FEM Analysis on Stiffened plates using ANSYS

A Thesis Submitted to

National Institute of Technology, Rourkela

In Partial fulfillment of the requirement for the degree of

Bachelor of Technology

In

Mechanical Engineering

By

DANDAPATI HAREESH **KINTALI VINOD**

109ME0572 **109ME0067**

Department of Mechanical Engineering

National Institute of Technology

Rourkela -769 008 (India)

FEM Analysis on Stiffened plates using ANSYS

A Thesis Submitted to

National Institute of Technology, Rourkela

(Deemed University)

In Partial fulfillment of the requirement for the degree of

Bachelor of Technology,

In

Mechanical Engineering

By

Dandapati Hareesh **Kintali Vinod**

109ME0572 **109ME0067**

Under the guidance of

Prof. Anirban Mitra

Department of Mechanical Engineering

National Institute of Technology

Rourkela -769 008 (India)

National Institute of Technology

Rourkela

CERTIFICATE

This is to certify that the thesis entitled "**FEM Analysis of Stiffened plates using ANSYS** " submitted to the National Institute of Technology, Rourkela (Deemed University) by **Dandapati Hareesh**, Roll No. **109ME0572** and **Kintali Vinod ,** Roll No. **109ME0067** for the award of the Degree of **Bachelor of Technology** in Mechanical Engineering is a record of bonafide research work carried out by him under my supervision and guidance. The results presented in this thesis has not been, to the best of my knowledge, submitted to any other University or Institute for the award of any degree or diploma.

The thesis, in my opinion, has reached the standards fulfilling the requirement for the award of the degree of **Bachelor of Technology** in accordance with regulations of the Institute.

Supervisor,

Prof. Anirban Mitra.

ACKNOWLEDGEMENT

It is a great pleasure to express my gratitude and indebtedness to my supervisor **Prof. Anirban Mitra** for his guidance, encouragement, moral support and affection through the course of my work.

I am also sincerely thankful to **Prof K.P. Maity**, Head of the Department of Mechanical Engineering, NIT Rourkela for the allotment of this project and also for his continuous encouragement.

Last but not the least I would like to extend my heartful gratitude to all other faculty members of Department of Mechanical Engineering, NIT Rourkela for their valuable advises and constant support at every stage of the completion of the project.

DANDAPATI HAREESH (109ME0572),

KINTALI VINOD (109ME0067),

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA-769008

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 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 .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.

CONTENTS

LIST OF FIGURES:

Chapter 1

INTRODUCTION

NEED FOR STIFFENED PLATES:

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 cost-effective 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.

Applications of Stiffened Plates:

1. In aerospace and marine constructions, where minimization of weight of the components is of paramount interest, stiffened plates find extensive application.
2. They are used in off-shore constructions like oil rigs, marine constructions such as ship and submarine hulls, decks and bridges of ships.
3. In aerospace applications, aircraft wings are made out of stiffened plates. Apart from that they are also utilized in making advanced rocket launching structures.
4. Stiffened plates are of common occurrence in the field of highway bridges and elevated roadways, where they are generally employed to build the bridge decks.
5. Among land based structures, lock gates, box girders, plate girders etc. are also some of the fields of application for such plates.
6. In some building structures floor slab systems are often made out of these components in order to avoid uncomfortable vibrations.
7. There are some examples of stiffened plates being used in railway wagons, cargo containers, goods vehicle sidewalls etc.

In fact, there are virtually innumerable engineering applications of these components, which only serve to underline their efficiency, as well as their importance.

AIM OF THE PRESENT WORK:

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 .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.

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 [4]. The formulation accepts eccentric and concentric stiffeners of different cross-sections.

Finite Element Method:

The **finite element method (FEM)** (its practical application often known as **finite element analysis** (FEA)) is a numerical technique for finding 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.

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.

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.

The analysis is done in a numerical way by the ANSYS program, a finite element package, which enables us to solve the linear and the nonlinear PDE's and thus the modulus of elasticity of the beam material is obtained . ANSYS is modeling and analysis software which helps in the modeling and analysis of required models, a FEM tool. It is used to analyse complex problems in mechanical structures, thermal processes, electrical fields, magnetics , computational fluid dynamics. ANSYS provides a rich graphics environment, which is used to display results of analysis that re performed .

