FREE VIBRATION ANALYSIS OF LAYERED BEAMS WITH DELAMINATION DAMAGE-AN ANSYS®-BASED FEM INVESTIGATION

By

Anahita Rastkar Ghorbanzad, BSc

Mechanical Engineering

University of Tabriz, Iran, 1996

A thesis presented to Ryerson University

in partial fulfillment of the requirements for the degree of

Master of Science in the program of

Aerospace Engineering

Toronto, Ontario, Canada, 2012,

Author's Declaration

I hereby declare that I am the sole author of this thesis. This is a true copy of the thesis, including any required final revisions, as accepted by my examiners.
I authorize Ryerson University to lend this thesis to other institutions or individuals for the purpose of scholarly research.

I further authorize Ryerson University to reproduce this thesis by photocopying or by other means, in total or in part, at the request of other institutions or individuals for the purpose of scholarly research.
I understand that my thesis may be made electronically available to the public.

Abstract

FREE VIBRATION ANALYSIS OF LAYERED BEAMS WITH DELAMINATION DAMAGE-AN ANSYS®-BASED FEM INVESTIGATION

Anahita Rastkar Ghorbanzad, Master of Applied Science, Aerospace Engineering
Ryerson University, Toronto, 2012

Identifying delamination has been a focal point for many researchers. The reason for this interest arises from criticality of delamination in a variety of industries: automotive, aerospace, and construction. Therefore, vibration-based damage identification method is applied to detect, locate and characterize the damage in a mechanical structure. In this method, natural frequency as a diagnostic tool to determine the integrity of a structure has been utilized.

The current research presents a FEM-based investigation into free vibrational analysis of defective layered beams with free mode delamination. It is shown that the size, type and location of delamination directly influence system non-dimensional frequencies.

Based on an existing 1D model, the investigation is extended to 2D modelling for single- and double-delamination cases. In each case, Fixed-Fixed and cantilevered beam configurations, both centred and off-centred delamination conditions are studied. Further, a 3D model is also developed for single delamination of a Fixed-Fixed beam. All simulation results show excellent agreement with the data available in the literature. The ANSYS® FEM-based modelling approach presented here is general and accurately predicts delamination effects on the frequency response of beam structures.

Acknowledgments

There are many to whom I owe the deepest of gratitude for the completion of this dissertation.

First and foremost, I would like to thank my advisor, Dr. Seyed M. Hashemi, for his patience and guidance throughout this study. His coaching has allowed me to view my research beyond academics and applicable to all areas in life.

I am grateful to my parents for their continued emotional support during my graduate studies.

I would also like to make special acknowledgments to my husband, Mehran Salmani, whose support and genuine interest in my graduate studies has helped me in countless intangible ways.

To all of you, I will always remember.

Anahita Rastkar

Table of Contents

List of Tables

List of Figures

Nomenclature

I	Moment of inertia
E	Modulus of Elasticity
EI	Bending Stiffness
A	Cross-Sectional Area
L	Beam Segment Length
x	Axial Coordinate
λ	Non-dimensional Frequency
v	Lateral Displacement
M	Bending Moment
a	Length of delamination
H	Beam Segment Height
a_1	Delamination Length
a_2	Delamination Length
a_0	Total delamination Length
at	overlapping delamination
Py	Axial force
V1	Shear force
V2	Shear force
V3	Shear force
Rc	Rigid connector and constrained mode
Rf	Rigid connector and Free mode
Sf	Soft connector and Free mode
Sc	Soft connector and constrained mode
ρ	Density

Chapter 1: Introduction

==

In recent years, mechanical structures are manufactured using composite materials. High specific stiffness and strength of the material is what makes these composites the material of choice in structures. These composite materials are favoured in the industry such as aerospace because of their strength and weight. Composites are fabricated using high-strength fiber filaments and embedding them in plastics, metal or ceramics. Composites are often costlier than conventional metals but their properties justify their use.

Glass reinforced plastic (GPR) is the most common plastic material. Polyester and epoxy resin, reinforced with glass is used in boats, footbridges and automobile body.

One of the most expensive composite materials is carbon fibres. They possess an increased stiffness with a high tensile strength. Their tensile, strength properties compare with steel but weigh about a quarter as much. Carbon fiber materials now compete with aluminium in aircraft structures. Airbus A320, saved over 850 kg in weight by using composite materials in aircraft structures such as elevators, rudder, vertical stabilizer and ailerons [1] (Figure 1). Mechanical properties of composite materials deteriorate due to possible damages. Delamination is one of the most common failures of composite materials. Such failures, especially in aerospace industry often result in fatal accidents.

National Transportation and Safety Board (NTSB) reports accidents involving single engine planes, gliders and large commercial aircrafts due to delamination. Flight 587 of American Airlines crashed on November 11, 2001 in New York due to delamination in vertical stabilizer and separation of rudder of the aircraft [17] (Figure 2).

On August 24, 1996, Sunquest Aviation plane Grob G115D, crashed in Florida, US. Eyewitnesses reported hearing the engine ‘revving up’ and saw pieces of airplane falling to the ground. Grob’s flutter analysis based on control connection stiffness, indicated signs of potential flutter under certain maintenance conditions. Oxidization was discovered on several of the elevator hinges displaying signs of de-bonding/delamination. The result was rudder flutter and in-flight breakup of the airplane [17]. Therefore, detecting delamination failures especially in early stages is of outmost importance to avoid the occurrence of such tragedies.

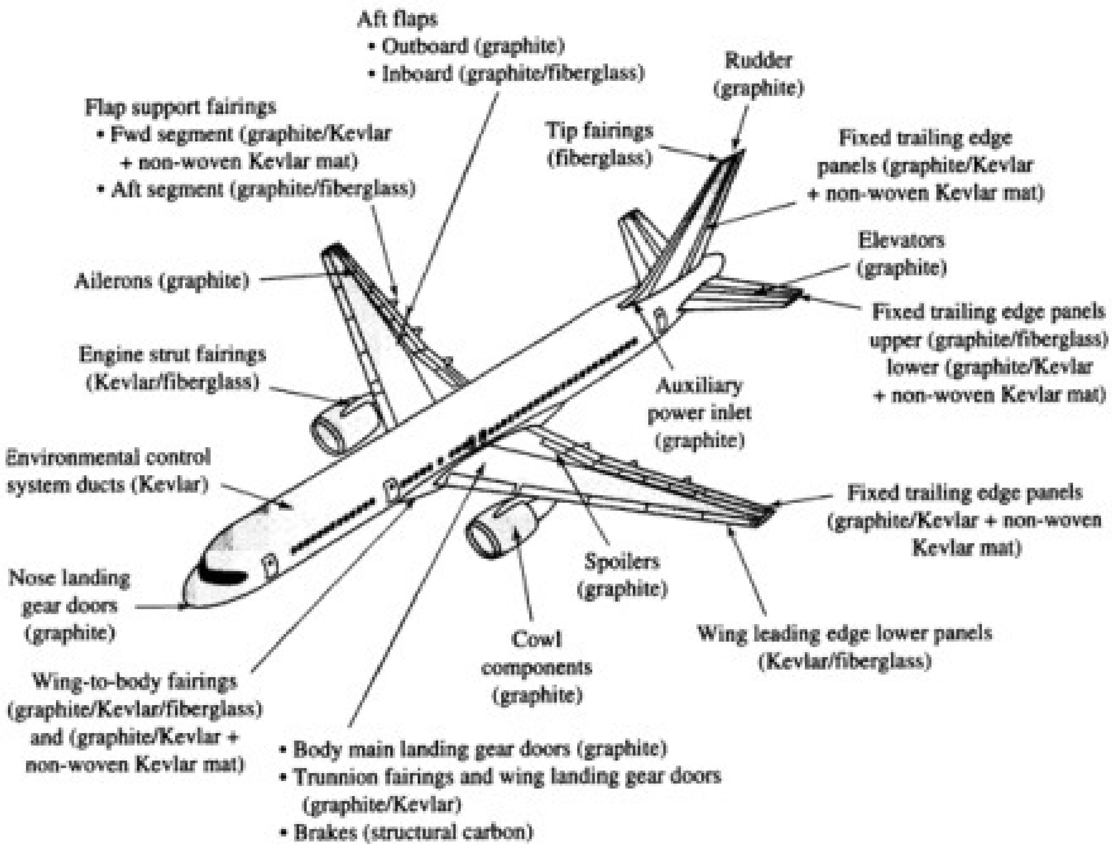


Figure 1- List of Composite parts in the main structure of Boeing 757-200 aircraft, adapted from [33]

Delamination is caused by improper or imperfect bonding, crack in material, chemical corrosion, and separation of joined tiles or broken fibers during manufacturing. Some of these failures may be attributed to in-service loads, which are caused by object impact or by fatigue [4].

When delamination occurs in a structure, the bending stiffness at that cross section of the material decreases and in turn the natural frequency of the mechanical system also decreases. A small change in the value of natural frequency is a great indicator to identify that delamination has occurred. Since frequency is proportional to the square root of stiffness, a small variance in frequency would mean a large damage to stiffness of the composite. It should be noted that the reduction in stiffness and natural frequency depends on the size and location of the de-bonding of material.

Figure 2- Core material of rudder of flight AAL587, adapted from [17]

The delamination analysis on structures can be categorized into:

- linear :

  A linear damage can be explained as a damaged structure which remains linear elastic after damage, while it was originally linear elastic as well.

- non-linear:

  In this case, the elastic structure behaves in nonlinear manner while initially behaved in linear elastic structure.

In this research, only linear elastic structure is discussed and nonlinear structure is beyond the scope of this thesis.

Linear methods can be divided in model based and non-model based categories. The focus of this study is on model based methods. In this method, material is assumed to be homogenized on cross section which will behave like an isotropic material. Another assumption which is incorporated in this study is the absence of torque and out of plane coupling for a layered slender beam. With these assumptions, Euler-Bernoulli classic beam theory can be used for this model.

Analysis of delamination in composite material has stirred an interest for many researchers and scholars over the years. The two possible conditions are “free mode” and “constrained mode”. In free mode, the delaminated layers deform without touching each other, ‘freely”, and each deformed layer has different displacement. In constrained mode, in opposition to free mode, the delaminated layers are considered to be in contact with one another. That would mean the delaminated layers have identical transverse displacement.

Researchers first concentrated on free mode and following that constrained mode was investigated. The research continued with investigation of constrained mode for biomaterial.
One of the pioneers of these researchers is Ramkumar [7]. In 1979, he introduced a model in which four Timoshenko beams were connected at the delamination edges to form a composite beam with one through width delamination. The results they obtained from this research were lower than experimental measurements.

In 1982, Wang [5] improved the analytical solution accomplished by Ramkumar [7]. In that research a coupling was introduced between flexural and axial vibration of delaminated segment of material. Free mode was first examined. By using an isotropic beam with splits and the classical beam model theory Euler-Bernoulli, Wang’s [5] calculations of natural frequencies were in line with experimental results. He proved Ramkumar’s [7] neglect of coupling effect was the reason for discrepancies between calculated and experimental data. In the same year, Mujumdar and Suryanarayan [8] used Euler-Bernoulli theory. In this study, as well as Wang, four joined isotropic beams were considered, and assuming coupling between bending and axial vibration exists. Wang [5] analyzed “free mode vibration” and Mujumdar and Suryanarayan [8] focused their study on “constrained” mode.

In 1992, Yin and Jane [31] and Chen and Goggin [32] researched vibrations of a delaminated beam with respect to buckled reference states using classical and beam theory shear deformable beam theory. They assumed delamination was closed during motion. In the same year, the research continued with Shen and Grady [38] who developed an analytical model which can predict vibration frequencies and modes of delaminated beams.

In 1994, Han, Zhang and Lu [9] presented finite element models using the Timoshenko beam theory. They showed results from their model by finite element are with excellent agreement with other solutions, and Shu and Fan [10] studied constrained mode models for biomaterial beams in 1995.

Lestari and Hangagud [30] in 1999 researched multiple delamination of a single beam using classical beam theory. In their study they simulated the open and closed behaviour between delaminated layers. In 2001, Krawczuk [29] investigated natural frequencies of cantilever beams with a delamination incorporating formulation method based on beam and column model.

Since 1995, Della and Shu [6, 11, 13, 23 and 24] have analyzed both "free mode" and "constrained mode" of delaminated multilayer beams. They mathematically proved that slenderness ratio has an influence on vibration behaviour of the beam. In their study, normalized axial stiffness and normalized bending stiffness were introduced to give a better indication of vibration behaviour of delaminated beams. In that study, the coupling between transverse and longitudinal vibration in an inhomogeneous beam was formulated and investigated. This progression continued when both free mode and constrained mode were investigated in the same research, introducing slenderness ratio and normalizing axial and stiffness and bending stiffness.

Y. Zou, L. Tong and G. P. Steven [2] focused on the model-based delamination detection methods for composite structures using vibrations. In this study finite element using formulation analysis was utilized.

In 2011, Erdelyi and Hashemi [4] studied finite element modeling for doubly delaminated composite beam. In that work, the laminated beam was considered to have double equal length delamination in some point along its span. He developed a finite element solution using Galerkin method. This method consisted of developing two separate element schemes and testing for sensitivity of each element. Erdelyi and Hashemi [4 and 26] also proposed a Dynamic Stiffness Matrix (DSM) model for free vibration analysis of doubly delaminated layered beams. In their paper, DSM was derived from general solution to the governing differential equation of motion and element's mass and inertia properties enclosed in a single matrix. Their FEM and DSM results showed excellent agreement with those available in the literatures.

With recent advancements in simulation software, the FEM modelling of defective structures has become easier and more popular. However, to the author's best knowledge, up to the present time, only limited FEM-based studies on fundamental frequency of fully clamped composite layered plate [34 and 35], composites beams using Timoshenko beam theory [39], and 1D beam model have been reported [25].

Johan Bellecave [25], using ANSYS® simulation software, confirmed the results obtained by Shu and Della [6, 10, 11 and 23], through 1D simulation. Bellecave [25] investigated only clamped-clamped, homogeneous beam with single delamination.

Obviously, 1D simulation was a great start to use ANSYS® software, but for practical purposes 2D and 3D simulations need to be investigated. The present study, not only expands the work accomplished by Bellecave, but also delves into 2D and 3D simulations for different boundary conditions. The cantilever and clamped-clamped beam configurations and data reported by Della & Shu [6, 10, 13 and 23] were investigated in this research. For both clamped-clamped and cantilevered boundary conditions, single and double delaminations are investigated. Further, to investigate the effect of delamination location on the natural frequencies of defective layered beams, both centred and off-centred delaminations are studied. These simulations are carried out in 2D. The comparison is made between the frequency data resulting from the FEM modeling and those available in the literature to show the effect of delamination size, type, and location, on the system frequencies.

Last but not least, clamped-clamped beam is also simulated in 3D for single delamination. The limitation of computer capabilities prevents the 3D simulation of double delamination configurations.

In the following chapter, for the convenience of the reader, a brief discussion on the development of analytical formulation and results, as reported by other researchers is first presented. A brief discussion is also provided regarding the effects that size, type and location of delaminations have on natural frequencies. Chapter 3 presents, a comprehensive structure of this report, all 2D and 3D simulations including a comparison with other literatures. These comparisons include the validity and proof of how and to what extent size, type and location of delaminations impact on natural frequencies in beam structure. Concluding remarks and future works are presented in Chapter 4. Finally, the report ends with Appendices A, B, and C, presenting, respectively, additional information about element and analysis types in ANSYS®, applied ANSYS® codes in this research, and additional supporting simulation illustrations.

Chapter 2: Theory

===

2.1 Vibration

Vibration is a mechanical condition where oscillation develops around an equilibrium point. These oscillations can be either periodic like a pendulum or random such as the movement of a tire on a road [1].

Vibration can be desirable like musical instruments or speakers in a sound system, or more often it is undesirable. The vibrations of engines, electrical motors are unwanted and basically it is a sign of energy being wasted.

2.1.1 Types of vibration [1]

- Free vibration

  This condition occurs when an initial input sets off a mechanical system and then the system is allowed to vibrate freely. An example of this type of vibration is when a string on a guitar is pulled and then let go to vibrate. In this type of vibration, the mechanical system vibrates at one or more of its natural frequency and then damp down to zero.
- Forced vibration

  When an alternate force is applied to a mechanical system it is called forced vibration. Earthquake is a good example of this type of vibration. In this type of vibration the frequency of vibration is the frequency of force applied. The order of magnitude of this vibration is dependent on the actual mechanical system.

The frequency at which a system vibrates naturally after it has been set in motion is called natural frequency. Therefore, if there is no interference, the number of times a system oscillates between its original position and the position it moved to after it was set into motion is the natural frequency of that system. For example, if a beam that is fixed on one end and a weight attached to its other end, (Figure 3), is pulled and then released, the beam oscillates at its natural frequency.

In this example, the natural frequency can be calculated using this formula:

$$f=1/2\pi\sqrt{(k/m)}$$

Where m is mass weight at the tip and k is the beam stiffness in lbs/in, or N/m.

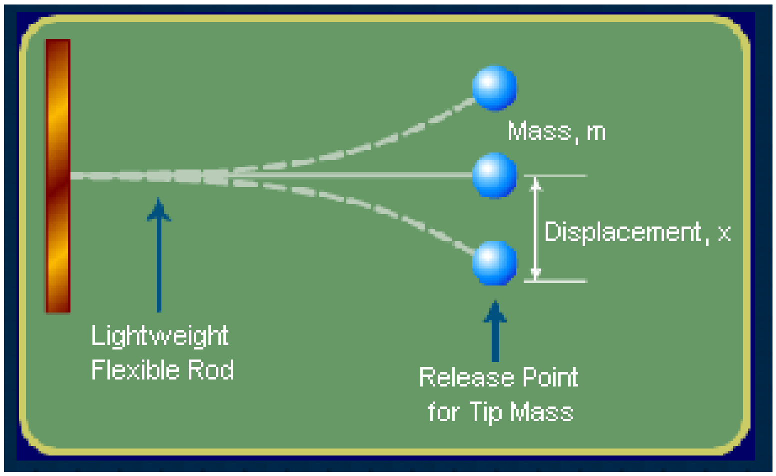


Figure 3- Natural Frequency, adapted from [16]

The study of natural frequencies and modes of free vibration of a system can be accomplished by one of the many well-established methods, namely Analytical and semi-analytical Methods, in-house codes based on (Numerical) Finite element Method (FEM) formulations, Software-based FEM Simulations, and Experimental Methods, etc.

In what follows, the application of analytical methods to the free vibration analysis of delaminated beams, as reported in the literatures [6 and 24], is first briefly discussed. The main focus of the present study is the application of the educational version of FEM-based commercial ANSYS® software to the free vibration analysis of various layered beam configurations with delamination(s).

2.2 Analytical Method

To narrow down the field for analytical method, the focus of this research will be on single and double delaminations.

2.2.1 Single delamination

Single delamination was studied by previous scholars in two conditions. First, free mode and also constrained mode. In this research, free mode is primarily considered. To explain analytical method in single delamination, and how it is determined, the work done by previous researchers will be illustrated. They reported the analytical solutions for “free mode”.

In their research, the beam has three spanwise origins: two integral and one delamination. The assumption was that the beam is divided into 4 segments.

Free mode

All four beams are treated as Euler–Bernoulli beams, therefore, the present solution is valid as long as L_i>>H_i , i=4, Figures 4, 5, 6, and 7 [6].

For simplification of the analysis, the following assumptions are considered:

(a) the delamination pre-exists;

(b) The two layers of the beam are linear elastic;

(c) the effect of coupling due to the difference in the material properties of the two layers is neglected; and

(d) the two layers are identical in densities, which is relevant only for the ‘free mode’ model

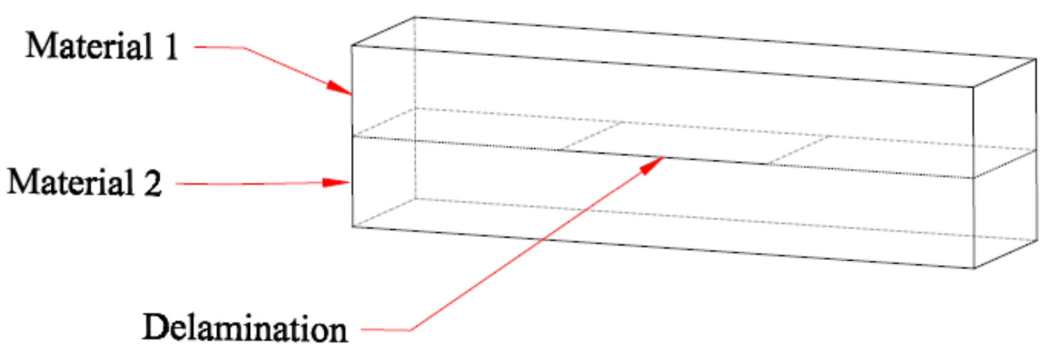


Figure 4- A beam consisting of two distinctive layers is delaminated along the interface

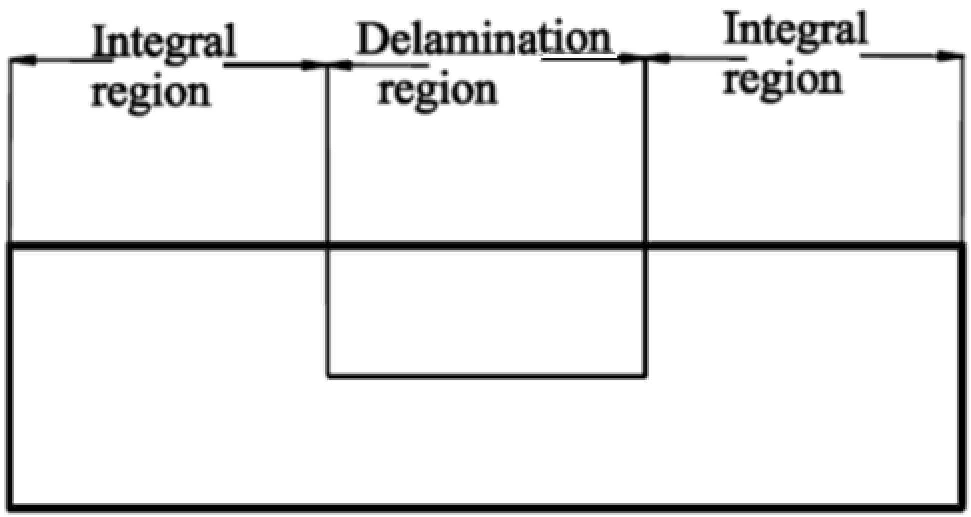


Figure 5- Front view of delaminated beam; showing three different origins

The delamination occurs between point $x=x_2$ to $x=x_3$ (Figure 6). The assumption is: there is no shear and axial deformation and also torsion is ignored.

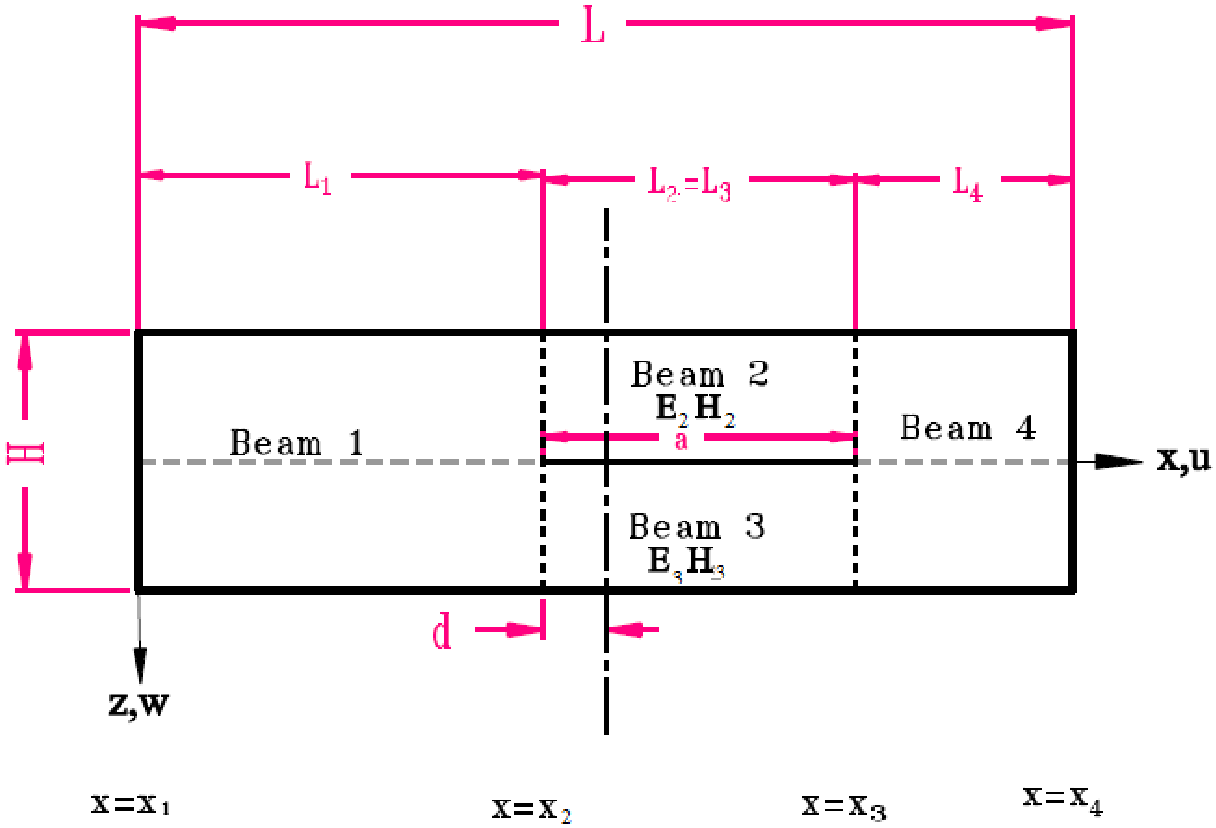


Figure 6- Beams 2 and 3 are connected at their ends to the integral beams 1 and 4

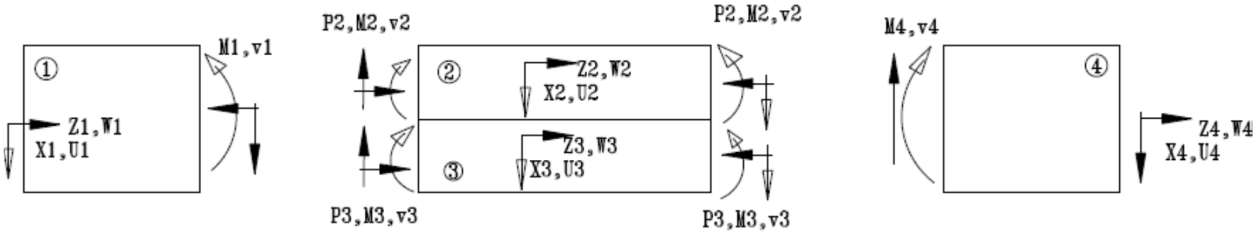


Figure 7- Diagram of forces and bending moment of the beam

The governing equation for free mode is:

$$EI_i \frac{d^4 w_i(x)}{dx^4} + \rho_i A_i \frac{d^2 w_i(x)}{dt^2} = 0 \tag{1}$$

For beam 1, we have [6 and 24]:

$$\left(\frac{E_2 H_2^3}{12} + \frac{E_3 H_3^3}{12} + \frac{E_2 H_2 H_3^2 + E_3 H_3 H_2^2}{4}\right) \times \frac{\partial^4 w_1}{\partial x^4} + (\rho_2 H_2 + \rho_3 H_3) \frac{\partial^2 w_1}{\partial t^2} = 0 \tag{2}$$

Substituting $w_i(x,t) = Wi(x)\sin(\omega t)$ in above formula [6 and 24]:

$$\left(\frac{E_2 H_2^3}{12} + \frac{E_3 H_3^3}{12} + \frac{E_2 H_2 H_3^2 + E_3 H_3 H_2^2}{4}\right) \frac{\partial^4 w_1}{\partial x^4} - (\rho_2 H_2 + \rho_3 H_3)\omega^2 W_1 = 0 \tag{3}$$

For beam 2 and 3, the governing equations are as follow:

$$\frac{E_2 H_2^3}{12} \times \frac{\partial^4 w_2}{\partial x^4} - \rho_2 H_2 \omega^2 W_2 = 0 \tag{4}$$

$$\frac{E_3 H_3^3}{12} \times \frac{\partial^4 w_3}{\partial x^4} - \rho_3 H_3 \omega^2 W_3 = 0 \tag{5}$$

The general solutions for above equations (3) to (5) are,

$$W_i(x) = \left[C_i \cos\left(\lambda_i \frac{x}{L_i}\right) \quad + S_i \sin\left(\lambda_i \frac{x}{L_i}\right) \quad + CH_i \cosh\left(\lambda_i \frac{x}{L_i}\right) \quad + SH_i \sinh\left(\lambda_i \frac{x}{L_i}\right) z\right] \tag{6}$$

where

$$\lambda_i{}^4 = \frac{\omega^2 \rho A_i}{EI_i} L^4 \tag{7}$$

λ_i is the non-dimensional frequency, and C_i, S_i, CH and SH are unknown coefficients in Equation 6, which can be obtained from four boundary conditions and twelve continuity conditions.

2.2.2 Double delamination

Overall, there are two possibilities with double delamination. One is overlapping delamination which is also known as “non-enveloped”, and “enveloped” delaminate, in which damaged laminates have the same centre on beam (Figures 8 and 9).

When the beam vibrates, different layers can vibrate freely and have different transverse deformations or may be vibrated together in a constrained mode. In current research, only free mode is considered.

