International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Structural Analysis of Stiffened Plates with Different Materials

H.K. Sreedhar[1], D. Harsha Vardhan[1]

[1]Dept. of Mechanical Engineering, PVKK Institute of Technology, Anantapur, Andhra Pradesh, India

ABSTRACT:

Stiffened plates are comprised of plate components, to which for the most part load is connected, and shaft components situated at discrete dispersion's in one or both headings. The present investigation manages limited component examination of Stiffened plate by different Materials. The current investigation covering structural and modal analysis of stiffened plate with steel, Unidirectional Composite material as Single ply and laminated composite material with 5 plies. The Finite element Analysis is carried by using FEM Solver Optistruct which is inbuilt in Hyper mesh. Optimization of results carried out to match the stiffness and natural frequencies with steel to substitute it.

Keywords: Mode shape, Composite, Stiffened plate, FEA. Etc.

Introduction:

In the present world, the demand for structures with high stiffness is increasing day by day. one of the ways to deal with it is by using stiffeners. Countless mechanical structures are composed of stiffened plates. These plates has many applications in aerospace and marine constructions, where minimization of weight of the components is of paramount interest, stiffened plates find extensive application. They are used in off-shore constructions like oil rigs, marine constructions such as ship and submarine hulls, decks and bridges of ships. In aerospace applications, aircraft wings are made out of stiffened plates. Apart from that they are also utilized in making advanced rocket launching structures. In fact, there are virtually innumerable engineering applications of these components, which Only serve to underline their efficiency, as well as their importance.

In the present world, the increasing demand of structurally efficient and significantly higher strength to weight ratio structures is mostly served by Stiffened plates. The present work deals with the structural behaviour of a stiffened plate under static uniform loads. Firstly, we will consider a geometrically nonlinear beam problem by analyzing the large deflections of a beam of linear elastic material, under the action of transverse load along its length. Under the action of these external loads, the beam deflects into a curve called the elastic curve. Firstly, the relationship between the beam deflection and the loads would be established using Ansys and then the results would be extended to perform analysis on Stiffened plates. The linear and nonlinear behaviour of the beams would be studied under static loading .and then we are using hypermesh software under that we are using optistruct analysis can be performed on the composite stiffened plate with single layer and with multi layers .

Husam Al-Qablan et al in 2010 carried Buckling of simply supported square orthotropic plates with multi-blade stiffeners is addressed herein. An approximate, semi-analytical model for such plates subjected to in-plane loading is derived results were validated using mathematica, they considered stiffened plate of 1000 mm X 1000 mm X 12 mm and compared results of steel and composite of unidirectional material and concluded that the buckling load is highly influenced by fiber orientation angles. The best performance was achieved using [45,-45, 90, 0, 0, 90,-45, 45] fiber orientations in the three loading cases.

Designs and response surface methodology in predicting the natural frequency of laminated composite plate under clamped boundary condition. The finite element analyses have been carried out using D-optimal design in the design of experiments technique by varying the fibre orientations, -450, 00, 450 and 900.optimal solution obtained and concluded that any number of plies can be applied by varying the ply thickness

To avoid the resonant behavior of the laminated structures, the results of the free vibration analysis for the laminated composite structures in the structural design are very important. Also, the composite structures whether used in civil, marine or aerospace are subjected to dynamic loads during their operation. Therefore, there exists a need for assessing the natural frequency

Therefore, for assessing the natural frequency of the laminated composite plate structures, the accurate mathematical model is required. However, free vibration analysis of laminated composite plates and shells has not received ample attention. Finite element method is especially versatile and efficient for the analysis of complex structural behaviour of the composite laminated structures.Ihsan Küçükrendeci and Ömer K. Morgül investigated Effect of elastic boundary conditions on the linear undamped free vibrations of a five layer symmetrical laminated rectangular plate. The free vibration analysis of composite laminated right triangular plates is analyzed based on Mindlin plate theory by using the finite element method. Pandit et al used a nine-noded isoparametric plate-bending element for the analysis of free undamped vibration of isotropic and fiber reinforced laminated composite plates. He studied the effects of the fiber orientation, boundary conditions, material properties, and geometric ratio a/h on the vibration frequencies. Khoa and Thinh developed a rectangular non-conforming element based on Reddy‟s higher order shear deformation plate theory to analyze the laminated composite plates. They concluded that, the size of the mesh and the convergence of the method is involved by thickness ratio (h/a).