BEHAVIOUR OF CANTILEVER BEAM UNDER UNIFORM LOADING:

Beams are common elements of many architectural, civil and mechanical engineering structures and the study of the bending of straight beams forms an important and essential part of the study of the broad field of mechanics of materials and structural mechanics. All undergraduate courses on these topics include the analysis of the bending of beams, but only small deflections of the beam are usually considered. In such as case, the differential equation that governs the behavior of the beam is linear and can be easily solved. But as the load increases non-linearity comes into play. This can be demonstrated using a very simple experiment i.e. bending of a cantilever beam. The mathematical treatment of the equilibrium of this system does not involve great difficulty. Nevertheless, unless small deflections are considered, an analytical solution does not exist, since for large deflections a differential equation with a nonlinear term must be solved. The problem is said to involve geometric nonlinearity. This type of nonlinearity is related to the nonlinear behavior of deformable bodies, such as beams, plates and shells, when the relationship between the extensional strains and shear strains, on the one hand, and the displacement, on the other, is taken to be nonlinear, resulting in nonlinear strain-displacement relationships.

Definition of static analysis:

A static analysis determines the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads.

A static analysis contains steady inertia loads, and time varying loads that can be assumed as static equivalent loads (such as static equivalent wind).

Static analysis is used to calculate the stresses, displacements, strains, and forces in structures caused by loads that do not contain inertia and damping effects. We assume that in steady loading the loads and the structure's response vary slowly with respect to time.

Various types of loading that can be applied in a static analysis are:

- Externally applied forces and pressure.
- Steady-state inertial forces (such as gravity).
- Nonzero displacements.
- Temperatures (for thermal strain).

Linear Vs. Nonlinear Analysis:

A static analysis can be either linear or nonlinear. We can allow all types of nonlinearities such as large deflections, plasticity, creep, stress stiffening, contact (gap) elements, hyper elastic elements etc.

Information about nonlinear analysis

Ansys employs the “Newton-Raphson” approach to solve nonlinear problems. In this, the load is subdivided into a series of load increments. Load increments are applied in several load steps.

A nonlinear analysis is organized into three level of operations:

- The “top” level consists of the load steps that you define explicitly over a “time” span. Loads are assumed to vary linearly within load steps (for static analysis).
- Within each load step, you can direct the program to perform several solutions (sub steps or time steps) to apply the load gradually.
- At each substep, the program will perform a number of equilibrium iterations to obtain a converged solution.

Chapter 2

LITERATURE SURVEY

Year	Author	Journal, Vol, Page	Theoretical/ Analytical/ Experimental/ Statistical	Software used	Method used	Material used	Findings
1992	D.Vennugopal Rao, A.K. Sheikh M.Mukhopadhyay	Vol 47 pg 987-993	Mathematical and theoretical	FORTRAN77	Newton-Raphson Iteration	Eight noded parametric plate	Formulation of a generalized formula
1999	A.K. Sheikh, M.Mukhopadhyay	Vol 76 , 765-785	Mathematical	N.A.	Spline finite strip	NA	
2011	Anirban Mitra, Prasanta Sahoo		Mathematical	FORTRAN	Energy methods	Aluminium	Dynamic behavior furnished in the form of back bone curves

Chapter 3

\

ANALYSIS

PROCEDURE FOR PERFORMING NON LINEAR ANALYSIS ON A CANTILEVER 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-> and the following values were fed as shown in the figure 3.1.1