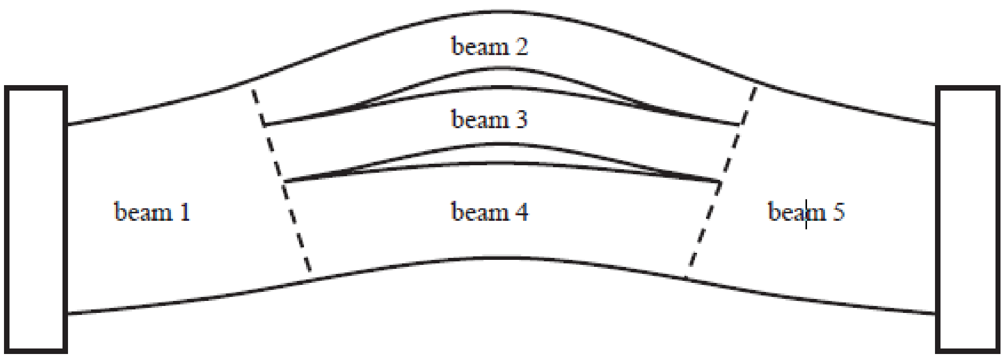


Figure 8- Free mode vibration on envelped delamination, adapted from [23]

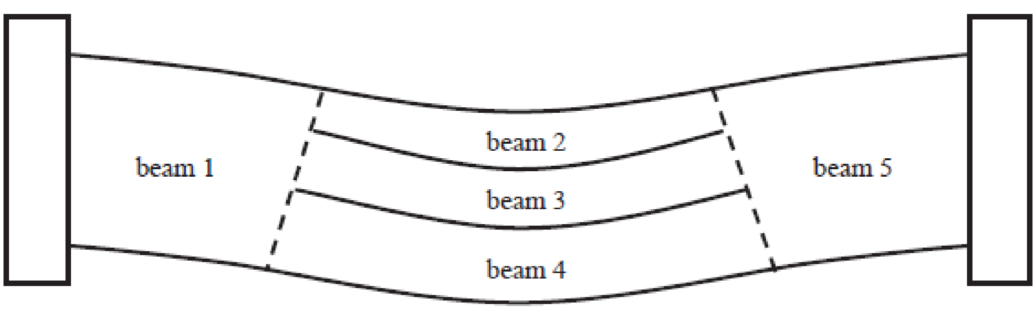


Figure 9- Constrained mode on envelped delamination, adapted from [23]

In this section a clamped-clamped homogeneous beam with double delamination is studied. In previous literatures, the beam was divided into four segments: two integral and two delaminations. Researchers considered the beam as having seven segments and they assumed sections interconnected, to form a single beam [6 and 24] (Figure 10).

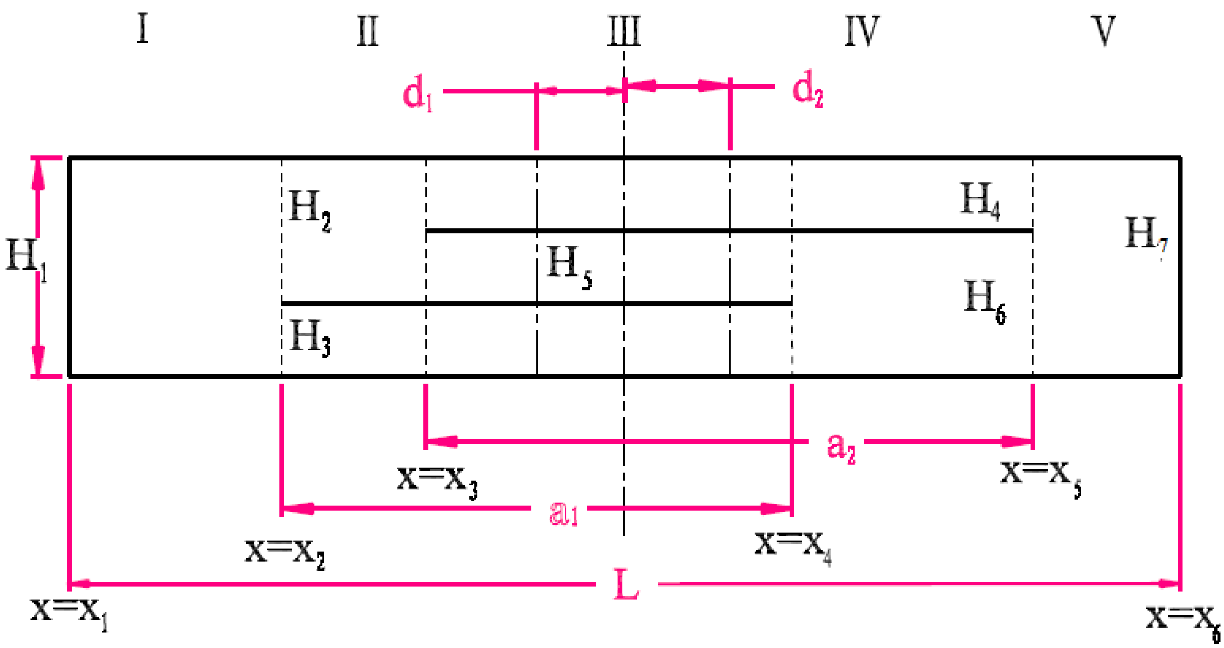


Figure 10 (a) – A model of a beam with two overlapping delamination, adapted from [24]

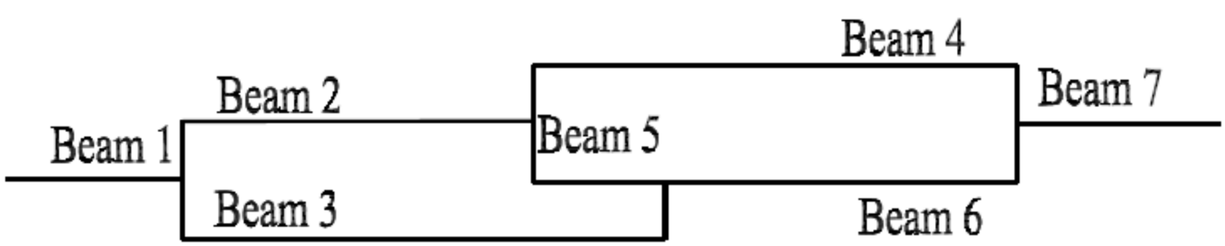


Figure 10 (b) – The delaminated beam is modeled by seven interconnected beams, adapted from [24]

Wang [5] studied a model where layers of delamination did not touch each other; Mujumdar and Suryanarayan [6] followed that study, but focused on layers touching each other. More recently, Della, Shu and Yapu Zhao [24] concentrated their efforts on rigid and soft connectors. Their finding was that the actual results are closer to the results of rigid connector (Figure 11).

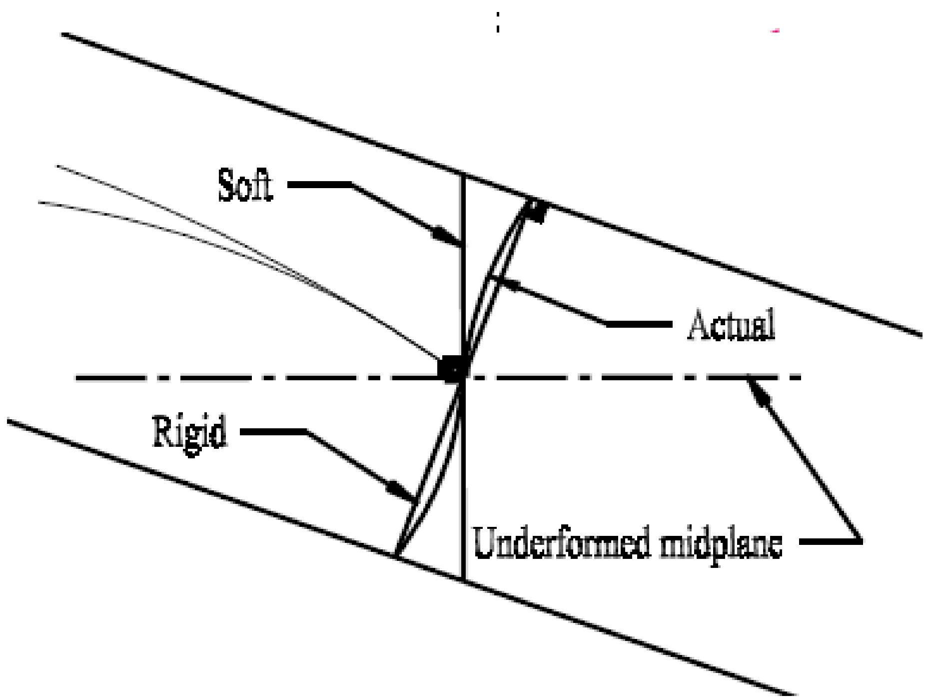


Figure 11- Graph of the actual, soft and rigid connectors on the deformed beam, adapted from [24]

To avoid stating repetitive formulations, the formulations for single delamination can also be applied for double delamination except, in this case, the number of unknown coefficients is 28, which can be determined by appropriate boundary and continuity conditions.

2.3 The Finite Element Method

The Finite Element Method (FEM) is a versatile numerical method frequently used to solve engineering problems. FEM utilizes the stress/strain equation for an element of whole structure. In this method a mechanical structure is broken down to a large number of substructures known as element. The Finite Element Method considers a number of compatible elements connected to each other. Using this technique any complex structure can be modelled as an assembly of simpler structures. The greatest advantage this method has over other methods is its generality with which natural frequencies and mode shapes can be determined. The challenge for this method is that it requires a large (memory) computer to be sensitive enough to achieve numerical output.

The global nodal forces "F" and the global nodal displacements "d" are related through use of the global stiffness matrix "K" by

$$\{F\} = [K]\,\{d\}$$

The above equation is called the global stiffness equation. It is the basic equation formulated in the stiffness or displacement method of analysis.

General Steps of the Finite Element Method (FEM) [37]

In this section general steps that are included in finite element method formulation, and solution to an engineering problem are outlined. These steps are used as guides in developing solutions for structural and non-structural problems.

Step 1: Select the Element Types: In this step body of a structure is divided into an equivalent system of finite elements and their nodes. The element that mostly resembles actual physical behaviour is chosen as a model.

Step 2: Select a Displacement Function

In this step a displacement function of each element is chosen. This function is represented within the element using its nodal values. Some of the examples of these functions are linear, quadratic and cubic polynomials.

Step3: Define the strain/Displacement and Stress/Strain Relationship

Stress/strain and strain/displacement relationship are important to develop equations for each finite element. For instance, in one-dimensional deformation of x, strain ε_x related to displacement u by:

$$\varepsilon_x = du/dx$$

The stress must be in relation with strain through a law called the constitutive law. Obviously, accurate results stem from the ability to determine the material behaviour. Hooke's law is often the governing law for stress analysis, and is:

$$\sigma_x = E\varepsilon_x$$

Step 4: Derive the Element Stiffness/Mass Matrix and Equations

The matrices that govern element stiffness and element equations were originally based on the concept of stiffness influence coefficients. This required a background in structural analysis. There are three methods to determine the stiffness matrix. They are Direct, Variational, and Weighted Residual Methods. In direct equilibrium method, the element stiffness matrix and element equations that relate nodal forces to nodal displacements are derived by using force equilibrium conditions for a basic element, in conjunction with force/deformation relationship.

The mass matrix of the system, representing the inertia effects in the dynamic analysis, is also derived and included in the element equations.

Step5: Assemble the Element Equations to Obtain the Global or Total Equations and Introduce Boundary Conditions (B.C.)

This step involves element nodal equilibrium equations that were developed in last step and implying them into global nodal equilibrium equations.

Step 6: Solve for the unknown Degrees of Freedom (DOF)

For a simple static analysis, equation below takes into account the boundary conditions and it is written in expanded form:

$$\begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \dots & K_{1n} \\ K_{21} & K_{22} & \dots & K_{2n} \\ \vdots & & & \vdots \\ K_{n1} & K_{n2} & \dots & K_{nn} \end{bmatrix} \begin{Bmatrix} d_1 \\ d_2 \\ \vdots \\ d_n \end{Bmatrix}$$

where 'n' is total number of unknown nodal degrees of freedom in structure.

These equations can be solved for displacement matrixes (d_1 to d_n) by using eliminating methods available.ds are the primary unknowns; because they are the first values calculated using the stiffness finite element method.

Step 7: Solve for the Element Strains and Stresses

Secondary quantities of strain and stress which are important can be obtained. These can be directly expressed as displacements determined in previous section.

Step 8: Interpret the Results

Obviously, the main objective is to interpret and analyze the results. The results are important in determining the location and size of delamination in a structure. These determinations are helpful in design and analysis decisions that have to be made.

When a beam is considered as a mechanical structure, as it is in this research, it is considered to be divided by imaginary lines into a series of elements, and these elements are connected at the intersections of these imaginary lines. These intersections are known as nodes. Any stress and strain at each element would be defined as the stress and strain displacements and forces at nodes. From these displacements, equations are developed individually for nodes and a series of these equations would be needed to cover the whole system (beam). To determine the stresses, strains and natural frequencies of the system, these equations must be solved simultaneously (i.e., assembly + application of the B.C.).

In dynamic/vibration modeling, the mass/inertia effects are also included in the dynamic equilibrium equation of the system. For free vibration analysis, finite element method leads to a linear eigenvalue problem,

$$([K]-\omega^2 [M])\{d\} = \{0\}$$

This is then solved to find natural frequencies (eigenvalues) and modes (eigenvectors) of the system. FEM displays great accuracy in lower modes of vibration, and the precision increases as the number of elements increases [26].

2.4 Simulation Method

In recent years, software technology has made significant progress, and with this advancement, simulation software has improved noticeably. One of the capable software that has been developed is ANSYS®. ANSYS® software, like in any other methods of problem solving, the information must be given and the problem is characterized. All conditions and properties are set, and the analysis and solutions are developed.

These solutions are usually examined by comparing them with baseline results. These baseline results are obtained by traditional analytical methods.

In order to get familiar with ANSYS® release 12.0 some of the characteristics are discussed here. The interface in this release is called “Mechanical APDL application”. This application provides a combination of power and user friendly software. This would translate into more flexibility options for users.

With this software, Design Model is developed to enable the user to edit geometry. Design Model is a tool to draw 2D sketches, upload 3D CAD models, and model 3D parts. Whether an intermediate or an experienced user, this software can offer the functionality and power that either user desires.

In summary, certain recommendations would be given to enable other researchers to continue the analysis for other related subjects. Each solution presented in ANSYS® serves as a template to be used as a model to solve similar types of simulations.

2.4.1 The Mechanical Application Approach

Mechanical applications can be used to carry out different types of structural, electromagnetic and thermal analyses. Using this application, model's environment and boundary conditions are defined, the problem is solved, and then results are visualized. In what follows, an overall understanding of the workflow logic involved in any mechanical application is briefly discussed. Overall steps to use the Mechanical Application are further explained in Appendix A.

These logical steps start with (1) *Creating Analysis System*, and continues with (2) *Defining Engineering Data*, (3) *Attaching Geometry*, (4) *Defining Part Behavior*, and finally is completed with (4) *Stiffness Behavior*.

In ANSYS® there are limited tools to create geometry. Therefore, it is a favourable to create geometry in CAD software and then import it to ANSYS®. Normally, the model will be automatically meshed at solving time, but the option also exists to mesh it before that. Determination of element size and type is dependent upon a number of factors such as, overall model size, complexity of structure, and body curvature. There are almost 200 different types of element in ANSYS® library (see Table A1, Appendix A). Selecting an element type from the large library of elements in ANSYS® could be intimidating to the user; however, this selection can be narrowed down by knowing the type of analysis, whether it is linear or nonlinear, and if it is for structural, modal or electrical analysis, etc. Elements are categorized into groups with similar characteristics such as: Shell, Plane, Solid, etc. Each element has two sections: Group type + number. The older version of elements have a lower number than the newest version. LINK1, BEAM3, SHELL181, SOLID187, etc. are the most modern in the element library. It is interesting to know that some simulation software such as NASTRAN® have a general type of element that is used for all types of analyses.

The strain on nodes can be linearly varied from one to another in linear elements because there is no mid-side node. These types of linear elements in ANSYS® are called "fully integrated", which means they react like semi-quadratic element, but not completely. In general, the simulation time for a linear element is shorter than a quadratic element which contains mid-side node. To choose an element some factors need to be considered:

- Quadratic elements function much better than linear elements for in-plane bending type problems. Even turning on several linear elements cannot be as good as quadratic elements.

- Linear elements tend to have more errors to non-perfect element shapes. Linear elements perform better in a mapped mesh as oppose to a free mesh.
- Contrary to in-plane bending, linear and quadratic elements perform the same in out-of-plane, plate type bending.
- If the elements are regularly shaped, linear bricks display good results. If the volume has a complex shape, which results in skewed elements, meshing should be done with 20 node bricks or tetrahedral elements. Because linear triangles are constant-strain elements they give poor results. In general, one quadratic element having three nodes is more accurate than two linear elements with three nodes along the same dimension.
- A finer mesh of linear elements, in non-linear analysis, is more accurate and stable than a rougher quadratic mesh with same number of nodes.

2D Solid Elements:

There may be confusion for ANSYS® users in the beginning asking, how 2D element can be a "solid". The answer is: By modeling a cross section in ANSYS®, 2D element can behave like solid part. In general, 2D solid elements are used in three types of analysis:

- Plane Strain*:* When a subject for modeling is thin sheet or very long with small thickness, 2D solid element provides the same result, which can be established by shell model. Stresses in the thickness direction are significant.
- Plane Stress: A plane stress analysis is a 2D model of a thin sheet with in-plane loading. Axisymmetric: In this type of analysis, we are modeling a cross section of a structure that is fully defined by rotating the cross section around a central axis. A pressure vessel or bottle would be examples of this kind of structure. The thickness direction here represents the circumferential (hoop) direction and those out-of-plane stresses are very significant (picture a tank under pressure).
- Axisymmetric*: when* a cross section of a structure is 360° rotated around central axis, this type of analysis should be considered because the thickness direction presents the circumferential direction, and out-of-plane stresses are important to be considered.

Considering the analysis made here is assumed to be linear, and because the ANSYS® package is educational, with the help of following chart, PLANE 42, PLANE 182, and SOLID 45 were selected for this research. For instance, Table 1 provides the most commonly used element types for stress analysis.

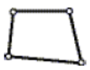
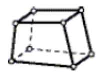
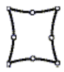
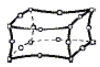
Element Order	2D Solid	3D Solid	3D Shell	Line Elements
Linear	 PLANE42 PLANE182	 SOLID45 SOLID185	 SHELL63 SHELL181	 BEAM3/44  BEAM188
Quadratic	 PLANE82/183  PLANE2	 SOLID95/186  SOLID92/187	 SHELL93	 BEAM189

Table 1- Samples of element types in ANSYS®, adapted from [27]

PLANE42: It is a basic 2D solid element (Figure12). This element can be used in Plane stress or strain. As it is shown in following figure, it is defined by four nodes having two degrees of freedom at each node: UX and UY. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

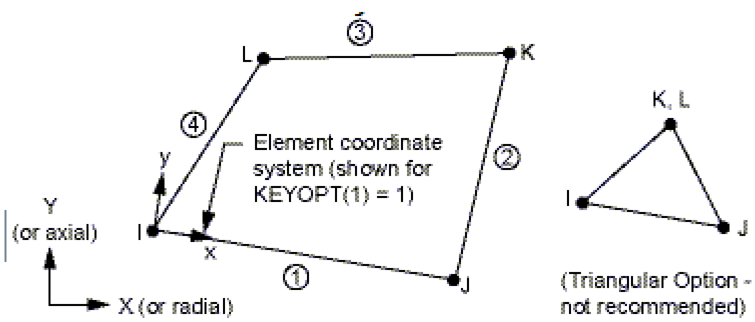


Figure 12- PLANE42 Geometry, adapted from [27]

PLANE42 Assumptions and Restrictions:

- The area of the element cannot be zero.
- This element is only applicable in XY plane as shown in Figure 12
- If K and L node are duplicated, a triangular element may be used in meshing
- For purpose of having a constant strain element result, any extra shape triangular elements is automatically deleted

PLANE42 Product Restrictions

- Plane 42 cannot be used in DAMP material
- KEYOPT(6) = 3 cannot be applied

PLANE82: 8 nodes version of PLANE42. A triangular is an option in this type of element.

PLANE182: The new version of PLANE42, which can be used for nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

3D Solid Elements:

These types of elements have their geometry fully defined by the element nodes. They are elements that are used to mesh volumes in ANSYS®. These volumes could be created in the ANSYS® preprocessor or imported from a CAD system. Hexahedral elements (bricks) can be used to mesh regularly shaped rectangular type volumes, and tetrahedral elements (tets) can be used to mesh any volume. Even linear bricks, if not distorted, can model a thin plate in out-of-plane bending with one element through the thickness since they have "extra shapes" turned on.

SOLID45: Basic Linear Brick. This element is used for the 3-D modeling. As it is shown in Figure 13, this element is identified by 8 nodes and each node has three degrees of freedom in UX, UY and UZ.

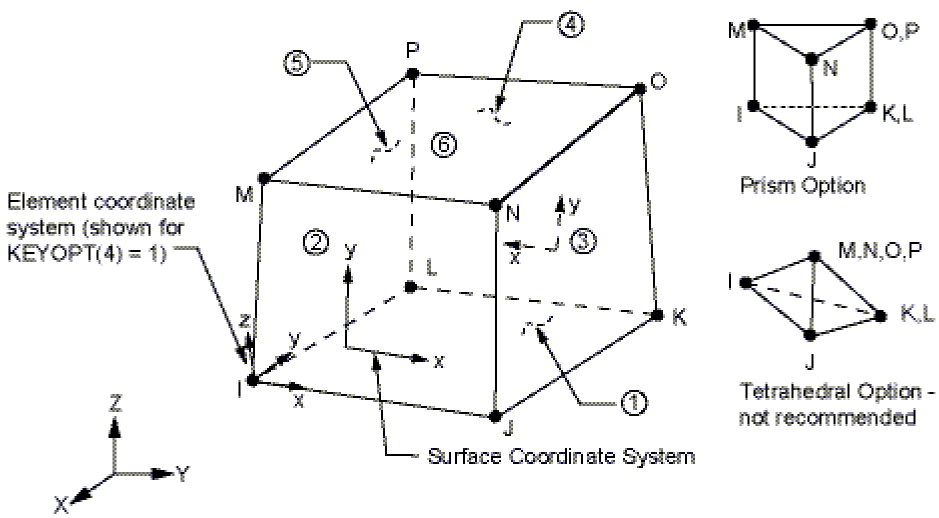


Figure 13- SOLID45 Geometry, adapted from [27]

SOLID45 has the capabilities of plasticity, creep, swelling, stress stiffening, large deflection, and large strain. An option of reduced integration with hourglass control is available.

SOLID45 Assumptions and Restrictions

- Zero volume elements are not allowed.

- Elements may be numbered either as shown in Figure 13: SOLID45 Geometry or may have the planes IJKL and MNOP interchanged.
- The element should not be twisted such that the element has two separate volumes. This occurs often when the elements are not numbered properly.
- All elements must have eight nodes.
- A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see Triangle, Prism, and Tetrahedral Elements).
- A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

SOLID45 Product Restrictions

- When used in the products listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.
- The DAMP material property is not allowed.
- The only special feature allowed is stress stiffening.
- KEYOPT (6) = 3 is not applicable.

SOLID95, SOLID185, SOLID186, SOLID92, and SOLID187 are other available solid elements in ANSYS®. A brief explanation is provided in Appendix A.

3D Shell Elements:

A shell element is simply a surface type element. It is a 2D element that is called 3D because it is not restricted to the XY plane like a 2D solid element; it can be located anywhere in three-dimensional space and it can be damaged out-of-plane. Shell elements are engineering "abstractions", because a geometric surface has no physical thickness. An ANSYS® real constant is used to give a theoretical thickness to a shell element. Shell elements can also be called plate elements, and are utilized to model panel type structures where the thickness is smaller than the other dimensions of the part. They are capable of carrying in-plane loads (also called membrane loads) and also out-of-plane bending moments and twisting. More information about SHELL63, SHELL93 and SHELL181 are presented in Appendix A.

Beam Elements:

Line elements are segments of shells and they are better known as a simple line, which exemplifies a long slender structure that can carry axial, bending, shear and twisting forces. Real constants describe the cross sectional properties of the beam element. Different types of beam elements and their applications are further explained in Appendix A.

2.4.2 Analysis Types

Several types of analyses in the Mechanical application can be performed using pre-configured analysis systems (see Create Analysis System). For doing more advanced analysis, Commands objects in the Mechanical interface can be used. This will allow the user to enter the Mechanical APDL application commands in the Mechanical application to perform the analysis.

There are various analysis types that can be performed in the Mechanical interface. For furether info, please find Depending on how experienced a user is, the results and features could vary. Due to the application of Modal analysis in this research, and the extensive usage of it throughout this thesis, a brief review of this analysis will be discussed below.

Modal Analysis in ANSYS®

Modal Analysis is a tool used to determine vibration characteristics or natural frequencies of a mechanical structure. It can also be used for dynamic analysis, harmonic response, and transient dynamic analysis. Modal analysis in ANSYS® is linear analysis. In this research natural frequencies and mode shapes are concentrated upon.

Process Involved in a Modal Analysis

The overall process for modal analysis includes the following steps:

1. Building the Model for a Modal Analysis

   To build a model for modal analysis, the following conditions must be considered: only linear behaviour is accepted and if non-linear elements are specified, ANSYS® would ignore the specification and treats it as linear.

2. Mesh Controls Overview

   As mentioned before, when Mechanical application is being used, a part or structure is automatically meshed at solve time. If it is desired, meshing can be done before solve time. To achieve this feat, select mesh from Tree Outline, right click on that, and then select Generate Mesh. In addition, mesh controls assist the user to refine and fine tune the mesh.

The element size is determined by the size of the bounding box, body curvature and complexity of the structure.

3. Apply loads and obtain the solution

   There are several mode-extraction methods that can be selected in ANSYS®. These include: Block Lanczos, Supernode, PCG Lanczos, reduced, unsymmetric, damped, and QR damped. Damping in the structure can be accomplished by the damped and QR damped methods. The QR Damped method also allows for unsymmetrical damping and stiffness matrices.

4. Visualize/Review the results

   Results from a modal analysis are written to a structural result file called, Jobname.RST. Results could include:

   Natural frequencies

   Expanded mode shapes

   Relative stress and force distributions (if requested).

   The results can be reviewed in POST1 [/POST1], the general postprocessor.

Chapter 3: ANSYS® Models

3.1 Introduction

With aid of ANSYS® simulation, the size, location and type of delamination in a beam structure can be varied to study their effects on, and the changes in, the system natural frequencies. As will be shown later in this chapter, in general, the larger the delamination, the lower the natural frequency. In addition, the location of the delamination also affects the natural frequencies of the defective system. Furthermore, if beam structure has more than one delamination, then natural frequencies decrease even more. As also reported by other researchers, the extent of defcet is also an important factor; after a certain delamination size the fundamental natural frequency decreases more sharply.

ANSYS®, as previously explained, is capable to do simulations in 1D, 2D and 3D environments. 1D FEM simulations are omitted here, as they have been previously investigated [25]. Therefore, this study concentrates its focus mostly on 2D and 3D simulations. These simulations include different frequencies and delamination conditions on two types of beams. These beams are further divided into different categories and each category is individually investigated.

Since this work covers a wide area of effects of delaminations and their effects on the natural frequencies of defective systems for a number of beam configurations, a flow chart is included for the reader convenience (Figure 14).

This is also to ease the usage of this research for future users and to have a better understanding of process involved in this research.

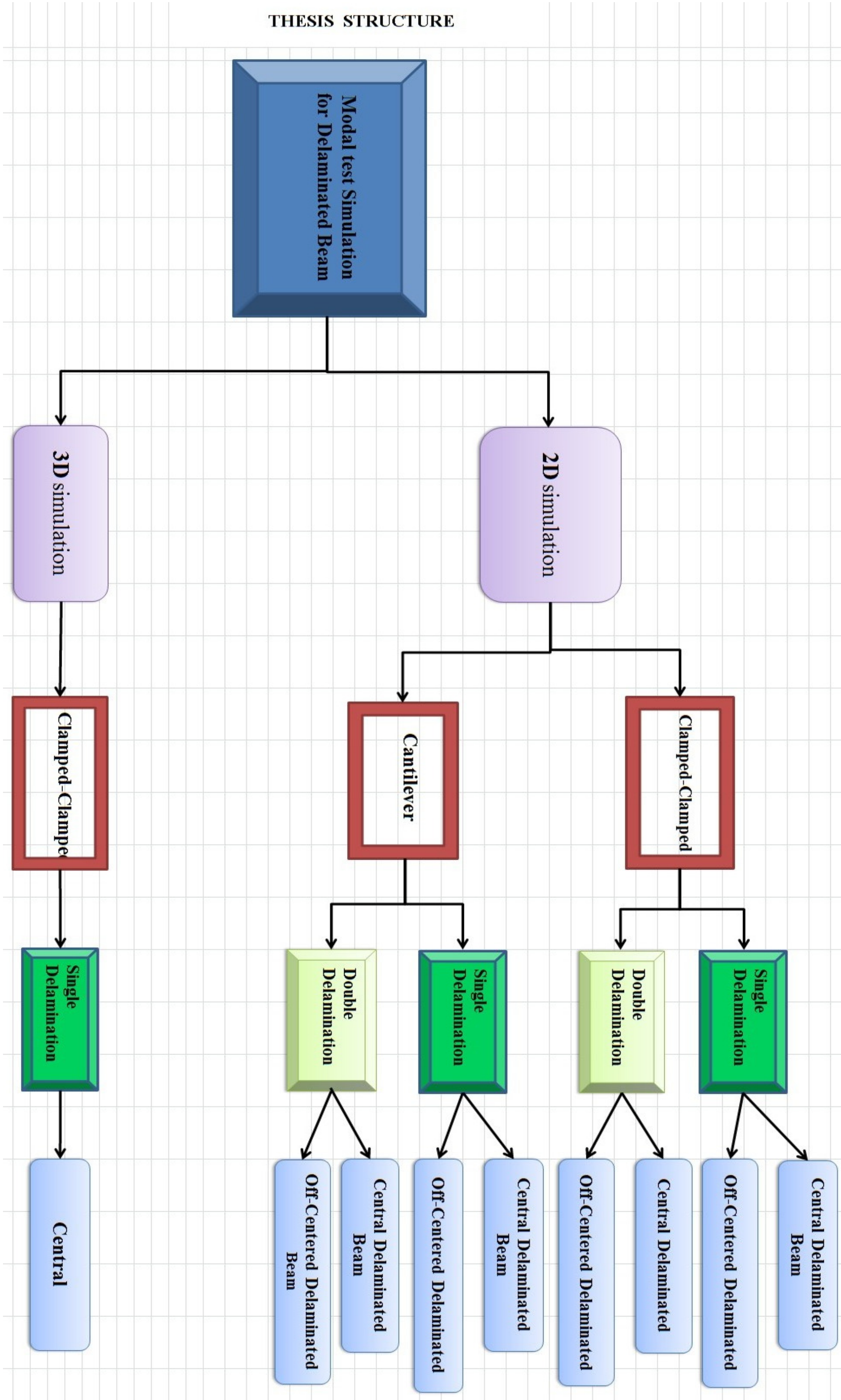


Figure 14- Flow chart of current thesis

3.2 Results and discussion

Contrary to other finite element software, ANSYS® provides the option to write a specific program or commands in Microsoft Word, Notepad or Word pad. This written Macro file can then be executed in ANSYS®.

