The test performed continuously for a period of 4-6 hours depending on the specifications of the specimen. As the test is conducted for more hours it is observed that there is some creep in the generation of data due to heating of strain gauges which were avoided by dynamic calibration technique.

RESULTS AND DISCUSSIONS

The analysis has been carried out by applying loads from 100 N-1000 N in steps of 100 N increment and the corresponding deflections, stresses and strains were recorded in the mid portions of the gauge length and graphs are drawn between them. Similarly for loads varying from 1000N to 3000N also analysis is done. The transducer with a circular cross section was selected and the diameter values of the gauge on which the strain values are to be calculated has been varied and the results were plotted in the graphs.

From the analysis it is found that the stress levels induced in the load cells are well below the yield stresses of the material selected, which strongly supports that the selected material and proposed dimensions will meet the design requirements of the flexural fatigue machine.

Also it is observed that fatigue analysis on the load cell resulted positively stating the load sensor will run dynamically for repeated number of operations.

Table Stress Results

ANALYSIS MODE	INDUCED STRESSES(VON MISSES)(mpa)	YIELD STRENGTH(mpa)
Static compression and tension for 1000N load	3.84049	5.515
Static compression and tension for 1500N load	2.76398	5.515
Static compression and tension for 2000N load	2.24812	5.515
Static compression and tension for 2500N load	1.89235	5.515

The following are the graphs showing different values of deflections at different loading conditions.