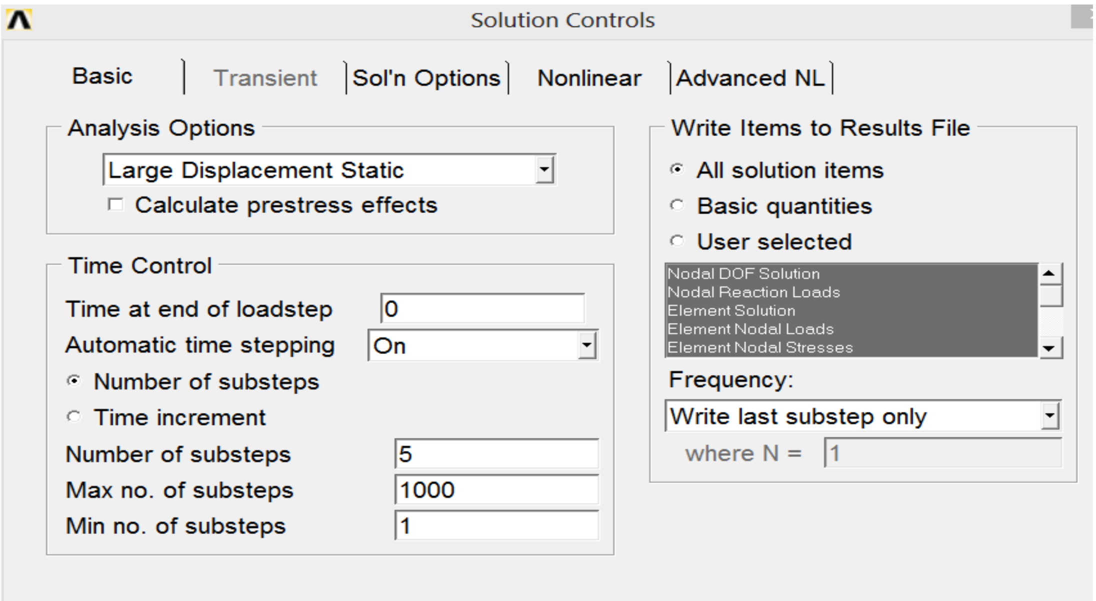


Figure 3.1 Values that are fed in Sol'n Control

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 the required node is selected from the list of nodes)

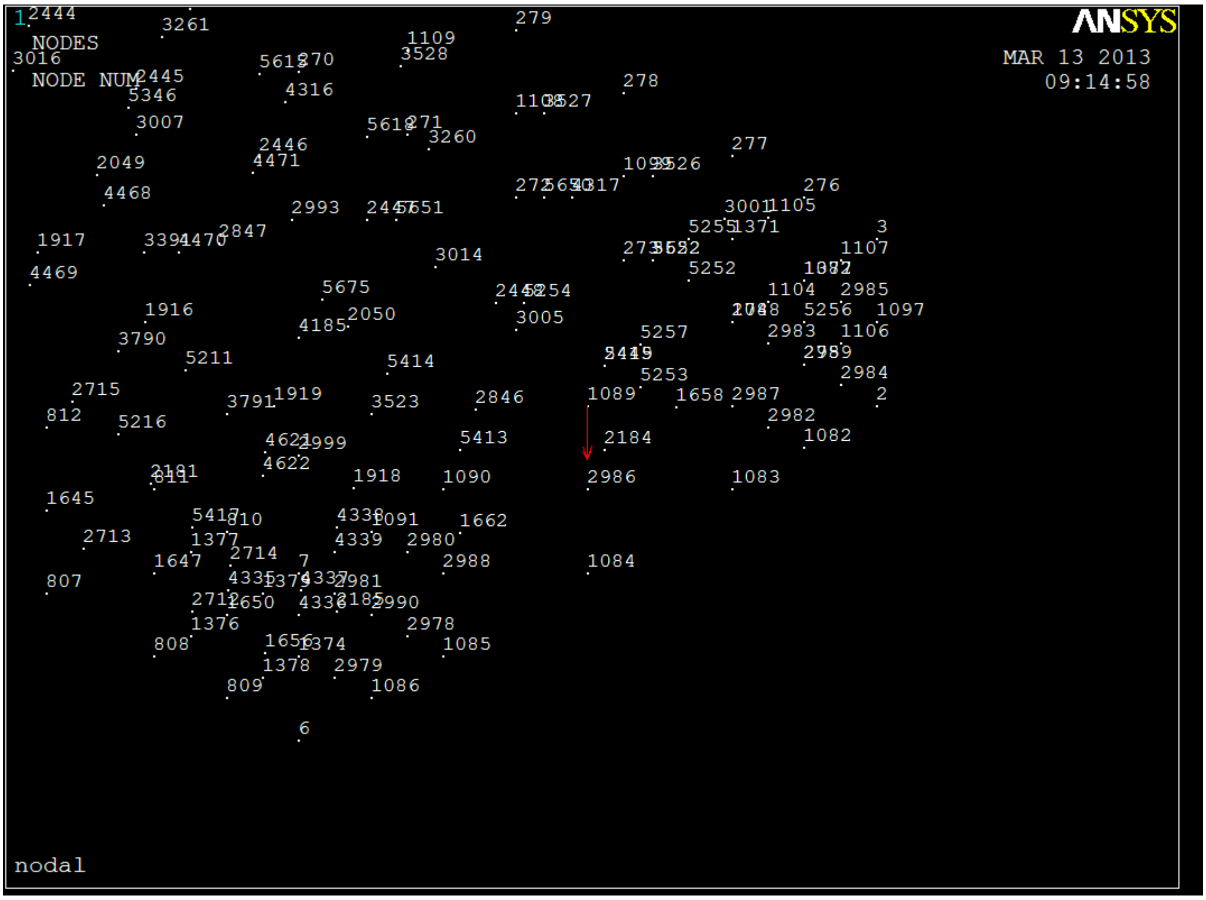


Fig 3.2 Nodal diagram of a cantilever beam

10. Solve ->Current LS

In order to obtain the deformed shape

Go to General Postproc ->Plot Results (select deformed shape)

PROCEDURE FOR PERFORMING NON LINEAR ANALYSIS ON A CANTILEVER BEAM (for Uniform Load):

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->and the following values were fed as shown in fig 3.1.