To that end, several programs that are developed specifically for this research have been included in appendix A at the end of this research. In this program some of the parameters had to be included as assumptions which are as follow:

1) The material is homogeneous and isotropic
2) No coupling has been employed
3) Free mode only is considered
4) Rigid connectors are applied

With these assumptions, modal test for different cases is carried out.

In present study, two types of models are created: 2D and 3D.

3.2.1 2D Model

For the finite element modeling of layered beam, PLANE182-2D-4Node element was used. In this case study the effects of delamination on vibration characteristic of a beam is illustrated. For simplicity, a single delamination is considered and these results can be extended to multiple delaminations as long as there is no separation between the layers of delamination at the mid-plane. For 2D modelling, a Macro file has been created to simulate single delaminated clamped-clamped and cantilever beams, with consideration that delamination can happen on either center or off-center of the beam (Appendix A).

3.2.1.1 Case study 1 - Single delamination (Clamped-Clamped beam)

A clamped-clamped homogenous beam model is developed where delamination is located in mid plane and centered position (Figure 15). In modeling, two layers are stitched as shown by green color in Figure 15.

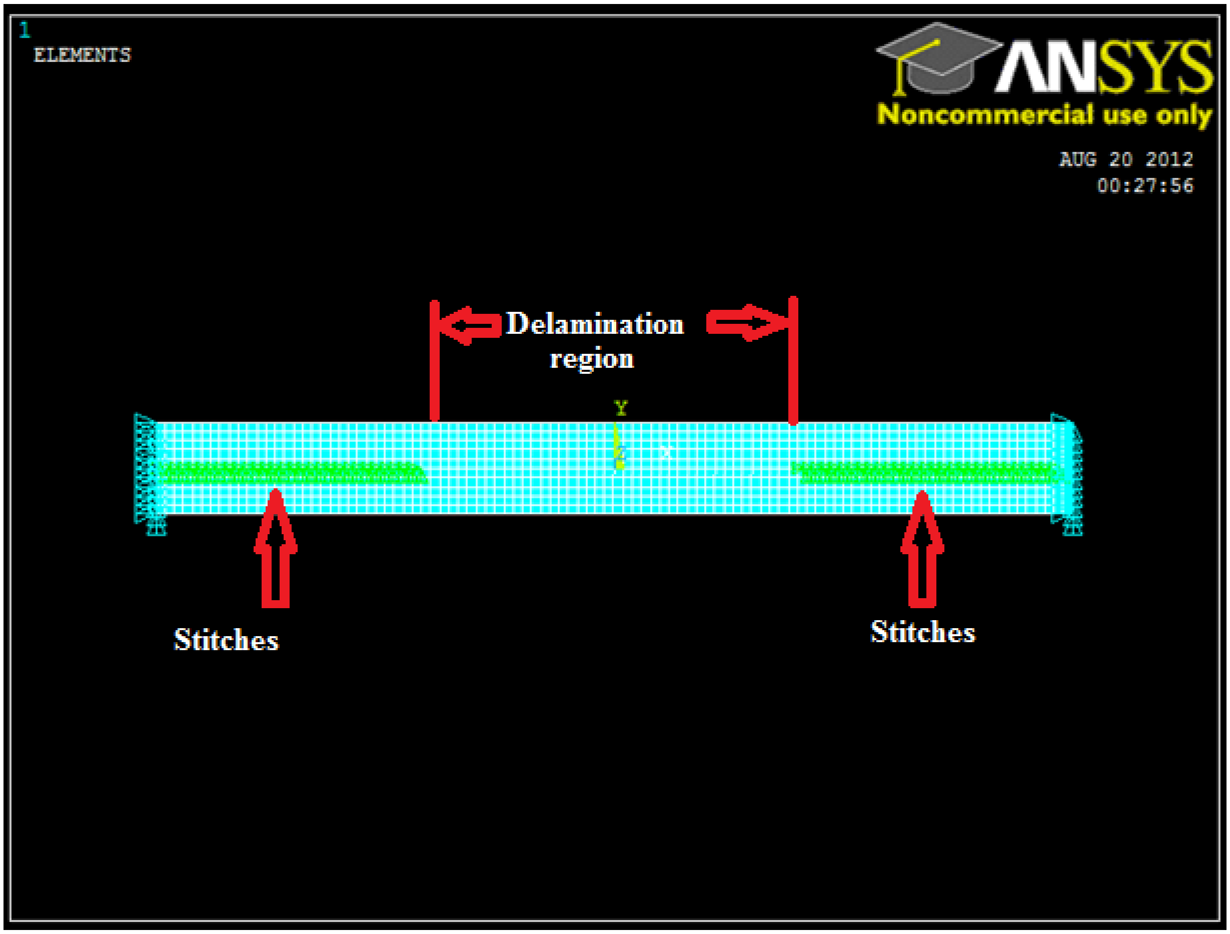


Figure 15- 2D clamped-clamped beam

The modal test was conducted with the following conditions:

Delamination length over Length of beam (a/L) = 0.40, Thickness of beam (H) =2x H_2 = 2x H_3 and homogenise E= E_2= E_3, UX=UY=0 for both ends. The element size is 0.1. The first set of modal test results are summarized in Table 2. These numbers are natural frequencies which were obtained for this beam from ANSYS®.

Mode No.	**Frequency** (Hz)	**Non-dimensional frequencies** $\lambda_i^{\ 2} = \omega L^2 \sqrt{\frac{\rho A_i}{EI_i}}$
1	0.27859	21.829
2	0.56021	43.896
3	0.84178	65.959

Table 2- First three modes of single centered delaminated clamped-clamped beam from ANSYS®

The corresponding mode shapes for above frequencies have been captured in Figures 16 to 18.

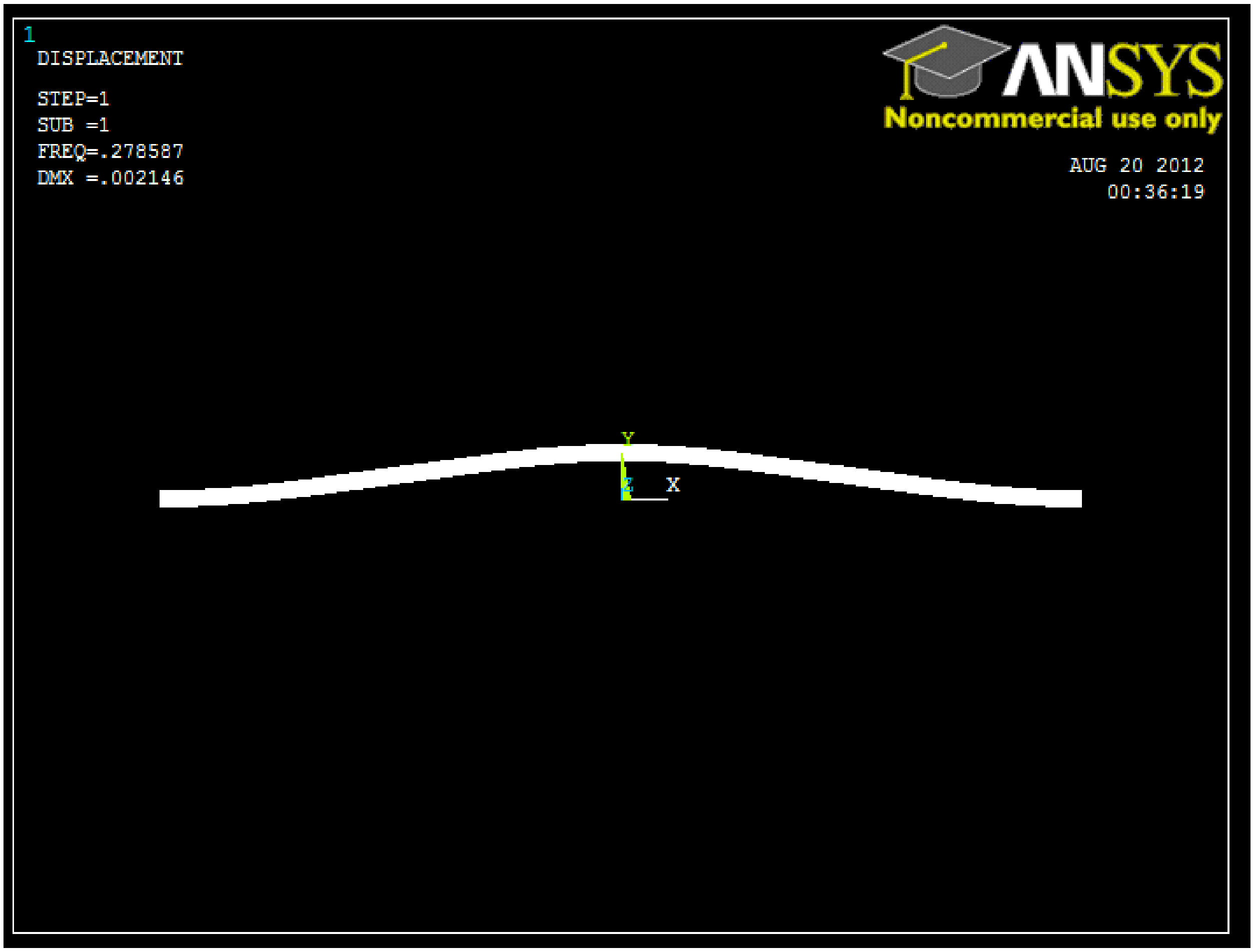


Figure 16- First mode of single centered delamination clamped-clamped beam

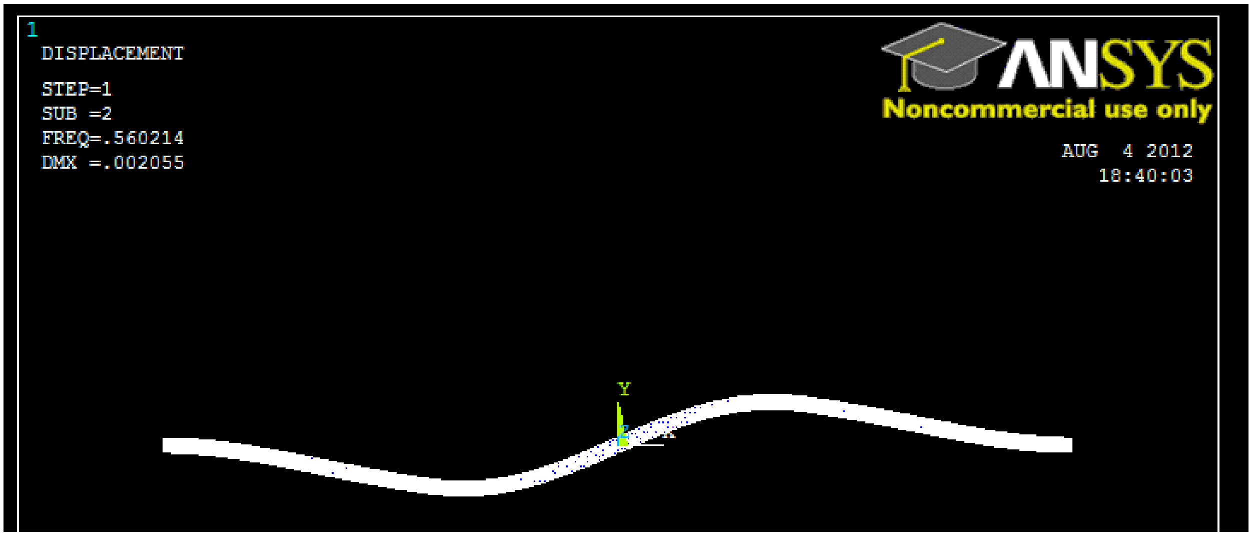


Figure 17- Second mode of single centered delamination clamped-clamped beam

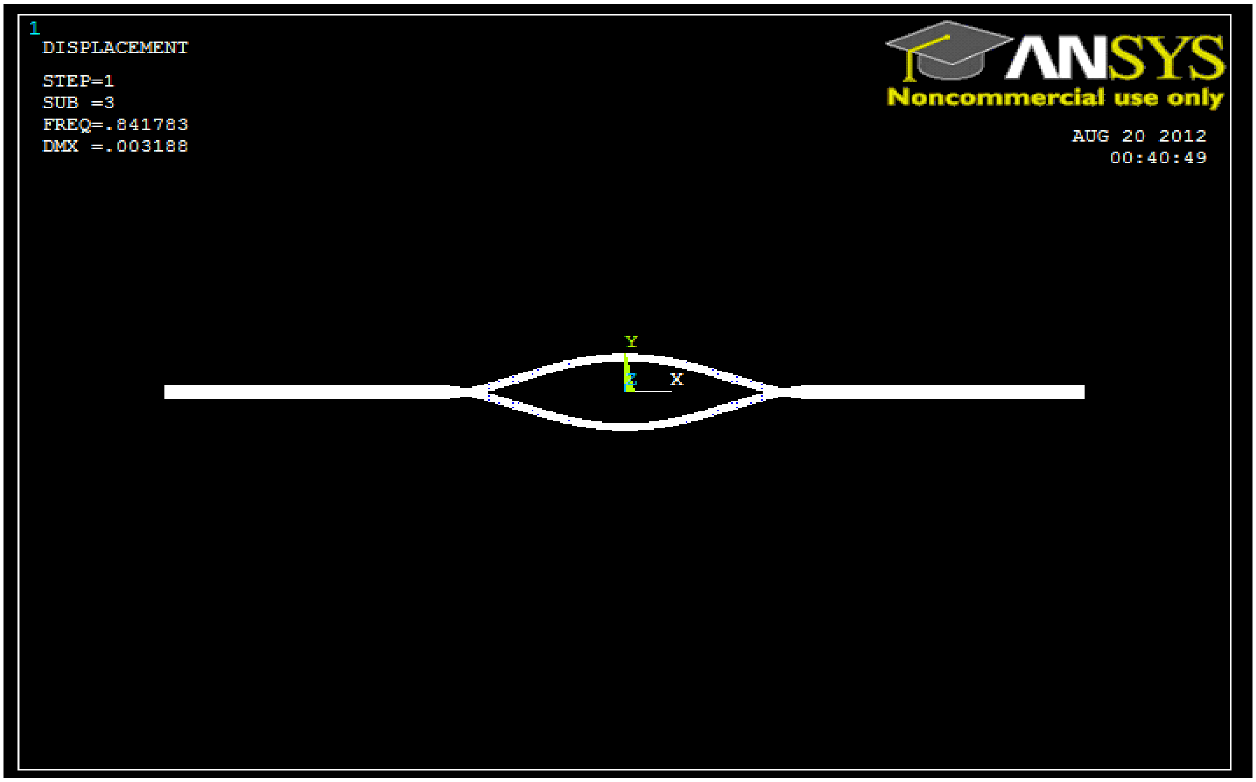


Figure 18 (a) - Third mode shape in interval 1 while upper and lower layers are twisting

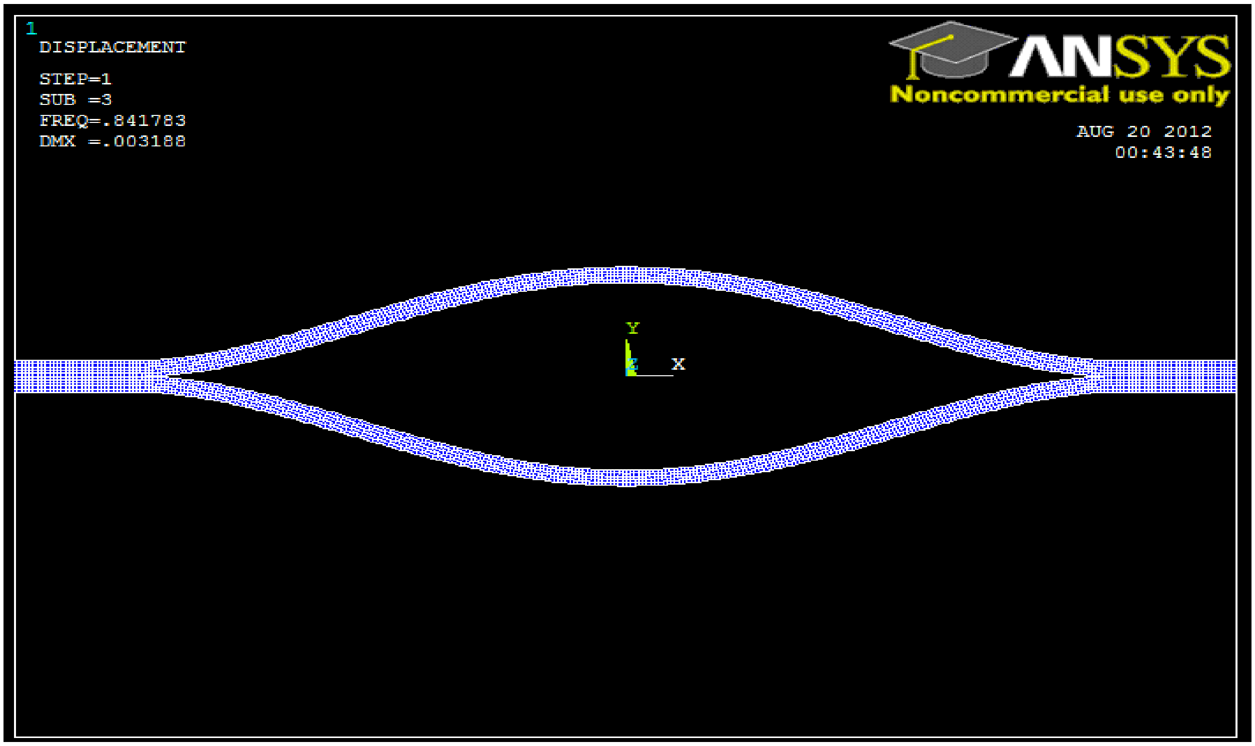


Figure 18 (b) - Third mode shape in interval 2 while upper and lower layers are opening

3.2.1.2 Result 1 - Single delamination (Clamped-clamped beam)

To further investigate the effects of delamination ratio of a/L on non-dimensional frequency of single centered delamination of a clamped-clamped beam, the aforementioned macro file was run several times, and in each case the length of delamination was varied from 0 to 0.6L.

The results obtained from this simulation are compared with those of Della and Shu [13] and Wang's [5] data to verify the presented ANSYS® model program (Table3). Comparing these results proves the fact that the length of delamination significantly impacts the non- dimensional frequency provided that a/L>0.4. However this impact lessens for the length values below 0.4.

	First Mode			Second mode		
a/L	Elem size 0.1	Della&Shu [13]	Wang [5]	Elem size 0.1	Della&Shu [13]	Wang [5]
0	22.396	22.37	22.39	61.588	60.76	61.67
0.1	22.395	22.37	22.37	60.624	60.76	60.76
0.2	22.375	22.36	22.36	55.700	55.97	55.97
0.3	22.250	22.24	22.24	48.829	49	49
0.4	21.829	21.83	21.83	43.896	43.87	43.87
0.5	20.866	20.89	20.88	41.649	41.45	41.45
0.6	19.260	19.3	19.29	41.213	40.93	40.93

Table 3- First and second non-dimensional frequencies comparison

The performance of present simulation is compared with developed analytical data by Wang [5], and Della-Shu [13], and errors for first and second modes are reported in Table 4.

First Mode			Second Mode		
Min. error	Max. Error	Ave. Error	Min. Error	Max. Error	Ave. Error
0.00%	-0.21%	0.09%	0.06%	1.36%	0.52%

Table 4- Result deviation between FEM model and reference data

The low percentage of errors displayed in Table 4, reiterates the fact that the simulation program in ANSYS® is functioning properly and is giving acceptable results. A better indication of the comparison made above between simulation data and analytical data, can be observed in Figures 19 and 20. The former, presents comparison between the ANSYS® results and those from Della-Shu [13] (1^{st} mode). The latter compares the ANSYS® results to those presented by Wang [5]

(2^{nd} mode). Figure 20, compares the results that were obtained from ANSYS® to the results from Wang [5].

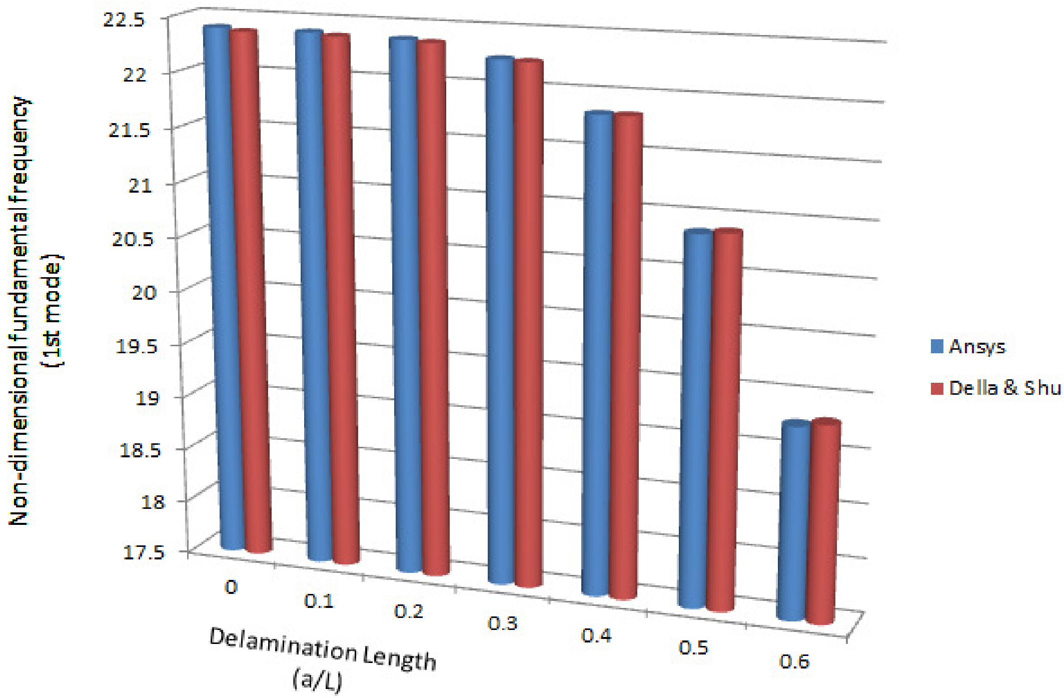


Figure 19- First non-dimensional frequencies comparison between Della-Shu [13] and ANSYS®

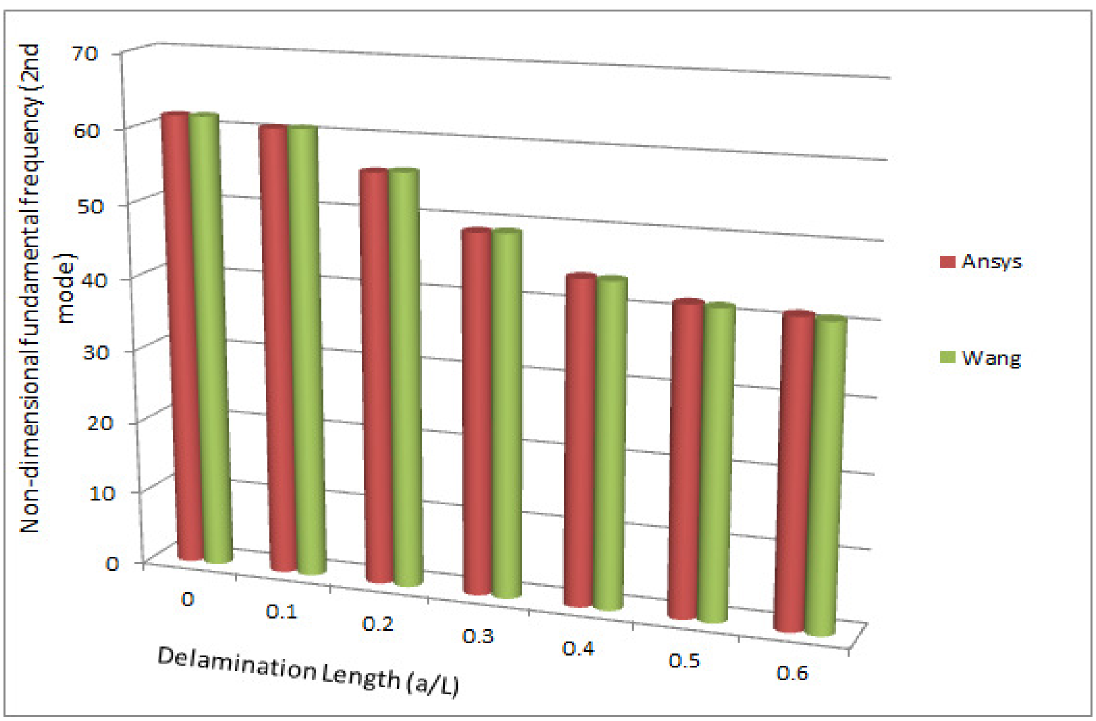


Figure 20- Second non-dimensional frequencies comparison between Wang [5] and ANSYS®

3.2.1.3 Result 2 - Single delamination (Clamped-clamped beam)

In order to verify the effect of length of the beam segments on the precision of ANSYS® analysis (provided L/H is very large), the frequency data was reproduced for different a/L values, by setting a/L=0.2 and changing L/H ratio (see Table 5). Three different sets of values, L/H=20, 40, and 60, were used in simulations and comparison was made between ANSYS® results and those reported by Della and Shu [13]. The data available in the literature [13] are based on Euler-Bernoulli beanding beam theory, which is valid for long and thin (slender/engineering) beams. As can be observed from the data in Table 5, when L/H ratio becomes significantly high, the effect on frequency becomes almost negligible; when L/H increases the error decreases. This is an indication that analyses are length-independent for large L/H ratio, i.e., Euler-Bernoulli beam.

Set #1	a/L	L/H	a/H 2	Lambda^2	L	a	Della&shu [13]	Variance
	0.200	20	8	22.11	20	4	22.36	-1.1%
	0.200	40	16	22.33	40	8	22.36	-0.13%
	0.200	60	24	22.37	60	12	22.36	0.07%

Set #2	a/L	L/H	a/H 2	Lambda^2	L	a	Della&shu [13]	Variance
	0.300	20	12	21.97	20	6	22.24	-1.23%
	0.300	30	18	22.14	30	9	22.24	-0.45%
	0.300	40	24	22.20	40	12	22.24	-0.17%
	0.300	60	36	22.25	60	18	22.24	0.04%

Set #3	a/L	L/H	a/H 2	Lambda^2	L	a	Della&shu [13]	Variance
	0.400	20	16	21.49	20	8	21.83	-1.57%
	0.400	30	24	21.70	30	12	21.83	-0.59%
	0.400	40	32	21.77	40	16	21.83	-0.27%
	0.400	60	48	21.83	60	24	21.83	0.00%
	0.400	75	60	21.85	75	30	21.83	0.08%

Table 5- Non-dimensional frequencies are independent from L/H ratio

3.2.1.4 Case study 2 - Single out of mid-plane delamination

Previous clamped-clamped beam example is repeated with the exception of delamination is out of mid-plane. H_2=0.67H, and H_3=0.33H. After simulating this model in different cases of a/L, the data is summarized in Table 6. It can be observed from the table that the location of delamination with respect to mid-plane is a determining factor in primary frequency. To clarify this matter more, if results from Table 3 are compared with results in Table 6, the primary frequency, or non-dimensional frequency, will decrease as delamination occurs closer to the edge in width direction of the beam.

a/L	L/H	a/H_2	Lambda
0.1	60	8.955224	4.732
0.2	60	17.91045	4.725
0.3	60	26.86567	4.699
0.4	60	35.8209	4.63
0.5	60	44.77612	4.472
0.6	60	53.73134	4.166

Table 6- Primary frequency of single out of mid plane delamination

3.2.1.5 Case study 3 - Single delamination (Cantilever beam)

In this section, the effects of delamination ratio, a/L, on the natural frequency of a cantilever beam with single off-centered and centered delaminations, are studied. The macro file was run several times, and in each case the length of delamination was varied from 0 to 0.6L.

A cantilever beam was developed and delamination is located in non-mid plane and centered position (Figure 21). The modal test was conducted with the following conditions:

The delamination length over length of beam (a/L) is 0.4, Thickness of beam2 (H_2) = 0.33H, d=0, Thickness of beam3 (H_3) = 0.67H and homogenise $E_1 = E_2 = E_3$. Boundary condition is: one end is clamped, UX=UY=0 and other end is free. The element size is 0.1.

