International Journal on Mechanical and Production Engineering
Volume 4: Issue 2 : April 2018, pp 13 – 15 www.aetsjournal.com ISSN (Online) : 2395 - 3500

FEM ANALYSIS OF STIFFENED PLATES USING ANSYS

P. DEENATHAYALAN , S.MATHIYAZHAGAN N.MATHIAZHAGAN

***Abstract*—** In the present world, the increasing demand of structurally efficient and significantly higher strength to weight ratio structures is mostly served by Stiffened plates. These structural elements can be defined as plates reinforced by a single or a set of beams or ribs on one or both sides of the plate. So, stiffened plates are made up of plate elements, to which generally loading is applied, and beam elements located at discrete spacings in one or both directions. The present work deals with the structural behavior 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 behavior of the beams would be studied under static loading .The simulation analysis is completed with a numerical analysis of the system using the ANSYS program, a comprehensive finite element package, which enables students to solve the nonlinear differential equation . ANSYS provides a rich graphics capability that can be used to display results of analysis on a high-resolution graphics workstation.

Keywords -- finite element method (FEM) , finite element analysis (FEA)

I. 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 structural elements can be defined as plates reinforced by a single or a set of beams or ribs on one or both sides of the plate. So, stiffened plates are made up of plate elements, to which generally loading is applied, and the beam elements are attached at discrete spacing in one or both directions. The material of the plate and stiffener can be identical or different. The main advantage of using stiffened.

plates is their significantly high stiffness to weight ratio compared to unstiffened plates [5]. Due to the increase in overall stiffness of the system, enhanced load carrying capacity and stability characteristics are obtained. So, it can be said that these are structurally efficient components that also have the added benefit of material savings and subsequent, economic and costeffective design. Another advantage of stiffened plates is that they can be fabricated with ease and simplicity. Hence, it is no surprise that such structural components find wide-spread utilization in modern branches of civil, mechanical, structural and construction engineering. Stiffened plates are subjected to various types of loading conditions in their working environment. For instance, in case of bridge decks the stiffened plates are under lateral or transverse loads. On the other hand, longitudinal bending of the ship hull exerts longitudinal in-plane axial compression on the plates. The loading conditions on a component can be broadly classified into two classes, static (invariant with time) and dynamic loading (varies with time). Hence, the design of the plates must be carried out keeping in mind these two aspects of loading. Considerations for static design include desired load carrying capacities and deformations within prescribed limits, while, design based on dynamic considerations deals with mechanical behavior with respect to time varying excitations.

P. Deenathayalan, PG Student , Meenakshi Ramaswamy Engineering College , Thathanur , Tamilnadu, India.
S.Mathiyazhagan , HOD, Mechanical, Meenakshi Ramaswamy Engineering College ,Thathanur,Tamil Nadu. India.
(Email id: mathiyazhagan4690@gmail.com)
Dr.N.Mathiazhagan , Principal, Meenakshi Ramaswamy Engineering College, Thathanur ,Tamilnadu, India.
(Email id: mathi.sharmi@gmail.com)

II. METHODOLOGY

The analysis is done using Finite Element Method and the simulation is done using ANSYS. 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 formulation accepts eccentric and concentric stiffeners of different cross-sections.

1) Finite Element Method

The **finite element method (FEM)** (its practical application often known as **finite elementanalysis** (FEA)) is a numerical technique forfinding approximate solutions to partial differential equations (PDE) and their systems, as well as (less often) integral equations. In simple terms, FEM has an in built algorithm which divides very large problems (in terms of complexity) into small elements which can be solved in relation to each other. FEM solves the equations using the Galerkin method with polynomial approximation functions. The solution is obtained by eliminating the spatial derivatives from the partial differential equation. This approximates the PDE with

- a system of algebraic equations for steady state problems .
- a system of ordinary differential equations for transient problems.

These equation systems are linear if the corresponding PDE is linear and vice versa. Algebraic equation systems are solved using numerical linear algebra methods. The ordinary differential equations that arise in transient problems are numerically integrated using techniques such as Euler's method or the Runge-Kutta method.

In solving PDE’s, the major problem is to create an equation that approximates the equation to be analysed, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, which have their respective pros and cons. The finite element method is considered to be the best way for solving PDE’s over complicated domains (like cars and oil pipelines), but when it (domain) changes (ex: a solid state reaction with moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For example in a frontal car crash simulation where we need more accurate results in the front of the car and hence we can reduce the simulation cost in the rear end. Another instance is in weather prediction (which is done numerically), where it is more important to have accurate predictions over highly developing nonlinear phenomena (unpredictable natural calamities which happen, like cyclones) rather than relatively calm environment.

2) Geometric Non-Linearity

Structures whose stiffness is dependent on the displacement which they may undergo are termed geometrically nonlinear. Geometric nonlinearity accounts for phenomena such as the stiffening of a loaded clamped plate, and buckling or 'snap-through' behavior in slender structures or components. Without taking these geometric effects into account, a computer simulation may fail to predict the real structural behavior.

3) Material Non-Linearity

Material Nonlinearity refers to the ability for a material to exhibit a nonlinear stress-strain (constitutive) response. Elasto-plastic, hyperelastic, crushing, and cracking are good examples, but this can also include temperature and time-dependent effects such as visco-elasticity or visco-plasticity (creep). Material nonlinearity is often, but not always, characterized by a gradual weakening of the structural response as an increasing force is applied, due to some form of internal decomposition.