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->Pressure->On Areas (where the required surface is selected).

10. Solve ->Current LS

In order to obtain the deformed shape

Go to General Postproc ->Plot Results (select deformed shape)

PROCEDURE FOR PERFORMING NON LINEAR ANALYSIS ON A STIFFENED PLATE:

A. **GEOMETRIC MODELLING**

B. **NON LINEAR ANALYSIS ON STIFFENED PLATES**

GEOMETRIC MODELLING:

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 -> Key points->In Active CS

The coordinates are entered in the pop up window.

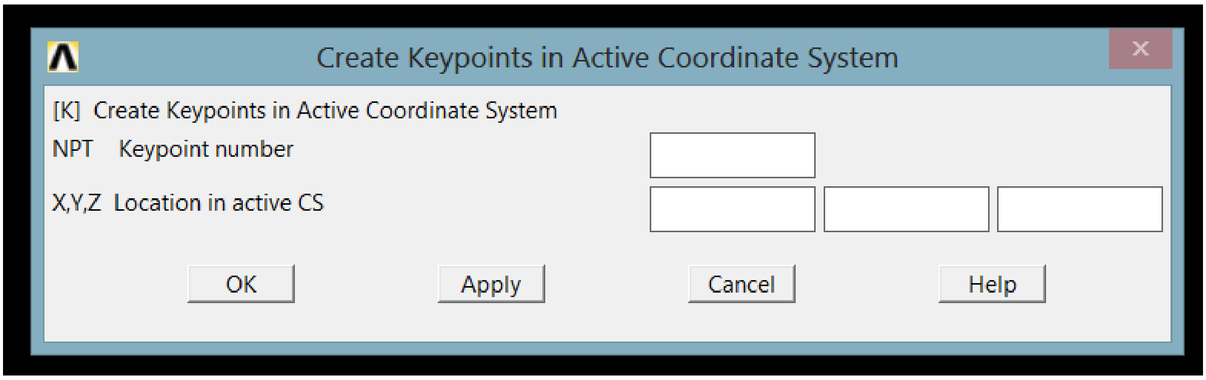


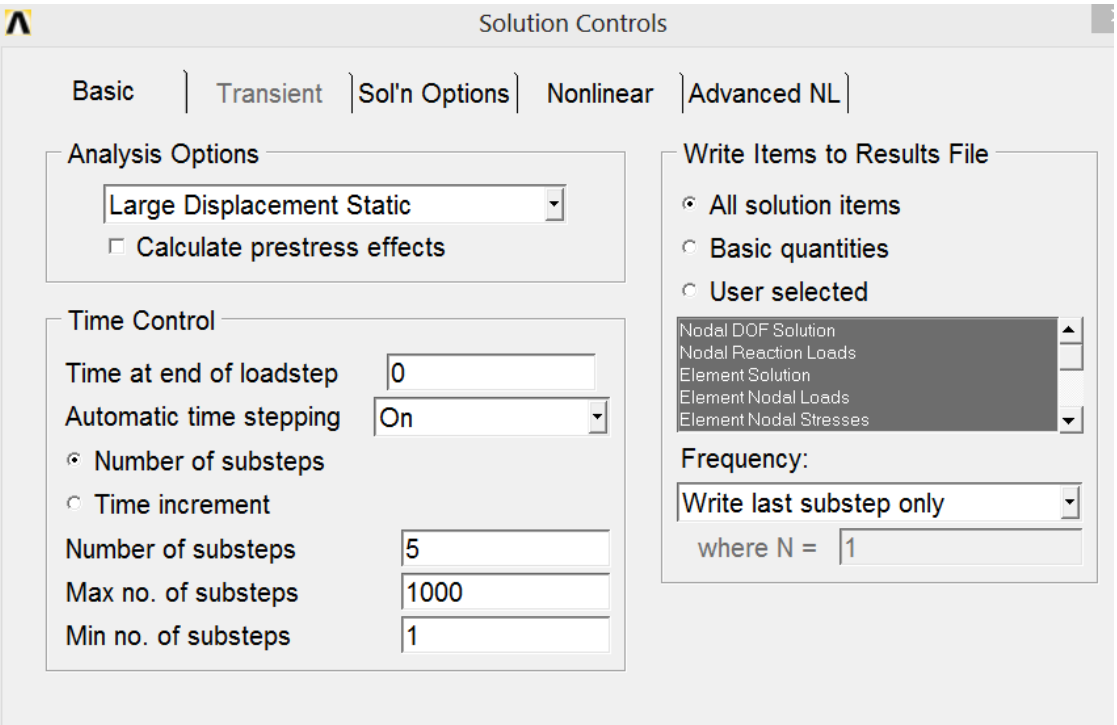
Fig 3.3 Selection of key points in active CS mode