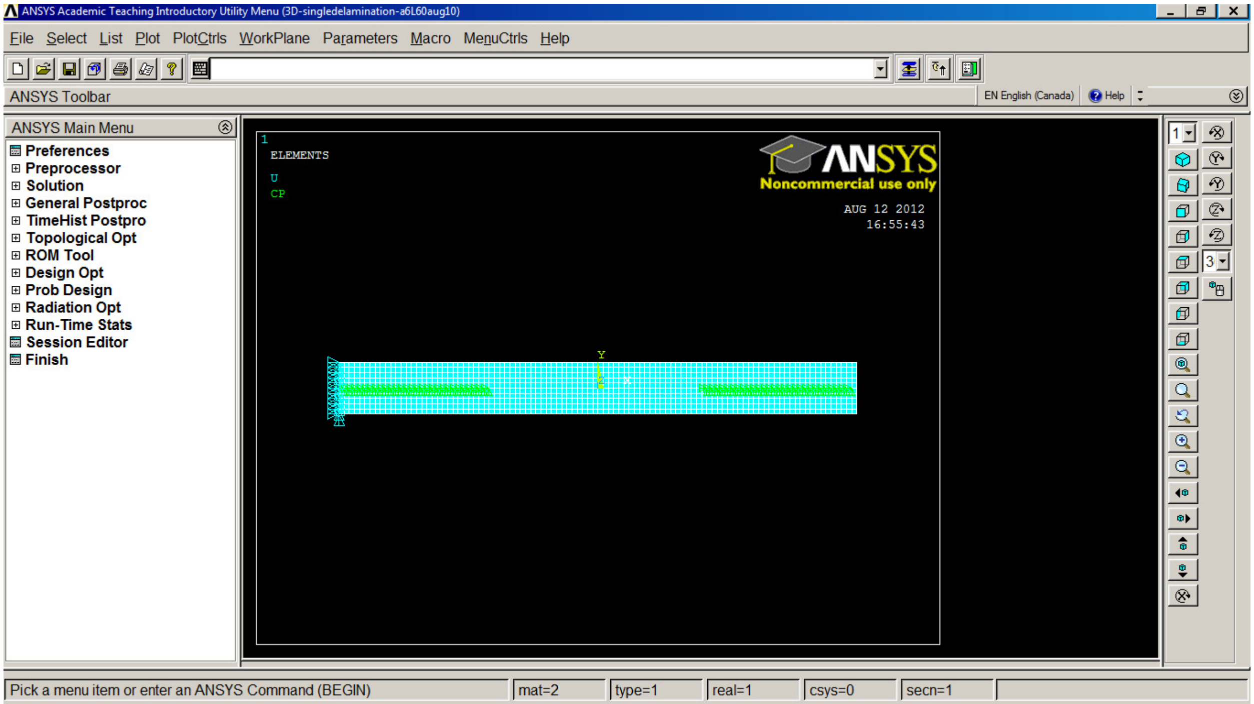


Figure 21- The cantilever beam with single centered delamination

In Figure 22, a comparison is made between clamped-clamped results and those of cantilever beam. It shows the impact delamination length has on primary frequency which is higher in clamped-clamped beam than cantilever beam for a>0.4L (Table 8).

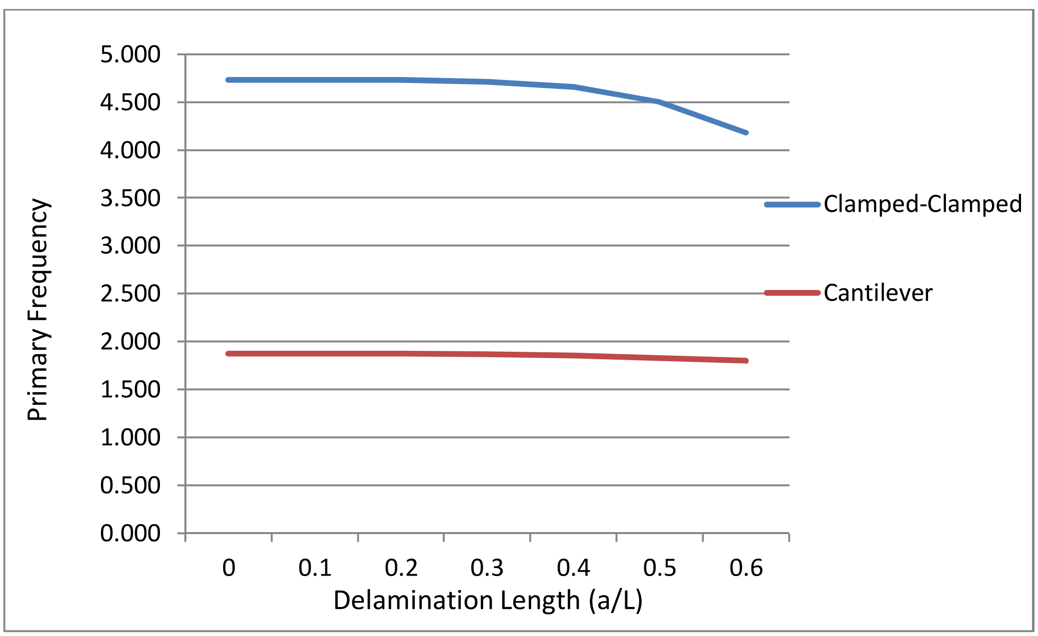


Figure 22- Influence of a/L on primary frequency

In the table below (Table 7), the numerical results for clamped-clamped and cantilever beams are tabulated.

There are three important results that can be observed from this table. First, the primary frequency for clamped-clamped and cantilever beams decreases as delamination length (a) increases. Secondly, primary frequency slightly decreases in a short delamination (a<0.4L) for both boundary conditions. Thirdly, for delaminations with long length (a>0.4L), the primary frequency sharply decreases in clamped-clamped beam rather than cantilever beam.

	a/L	**L/H**	**a/H_2**	**Primary Freq.**
clamped-clamped Central delamination	0	60	0	4.732
	0.1	60	8.955224	4.732
	0.2	60	17.91045	4.730
	0.3	60	26.86567	4.714
	0.4	60	35.8209	4.657
	0.5	60	44.77612	4.502
	0.6	60	53.73134	4.183
Cantilevered Central delamination	**a/L**	**L/H**	**a/H_1**	**Primary Freq.**
	0	60	0	1.877
	0.1	60	18.18182	1.877
	0.2	60	17.91045	1.874
	0.3	60	26.86567	1.866
	0.4	60	35.8209	1.851
	0.5	60	44.77612	1.829
	0.6	60	53.73134	1.798

Table7- Comparison of clamped-clamped & cantilever beam

When the simulation results displayed in Figure 22 are compared with those reported by Shu [11] in Figure 23, the similarities are noticeable. This proves the fact that the new model created for cantilever beam is working properly.

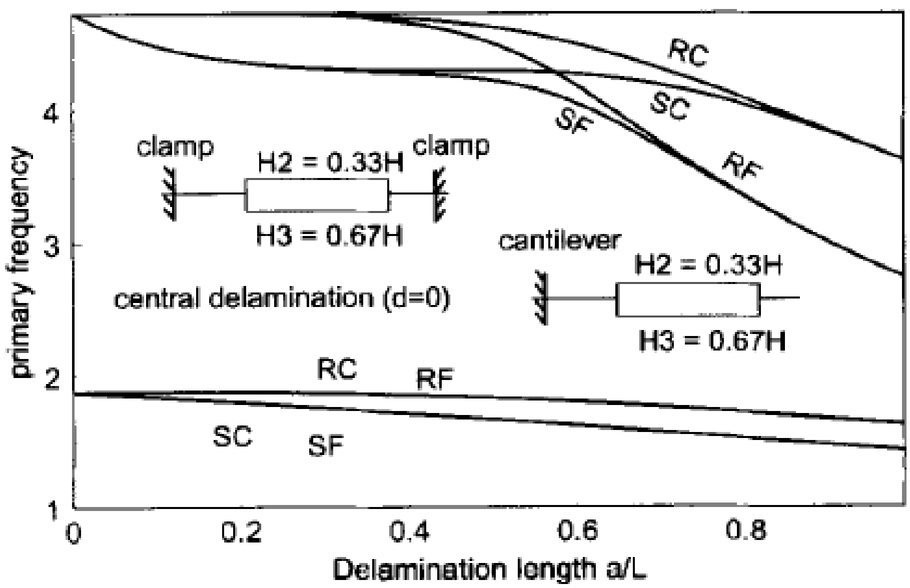


Figure 23- Test results adapted from Shu [11]

3.2.1.6 Case study 4 - Single off-centred delamination

For this part of the thesis, and to find out about the influence of off-centered delamination on primary frequency, macro files are updated from centred to off - centered position. Therefore, the files in Results 3.2 and 4 have been adapted to simulate off-centered delamination of clamped-clamped and cantilever beam, d=6 (Figures 24 and 25).

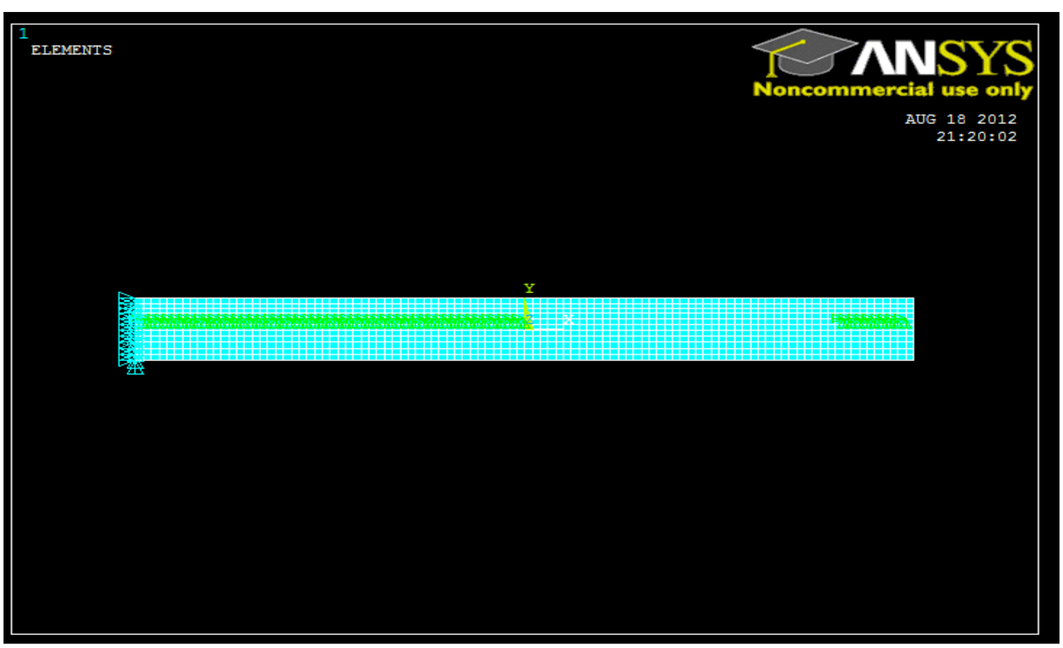


Figure 24- Single off-centred delamination cantilever beam

After running the updated programs, the following results are obtained for clamped-clamped and cantilever beam:

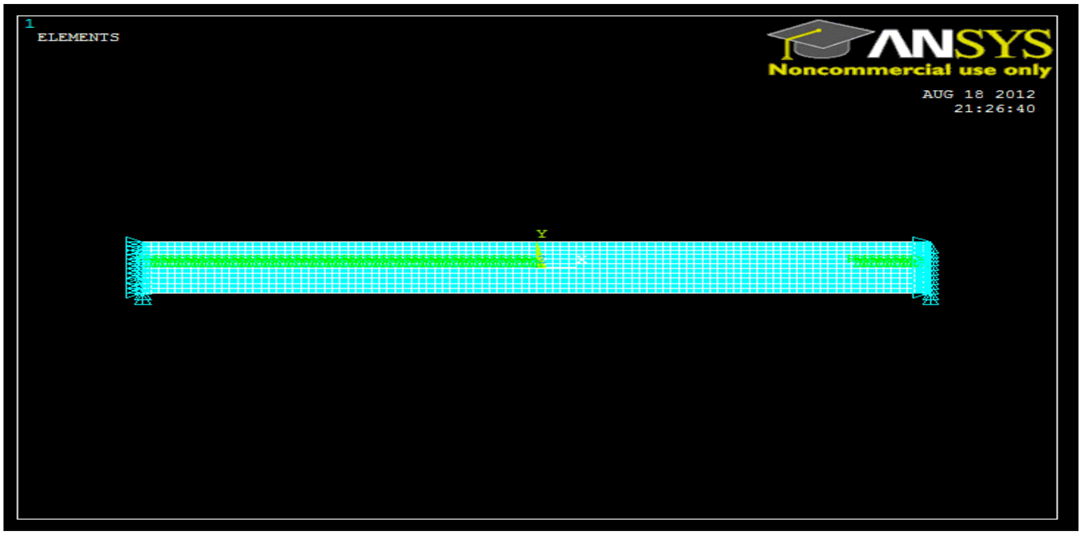


Figure 25- Single off-centred delamination clamped-clamped beam

When data from Table 8 and figure 26 are compared, it can be concluded that the primary frequency decreases similar to single centered delamination for both types of boundary conditions. However, in off-centered delamination, there is a slightly sharper decrease at a/L≥0.3 compared to centered delamination.

	a/L	L/H	a/H_2	Primary Freq.
Clamped-Clamped Not central delamination	0	60	0	4.732
	0.1	60	8.955224	4.732
	0.2	60	17.91045	4.725
	0.3	60	26.86567	4.699
	0.4	60	35.8209	4.630
	0.5	60	44.77612	4.472
	0.6	60	53.73134	4.166
Cantilever Not Central delamination	0	60	0	1.877
	0.1	60	8.955224	1.877
	0.2	60	17.91045	1.874
	0.3	60	26.86567	1.867
	0.4	60	35.8209	1.854
	0.5	60	44.77612	1.834
	0.6	60	53.73134	1.806

Table 8- Comparison of clamped-clamped & cantilever beam

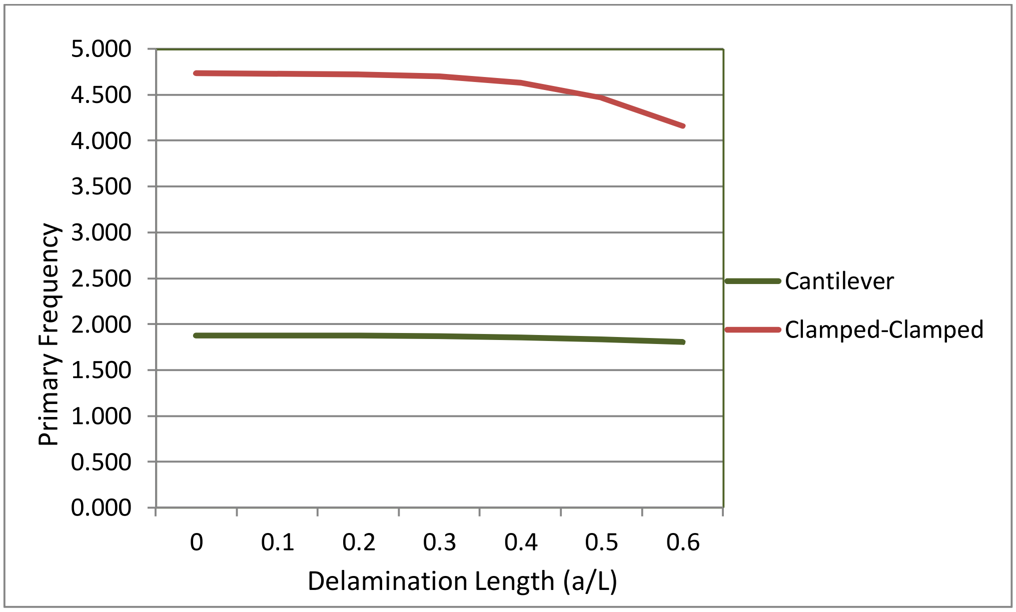


Figure 26- Influence of a/L on primary frequency.

3.2.1.7 Case study 5 - Double centered delamination

There are a number of possible configurations for double delamination of a beam. These possibilities, including centered, off-centered, mid-plane, overlapped, and enveloped are captured below in Figure 27.

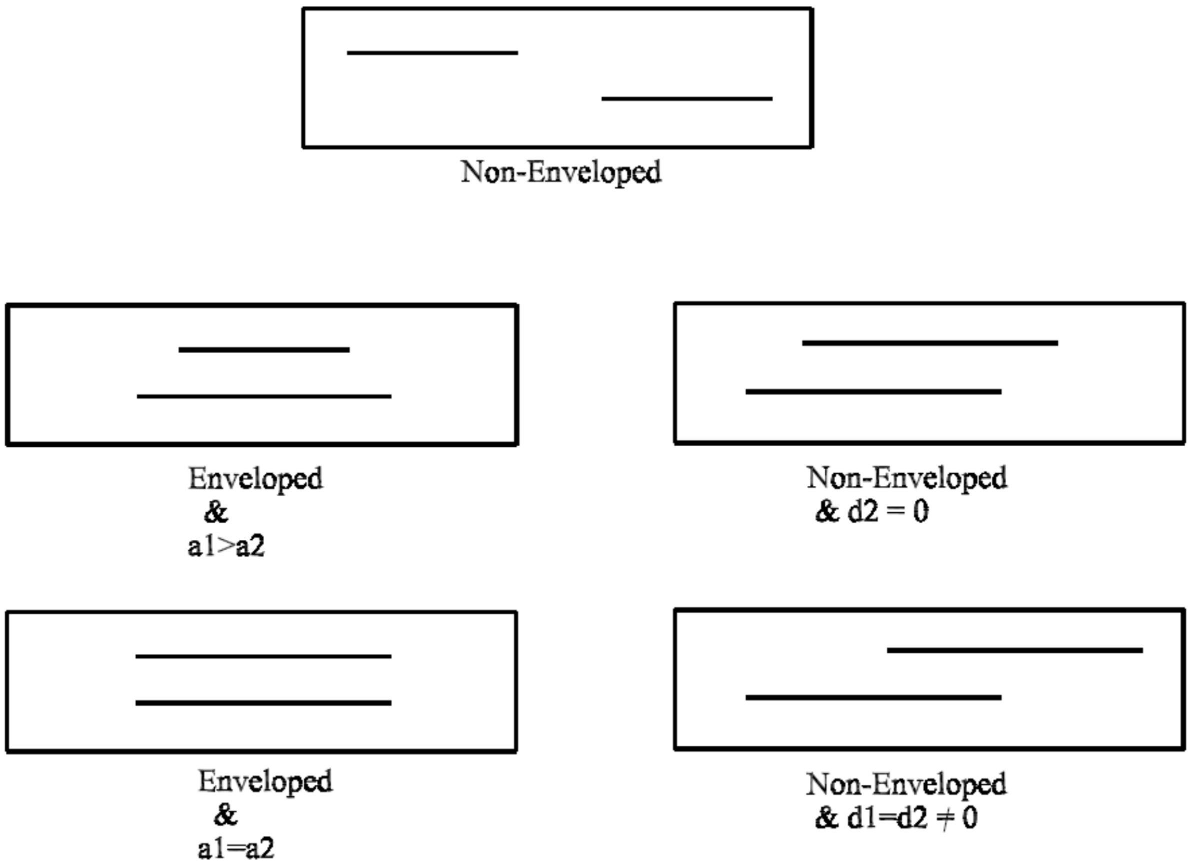


Figure 27- Different possibilities of double delamination

To gain a better understanding of the effects of double delamination on vibration of beam, 2D model as shown in Figure 28 is created in ANSYS®. The macro file created for this investigation, and certain parameters as followed were considered:

a) Material type: isotropic homogenous (E_5= E_4= E_3=E)
b) Element type: Plane 182
c) Element size=0.1
d) Boundary condition: Clamped-clamped beam
e) Analysis type: Free mode
f) Center of delamination is center of beam (d_1=d_2=0)

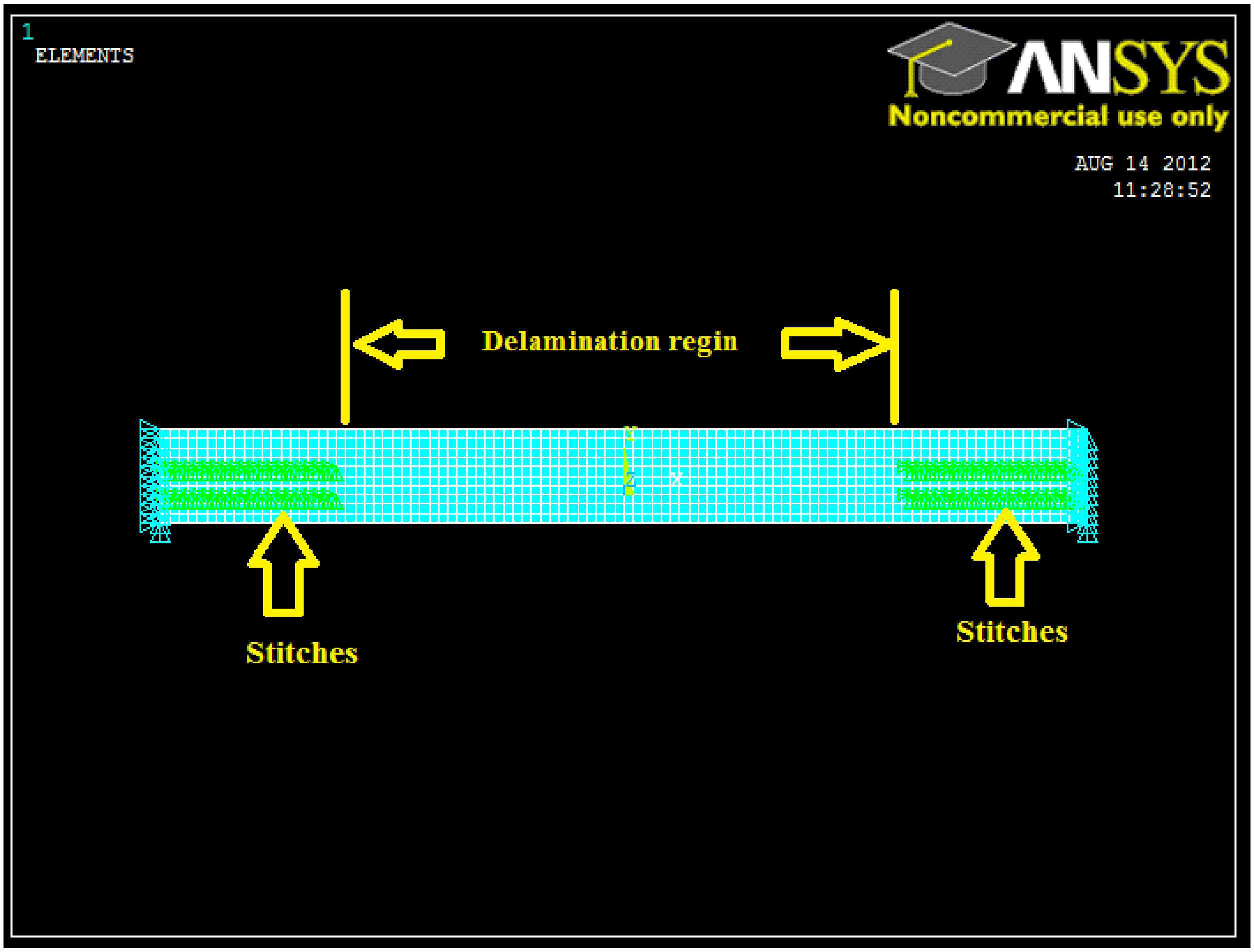


Figure 28- Double delamination

Various models were created and modal test was run for each case. All results confirm that an increase in delamination length ratio will cause a decrease in primary frequency (Table 9). The outcome of simulation of Double central delaminated beam is compared with Erdelyi [4] and Della, Shu and Yapu Zhao's [24] results in table 9. The results achieved by ANSYS® simulations are in agreement with previous analytical literatures [4, 24]. Therefore, it can be concluded that the new macro file for double delamination is a suitable template for further investigation of other configurations of double delamination of beams. These beams, for example, could have different thicknesses, lengths, delamination sizes and locations, etc.

	a_2/L	L/H	a_1/H_5	a_2/H_3	a_2/H_4	L	Lambda from present study	Hashemi & Erdelyi [4]	Della & Shu [24]	Variance
First Mode	0.2	60	40	24	40	60	**4.727**	4.725	4.75	-0.5%
	0.3	60	60	36	60	60	**4.692**	4.707	4.7	-0.2%
	0.4	60	80	48	80	60	**4.574**	4.575	4.55	0.5%
	0.5	60	100	60	100	60	**4.322**	4.315	4.3	0.5%
Second Mode	0.2	60	40	24	40	60	**7.028**	7.045	7.05	-0.3%
	0.3	60	60	36	60	60	**6.339**	6.335	6.37	-0.5%
	0.4	60	80	48	80	60	**5.979**	5.965	5.95	0.5%
	0.5	60	100	60	100	60	**5.577**	5.845	5.8	-3.8%

Table 9- Comparing new program with analytical results and other literatures

3.2.1.8 Case study 6 - Double non-enveloped delamination

Double non-enveloped delamination model ANSYS® is created with following assumptions:

The beam in homogenous, and the boundary condition is set as UX=UY=0, for both ends.

Furthermore, $a_t/L= 0.6$, $d_1=d_2$, $a_1=a_2$, $a_1+a_2= a_0+a_t$ and $H_3=0.5H$, $H_4=0.2H$, $H_5=0.3H$ (Figure 29).

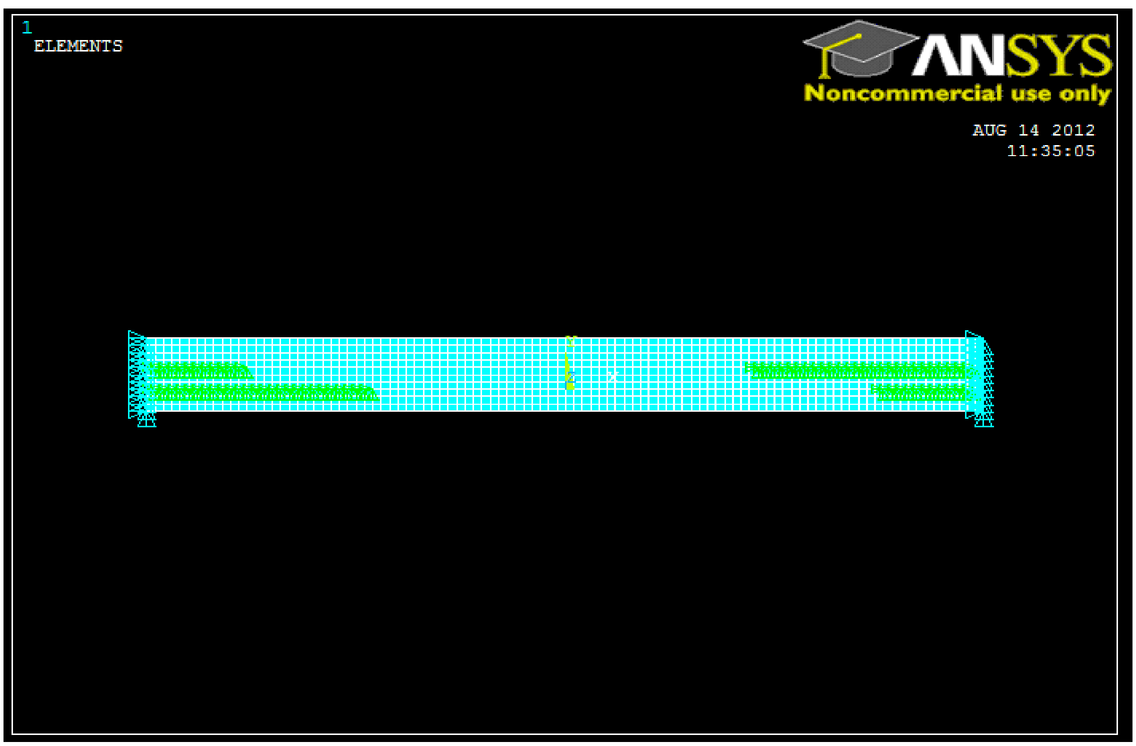


Figure 29- Enveloped delamination with $a_1+a_2= a_0+a_t$ condition

In figure 30, influence of delamination length on the fundamental natural frequency of a clamped-clamped beam is displayed. Total axial delamination length is shown as a_t, which represents the length of the entire delamination region, including overlapping segment. The length of overlapping delamination section is denoted as a_t. Frequencies are compared as the ratio of fundamental frequency, ω, with respect to the natural frequency, ω_0, of an intact beam. In what follows, two delamination configurations are studied. In the first case, both delaminations are assumed to be of equal length, i.e., $a_1=a_2$ and $a_t + a_0 = a_1+a_2$. The overlapping segment, a_0, is assumed to be at the center of the beam.

The second case studied here is a non-overlapping delamination, where $a_0=0.0$. Also, d_1 is assumed to be equal to d_2, where d_1 and d_2 are the distance of each delamination from center of the beam.

For the free mode delamination, which is the focus of the present study, the frequency ratio ω/ω_0 decreases slightly when a_t/L is less than 0.3. This decrease remains insignificant until the value of a_t/L reaches 0.4. At this time, frequency decreases significantly and it continues to decrease as a_t/L increases. Another noteworthy fact is that the fundamental frequency decreases as the length of overlapping delamination, a_0, increases. Observation of this simulation is illustrated in table 10 and figure 30.

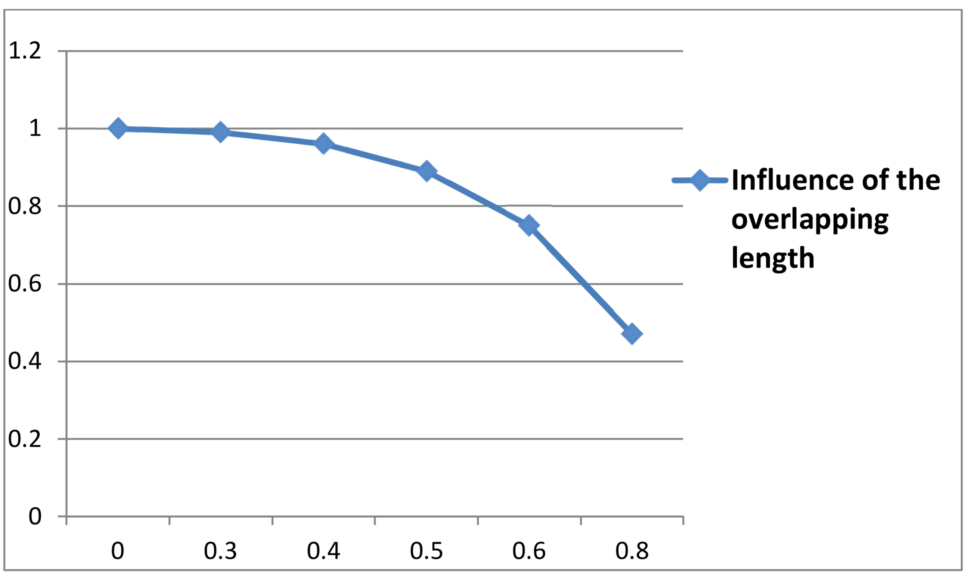


Figure 30- Influence of overlapping length on the fundamental frequency of a clamped-clamped beam

Fundamental Frequency ω/ω0	Total axial Delamination length (at/L)
0.47	0.8
0.75	0.6
0.89	0.5
0.96	0.4
0.99	0.3
1	0

Table 10- Influence of overlapping length on fundamental frequency of clamped-clamped beam

Similar to Della, Shu and Zhao [24], the results obtained in this simulation show that the fundamental frequency ratio ω/ω_0 decreases sharply after $a_t/L = 0.4$ for free mode delamination model (Figure 31).

