Vibration Analysis of Hybrid Composite Laminated Beam with Crack in Ansys Parametric Design Language

Amit Ghansela[1], Ajay Singh[2]
[1]*Roorkee, Haridwar (U.K), India*
[2]*Asst. Prof. RIT, Roorkee, Haridwar (U.K), India*

Abstract- Composite cables and beams are an integral part of many structures and are widely used in high-performance machines, airplanes and racing models. Cracking is a type of damage that occurs frequently in building elements and can cause significant structural damage. The effect of cracks on the electrical properties of the structure, such as natural frequencies and vibration modes, has been studied in many studies. However, parametric studies such as the effects of the geometry of the composite beam, the fracture field and the support of its natural frequencies are rarely encountered in the literature. In this review, all studies were performed on hybrid laminated composite beams made of three materials using finite elements, consisting of boron-epoxy, aramid-epoxy and s-glass-epoxy mixture. He probed the numbers to study the response of compound power lines in free lateral vibration.

Keywords- Free Vibration, Forced vibration, undamped & damped vibration, linear & Nonlinear vibration, crack, ANSYS APDL.

INTRODUCTION

In any industrial, constructional and structural application materials play vital role to get desired results, properties, especially for application oriented features. From the last decennaries, in the field of infrastructure, demand of the specific material is increasing, for application oriented features like corrosion resistivity, high strength under cyclic loading conditions, for good design pliability, chemical resistivity and for many application specified properties. These needs make composite a vital material for constructional and structural application in infrastructure as well as in other engineering applications like in aviation, naval vessels and aerospace applications.

These advanced materials have wide range of desired properties which are necessary for structural application as well as other engineering applications like high range of strength to weight ratio and high degree of freedom in case of fabrication and forming capability. Advance material like composites also provide good compatibility with rapid prototyping technique which shows good future scope for both composites as well as rapid prototyping.

Infrastructural development is a main key for the development of any nation because development of nation depends on industries, roads and services of the nation so in the era of industrial revolution the discovery of new materials was important. Humans first started their work with metals which were extracted by ores which are found in the earth crust. As the developments in metallurgy take place, new materials in the form of metal alloys came in picture but with some restrictions like metal can't give desired properties and if two metals can give desired properties but the alloy of them is impossible according to Hume-Rothery rule which took human on the gate of new invention or for advanced.

Composites are those advance materials In most of the composite materials, there are two phases first one is reinforcement and second one is matrix. Reinforced cement concrete (R.C.C.) beam, tires of automobile, wood, boron epoxy, s- glass epoxy are the absolute samples.

OBJECTIVE OF STUDY

In any structural component, the application of the composite material specially laminated composite materials is the unique value of its natural frequency on which that particular part start to vibrate without the application of any external load and it becomes

important to the designer to know all possible modes to prevent or eliminate the condition of resonance. After reviewing, previous carried out researches on vibration analysis of composite beam, especially with the help of finite element solver like ANSYS has a great scope and endeavour. Hence, the present study of vibration analysis on hybrid laminated composite beam, is motivated by the all past carried out researches, which are done on structural member of simple laminated materials, in which effect of some important parameters like load constraints, ply angle of lamina and fibre volume fraction is studied with and without crack in the structure

METHODOLOGY

An ANSYS modal analysis has been widely used to determine the Eigen where it comes under the category of dynamic.

analysis in ANSYS. Natural frequencies and mode shapes are the most important and useful dynamic parameters of any structure which directly influence the designing parameters with respect to safe design conditions. Basically, mode shapes and natural frequencies are the beginning/starting point of harmonic analysis in mode superposition method.

In ANSYS, modal analysis is a part of linear analysis, and it comes as it's under. In present investigation/case study mode extraction method has been used by us which incorporates with Sub space, Block Lanczos, Power Dynamics, Reduced, Unsymmetrical, Damped and QR damped where it is found that QR damped method provides the exposure of damping in any construction usually.

METHODS

As a common practice, generally almost engineering problems are based on mathematical prototype of real stick life problems but in several situation it is not possible to determine the exact solution of concerned problems, hence finite element method is considered to get an appropriate solution of concerned problems. Finite element method is purely numerical methodology to get an appropriate solution of various cases such as structural member, heat flow analysis, fluid mechanics, thermal process and electromagnetism.