5. Modeling->Operate->extrude ->Areas->along Normal (required dimensions were fed)

NON LINEAR ANALYSIS ON STIFFENED PLATES:

6. Meshing ->Mesh Tool (smart size and the shape of the mesh were chosen and then the element was meshed)

7. Go to Solution ->analysis Type -> Sol’n Control->and the following values were fed as



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 the required node is selected from the list of nodes)

Chapter 4

RESULTS AND DISCUSSIONS

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 Anirban Mitra, Prasanta Sahoo and Kashinath Saha[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.

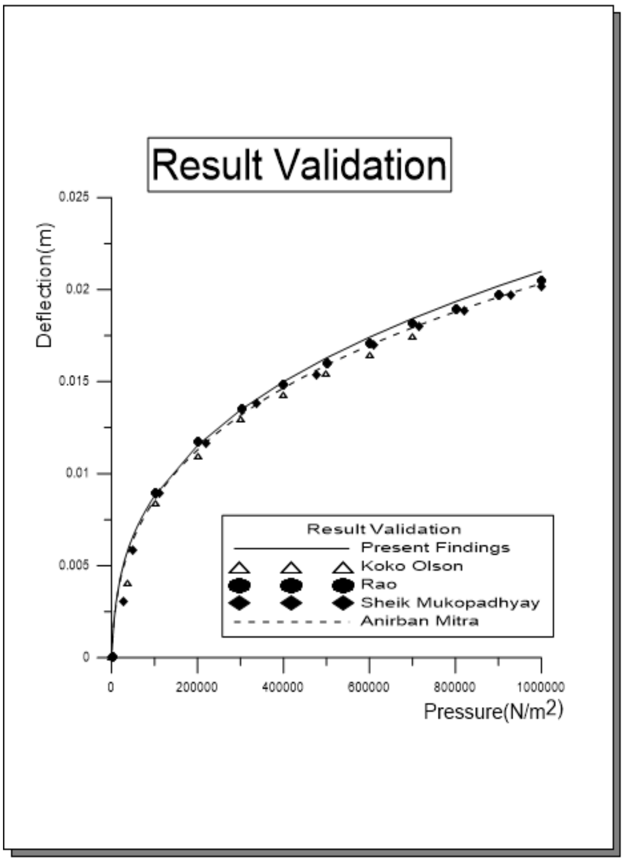
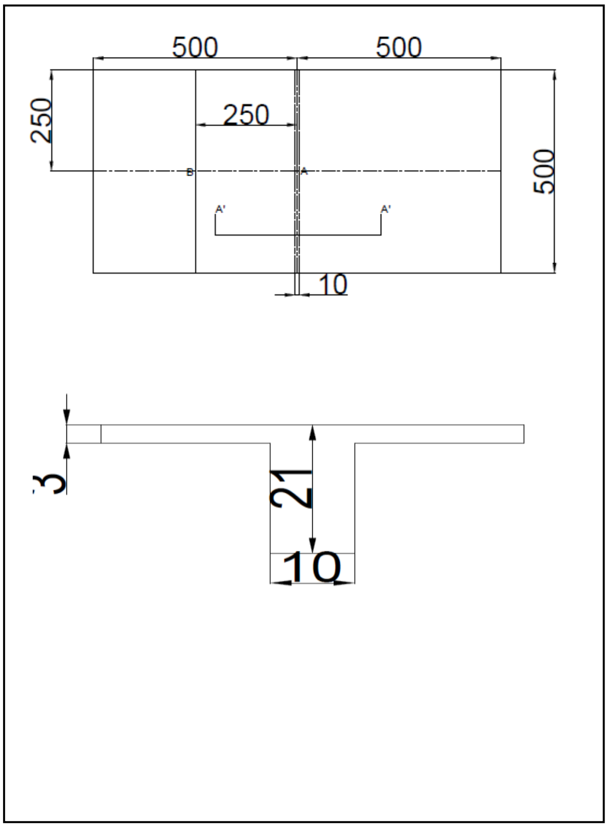


Fig 4.1 Validation result for the
Load Vs. Deflection in a
uniformly distributed load on a stiffened plate



4.2 AutoCAD diagram of the stiffened plate

2.BEHAVIOUR OF STIFFENED PLATES W.R.T. TO CHANGE IN STIFFENER POSITION:

The position of the stiffener was varied from the center and their respective results were studied. The distance of the stiffener was varied in terms of 0.1L (L being the length of the plate).