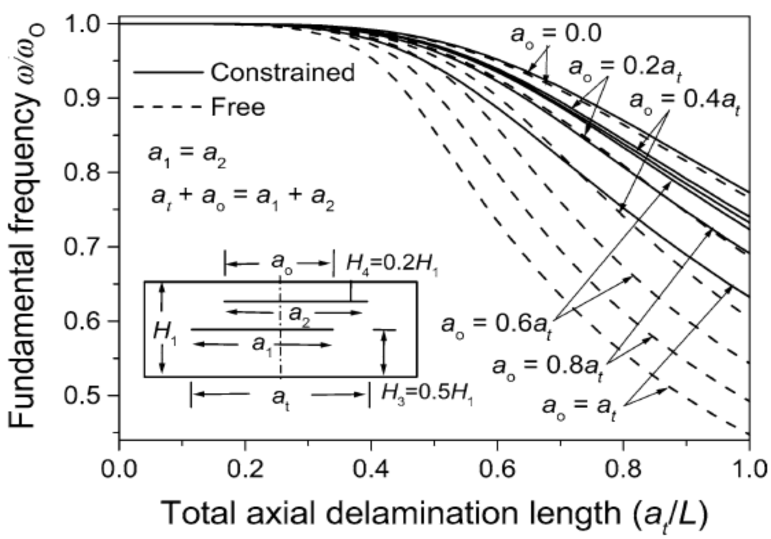


Figure 31- Relation between fundamental frequency and delamination length, adapted from [24]

However, there are some discrepancies for values higher than 0.4, which provides opportunity for further studies in future.

3.2.1.9 Case study 7 - Impact of Size, location and type of delamination

The following table is created based on reported data in previous sections of this thesis. From the table, it can be observed that when size of delamination increases, the non-dimensional frequencies decrease, especially when a/L≥ 0.4.

In addition, in case of more than one delamination, the non-dimensional frequency of defective beam structure is affected more than a single-delamination configuration.

As presented in the Table 11, when defect occurs closer to the edge of beam (out of mid-plane), the slope of decrease of non-dimensional frequencies is sharper than mid-plane delamination.

		Single Delamination		Double Delamination
		0.33H-0.67H	0.5H-0.5H	0.3H-0.3H-0.4H
Central delamination Clamped-Clamped	a/L	Primary Freq.	Primary Freq.	Primary Freq.
	0.2	4.730	4.730	4.727
	0.3	4.714	4.717	4.692
	0.4	4.657	4.672	4.574
	0.5	4.502	4.568	4.322
	0.6	4.183	4.389	3.958

Table 11- Impact of Size, location and type of delamination

In summary, each of the size, type and location of delamination has a direct influence on non-dimensional frequencies.

3.2.1.10 Case study 8 - Effects of element size in simulation results

The size of element is an important factor in FEM-based simulation, affecting the results precision. Selecting too many or too few elements would have a diverse effect on simulations performed. Based on current research and the data tabulated in Table 12, in 2D simulation, size of element should not exceed 0.17% of length of beam, as the variance between simulation data and mathematical results grows larger when the element size is greater than 0.0017L.

Single Central delamination clamp-clamp beam									
a/L	L/H	a/H_2	f	$\omega=2pf$	λ^2	Wang[5]	variance	Elem. Size	QTY Element
0.4	60	48	0.279	1.7504	21.829	21.83	0.00%	0.17L%	6000
0.4	60	48	0.281	1.7646	22.007	21.83	-0.80%	0.33L%	1800
0.4	60	48	0.285	1.7887	22.307	21.83	-2.14%	0.5L%	800

Table 12- Effects of element size in simulation result

In addition, when the mesh is coarse (QTY=800), the penetration in mode shape cannot be captured as well as in fine mesh (QTY=6000), Figures 32 and 33.

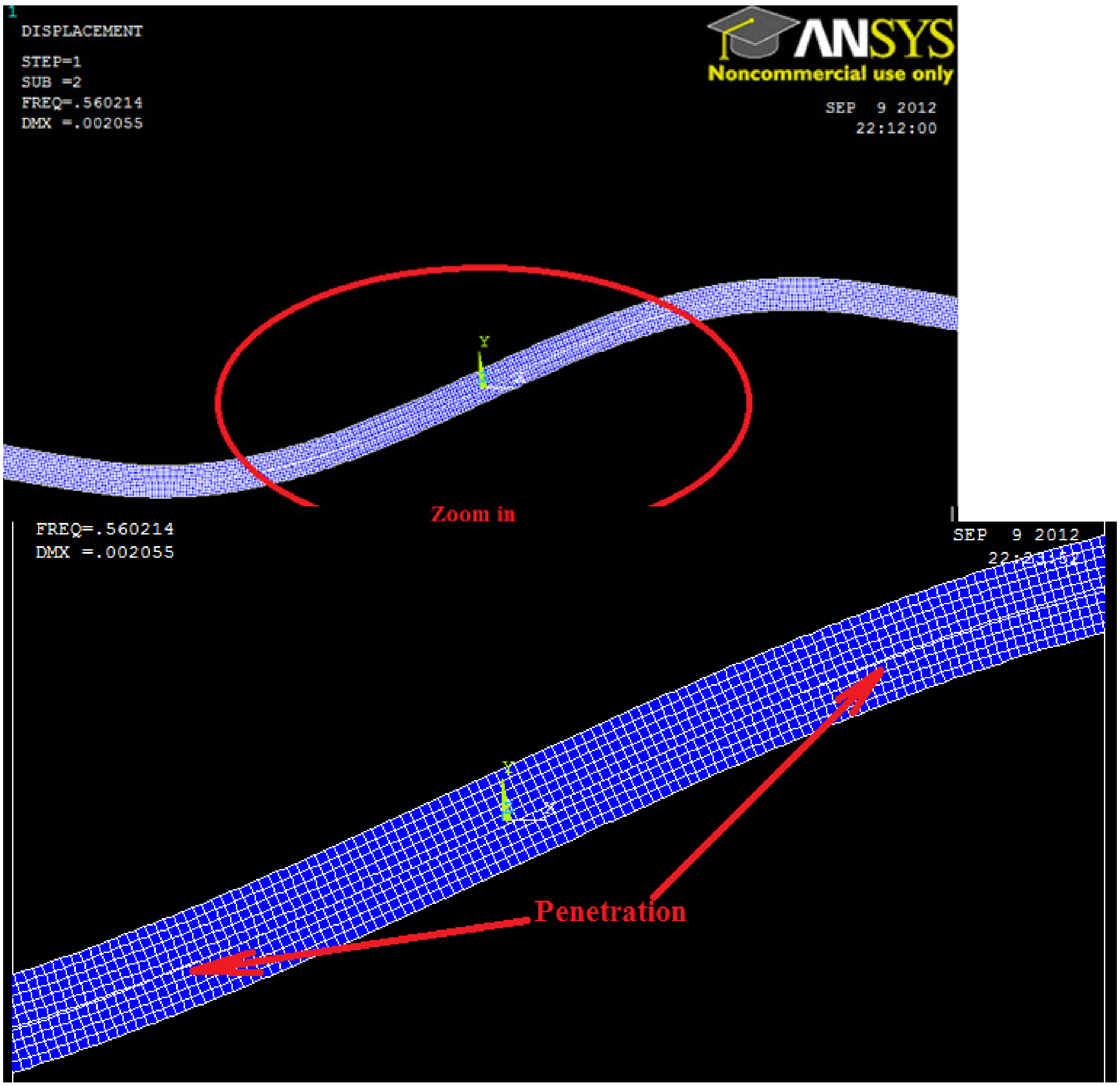


Figure 32- Penetration in fine mesh

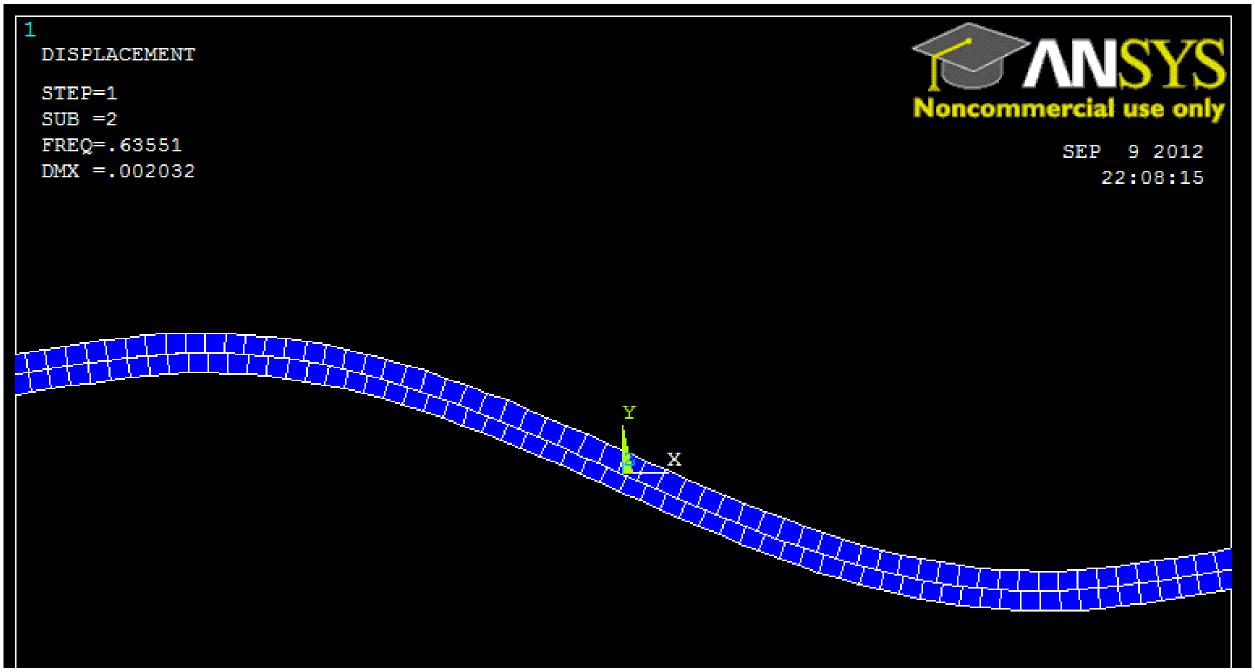


Figure 33- Penetration in coarse mesh

3.2.1.11 Case study 9 - Influence of element type

As it was mentioned earlier in this report, selection of element type depends on the nature of analysis preformed, in this case liner modal analysis, and also accuracy and calculation time. Therefore, Plane182 element was selected for this research. However, it should be noted that Plane182 is not the only option; there are other types of elements that one could use and produce more accurate results. However, the time for simulation would have been longer, but in the end the results are not different significantly.

For comparison purposes, Shell63 element was also examined in a 2D simulation. A modal test was conducted with both Plane182 and Shell63 elements. The average difference in results for the first four modes was found to be very small, i.e., around 0.25%. A model created using Shell163, however, needed more time to solve. Therefore, to save in simulation time without compromising the precision of results, Plane182 was selected for all 2D simulation tests in this research.

3.2.2 3D modeling

Even though 2D modelling is found to be quite sufficient for investigating effects of delamination on natural frequency, but 3D modelling is probably more practical in today's industries. It would also pave the road for a more realistic modelling/simulation and analysis of defective 3D components. Therefore, in what follows, an ANSYS® FEM-based 3D modelling, for the free vibration analysis for delaminated layer beam will be examined. Single delaminated clamped-clamped beam is modeled in ANSYS®. Assumption: beam is homogeneous, free mode test is run by new 3D program in ANSYS® (Figures 34 and 35).

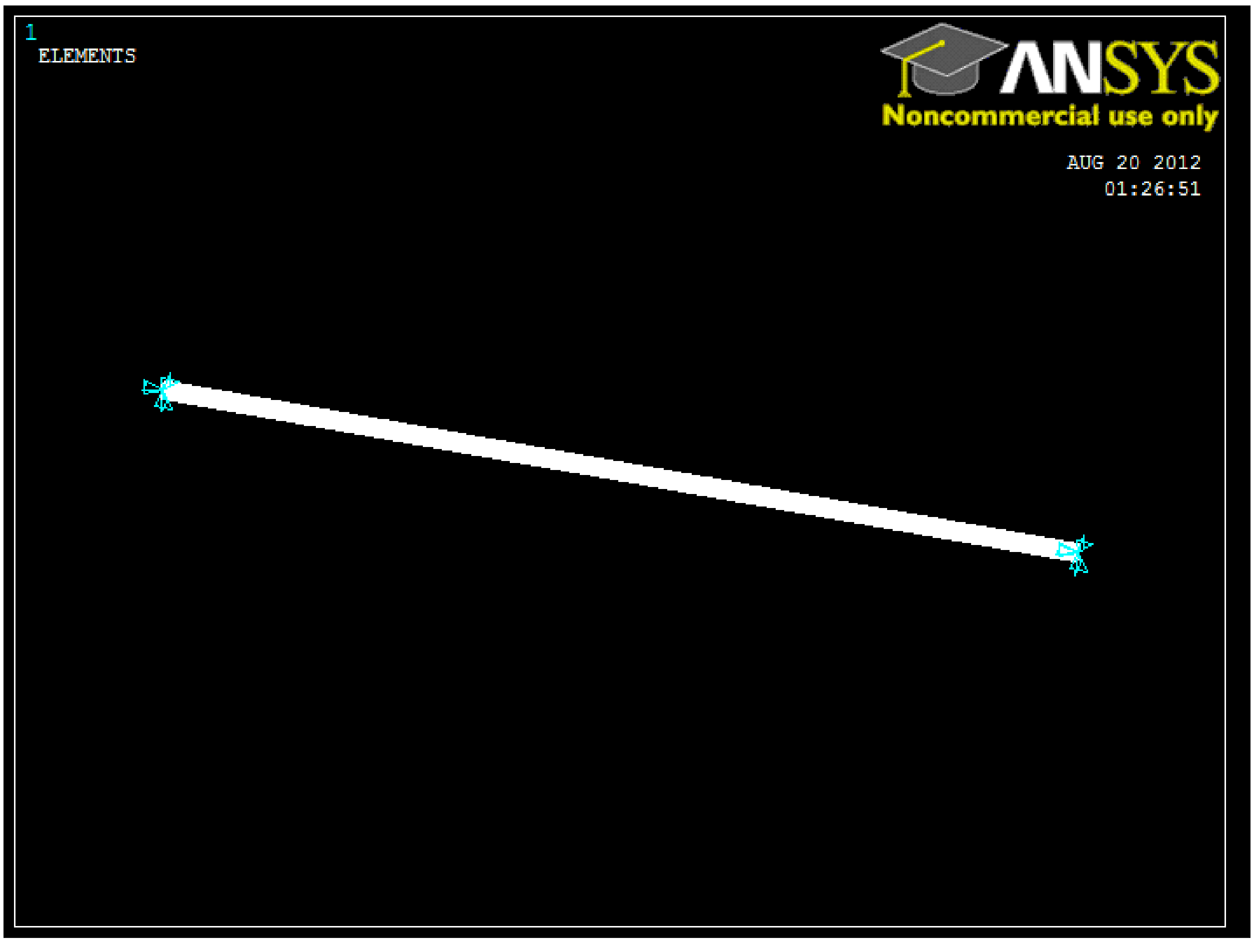


Figure 34- 3D single delaminated beam

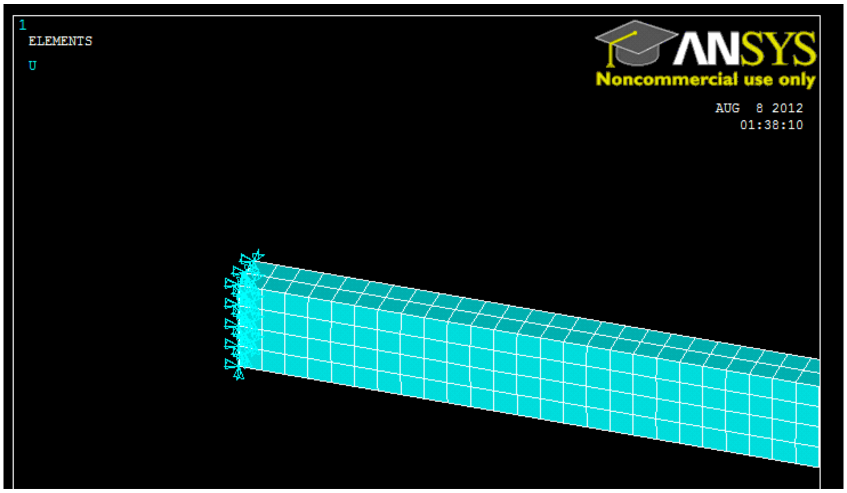


Figure 35- Close look at 3D model

The advantage of having simulations in 3D modeling as opposed to 2D is that more frequencies along the other axes could also be captured in 3D simulation which is an important issue in design and predicting delamination. It should be noted that these frequencies cannot be captured as easily in 2D modeling.

To have a detailed report of how 3D simulation is carried out in this research, the logical steps will be explained in the form of an example as follow:

3.2.2.1 Case study 10 - 3D single centered delamination beam

This is a uniform beam along its length and its cross-sectional area and it is fixed at both ends, clamped-clamped. The investigation focuses on the effects of single delamination on the system natural frequency and its mode shapes.

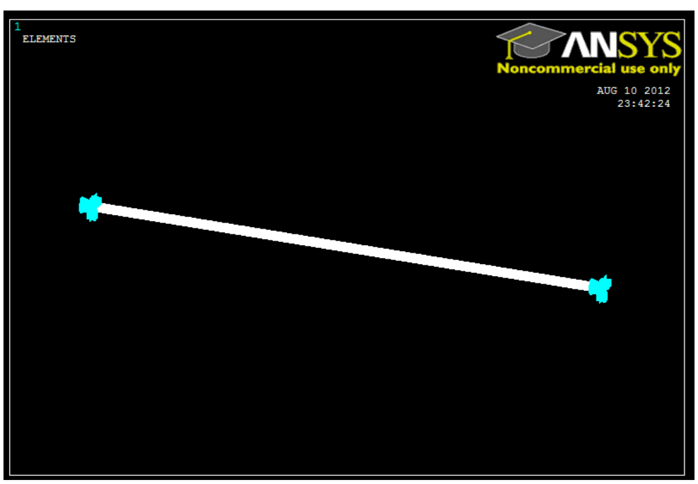


Figure 36- 3D model of delaminted beam

Approach and Assumptions

Assumption is that both sides of the beam are fixed so that it has no degrees of freedom. The beam is isotropic so the material properties are constant.

First a 2D model of length of the beam is created using solid modeling. This area is then extruded across the thickness of the beam to form a 3D solid model.

Assumptions:

Applicable ANSYS® Products	ANSYS® ED (ANSYS® Mechanical)
Discipline:	Structural
Analysis Type:	Modal
Element Types Used:	PLANE42 and SOLID45
ANSYS® Features Demonstrated:	extrusion with a mesh

Summary of Steps

Following is a list of necessary steps and a description of each step completed to achieve the end results in Table 8.

a) Input Geometry

For this step, any geometry file which is saved in IGS format can be imported to ANSYS®.

b) Define Materials

In order to introduce the material that is specific to any given example, the preference will be set as follow:

"Structural", "Linear", "Elastic", "Isotropic" for each layer and beam is homogenous. $E=E_1= E2$.

c) Generate Mesh

The first step of meshing the model is defining element type. In this practice, there are two types of elements: 2-D element (PLANE 42) and 3D element (SOLID 52) should be defined.

Length area of beam is meshed with 2D elements. Then extrude the area to create a 3-D volume. The mesh will be "extruded" along with the geometry so 3-D elements will automatically be created in the volume.

The size of each element is identified as 0.1.

In extrusion process, "Element type number should be selected SOLID45. Also, the number of "Elem divs" is 5 in this problem. And "Offsets for extrusion" = 0, 0, 0.5.

The ANSYS® software used in this thesis is ANSYS® ED, which is the educational version. Because of limitations with this version, 4-node PLANE 42 element is used instead of 8-node PLANE82.

In designing this problem, the maximum node limit of ANSYS® ED was taken into consideration. That is why the 4-node PLANE42 element, rather than the 8-node PLANE82 element was used.

SOLID45 to run this problem in ANSYS® ED will produce this warning message. If ANSYS® ED is not being used, then SOLID95 (20-node brick) can be used as element type 2.

d) Apply Boundary conditions

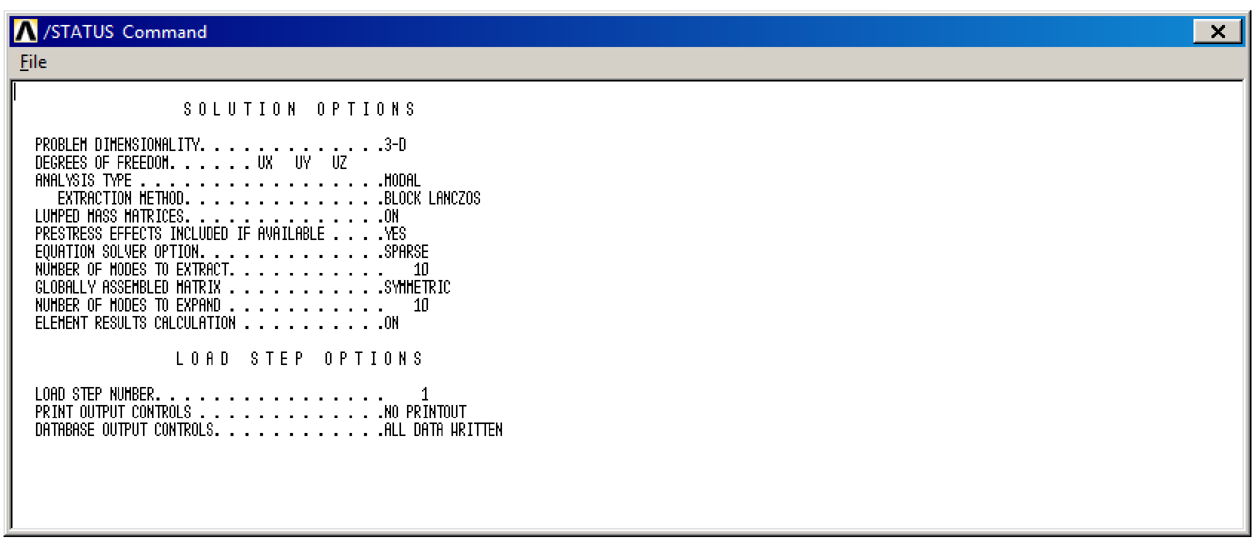
In this particular example, the beam is clamped-clamped. Therefore, the displacement of nodes at both ends of beam are 0 (UX=UY=UZ=0).

e) Obtain Solution

The next step is specifying analysis type. In this example, a modal analysis type is chosen: Block Lenczoc.

10 modes to extract are identified and also number of modes to expand is 10 in this problem.

After meshing and applying boundary conditions, the model is ready to be solved.



```
/STATUS Command
File

                 S O L U T I O N   O P T I O N S

PROBLEM DIMENSIONALITY. . . . . . . . . . . . .3-D
DEGREES OF FREEDOM. . . . . . UX   UY   UZ
ANALYSIS TYPE . . . . . . . . . . . . . . . . .MODAL
   EXTRACTION METHOD. . . . . . . . . . . . . .BLOCK LANCZOS
LUMPED MASS MATRICES. . . . . . . . . . . . . .ON
PRESTRESS EFFECTS INCLUDED IF AVAILABLE . . . .YES
EQUATION SOLVER OPTION. . . . . . . . . . . . .SPARSE
NUMBER OF MODES TO EXTRACT. . . . . . . . . . .    10
GLOBALLY ASSEMBLED MATRIX . . . . . . . . . . .SYMMETRIC
NUMBER OF MODES TO EXPAND . . . . . . . . . . .    10
ELEMENT RESULTS CALCULATION . . . . . . . . . .ON

                 L O A D   S T E P   O P T I O N S

LOAD STEP NUMBER. . . . . . . . . . . . . . . .    1
PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . . . . . . . .ALL DATA WRITTEN
```

f) Review Results

The list of natural frequencies (Table 13), and first 3 mode shapes for this 3D beam (a=0.5L , H_2= H_3=0.5H, and E_2= E_3=E) are obtained for this problem (Figures 37, 38, 39, and 40).

As it can be observed, all natural frequencies on all planes are captured by one simulation.

3D simulation		
Mode No.	Frequency from ANSYS® (Hz)	Plane
1	0.24803	Y-Z
2	0.27980	X-Y
3	0.55174	X-Y
4	0.68039	Y-Z
5	0.83676	X-Y
6	1.1717	Y-Z
7	1.3302	X-Y
8	1.4635	X-Y
9	1.9179	Y-Z
10	2.1933	X-Y