Patel et al. was used the finite element method for analyzing the free vibration of laminated anisotropic composite conical cylindrical shell structures. A simple two-noded shear flexible axi-symmetric shell element based on field consistency approach is employed. Rakesh Kumar et al. Employed a C0 isoparametric finite element formulation based on a shear deformable model of higher-order theory using a higher order facet shell element to study the free vibration analysis of composite and sandwich laminates. They also studied the parametric effects of degree of orthotropy, length to-thickness ratio, plate aspect ratio, number of layers and fibre orientation on the frequency and mode shapes. Reddy et al. used the finite element method and ANN to optimize the stacking sequence of laminated composite plates using Distance based optimal design in the design of experiments.

Methodology:

The analysis is done using Finite Element Method and the analysis is done using HYPERMESH. The advantage of using the FEM methodology is that unlimited number of stiffeners can be added to the model, which can be placed at any direction inside the plate element. The advantage of using optistruct in hypermesh software is that it is very easy to analysis and it will more useful for future extensions.

Static analysis is used to determine the displacements stresses, stains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non-zero)displacements, temperatures (for thermal strain). A static analysis can be either linear or non linear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps

- Building the model
- Obtaining the solution
- Reviewing the results.

Build the Model:

In this step we specify the job name and analysis title use PREP7 to define the element types, element real constants, material properties and model geometry element type both linear and non- linear structural elements are allowed. The ANSYS elements library contains over 80 different element types. A unique number and prefix identify each element type. E.g. BEAM 94, PLANE 71, SOLID 96 and PIPE 16

Material Properties:

Young's Modulus (EX) must be defined for a static analysis. If we plan to apply inertia loads (such as gravity) we define mass properties such as density (DENS). Similarly if we plan to apply thermal loads (temperatures) we define coefficient of thermal expansion (ALPX).

Geometrical definitions:

There are four different geometric entities in pre processor namely key points, lines, area and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

Model Generations:

Two different methods are used to generate a model:

- Direct generation.
- Solid modeling

With solid modeling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then instruct ANSYS program to generate all the nodes and elements automatically. By contrast, with the direct generation method, we determine the location of every node and size shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc.) the direct generation method essentially a hands on numerical method that requires us to keep track of al the node numbers as we develop the finite element mesh. This detailed book keeping can become difficult for large models, giving scope for modeling errors. Solid modeling is usually more powerful and versatile than direct generation and is commonly preferred method of generating a model.

Mesh Generation:

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called Nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as mesh.

Boundary conditions and loading:

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are assigned set ID. This helps the user to keep track of load cases.

Solution:

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements are stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non linear static analysis, transient dynamic analysis, etc.

RESULTS AND DISCUSSIONS

Finite element analysis is carried by using Optistruct solver, modeling meshing is performed in hyper mesh.

Analysis-1

Modal	:	Stiffened plate
Dimensions	:	1000 mm X 1000 mm X 5 mm
Material	:	Steel
Boundary conditions	:	Fixed at four corners
Material	:	Steel
Type	:	Isotropic
Density	:	7850 kg/m3
Poisson Ratio	:	0.3
Young's Modulus (E)	:	210 M Pa
Type of Element	:	Hex or Brick
Elemental Property	:	P shell
Load	:	Uniform pressure