(a)LOAD VS MAXIMUM DEFLECTION:

From the above graph it can be stated that the stiffness of the plate increases as the stiffener approaches the center of the plate. This can be said so as the deflection of the plate has been decreasing, as the stiffener approaches the center of the plate.

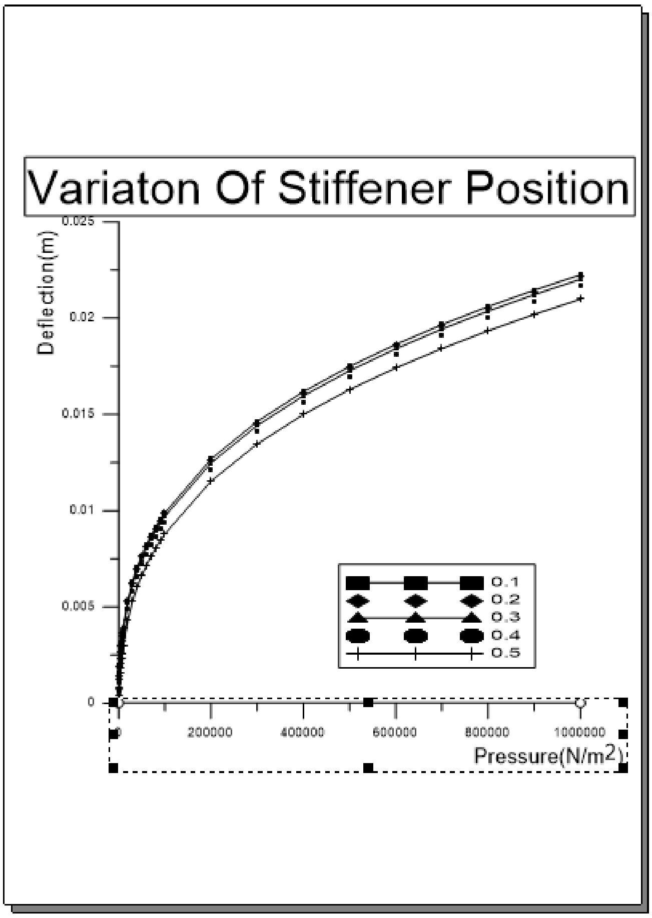


Fig 4.3 Load Vs Deflection graph with variation of stiffener position

(b) 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.

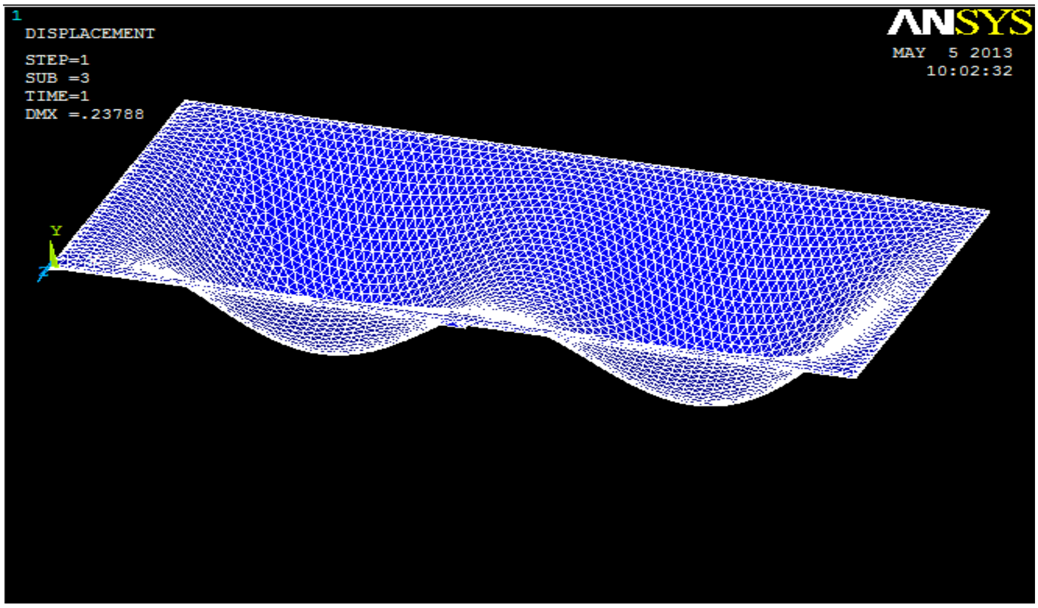


Fig 4.4 Deflection profile of the stiffened plate when the stiffener is at centre