Table 13- Modal test results from 3D beam

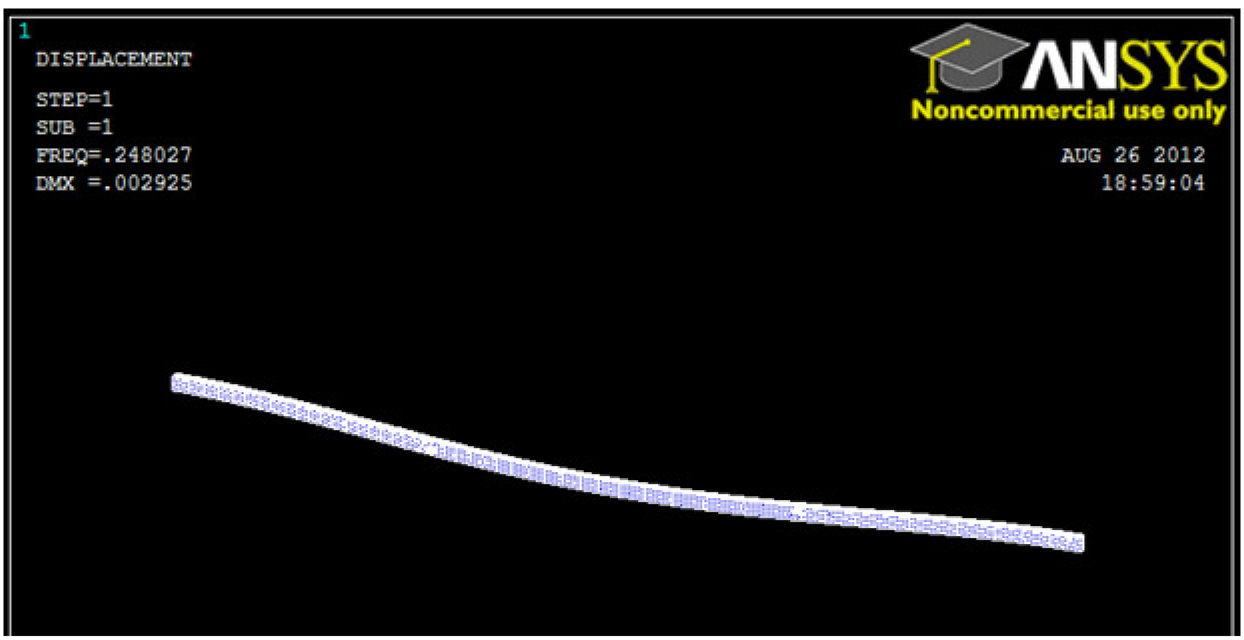


Figure 37- First Mode in X-Z plane

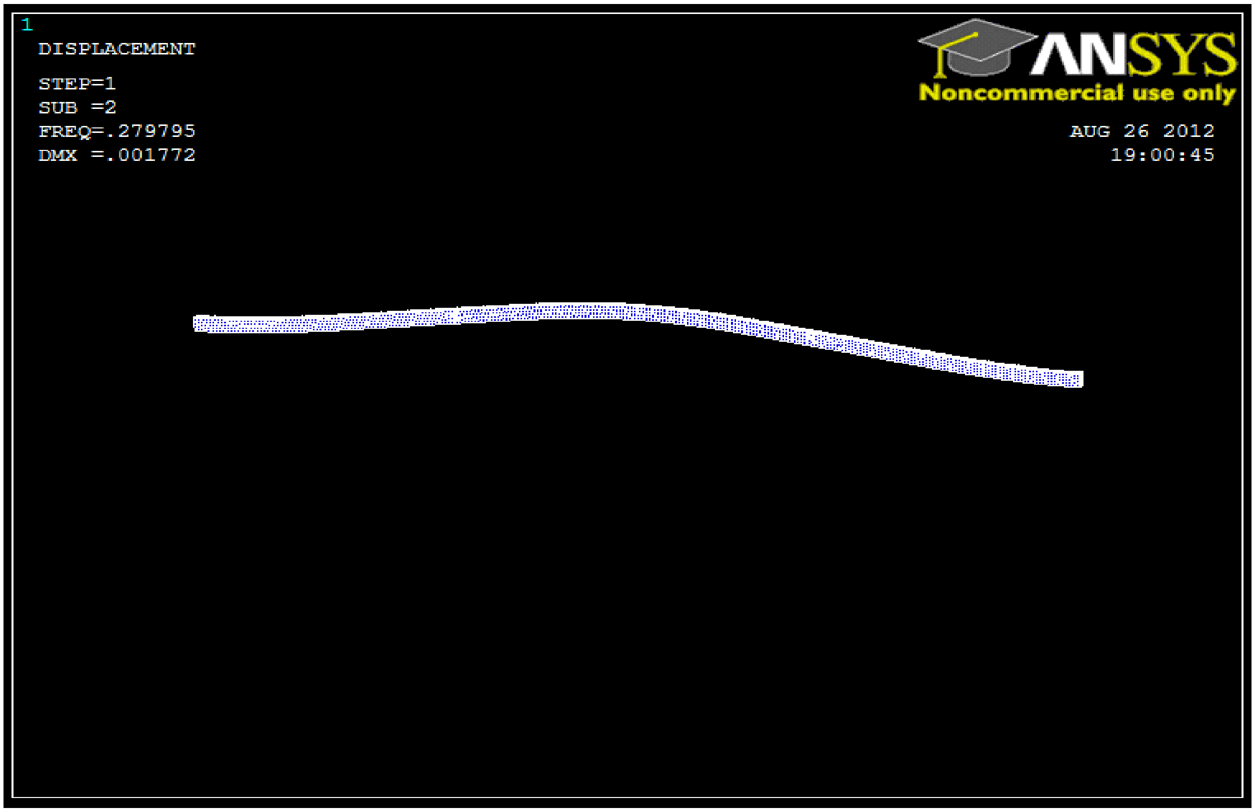


Figure 38- Second mode in X-Y plane

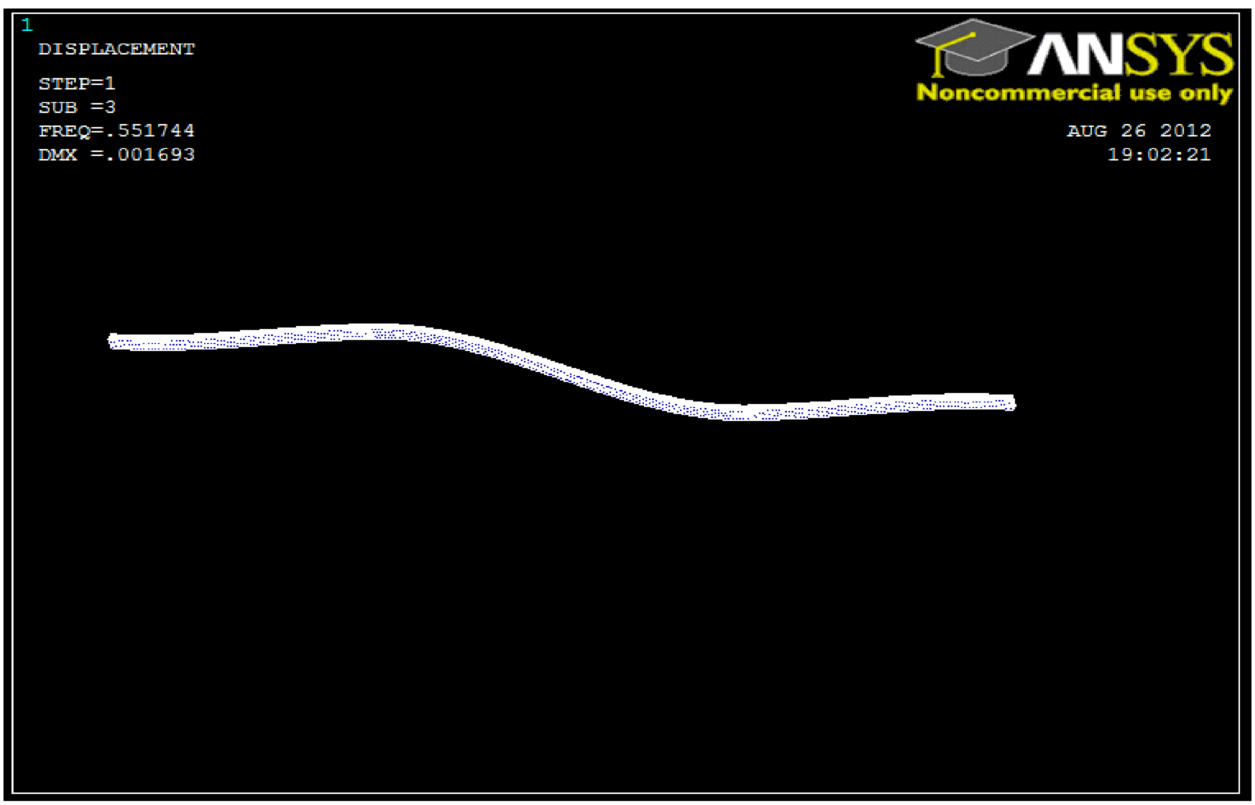


Figure 39- Third mode in X-Y Plane

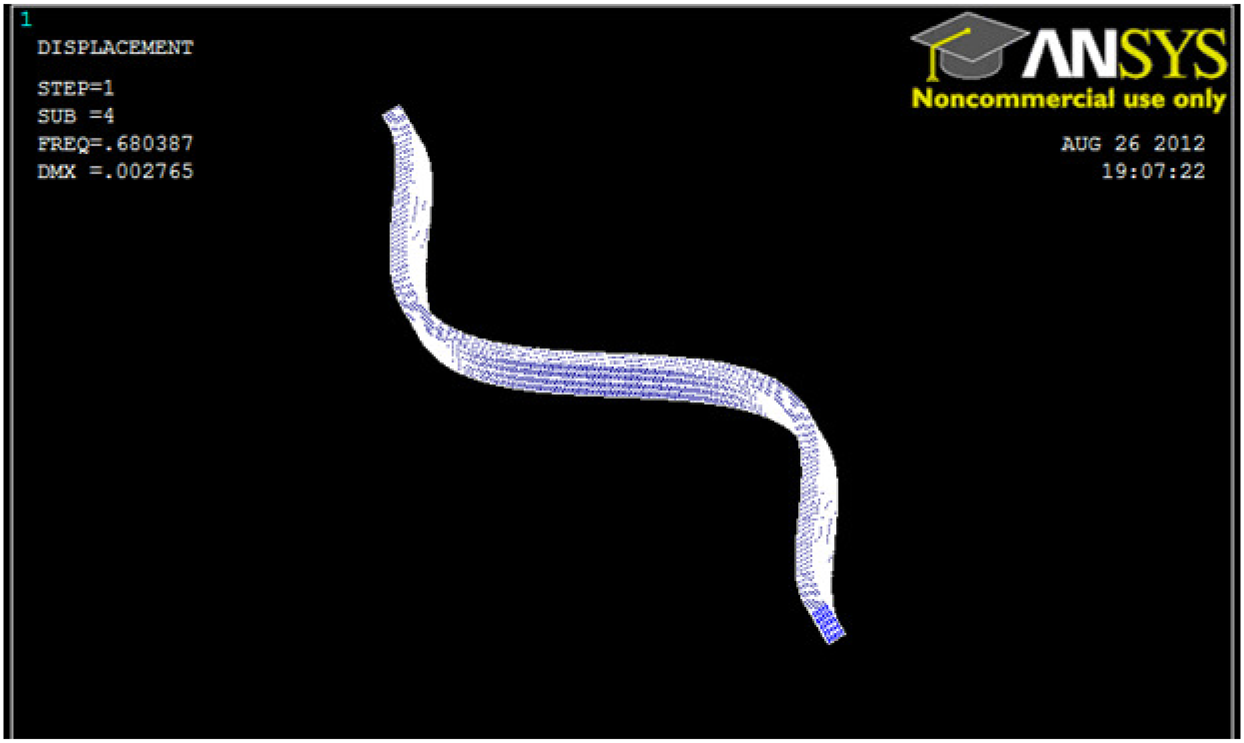


Figure 40- Forth mode in X-Z plane

In addition, the non-dimensional frequency parameter (λ^2) results from 3D simulation are compared with Della-Shu [13] to verify the presented ANSYS® program in Table 14.

a/L	**L/H**	**a/H$_2$**	**λ^2 from 3D simulation**	**Della & Shu** [13]	**Variance**
0.00	60	0	22.394	22.37	0.11%
0.10	60	12	22.398	22.37	0.13%
0.20	60	24	22.431	22.36	0.32%
0.30	60	36	22.339	22.24	0.44%
0.40	60	48	21.924	21.83	0.43%
0.50	60	60	20.907	20.89	0.08%
0.60	60	72	19.206	19.3	-0.49%

Table 14- Comparison of 3D simulation with analytical results [13]

The variance between simulation and analytical methods is less than 0.5%. This result proves that 3D simulation could not only be confidently used as an alternative to analytical methods, it also captures more natural frequencies, representing the beam vibration along the axis perpendicular to the original axis.

Finally, to have a better overall view of simulation in 1, 2 and 3D, the following tables are created:

a/L	L/H	a/H_2	3D simulation (Present study)	2D simulation (Present study)	1D simulation [25]	Della & Shu [13]
0.000	60	0	22.394	22.396	N/A	22.37
0.100	60	12	22.398	22.395	22.2588	22.37
0.200	60	24	22.431	22.375	22.2414	22.36
0.300	60	36	22.339	22.250	22.1268	22.24
0.400	60	48	21.924	21.829	21.7298	21.83
0.500	60	60	20.907	20.866	20.8025	20.89
0.600	60	72	19.203	19.260	19.2283	19.30

Table 15- Comparison of 3D simulation with 1D [25], 2D and analytical results [13]

a/L	L/H	a/H_2	Variance Between 3D & Della-Shu [13]	Variance Between 2D & Della-Shu [13]	Variance Between 1D [25] & Della-Shu [13]
0	60	0	0.11%	0.12%	N/A
0.1	60	12	0.13%	0.11%	-0.50%
0.2	60	24	0.32%	0.07%	-0.53%
0.3	60	36	0.45%	0.04%	-0.51%
0.4	60	48	0.43%	0.00%	-0.46%
0.5	60	60	0.08%	-0.11%	-0.42%
0.6	60	72	-0.50%	-0.21%	-0.37%

Table 16- Variance between 1D [25], 2D, 3D simulations and analytical results [13]

It can be concluded from Table 16 that 2D simulations results match best with analytical results from previous literatures. Also, the results obtained from 3D simulations are comparable to those of 2D, and any variance in 3D simulation is negligible. However, 3D simulation is more realistic because it takes into account frequencies in all planes, while in 2D simulations only frequencies on a particular plane is considered and frequencies on other planes are neglected. Therefore, 3D simulation should be used for any investigations on free vibration of delaminated beam because it captures all possible frequencies on defected beam in all planes in one simulation process.

3.2.3 Additional modal test results

To get a better understanding and to gain a better insight into the problem of defective beam structures with delamination(s), a number of additional modal tests have also been performed. These tests and the results have been included in Appendix C. However, the author was not able to find any existing similar data in the literature that these results could be validated against. In the future, if this data becomes available, the results here can be used as reference.

Chapter 4: Concluding Remarks and Future Work

==

Delamination causes weakening of strength and the integrity of a mechanical structure, resulting in a final failure. It also reduces the stiffness of the structure, which will have a direct impact on the system vibration and stability characteristics. There have been some analytical approaches, and limited number of other method, used to investigate the vibrational response of delaminated layered beams. However, to the best of author's knowledge, there has been no extensive study reported on the FEM-based modal analysis of such systems, exploiting commercial software and applications.

This research, presented an ANSYS® FEM-based study of the delamination effects on the vibration characteristics of a beam-type structure. The works and results reported by Shu and Della [6, 11, and 23], Della, Shu and Zhao [24], Shu and Fan [10], and Wang [5] were further examined. A brief explanation of their mathematical formulation and solution methods was offered, and the reported results were also discussed. The results obtained in this research illustrate that the effect of the delamination on frequencies depends on the size, type, and location of delamination(s). These effects are also dependent upon the type of boundary conditions and vibration mode.

It was shown that delamination leads to new vibration modes and frequencies proper to the defective structure, which are different form the ones commonly encountered in the case of intact beam configurations. The change in the magnitude of such frequencies depends upon the type, size and location of delamination, as well as the system boundary conditions.

With the aid of ANSYS®, and exploiting 1D and 2D modelling capabilities of the software, various delaminated layered beam configurations reported in the literature were simulated, and the natural frequency/mode results were verified against the existing data. Furthermore, a 3D model was also simulated. It was found that frequencies of vibration in other planes can be captured in one single simulation while these frequencies were neglected in previous research. In order to capture all the frequencies in analytical methods, calculations must be carried out for each plane separately. Obviously, repeating calculations is timely and ANSYS® FEM-based 3D simulation and modal analysis offer an alternative to the existing analytical method, which can save this time.

The following is a list of all simulations carried out in this research. These simulations represent the relationship that exists between delamination and frequency in different boundary conditions:

No. of options		See Result /Case study No.
1	2D single centered delamination Cantilever beam	C.S. 3
2	2D single off-centered delamination Cantilever beam	R. 3
3	2D single centered out of mid-plane delamination Cantilever beam	C.S. 2
4	2D double centered delamination Cantilever beam	Future work
5	2D double off-centered delamination Cantilever	Future work
6	2D single centered delamination clamped-clamped beam	C.S. 1
7	2D single off-centered delamination clamped-clamped beam	R. 3
8	2D single centered out of mid-plane delamination clamped-clamped beam	C.S. 2
9	2D Double enveloped delamination clamped-clamped beam	C.S. 4
10	2D Double non-enveloped delamination clamped-clamped beam	C.S. 5
11	3D-Single centered delamination clamped-clamped beam	C.S. 6

In summary, the present work paved the road to ANSYS® commercial software and its application in the free vibration analysis of a number of delaminated beam configurations.

In the future, all 2D defective configurations with boundary conditions not covered in this study can be studied. As mentioned in above table, items number 4 and 5 can be conducted in the future and be included in this study. Furthermore, experimental tests for the above conditions will add more credentials to the simulation. It is worth nothing that high accuracy and time savings are critical factors in any industry. To this end, incorporating ANSYS® software or any simulation method can benefit all industries and will be a more trusted tool in the future.

References:

1. Structural Vibration: Analysis and Damping- C. E Beards , 1996, *Halsted Press an imprint of John Wiley &Sons Inc*, ISBN 0 340 64580 6
2. Vibration-based Model-dependent Damage (Delamination) Identification and Health Monitoring For Composite Structures--A Review-Y,Zou, L. Tong and G.P.Steven , 2000, *Journal of Sound and Vibration* , 2(230), 375-378
3. In-Flight Separation of Vertical Stabilizer-NTSB/AAR-04/04, 2004, *National Transportation Safety Board*
4. Dynamic Finite Element Modeling of Doubly Delaminated Composite Beams- Nicholas H. Erdelyi, Seyed M. Hashemi, 2012, *Modelling and Simulation in Engineering,* 492415 (2012), 1-8
5. Vibration of Split Beams-Wang JTS, Liu YY, Gibby JA, 1982 *Journal of Sound and Vibration,* 4 (84) 491–502
6. Vibration of Delaminated Multilayer Beams- Christian N. Della, Dongwei Shu, 2006, *Composites Part B: Engineering* , 2-3 (37) 227–236
7. Free Vibration Frequencies of a Delaminated Beam- Ramkumar RL, Kulkarni SV, Pipes RB., 1979, *Society of Plastics Industry*, 22-E1–5
8. Flexural vibrations of beams with delamination- P.M. Mujumdar and Suryanarayan, 1988, *Journal of Sound and Vibration*, 125 (3) , 441-461
9. Effect of Vibration on Delaminated Beams-Amit Kumar Singh, 2010, *Thesis submitted to National Institute of Technology Rourkela*,769008
10. Free Vibration of Biomaterial Split Beam- D.Shu and H.Fan, 1996, *Composites: Part B*, 1(27B)79–84
11. Vibration of Sandwich Beams with Double Delaminations- D.Shu, 1995, *Composites Service and Technology*, (54) 101-109
12. Free Vibration Analysis of Delaminated Composite Beams- Jaehong Lee, 2000, *Computers and Structures*, 2 (74) 121-129
13. Free Vibration Analysis of Multiple Delaminated Beams under Axial Compressive Load- Christian N. Della and Dongwei Shu, 2009, *Journal of Reinforced Plastics and Composites,* 11(28) 1356-1381

14. Delamination, A Damage Mode in Composite Structures,-A.C Garg, 1998, *Engineering Fracture Mechanics*, 29(5) 557-584

15. Characterization and Analysis of Delamination Fracture- Tay, T. E., 2003, *Applied Mechanics Reviews*, 1(56) 1-32.

16. http://www.cs.wright.edu/~jslater/SDTCOutreachWebsite/nat_frequency.htm

17. http://aviation-safety.net/wikibase/wiki.php?id=42405 & http://www.yachtsurvey.com/composite_troubles_in_aircraft.htm

18. Detection of Structural Damage Through Changes in Frequency-Salawu, O. S., 1997, *Engineering Structures,* 19(9) 718-723

19. A Summary Review of Vibration-Based Damage Identification Methods- Doebling, S. W., Farrar, C. R., and Prime, M. B., 1998, *Los Alamos National Laboratory* (30)91–105

20. Identification and Health Monitoring of Structural and Mechanical Systems From Changes in their Vibration Characteristics: A Literature Review- Scott W. Doebling, Charles R. Farrar , Michael B. Prime, Daniel W. Shevitz ,1996, *Los Alamos National Laboratory* 10 (213)1762-1770

21. Free Vibration of Delaminated Composite Sandwich Beams- Jian Shiun Hu and Chyanbin, 1995, *AIAA Journal*, 10(33)

22. Free vibration Analysis of Beams on Elastic Foundation- Thambiratnam, D. and Y. Zhuge, 1996, *Computers & Structures* , 6(60)971-980

23. Vibration of Beams with Double Delaminations- Christian Della and Dongwei Shu, 2005, *Journal of Sound and Vibration*, 3(282) 919 - 935

24. Vibration of Composite Beams With Two Overlapping Delaminations- Christian N. Della, Dongwei Shu , Yapu Zhao, 2005, *Acta Mechanica Sinica*, 1(21)47 – 55

25. Beam Delamination Modeling in ANSYS®- Johan Belecave, 2011, *Internal report, Department of Aerospace Eng., Ryerson university, Toronto, Canada*

26. Dynamic Stiffness Matrix for Free Vibration Analysis of Doubly Delaminated Layered Beams- Seyed M. Hashemi and Nicholas H.Erdelyi, 2011, *International Conference on Acoustic and Vibration 21-22 Dec 2011*

27. ANSYS® 12.0.1 library

28. The Finite Element Method**,** fifth edition volume1: the Basis- 0**.** c**.** Zienkiewicz**,** 1978, *McGraw-Hill*, ISBN 0 7506 5049 4

29. Analysis of Natural Frequencies of Delaminated Composite Beams Based on Finite Element Method- Krawczuk M, Ostachowicz W, Zuk A., 1996, *Structural Engineering and Mechanics* (4)243-55.

30. Health Monitoring of Structures - Multiple Delamination Dynamics in Composite Beams- Wahyu Lestari' and Sathya Hanagud,1999, *American Institute of Aeronautics & Astronautics, A99-24831*

31. Vibration of a Delaminated Beam-plate Relative to Buckled States- Yin WL, Jane KC.,1988, *In Proceedings of AIAA/ASME/ASCE/AHS/ASC 29th SDM Conference*, Williamsburg, VA, 860-870.

32. Free Vibration of a Compressed Delaminated Plate- Chen HP, Goggin P., 1992, *Proceedings of AIAA/ASME/ASCE/AHS/ASC 33rd SDM Conference*, Dallas, TX, 1830-1840.

33. http://www.mrl.columbia.edu/ntm/level1/ch05/html/l1c05s03.html

34. Fundamental Frequency of Fully Clamped Composite Sandwich Plate- E.V. Morozov and A.V. Lopation, 2010, *Journal of Sandwich Structures & Materials,* 12 (5) 591-619

35. Elasto-Plastic Finite Element Analysis of Fully Clamped Laminated Thermoplastic Composite Plates- Karakuzu, Ramazan; Yucel, U Kerim and Aslan, Züleyha,2003, *Journal of Reinforced Plastics and Composites,* 6(22) 549 – 561

36. Vibration-based Analysis of Damage in Composites- K. Oruganti, M. Mehdizadeh , S. John , I. Herszberg, 2009, *Institute of Materials Engineering Australasia Ltd* , (33)496-504

37. A First Course in the Finite Element Method, Fourth Edition- Daryl L. Logan, 2007, *Nelson, a division of Thomson Canada Limited*, ISBN: 0-534-55298-6

38. Free Vibrations of Delaminated Beams- M.H. Shen and J.E. Grady, 1992. *AIAA Journal* 30(5), 1361-1370

39. Vibration analysis of delaminated composite beams using analytical and FEM models- H.Callioglu and G.Atlihab, Indian Journal of Engineering & Material Sciences (18),7-14

Appendix A: Element Types in ANSYS®

==

The reason for having a variety of elements in ANSYS® is:

First of all there is a logical plan from the beginning of the simulation. For instance, if structural analysis is required, there is no need to use an element with thermal degrees of freedom or vice versa. It should be noted that ANSYS® has some elements that are multi-functional and are capable to do either analysis.

As it is well published, ANSYS® is a huge program. It is quite capable in many physics fields. That would require ANSYS® to have elements for electric fields, electromagnetics, fluids, acoustics, and explicit dynamics analysis.

Secondly, the new version of ANSYS® has been issued in the past two decades, and new types of elements are introduced for researches. However, ANSYS® is keeping the old type of elements as well, which were used in modeling in the past, for compatibility with older models.

The following element types are supported by FE Modeler.

Shape Category	The Mechanical APDL Application Element Type
Point	MASS21[1], MASS71 MASS166 FOLLW201
2D Linear Line	LINK1[1], BEAM3[1], INFIN9 CONTAC12 BEAM23[1], LINK32[1], SHELL150 BEAM54[1], SHELL163 FLUID129 SHELL208 COMBI214, SURF251
3D Linear Line	BEAM4[1], COMBIN7, LINK8[1], LINK10[1], LINK11[1], COMBIN14[1], PIPE16, PIPE18, PIPE20, BEAM24, LINK31, LINK33[1], LINK34[1], COMBIN37[1], FLUID38, COMBIN39, COMBIN40, BEAM44[1], CONTAC52, PIPE59, PIPE60, LINK68[1], CIRCU94, FLUID116, CIRCU124, CIRCU125, TRANS126, FLUID138, LINK160[1], BEAM161, COMBI165, LINK167[1], PRETS179, LINK180[1], MPC184, BEAM188[1]
2D Quadratic Line	SURF151[2], SURF153[2], SHELL209
3D Quadratic Line	SURF156[2], BEAM189[1]
2D Linear Triangle	TRANS109
2D Linear Quadrilateral	PLANE13[2], PLANE25[2], FLUID29[2], PLANE42[2], PLANE55[2], PLANE67[2], PLANE75[2], FLUID79, FLUID81, INFIN110[2], FLUID141[2], PLANE162[2], PLANE182[2], INTER192, INTER202
3D Linear Quadrilateral	SHELL28[1], SHELL41[1,2], [2], INFIN47[2], SHELL57[1,2], SHELL63[1,2], INFIN115[2], FLUID130[2], SHELL131[2], FLUID136[2], SHELL143[1,2], SHELL157[1,2], SHELL163[1,2], SHELL181[1,2], SURF252[2]
2D Quadratic Triangle	PLANE2, PLANE35, PLANE146
2D Quadratic Quadrilateral	PLANE53[2], PLANE77[2], PLANE78[2], PLANE82[2], PLANE83[2], INFIN110[2], HF118[2], PLANE121[2], PLANE145[2], PLANE183[2], INTER193, INTER203, PLANE223[2], PLANE230[2]
3D Quadratic Quadrilateral	SHELL132[2], FLUID136[2], SHELL150[2], SURF152[2], SURF154[2], SHELL157[1,2], SHELL281[1,2]
Linear Hexahedral	SOLID5[2], FLUID30[2], SOLID45[2], SOLID62[2], SOLID64[2], SOLID65[2], SOLID69[2],SOLID70[2], FLUID80, SOLID96[2], SOLID97[2], INFIN111[2],FLUID142[2], SOLID164[2],SOLID185[2], SOLID190[2], INTER195
Quadratic Tetrahedral	SOLID87, SOLID92, SOLID98, HF119, SOLID123, SOLID127, SOLID148, SOLID168, SOLID187, SOLID227, SOLID232
Quadratic Hexahedral	SOLID90[2], SOLID95[2], INFIN111[2], SOLID117[2], HF120[2], SOLID122[2], SOLID128[2], SOLID147[2], SOLID186[2], INTER194 , INTER204 , SOLID226[2], SOLID231[2]
Meshing Facet	MESH200
Contact	TARGE170[3], TARGE173[3], TARGE174[3], TARGE175[3]

Table A1- Different types of elements in ANSYS®, adapted from [27]

“3D Shell Elements:
SHELL63: This is an older shell element but still very widely used. It can analyze large deflections but not plasticity.

SHELL93: Quadratic version of SHELL63. In addition it can analyze plasticity.

SHELL181: Newest linear shell element. Has plasticity and can be used to model laminated composites and sandwich panels.

Beam Elements

Line elements are an engineering abstraction from shells; they are physically known as a simple line, which represents a long slender structure that can carry axial, bending, shear and twisting forces. Real constants define the cross sectional properties of the beam element.

BEAM3: Simple 2D beam element. This element must be created in XY plane.

BEAM4: Beam element that can be modeled in 3D space. Has large deflection.

BEAM44: Tapered beam element. Can use the beam tool to define a section and have real constants automatically calculated.

BEAM188: Timoshenko beam that includes shear deformation. Can handle plasticity and can define a section with the beam tool. It can have a built-up cross section with more than one material. By default this is a true linear element. You need a lot of them to give the same results as one BEAM3/4/44/189. In release 8.0 turn on KEYOPT (3) = 2 to give an internal quadratic shape function to eliminate this problem. Also, it can display stresses on the actual beam shape as if it was solid.

BEAM189: Three node quadratic version of BEAM188. This is the ultimate beam element that can do everything well” [27].

Establish Analysis Settings:

Every specific analysis requires specific settings to achieve various solution options. For more customized option, researcher must refer to specific analysis type. The values can be selected in this section or default values may be used.

1. Define Initial Conditions:

   This step depends on analysis type selected.

2. Apply Loads and Supports:

Loads and support types are applied based on the type of analysis. In any application these may be different. For stress analysis, pressure and forces are considered while in thermal analysis, temperatures and convections are involved.

3. Solve:

In this step, the initial stages of solution process starts. The solution can be obtained via local or remote machine which is sufficiently powerful.

Some applications such as non -linear solutions may take a significant amount of time to complete. For convenience, a status bar is provided to display overall progress. More detailed information can be accessed from Solution Information Object.

4. Review Results:

In Mechanical Application section results are available in post processing. Depending on the different systems, various results may be observed. In a structural analysis, the results from equivalent stress may be interesting while in thermal analysis, temperature or heat flux may be desired.

To add result objects in the Mechanical application:

1. Highlight a Solution object in the Tree.
2. Select the appropriate result from the Solution context toolbar or use the right- click option.

To review results in the Mechanical Application:

1. Click on a result object in the Tree.
2. After the analysis has been completed, the results can be reviewed and interpreted as follow:

- Contour results - Displays a contour plot of a result such as stress over geometry.
- Vector Plots - Displays certain results in the form of vectors (arrows).
- Probes - Displays a result at a single point of time, or as a variation over time, using a graph and a table.
- Charts - Displays different results over time, or displays one result against another result, for example, force vs. displacement.
- Animation - Animates the variation of results over geometry including the deformation or delamination of the structure.
- Stress Tool - to evaluate a design using various failure theories.

- Fatigue Tool - to perform advanced life prediction calculations.
- Contact Tool - to review contact region behavior in complex assemblies.
- Beam Tool - to evaluate stresses in line body representations.

Analysis Types:

Several types of analyses in the Mechanical application can be performed using pre-configured analysis systems (see Create Analysis System). For doing more advanced analysis, Commands objects in the Mechanical interface can be used. This will allow the user to enter the Mechanical APDL application commands in the Mechanical application to perform the analysis.