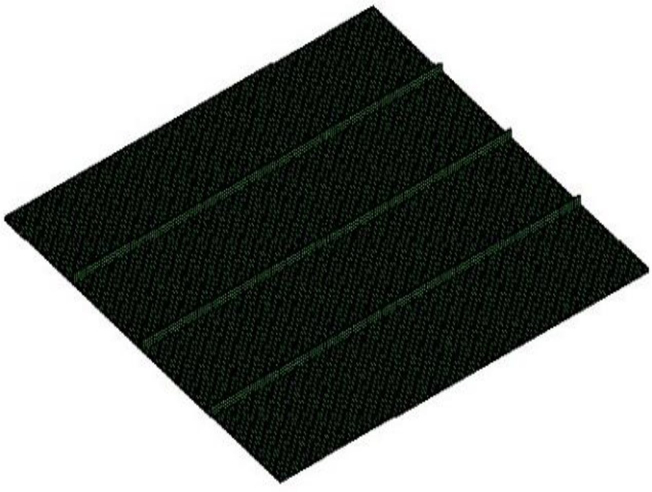
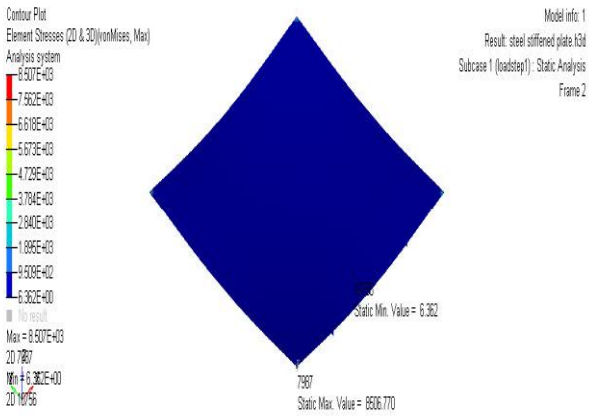


Figure 1 Meshed Modal of stiffened plate

Structural analysis Output of steel stiffened plate:



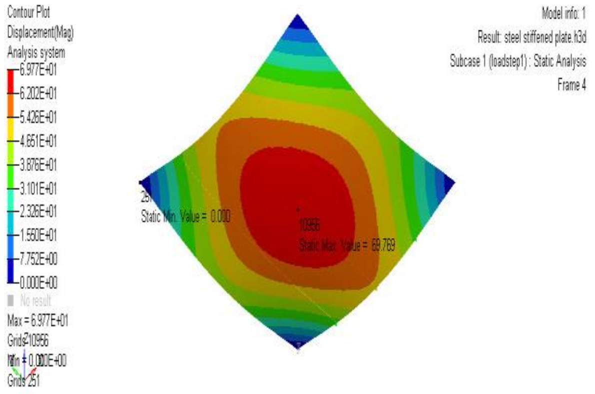


Figure 1 Stress Distribution in Steel Stiffened Plate

Figure 2 Deformation in Steel Stiffened Plate

It has shown that maximum Deformation of plate will occur at centre and maximum stress will occur at the supports, maximum deformation will be around 70mm and maximum stress will be around 85 MPa.

Analysis-2 Closed view of Meshed Modal

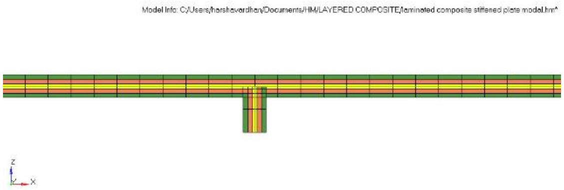
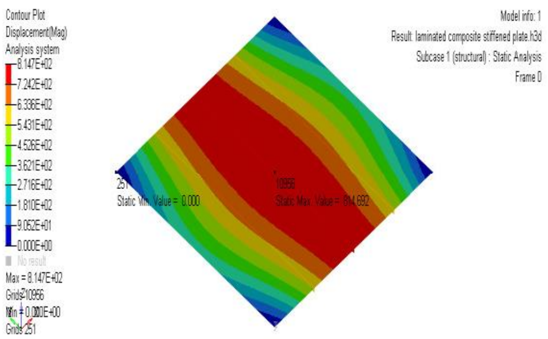


Figure 4 Closed view of Meshed Modal of laminated composite stiffened plate

In current work composite considered with 5 plies of uniform thickness and ply orientations and properties are shown in following table, glass fibers with epoxy matrix are considered for composite material.