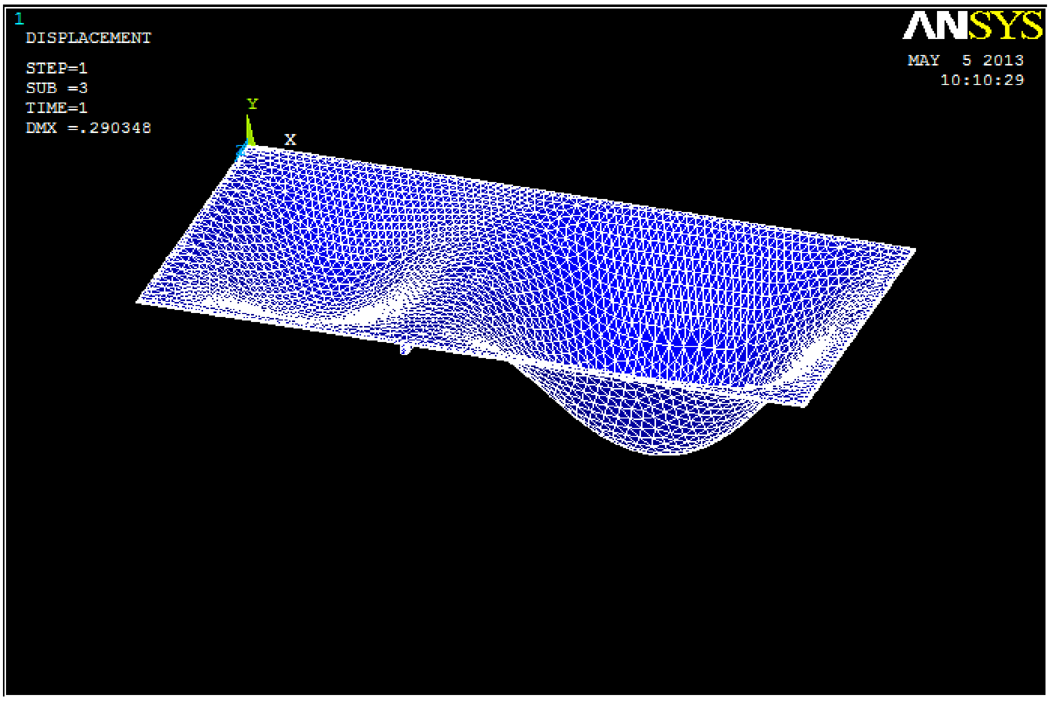


Fig 4.5 Deflection profile of the stiffened plate when the stiffener is shifted towards left from the centre

(C) W_{MAX}/T_P vs Stiffener position:

Here, W_{max} : is the maximum observed deflection

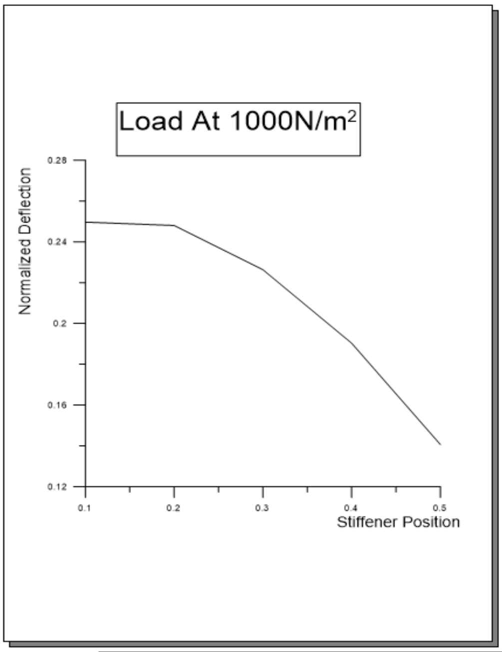
T_p is the thickness of the plate.

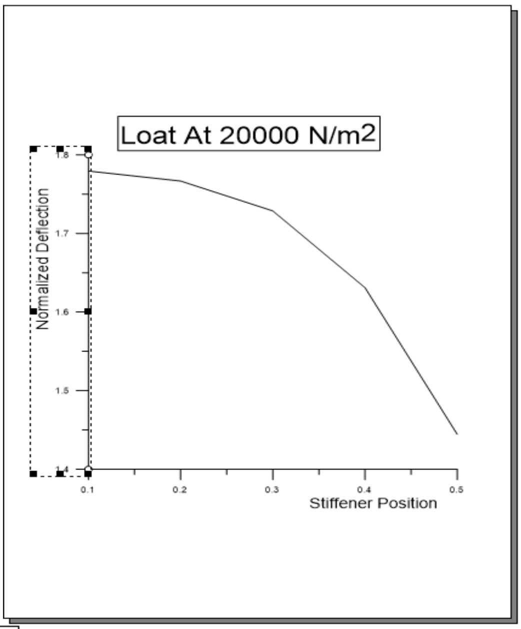
For a particular load (uniform), the values of W_{max}/T_p are calculated and the resulting graphs are plotted.