There are various analysis types that can be performed in the Mechanical interface. Depending on how experienced a user is, the results and features could vary. These analyses include:

- Electric Analysis
- Explicit Dynamics Analysis
- Harmonic Response Analysis
- Linear Buckling Analysis
- Magneto-static Analysis
- Modal Analysis
- Random Vibration Analysis
- Response Spectrum Analysis
- Shape Optimization Analysis
- Static Structural Analysis
- Steady-State Thermal Analysis
- Thermal-Electric Analysis
- Transient Structural Analyses
- Transient Thermal Analysis
- Special Analysis Topics
- Wizards

Appendix B: ANSYS® Templates

==

ANSYS® MACRO FOR THE 2D SINGLE CENTERED DELAMINATION MODELING

```
!--------------------------------------------- 2D -----------------------------------------------------
! INPUT TO THIS MACRO:
! ARG1 - L length of the beam
! ARG2 - H1 height of the bottom beam
! ARG3 - E1 young modulus of the bottom beam
! ARG4 - DENS1 density of the bottom beam
! ARG5 - H2 height of the top beam
! ARG6 - E2 young modulus of the top beam
! ARG7 - DENS2 density of the top beam
! ARG9 - A1 length of the delamination
! ARG10 - PREC element size

!----------------------------------------------------------------------------------------------------------
KEYW,PR_STRUC,1
!-------------------------------------------------DEFINITION OF PARAMETERS---------------------
*SET,L,ARG1
*SET,H1,ARG2
*SET,H2,ARG5
*SET,H,H1+H2
*SET,DENS1,ARG4
*SET,DENS2,ARG7
*SET,I1,H1**3/12
*SET,I2,H2**3/12
*SET,E1,ARG3
*SET,E2,ARG6
*SET,D1,0
*SET,A1,ARG9
*SET,PREC,AR10
*ASK,MOD,'1 - cantilever beam 2 - clamped-clamped'
!----------------------DEFINITION OF ELEMENTS----------------------------------------------------
/PREP7
ET,1,PLANE182
KEYOPT,1,3,3
!--------------------DEFINITION OF REAL CONSTANTS--------------------------------------------
R,1,1,
```

```
!----------------------------------------------DEFINITION OF MATERIAL PROPRIETIES----------
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,E1
MPDATA,PRXY,1,,0
MPDATA,DENS,1,,DENS1
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,E2
MPDATA,PRXY,2,,0
MPDATA,DENS,2,,DENS2
!--------------------------------------------------------MODELING---------------------------------------
/PREP7
RECTNG,-L/2,L/2,H1/2-H2/2,-H/2
RECTNG,-L/2,L/2,H1/2-H2/2,H/2
!---------------------------------------------------------MESHING----------------------------------------
TYPE,1
MAT,1
REAL,1
 ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,1
TYPE,1
MAT,2
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,2
!----------------------------------------------------CONSTRAIN NODES------------------------------
*DO,i,-L/2+PREC,L/2-PREC,PREC
*IF,i,LT,D1-A1/2,OR,i,GT,D1+A1/2,THEN
NSEL,S,LOC,X,i-0.0001,i+0.0001
NSEL,R,LOC,Y,H1/2-H2/2-0.0001,H1/2-H2/2+0.0001
CP,NEXT,UX,ALL
```

```
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
*IF,MOD,EQ,1,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
*IF,MOD,EQ,2,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,ALL,,,,,
ALLSEL,ALL
NSEL,S,LOC,X,L/2-0.0001,L/2+0.0001
D,ALL,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
!---------------------------------------------------------SOLUTION--------------------------------------
ANTYPE,2
MODOPT,LANB,10
EQSLV,SPAR
MXPAND,10, , ,0
LUMPM,0
PSTRES,0
MODOPT,LANB,10,0,0, ,OFF
!lauching the solver
/SOLU
/STATUS,SOLU
SOLVE
FINISH
```

ANSYS® MACRO FOR THE 2D SINGLE OFF-CENTERED DELAMINATION MODELING

```
!-------------------------------------------------2D ---------------------------------------------------------
! INPUT TO THIS MACRO:
! ARG1 - L length of the beam
! ARG2 - H1 height of the bottom beam
! ARG3 - E1 young modulus of the bottom beam
! ARG4 - DENS1 density of the bottom beam
! ARG5 - H2 height of the top beam
! ARG6 - E2 young modulus of the top beam
```

```
! ARG7 - DENS2 density of the top beam
! ARG8 - D1 position of the delamination from the middle of the beam
! ARG9 - A1 length of the delamination
! ARG10 - PREC element size

!--------------------------------------------------------------------------------------------------------------
KEYW,PR_STRUC,1
!-------------------------------------------------DEFINITION OF PARAMETERS----------------------
*SET,L,ARG1
*SET,H1,ARG2
*SET,H2,ARG5
*SET,H,H1+H2
*SET,DENS1,ARG4
*SET,DENS2,ARG7
*SET,I1,H1**3/12
*SET,I2,H2**3/12
*SET,E1,ARG3
*SET,E2,ARG6
*SET,D1,ARG8
*SET,A1,ARG9
*SET,PREC,AR10
*ASK,MOD,'1 - cantilever beam 2 - clamped-clamped'
!-------------------------------------------------DEFINITION OF ELEMENTS--------------------------
/PREP7
ET,1,PLANE182
KEYOPT,1,3,3
!------------------------------------------------DEFINITION OF REAL CONSTANTS-----------------
R,1,1,
!----------------------------------------------DEFINITION OF MATERIAL PROPRIETIES----------
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,E1
MPDATA,PRXY,1,,0
MPDATA,DENS,1,,DENS1
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,E2
MPDATA,PRXY,2,,0
MPDATA,DENS,2,,DENS2
!----------------------------------------------------------MODELING--------------------------------------
```

```
/PREP7
RECTNG,-L/2,L/2,H1/2-H2/2,-H/2
RECTNG,-L/2,L/2,H1/2-H2/2,H/2
!-----------------------------------------------------------MESHING----------------------------------------
TYPE,1
MAT,1
REAL,1
 ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,1
TYPE,1
MAT,2
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,2
!-------------------------------------------------------CONSTRAIN NODES-------------------------------
*DO,i,-L/2+PREC,L/2-PREC,PREC
*IF,i,LT,D1-A1/2,OR,i,GT,D1+A1/2,THEN
NSEL,S,LOC,X,i-0.0001,i+0.0001
NSEL,R,LOC,Y,H1/2-H2/2-0.0001,H1/2-H2/2+0.0001
CP,NEXT,UX,ALL
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
*IF,MOD,EQ,1,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
*IF,MOD,EQ,2,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,,ALL,,,,,
```

```
ALLSEL,ALL
NSEL,S,LOC,X,L/2-0.0001,L/2+0.0001
D,ALL,,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
!-----------------------------------------------------------SOLUTION--------------------------------------
ANTYPE,2
MODOPT,LANB,10
EQSLV,SPAR
MXPAND,10, , ,0
LUMPM,0
PSTRES,0
MODOPT,LANB,10,0,0, ,OFF
!lauching the solver
/SOLU
/STATUS,SOLU
SOLVE
FINISH
```

ANSYS® CODE FOR 2D DOUBLE ENVELOPED DELAMINATION MODELING H3=0.3, H4=0.3 and H5=0.4, E3=E4=E5, D1=D2=0,a1=6 a2=6

```
!--------------------------------------------------DEFINITION OF PARAMETERS----------------------
*SET,L,60
*SET,H1,0.3
*SET,H2,0.3
*SET,H3,0.4
H=H1+H2
*SET,DENS1,10000
*SET,DENS2,10000
*SET,DENS3,10000
*SET,I1,H1**3/12
*SET,I2,H2**3/12
*SET,I3,H3**3/12
*SET,E1,10000000000
*SET,E2,10000000000
*SET,E3,10000000000
!definition of the delamination
*SET,D1,0
*SET,A1,6
```

```
*SET,D2,0
*SET,A2,6
*SET,PREC,0.1
*ASK,MOD,'1 - cantilever beam 2 - clamped-clamped'
!-------------------------------------------------DEFINITION OF ELEMENTS--------------------------
/PREP7
ET,1,PLANE182
KEYOPT,1,3,3
!-------------------------------------------------DEFINITION OF REAL CONSTANTS-----------------
R,1,1,
!-----------------------------------------------DEFINITION OF MATERIAL PROPRIETIES----------
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,E1
MPDATA,PRXY,1,,0
MPDATA,DENS,1,,DENS1
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,E2
MPDATA,PRXY,2,,0
MPDATA,DENS,2,,DENS2
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,E3
MPDATA,PRXY,3,,0
MPDATA,DENS,3,,DENS3
!-----------------------------------------------------------MODELING-----------------------------------------------------------------
/PREP7
RECTNG,-L/2,L/2,H1/2-H2/2,-H/2
RECTNG,-L/2,L/2,H1/2-H2/2,H/2
RECTNG,-L/2,L/2,(H1/2-H2/2)+H2,(H/2)+H3
!-----------------------------------------------------------MESHING----------------------------------------
TYPE,1
MAT,1
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,1
```

```
TYPE,1
MAT,2
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,2
!layer 3
TYPE,1
MAT,3
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,3
!--------------------------------------------------------CONSTRAIN NODES------------------------------
*DO,i,-L/2+PREC,L/2-PREC,PREC
*IF,i,LT,D1-A1/2,OR,i,GT,D1+A1/2,THEN
NSEL,S,LOC,X,i-0.0001,i+0.0001
NSEL,R,LOC,Y,H1/2-H2/2-0.0001,H1/2-H2/2+0.0001
CP,NEXT,UX,ALL
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
*DO,j,-L/2+PREC,L/2-PREC,PREC
*IF,j,LT,D2-A2/2,OR,j,GT,D2+A2/2,THEN
NSEL,S,LOC,X,j-0.0001,j+0.0001
NSEL,R,LOC,Y,(H1/2-H2/2)+H2-0.0001,(H1/2-H2/2)+H2+0.0001
CP,NEXT,UX,ALL
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
!choice of a cantilever beam study
!then only one end is blocked
```

```
*IF,MOD,EQ,1,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
!choice of a clamped-clamped beam study
!then the two ends are blocked
*IF,MOD,EQ,2,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,,ALL,,,,,
ALLSEL,ALL
NSEL,S,LOC,X,L/2-0.0001,L/2+0.0001
D,ALL,,,,,,ALL,,,,,
ALLSEL,ALL
*ENDIF
!-----------------------------------------------------------SOLUTION---------------------------------------
ANTYPE,2
MODOPT,LANB,10
EQSLV,SPAR
MXPAND,10, , ,0
LUMPM,0
PSTRES,0
MODOPT,LANB,10,0,0, ,OFF
/SOLU
/STATUS,SOLU
SOLVE
FINISH
```

ANSYS® CODE FOR 2D DOUBLE NON-ENVELPOED DELAMINATION MODELING

H3=0.3, H4=0.3 and H5=0.4, E3=E4=E5,D1=3 D2=-3,a1=6 a2=6

```
!--------------------------------------------------DEFINITION OF PARAMETERS----------------------
*SET,L,60
*SET,H1,0.3
*SET,H2,0.3
*SET,H3,0.4
H=H1+H2
*SET,DENS1,10000
*SET,DENS2,10000
*SET,DENS3,10000
```

```
*SET,I1,H1**3/12
*SET,I2,H2**3/12
*SET,I3,H3**3/12
*SET,E1,10000000000
*SET,E2,10000000000
*SET,E3,10000000000
!definition of the delamination
*SET,D1,3
*SET,A1,6
*SET,D2,-3
*SET,A2,6
*SET,PREC,0.1
*ASK,MOD,'1 - cantilever beam 2 - clamped-clamped'
!------------------------------------------------------DEFINITION OF ELEMENTS----------------------------
/PREP7
ET,1,PLANE182
KEYOPT,1,3,3
!----------------------------------------------------DEFINITION OF REAL CONSTANTS------------------
R,1,1,
!--------------------------------------------------DEFINITION OF MATERIAL PROPRIETIES----------
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,E1
MPDATA,PRXY,1,,0
MPDATA,DENS,1,,DENS1
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,2,,E2
MPDATA,PRXY,2,,0
MPDATA,DENS,2,,DENS2
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,3,,E3
MPDATA,PRXY,3,,0
MPDATA,DENS,3,,DENS3
!---------------------------------------------------------------------------------------MODELING--------------------------------------------------------------------------------------------
/PREP7
RECTNG,-L/2,L/2,H1/2-H2/2,-H/2
RECTNG,-L/2,L/2,H1/2-H2/2,H/2
RECTNG,-L/2,L/2,(H1/2-H2/2)+H2,(H/2)+H3
!----------------------------------------------------------------MESHING-------------------------------------------
```

```
TYPE,1
MAT,1
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,1
TYPE,1
MAT,2
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,2
!layer 3
TYPE,1
MAT,3
REAL,1
ESYS,0
SECNUM,
ESIZE,PREC,0,
MSHAPE,0,2D
MSHKEY,0
AMESH,3
!-------------------------------------------------------CONSTRAIN NODES------------------------------
*DO,i,-L/2+PREC,L/2-PREC,PREC
*IF,i,LT,D1-A1/2,OR,i,GT,D1+A1/2,THEN
NSEL,S,LOC,X,i-0.0001,i+0.0001
NSEL,R,LOC,Y,H1/2-H2/2-0.0001,H1/2-H2/2+0.0001
CP,NEXT,UX,ALL
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
*DO,j,-L/2+PREC,L/2-PREC,PREC
*IF,j,LT,D2-A2/2,OR,j,GT,D2+A2/2,THEN
```

```
NSEL,S,LOC,X,j-0.0001,j+0.0001
NSEL,R,LOC,Y,(H1/2-H2/2)+H2-0.0001,(H1/2-H2/2)+H2+0.0001
CP,NEXT,UX,ALL
CP,NEXT,UY,ALL
ALLSEL,ALL
*ENDIF
*ENDDO
!choice of a cantilever beam study
!then only one end is blocked
*IF,MOD,EQ,1,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,ALL,,,,
ALLSEL,ALL
*ENDIF
!choice of a clamped-clamped beam study
!then the two ends are blocked
*IF,MOD,EQ,2,THEN
NSEL,S,LOC,X,-L/2-0.0001,-L/2+0.0001
D,ALL,,,,,ALL,,,,
ALLSEL,ALL
NSEL,S,LOC,X,L/2-0.0001,L/2+0.0001
D,ALL,,,,,ALL,,,,
ALLSEL,ALL
*ENDIF
!-----------------------------------------------------------SOLUTION-------------------------------------
ANTYPE,2
MODOPT,LANB,10
EQSLV,SPAR
MXPAND,10, , ,0
LUMPM,0
PSTRES,0
MODOPT,LANB,10,0,0, ,OFF
/SOLU
/STATUS,SOLU
SOLVE
FINISH
```

Appendix C: Various examples of simulations

==

Additional supporting simulation data:

This simulation is for 2D double enveloped delamination of a cantilever beam.

The assumptions are as follow:

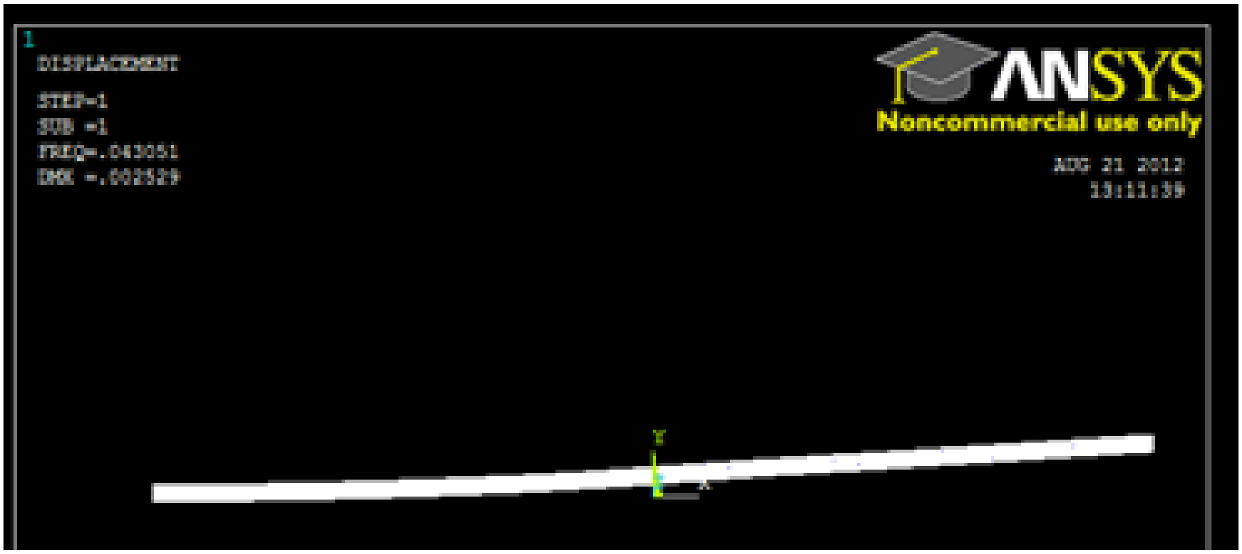
H3=0.3, H4=0.3 and H5=0.4, E3=E4=E5, d1=d2=0, a1=6 a2=12

```
/STATUS Command
File

                    S O L U T I O N   O P T I O N S

PROBLEM DIMENSIONALITY. . . . . . . . . . . . .2-D
DEGREES OF FREEDOM. . . . . . UX   UY
ANALYSIS TYPE . . . . . . . . . . . . . . . . .MODAL
   EXTRACTION METHOD. . . . . . . . . . . . . .BLOCK LANCZOS
EQUATION SOLVER OPTION. . . . . . . . . . . . .SPARSE
NUMBER OF MODES TO EXTRACT. . . . . . . . . .    10
GLOBALLY ASSEMBLED MATRIX . . . . . . . . . . .SYMMETRIC
NUMBER OF MODES TO EXPAND . . . . . . . . . .    10
ELEMENT RESULTS CALCULATION . . . . . . . . . .OFF

                    L O A D   S T E P   O P T I O N S

LOAD STEP NUMBER. . . . . . . . . . . . . . . .     1
PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . . . . . . . .ALL DATA WRITTEN
```

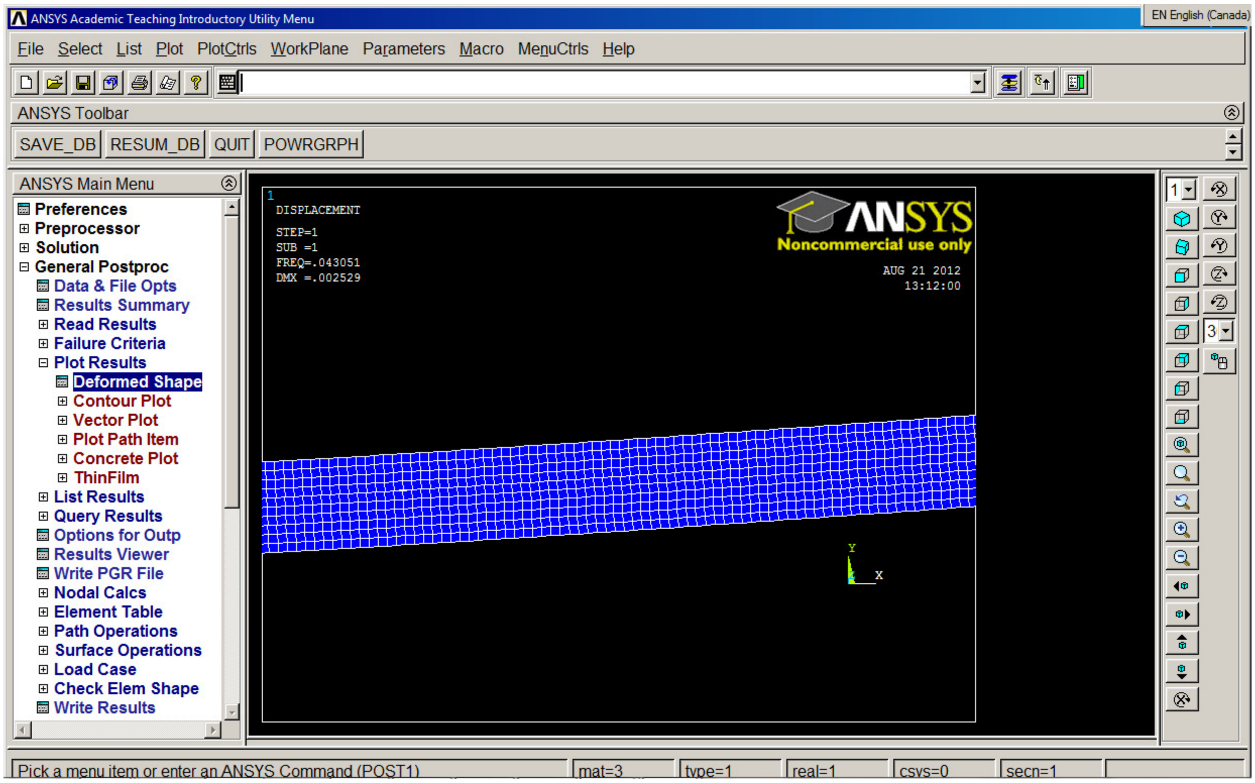
First 10 modes from ANSYS® for this beam

```
SET,LIST Command
File

 *****  INDEX OF DATA SETS ON RESULTS FILE  *****

  SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
    1 0.43051E-01        1          1         1
    2 0.27543            1          2         2
    3 0.49081            1          3         3
    4 0.92388            1          4         4
    5  1.0151            1          5         5
    6  1.2491            1          6         6
    7  1.7496            1          7         7
    8  2.5297            1          8         8
    9  2.5436            1          9         9
   10  2.8605            1         10        10
```

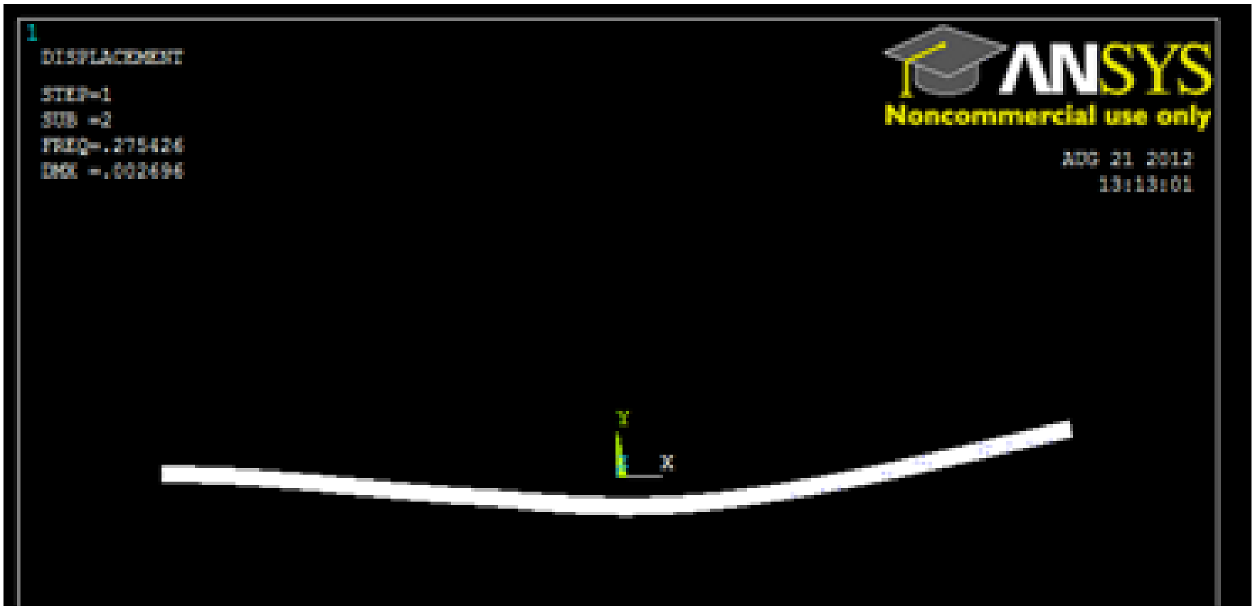
First mode



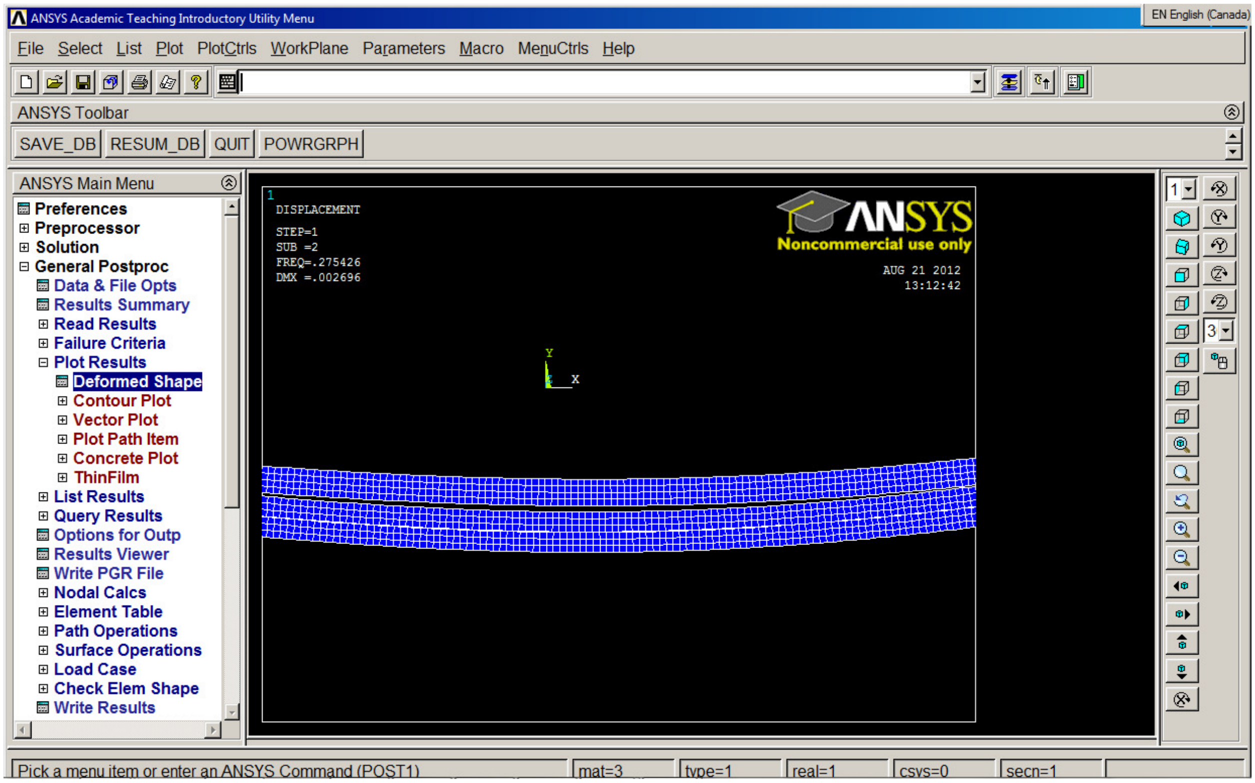
Close look on delamination area in first mode



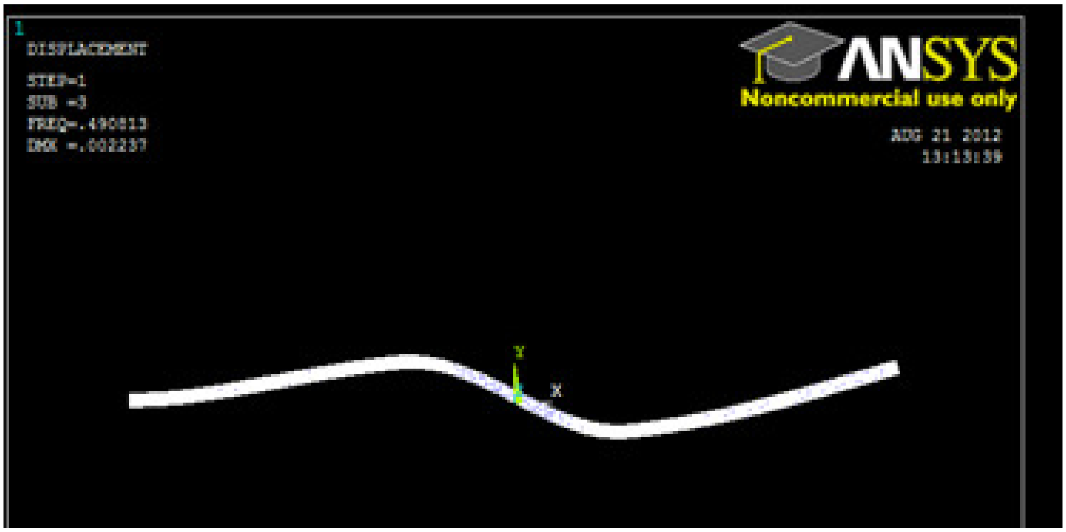
Second Mode



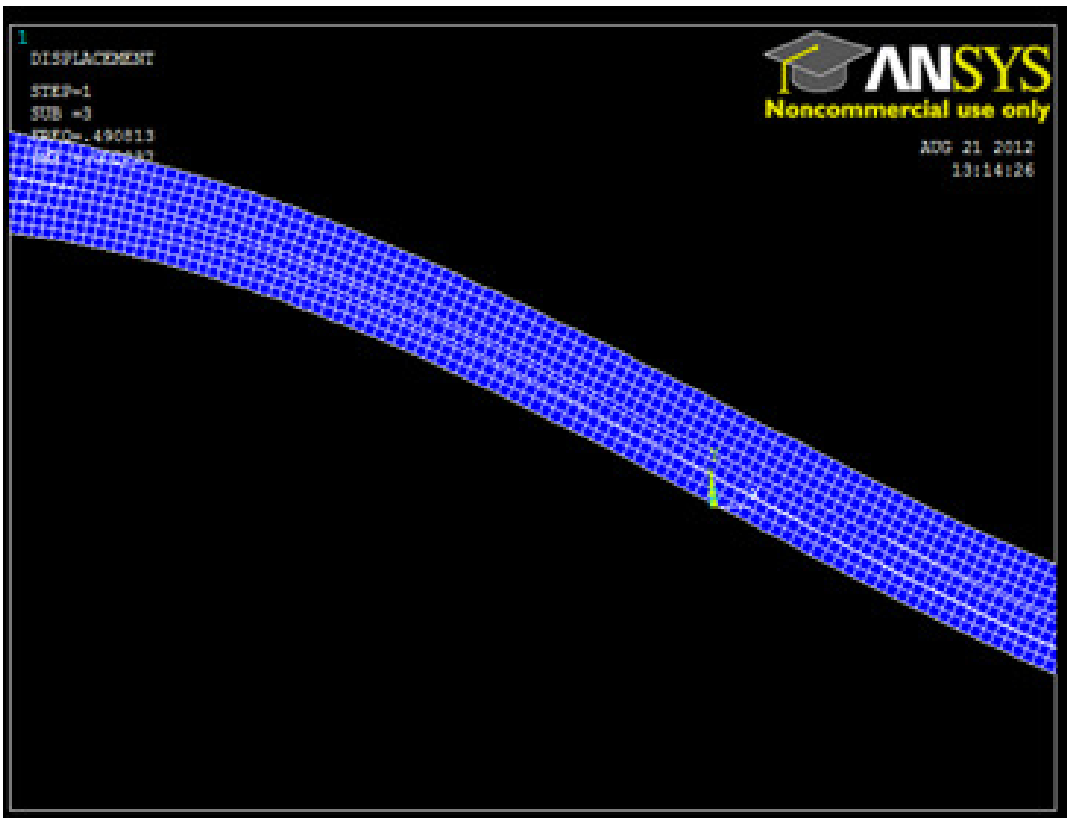
A close look at delamination area is shown in the following picture



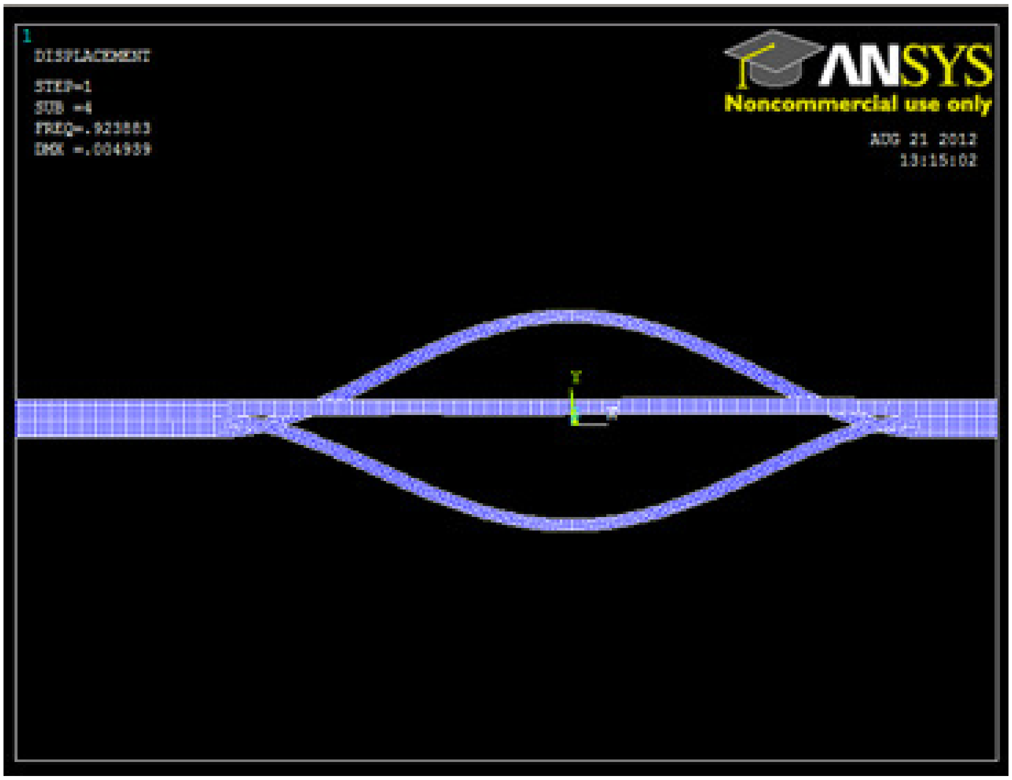
The 3rd mode shape of this beam is as follow