III. PROCEDURE FOR PERFORMING NON LINEAR ANALYSIS ON ACANTILEVER BEAM:

1. Go to Preferences->select structural
2. Then Pre-processor->Element Type ->Add/Edit->Select Solid tet 10 node 187 from the list of the elements
3. Material Properties ->Material Models -> select Structural-> linear ->elastic->isotropic-> values of EX and PRXY were given (71.7e09, 0.33 respectively).
4. Modeling ->Create -> Areas -> rectangular -> By dimensions
5. Modeling->Operate->extrude ->Areas->along Normal (required dimensions were fed)
6. Meshing ->Mesh Tool (smart size and the shape of the mesh were chosen and then the structure was meshed)
7. Go to Solution ->analysis Type ->Sol’n Control->
8. Define Loads ->Structural-> Displacement ->On areas (the face which are to be fixed are selected and the displacement value is set to 0)
9. Define loads ->Structural->force/moment->On Nodes (where te required node is selected from the list of nodes)

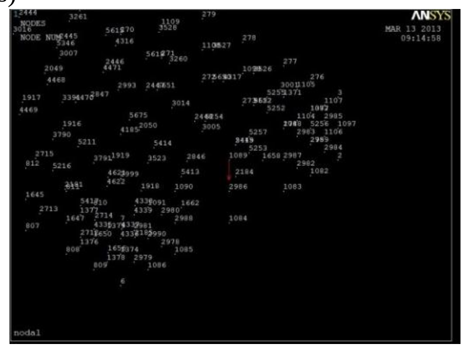


Figure1 Nodal diagram of a cantilever beam

IV. RESULTS AND DISCUSSIONS

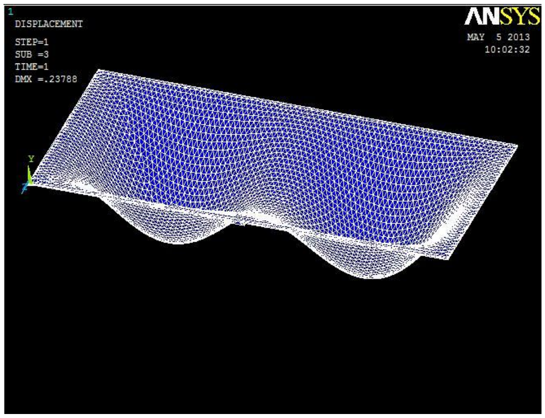
1) Validation Plot

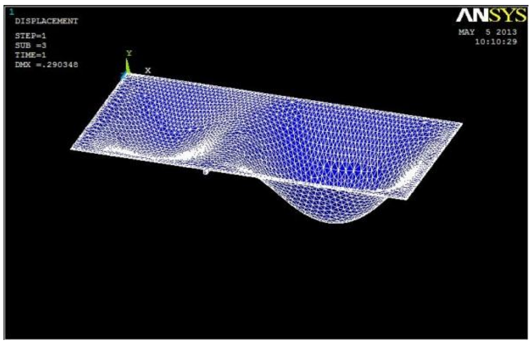
Validation of the present method has been carried out by comparing the results available in literature. The results of the static analysis including the nonlinear results are validates with those of Sheik and Mukhopadhyay[1], Rao et al[3], Korko and Olson[2], and AnirbanMitra, PrasantaSahoo and KashinathSaha[4]. The dimensions and the geometry of the clamped stiffened plate which were subjected to transverse loading is as shown in the figure () .The plots for maximum deflection were compared, and it is seen that they are in good agreement with the already established results.

These results were observed on a stiffened plate with a single stiffener though the middle of the plate. The physical properties of the plate were assumed to be $E_s = E_p$ =71.7GPa, υ= 0.33. The stiffened plate was analyzed under CCCC condition , where C denotes Clamping condition. In this case all the four faces were clamped.

2) Deflection Profile

From the above figures we can infer that the region of the plate which is nearer to the stiffener has a better stiffness, when compared to the rest of the plate. This can be said from the various deflections that are occurring in the different portions of the plate.





1. The Results exhibit hardening type nonlinearity. Stiffness of the system increases with deflection. It shows the effect of stretching of mid-plane of the plate (geometric nonlinearity).
2. From the discussions regarding the position of the stiffener, it can be safely said that the maximum stiffness or the lowest deflection can be obtained when the stiffener is placed at the center of the plate.
3. Change of the stiffener geometry (while maintaining the cross-sectional area constant) apparently doesn't have significant effect on the stiffness of the plate.

References

[1] Sheik, A.H. and Mukhopadhyay, MGeometric nonlinear analysis of stiffened plates by spline finite strip method. Computers & structures, 76 (3),2000,765-785
[2] Koko, T.S. and Olson, M.D. Nonlinear analysis of stiffened plates using super elements , International Journal Of Numerical Methods in Engineering, 31(2),1991, 319-343
[3] Rao, D.V. Sheikh, A.H. and Mukhopadhyay, A finite element large displacement analysis of stiffened plates. Computers & Structures,47 (6),1993,987-993.
[4] Roberto Ojeda, B GangadharaPrusty, A new approach for the large deflection finite element analysis of isotropic and composite plates with arbitrary orientated stiffeners. Finite elements in Analysis and design,43, 2007, 989-1002