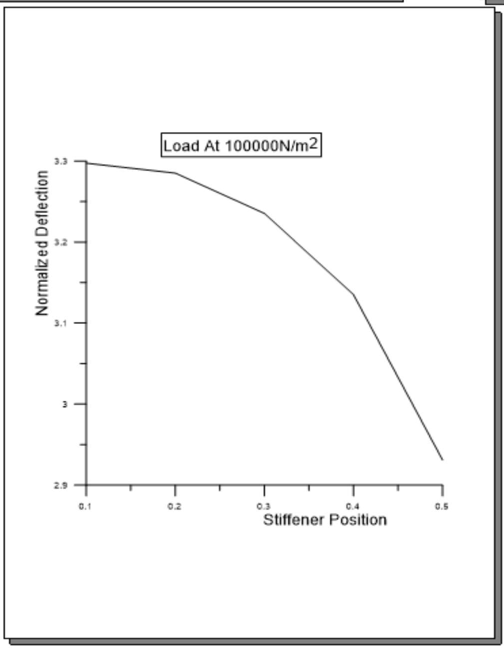
For Load = 1000N/m^2

For Load =20000N/m^2

For Load=100000N/m^2







In all the cases it can be observed that the maximum deflection of the plate reduces as the stiffener approaches the center of the plate. Hence we can say that maximum amount of stiffness occurs when the stiffener is at the center of the plate.

Fig 4.6, 4.7, 4.8 are the Graphs for stiffener position Vs. Normalized Deflection for 1000, 20000, 100000 (N/m^2).

3. VARIATION OF STIFFENER GEOMETRY:

Keeping the volume of the stiffener constant, the thickness of the stiffener is varied. Doing so, the maximum deflection of plate was noted. The values of T_s / T_p were varied from 0.1 to 0.5 and the corresponding results were plotted.

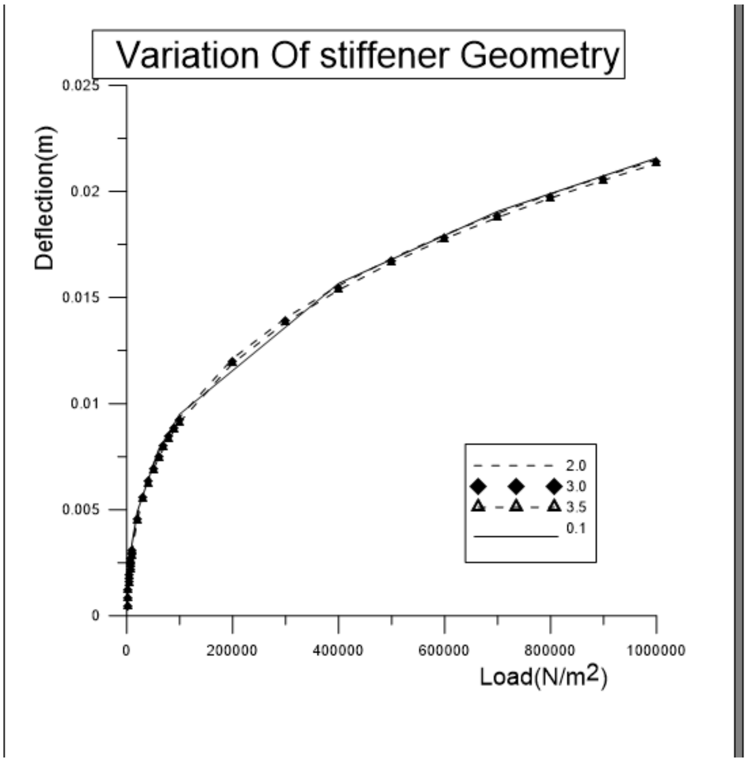


Fig 4.9 Load Vs deflection graph with variation in stiffener geometry

From the above plot we can say that the change in the stiffener geometry has hardly changed the stiffness of the plate.

Chapter 5

CONCLUSION

Conclusion :

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.

Chapter 6

REFERENCES

References .

[1].Sheik, A.H. and Mukhopadhyay, M Geometric 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 Gangadhara Prusty, 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

[5]. Nonlinear finite element method models for ultimate strength analysis of steel stiffened-plate structures under combined biaxial compression and lateral pressure actions ,Thin walled structures, 47, 2009 ,1008-1017

[6]. https://www.sharcnet.ca/