Finite element method endows an appropriate solution with respect to engineering problem which are governed by the application of partial differential equations theoretically. In this methodology, the entire/major problem is echeloned into minor elements, which are defined in term of finite elements. Further where these elements get connected to each other are referred as nodes. These elements can be of linear, rectangular and triangular geometry. Accuracy of the method is directly proportional to number of nodes, so for to get more accurate result number of elements should be taken as more.

There are several finite element solvers available worldwide for simulation and analysis of complex problems to bring them easy such as like Abαqus, Nαstran, Femαp, RFEM, and Autodesk simulation, FEBio, Strand7 and ANSYS.

Whereas ANSYS is one among the best, popular and user friendly software applications for simulation, which is widely used as finite element solver in many industries worldwide. It is to notify that ANSYS provides a wide range solution for various engineering problems regarding computational fluid dynamics, structural and thermal etc. with the application of finite element analysis software various differential governing equations can be easily solved after breaking the whole problem into X number of elements and similarly ANSYS works too. ANSYS software can solve the combined effects of various types of applied boundary conditions, which usually providing great exposure in the concerned modelling and awaited results of the analysing problem.

Whereas, it is also notify that some features of ANSYS batch language are coinciding with FORTRAN programming language. Beyond this ANSYS provides two category solver to solve the problem first one is graphical user interface (GUI) and second by the application of ANSYS commands and batch files. In present analysis with respect to requirement GUI is used, As we compare in between them GUI provides general approach of work but consumes more time as compared to ANSYS commands and batch files, which are time saving but complex. It is found that even GUI also has some disadvantages like it require as user to save complete problem file which includes data of modelling and meshing in e *.db file format and results in f*.rst file format, which requires large space of system memory

50 megabytes or more and other one is it becomes more time taking if user needs to repeat some operations. But when we talk about the batch processing and ANSYS commands it provides advantage of easy modification in running analysis without taking much time as it happens in GUI. Also batch files contain whole data of problem which contains entire modelling, meshing and results in 10-100k which makes it more advantageous.

MODEL OF BEAM

All the fibres are assumed to be oriented at an angle α.

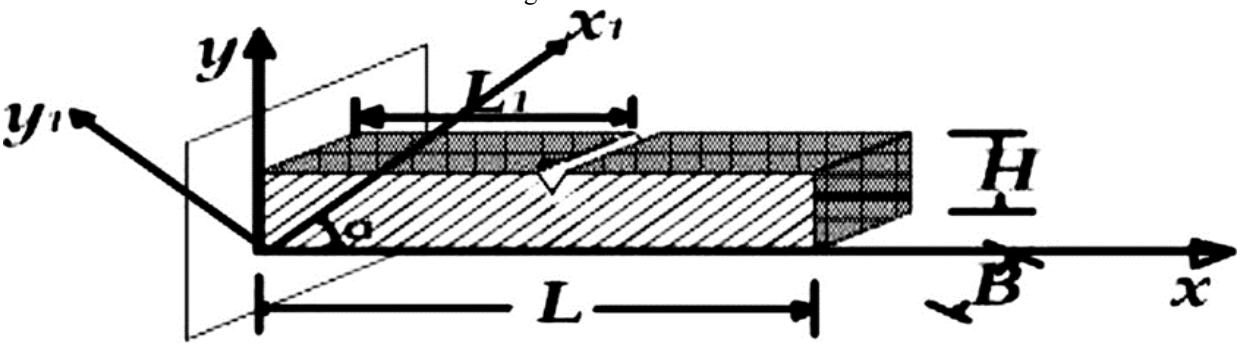

Fig. 3.1 Schematic Labelled Design of Laminated Composite Beam with Transverse Surface Crack in Clamped-Free (C-F) configuration.

Steps of Simulation in ANSYS APDL
All type of analysis which are
given below.

1. Pre-processing
2. Solution stage
3. Post-processing

First of all, at the starting of analysis in ANSYS APDL, user has to specify the preference like structural, thermal or ANSYS Fluid. These options are provided at the preference option in ANSYS APDL. After that user has to select the element type for discretization of problem, to break the whole problem in to finite elements like solid, shell, solid shell, beam, structural mass, structural link etc. Each type of element again have a subset of elements according to degree of freedom of that element.