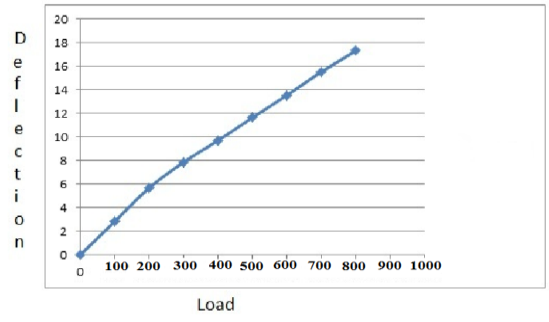


Figure Load v/s Deflection for 20mm dia

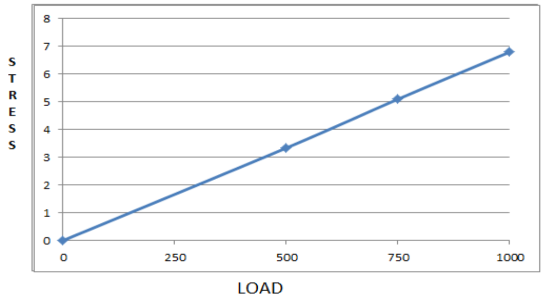


Figure Load vs Stress for 20mm dia

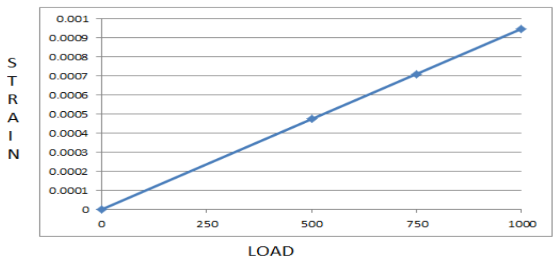


Figure load vs Strain for 20mm dia

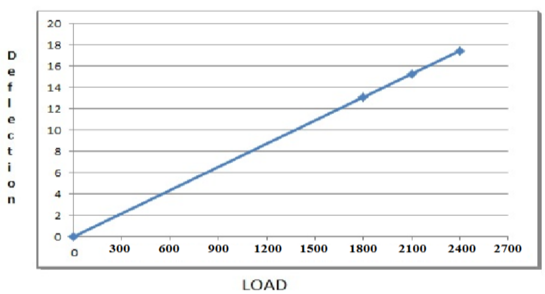


Figure Load v/s Deflection for 50mm dia

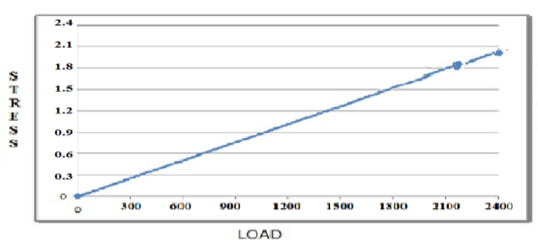


Figure Load vs Stress for 50mm dia

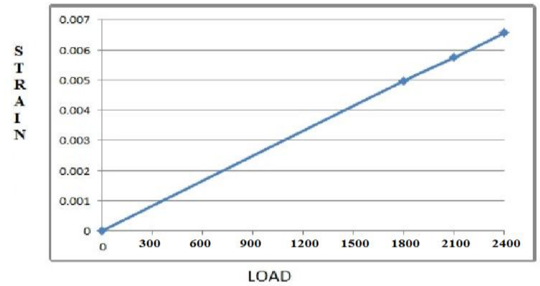


Figure Load vs Strain for 50mm dia

CONCLUSION

The FEA analysis carried out on various load cells which are having measureable load ranges of 0 - 1000N, 0 -1500N, 0 - 2000N, 0-2500N, clearly shows the linear elongation and the load v/s strain is observed to be constantly linear. Hence it can be concluded that the load cell body can be accurately calibrated for electronically measuring the dynamic loads provides a scope for recording the data accurately.The selection of Load cell material for making the Load cell bodies is meeting the present load range measurements.

The care taken while designing the load cell bodies considering fatigue as a critical design parameter has been already proved in the previous experiments that the maximum load falling on the Pillar type load cell has been maintained well below the endurance limit of the load cell body material.

It is presumed that the load cell will provide reliable service while conducting Flexural Fatigue experiments. The same Data acquisition system and signal conditioning system can be successfully used with the already existing electronic hardware, because of that there is no danger of failure of strain gauges since the strain levels has been maintained same.

REFERENCES

1. James H Williams Jr., Hursit Yuce and Samson S Lee (1982), "Ultrasonic and Mechanical characterizations of Fatigue States of Graphite Epoxy Composite Laminates", NASA Contractor Report 3504, Grant NSG-3210, January.

2. Van Paepegem W and J Degrieck (2001), "Effects of Load Sequence and Block Loading on the Fatigue Response of Fiberreinforced Composites", *Mechanics of Advanced Material and Structures*, Vol. 9, No. 1, pp. 19-35.

3. Yongming Liu and Sankaran Mahadevan (2009), "Efficient Methods for Time- Dependent Fatigue Reliability Analysis", *AIAA Journal*, Vol. 47, No. 3, pp. 494-504.

4. Akshantala, N. V., and R. Talreja. "A Mechanistic Model for Fatigue Damage Evolution in Composite Laminates." Mechanics ofMaterials, vol. 29, pp. 123-140,1998.

5. Arnold, S. M., and T. E. Wilt. "A Deformation and Life Prediction of a Circumferentially Reinforced SiC/Ti-15-3 Ring." Reliability, Stress Analysis and Failure Prevention, ASME-DE, vol. 55, pp. 231-238, American Society of Mechanical Engineers, Philadelphia, PA, 1993.

Coffin, C. F., Jr. (editor). Manual on Low Cycle Fatigue Testing. ASTM-STP-465, American Society for Testing of Materials, Philadelphia, PA, 1969.

Drucker, D. C. Introduction to Mechanics ofDeformable Solids. New York: McGraw-Hill, 1967.

Gao, Z., and K. L. Reifsnider. "Micromechanics of Tensile Strength in Composite Systems." Composite Materials: Fatigue and Fracture, vol. 4, pp. 453-470, ASTM-STP-1156, W. W. Stinchcomb and N. E. Ashbaugh (editors), American Society for Testing of Materials, Philadelphia, PA, 1993.

Hwang, W., and K. S. Han. "Fatigue of Composites-Fatigue Modulus Concept and Life Prediction." Journal ofComposite Materials, vol. 20, pp. 154-165, 1986.

Lorenzo, L., and H. T. Hahn. "Fatigue Failure Mechanisms in Unidirectional Composites."Composite Materials: Fatigue and Fracture, ASTM-STP-907, pp. 210-232, H. T. Hahn (editor), American Society for Testing of Materials, Philadelphia, PA, 1986. Pook, L. P. "On Fatigue-Crack Paths." International Journal of Fatigue, vol. 17, no. 1, pp. 5-13,1995.