Structural analysis Output of Laminated Composite stiffened plate:



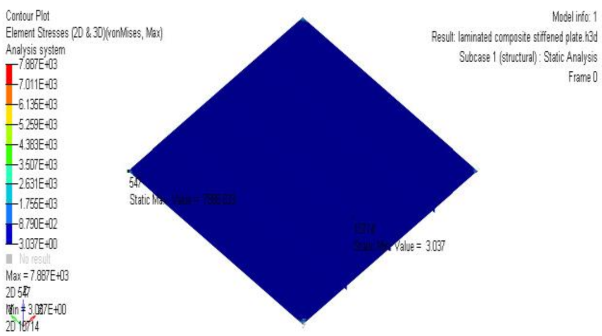


Figure5. Deformation of Composite Stiffened plate

Figure 6 Stress Distribution of Composite Stiffened Plate

When we compare the deformation and Vonmises stress in Steel and composite, composite showed that bit weaker than steel, but in the view of corrosion and manufacturing feasibility we can adopt the composite in place of steel

Conclusion

The Stiffened Plate is modeled and Performed structural and modal analysis in Hyper mesh by varying the material of the plate with fixed supports. The Following Conclusions were obtained

- Structural analysis is carried on the steel and composite plate it is clear that steel plate has less deformation and more stress inclusion in the plate for the applied pressure. Even though composite plate has more deformation since it has less stress inclusion it can be replaced.
- From Modal analysis natural frequency of composite is almost 50% of steel, since steel is possessing more stiffness it will have more natural frequencies and we can say by using polymer composite reaching 50% of Steel and we can replace them
- In both aspects from the simulated data it is suggested to replace the steel stiffened plate with 5 plies laminated Composite.

References:

1. *Husam Al-Qablan et al in 2010 Stability Analysis of Composite Panels with Stiffeners and Circular Cutouts in Jordan Journal of Civil Engineering,* Volume 4, No. 2, 2010 Pp 119-131.
2. B. Sidda Reddy et al in 2013 Vibration Analysis Of Laminated Composite Plates Using Design Of Experiments approach International Journal of Scientific Engineering and Technology (ISSN : 2277-1581) Volume No.2, Issue No.1, pp : 40-49
3. A. Ramanjaneya Reddy, B. Sidda Reddy and K. Vijaya Kumar Reddy, Application of design of experiments and artificial neural networks for stacking sequence optimizations of laminated composite plates
4. Won-Hong Lee.Sung-Cheon Han, Free and forced vibration analysis of laminated composite plates and shells using a 9-node assumed strain shell element. Int. J. comput.Mech.(2006), Vol.39,pp41-58, DOI:10.1007/s00466-005-0007-8.
5. Ihsan Küçükrendeci and Ömer K. Morgül Effect of elastic boundary conditions on free vibration of laminated composite plates, Scientific Research and Essays Vol. 6(19), pp. 3949-3958, 8 September, 2011,
6. Zhiqing Zou, free vibration analysis of composite laminated right triangualar plates with the finite element method, Mechanics of composite Materials and Structures, Volume 5, Issue 1, 1998, pages 25-39
7. M. K. Pandit, S. Haldar, M. Mukhopadhyay, Free Vibration Analysis of Laminated Composite Rectangular Plate Using Finite Element Method, Journal of Reinforced Plastics and Composites January 2007 vol. 26 no. 1 69-80, doi: 10.1177/0731684407069955.
8. Rakesh Kumar Khare , Tarun Kant,and Ajay Kumar Garg, " Free vibration of composite and sandwich laminates with a higher-order facet shell element," Journal of Composite Structures, Vol. 65, 2004, pp.405–418.
9. Ngo Nhu Khoa. and Tran Ich Thinh. 2007. Finite Element Analysis of Laminated Composite Plates Using High Order Shear Deformation Theory. "Vietnam Journal of Mechanics", VAST, Vol. 29, No. 1, pp.47 – 57.
10. B. P. Patel, M. Ganapathi and S. Kamat, "Free vibration characteristics of laminated composite joined conical-cylindrical shells," Journal of Sound and vibration,Vol. 237, No.5, 2000, pp. 920-930