Next step is assigning the real constants, as we know not all elements requires real constants like thickness, moment of inertia etc. But some elements usually require the values of appropriate real constants according to their behaviour. Further, next are material model and elastic constants in this section user which needs to assign the properties of material such as in case of isotropic material only two independent elastic constants are required first one is Young's modulus second one is Poisson's ratio. For modelling, ANSYS provides great flexibility to user. Even after by following create and operate sub commands modelling of problem can be done. Meshing is also the essential part of pre- processing in this section, complete problem as in present problem of composite beam is discretized into small elements, known as finite elements. Mesh tool provides the flexibility for meshing like in ANSYS APDL where meshing can be done automatically by the element length and according to number of elements required. It provides both type of meshing attributes 2D and 3D. Refinement can be done by the use of mesh tool, generally refinement is done at the place of sudden change in geometry like in beam there is a crack so this area requires more attention than others.

It should be keep in mind that Boundary conditions are the essential part of problem without applying boundary condition ANSYS can't solve the problem. There are two types of boundary conditions, first one is essential boundary condition and second one is natural boundary condition. As in previous paragraph, it is already mentioned that from the 'preference' to 'apply boundary conditions' all the sub steps comes under the step of pre- processing. Now second step is solution stage in this again we can provide some boundary condition. It provides flexibility at the time of applying boundary conditions. Now by the help of meshing and

boundary conditions ANSYS forms the equations and in this step can be done by the use of solve sub-command solution can be done.

To see the results of analysis ANSYS need to go for a step called post-processing, in this step by the use of nodal solution we can see the solution at each node. Post- processing provides great flexibility in reviewing the results. Post-processing contains some sub-commands like list results, failure criteria, result summary etc.

CONCLUSION

The present investigation of vibration analysis of hybrid composite beam with transverse crack is done by the use of the finite element method. Present analysis give a good exposure of dynamic behaviour of hybrid laminated composite structures which are generally assumed as orthotropic materials in nature.

The following outcomes are concluded by the present investigation of vibration response of beam of composite material with transverse open crack also known as v-notch.

It is clear that natural frequency decreases with the increase in ply angle for crack- less unidirectional hybrid cantilever composite cantilever beam and have maximum value at 0° and minimum at 90°.

- Natural frequency increases with the increase in Fibre volume fraction for crack-less unidirectional hybrid cantilever composite cantilever beam and have maximum value at 0.9 and minimum at 0.1.
- In the present analysis, it is clear that in first case, a cantilever beam is taken also known as clamped free configuration, the value of the natural frequencies is lower than second case in which a fixed beam is taken also known as clamped-clamped configuration.

The result of present study can be utilized for many applications where vibrations are main designing parameter. This work can be extended in following ways.

1. Vibration analysis for beams, having non-prismatic beam.
2. Vibration analysis of hybrid composite structure.
3. Vibration analysis of composite structures with inclined cracks.
4. Determination of dynamic stability of beam with multiple cracks.

Vibration analysis of structures of some advance material such as fibre metal laminates and functionally graded material with cracks of different geometries by practical and by the help of finite element method

REFERENCE

[1] Ali, A.T.Z. Aswan, A.J. Faizal, M. and Nisreen, A. 2009. Free vibration Analysis and Dynamic Behaviour for Beams with cracks. *International Journal of Science Engineering and Technology.*

[2] Ahmad, Faraz & Saini, Nitish & Danu, Sunita & Misra, Anadi. (2019). Vibration Analysis of Fiber Metal Laminate Beam in ANSYS Parametric Design Language. *International Journal of Applied Engineering Research. 14. 149-155.*

[3] Asp, L.E. Berglund, L.A. and Talreja, R. 1996. Effects of fibre and interphase on matrix- initiated transverse failure in polymer composites. *Composites Science and Technology*, 56(6):657-665.

[4] Bao, G. Ho, S. Suo, Z. and Fan, B. 1992. The role of material orthotropy in fracture specimens for composites. *International Journal of Solids and Structures*, 29(9):1105-1116.

[5] Barbero, E.J. 2013. Finite element analysis of composite materials using ANSYS®. CRC press.

[6] Bovsunovsky, A.P. and Surace, C. 2005. Considerations regarding superharmonic vibrations of a cracked beam and the variation in damping caused by the presence of the crack. *Journal of Sound and Vibration*, 288(4):865-886

[7] Chondros, T.G. Dimarogonas, A.D. and Yao, J. 1998. Longitudinal vibration of a continuous cracked bar. *Engineering Fracture Mechanics*, 61(5):593-606

[8] Danu, Sunita & Saini, Nitish & Khan, Yasar & Sharma, Harsh. (2019). Composites for Vibro-Acoustics-A Review. *International Journal of Applied Engineering Research. 14. 6-11.*