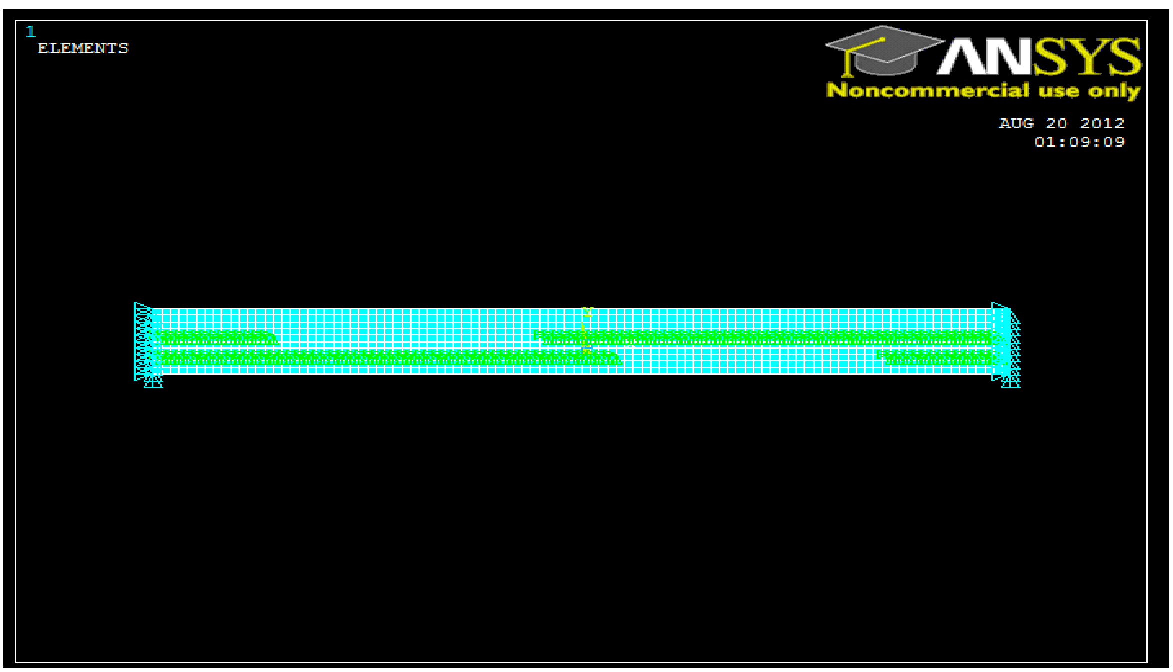
Close look on 3rd shape mode

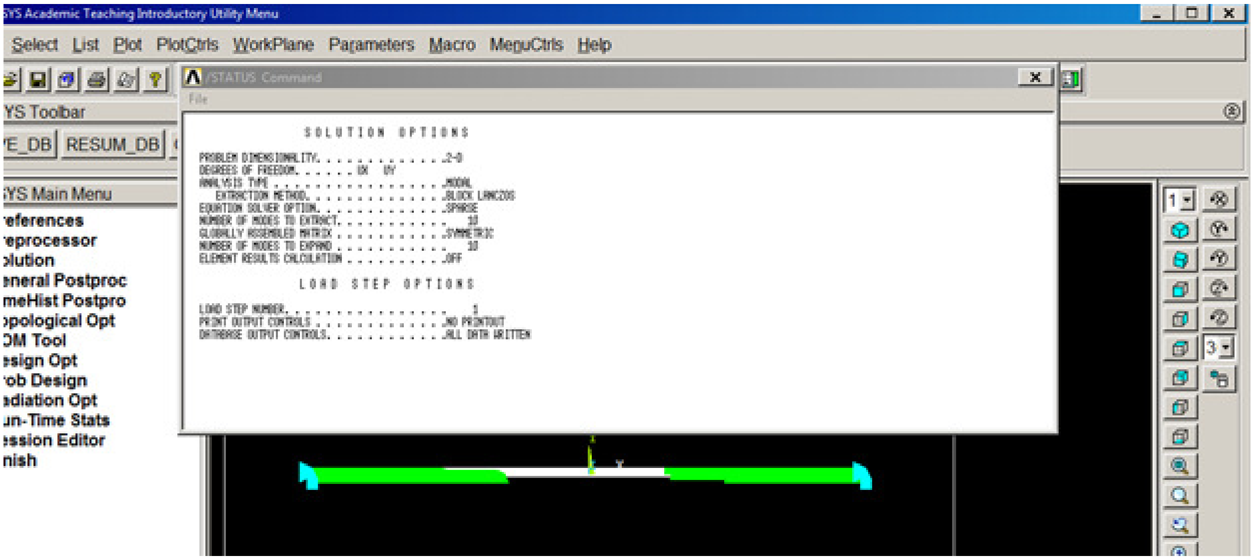


4th mode shape

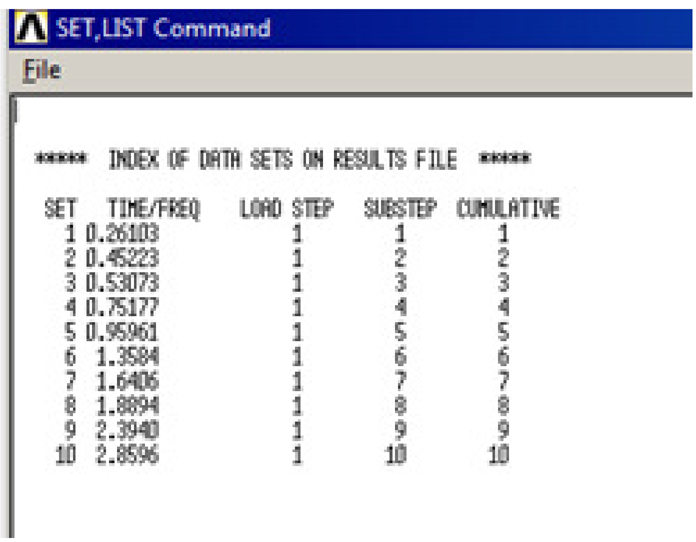


This simulation is for double non –enveloped delaminated of Clamp-Clamp beam with following assumptions:
a/L=0.4, Off center d1=-d2=3, Out of mid plane H_2=H_5=0.3H and H_3=0.4, a/H_3= 60, L/H= 60 and Homogenous E=E_1=E_2





First 10 modes obtained from this simulation:



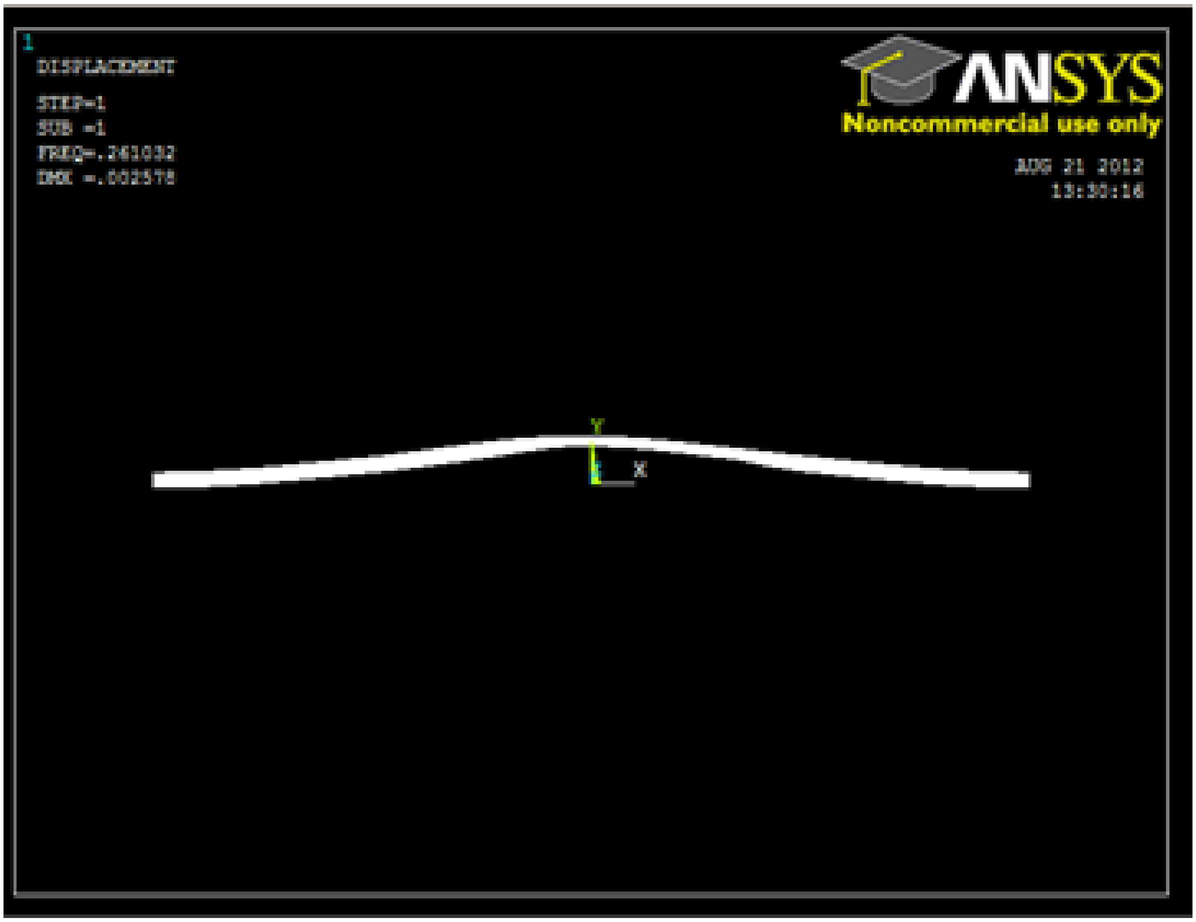
SET,LIST Command

File

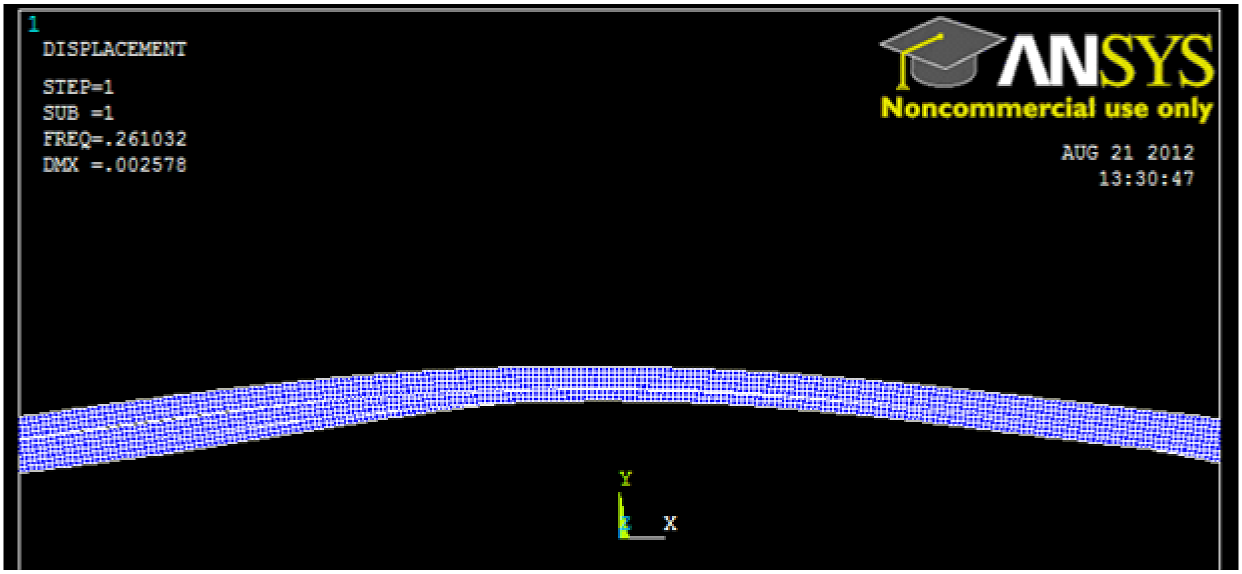
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.26103	1	1	1
2	0.45223	1	2	2
3	0.53073	1	3	3
4	0.75177	1	4	4
5	0.95961	1	5	5
6	1.3584	1	6	6
7	1.6406	1	7	7
8	1.8894	1	8	8
9	2.3940	1	9	9
10	2.8596	1	10	10

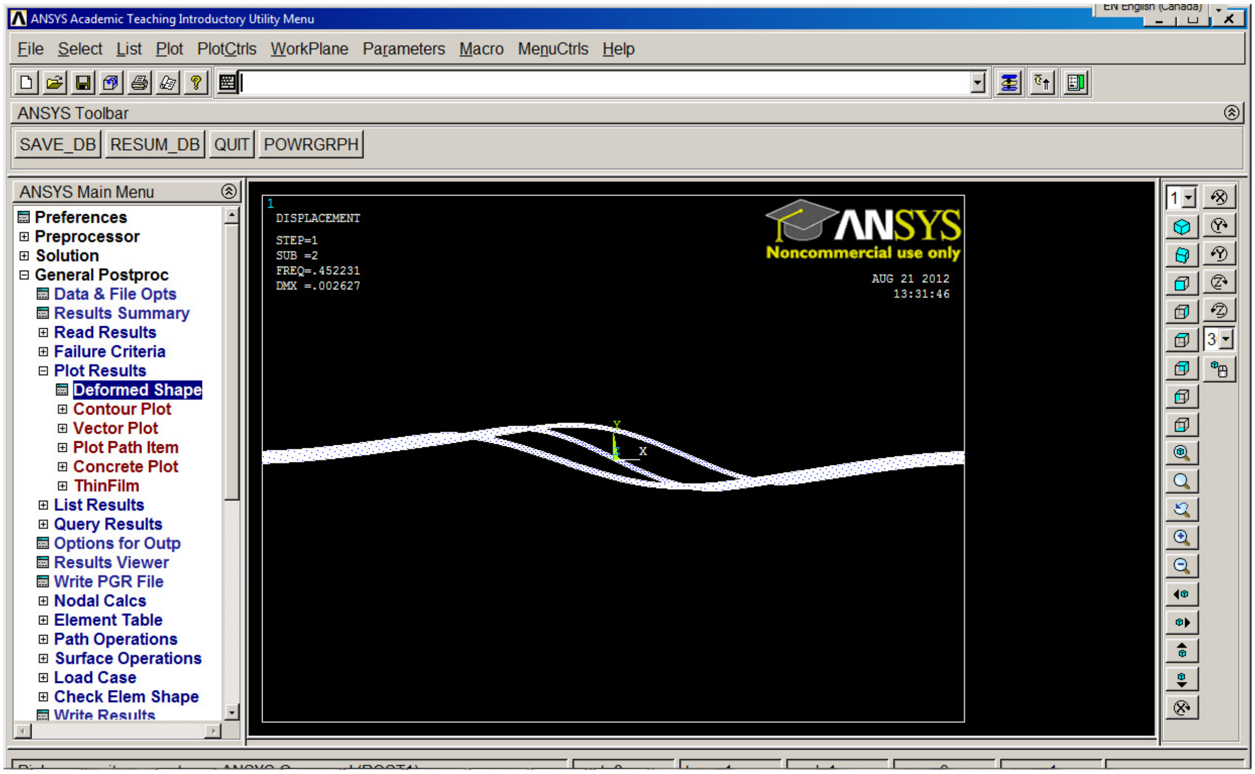
First mode



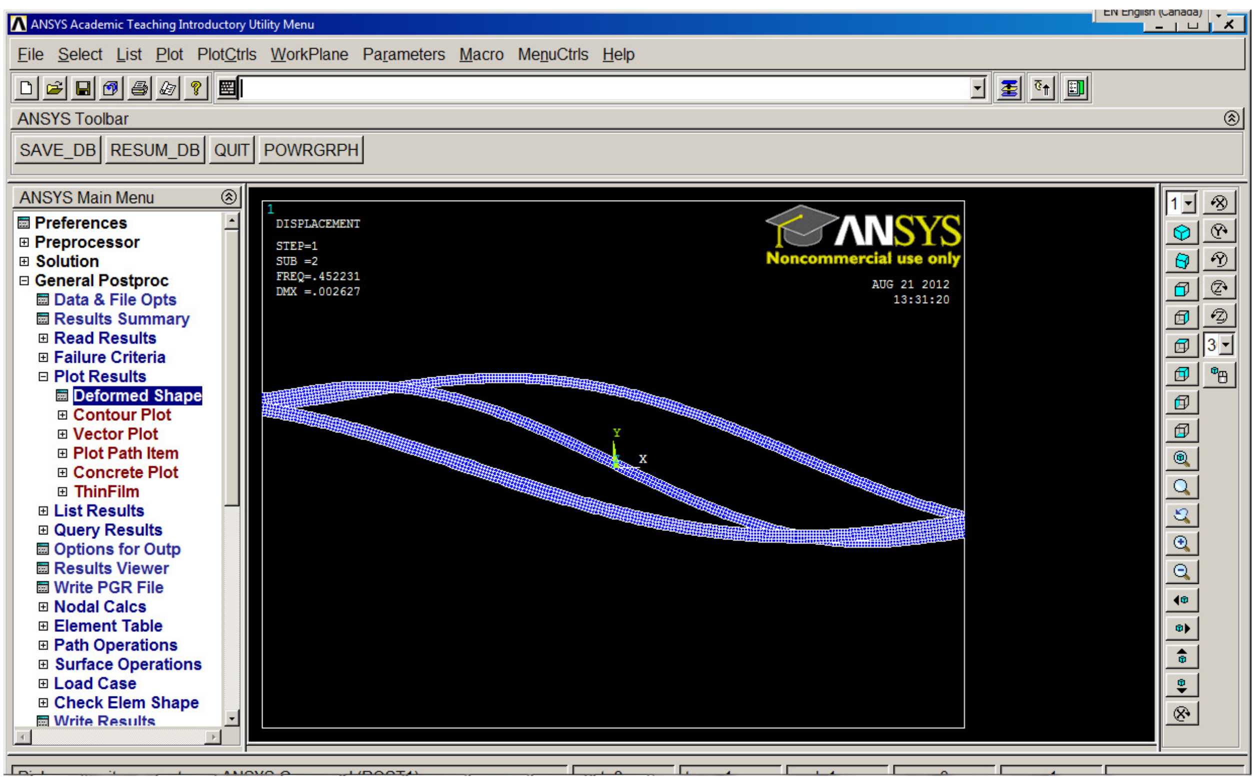
Close look at mid-point of first mode



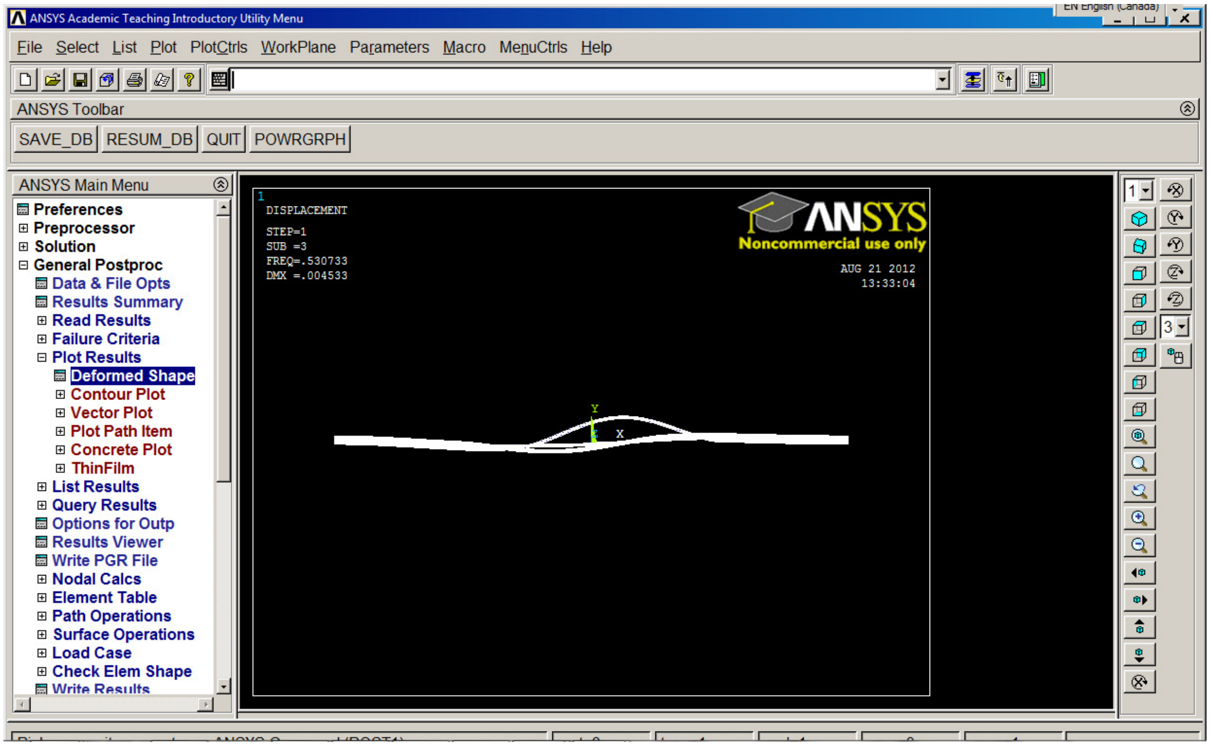
Second mode

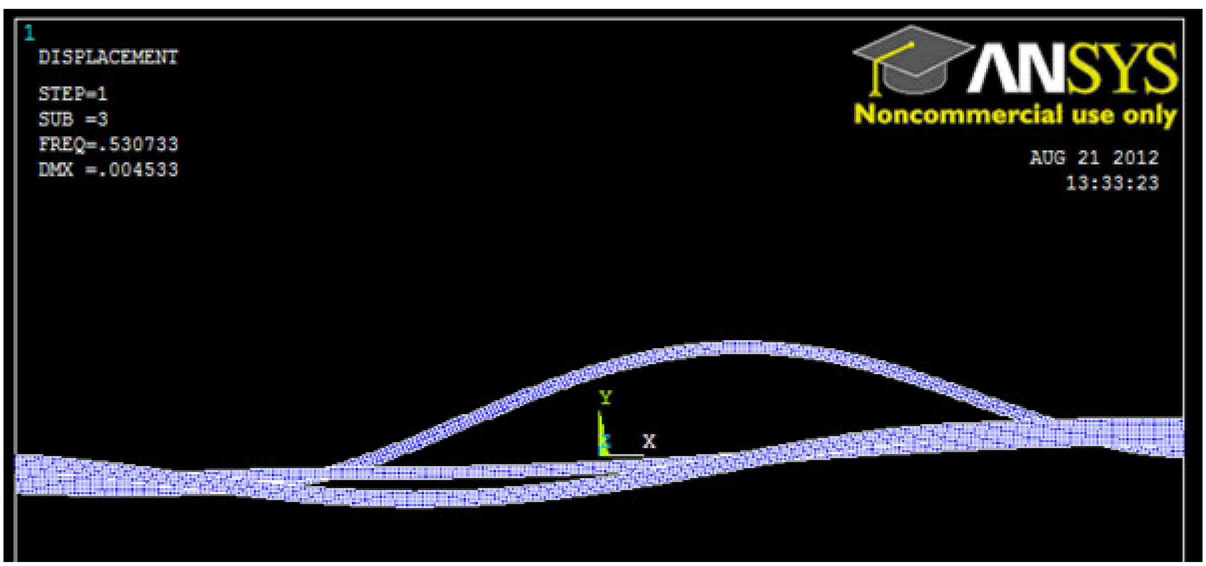


Zoom in on middle of the beam in second mode

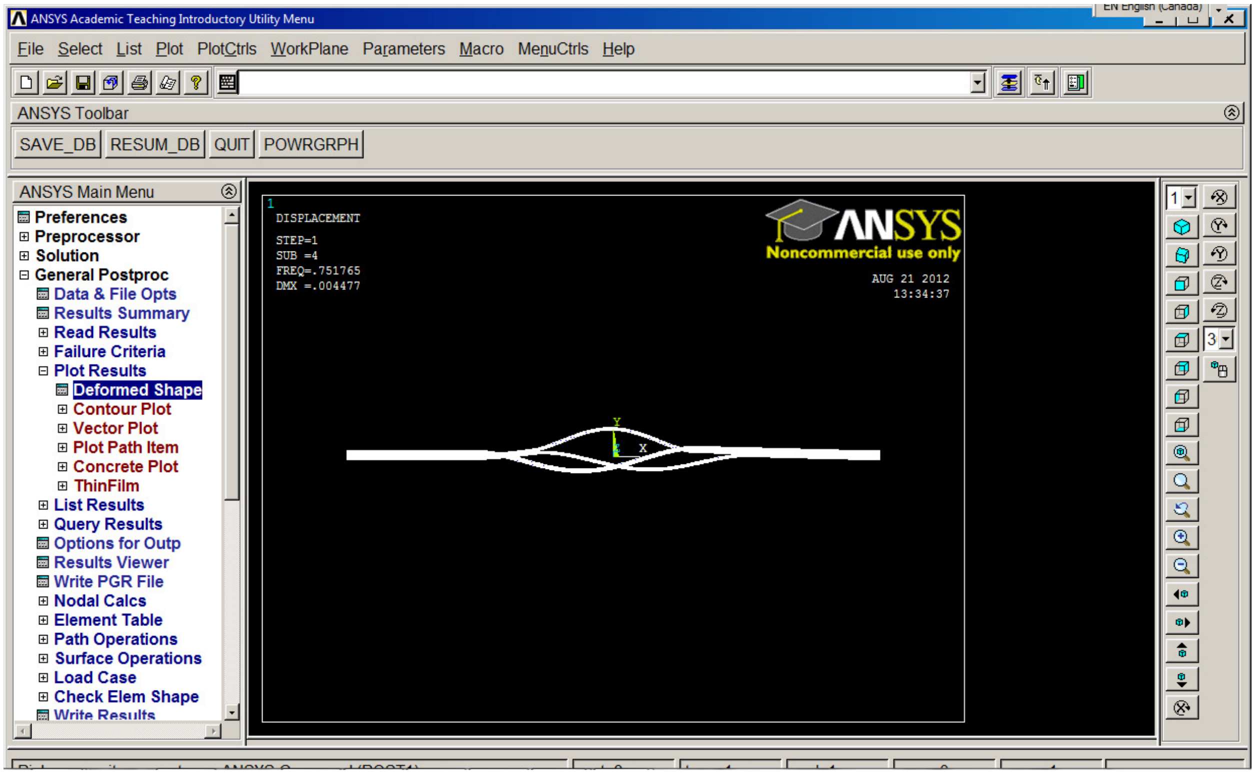


3rd mode of double non enveloped beam



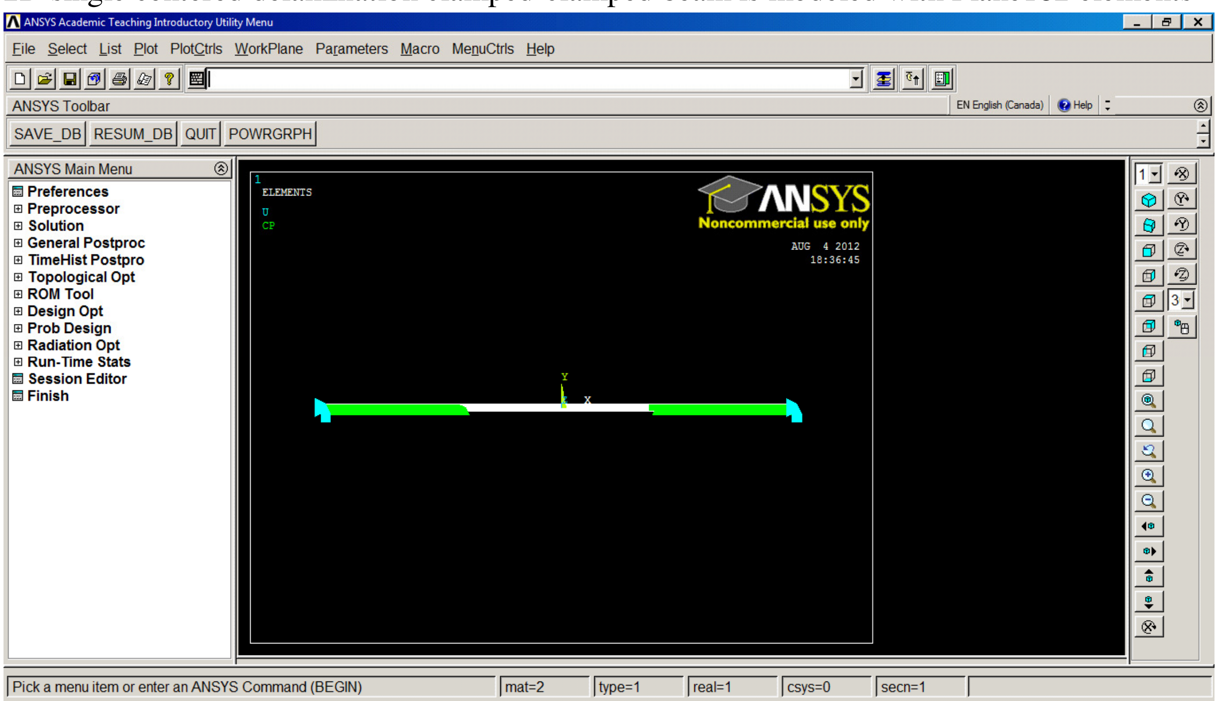


Fourth mode of this modal test



2D Clamped-Clamped Beam

2D single centered delamination clamped-clamped beam is modeled with Plane182 elements



Modal test for first 10 modes is done on this 2D model

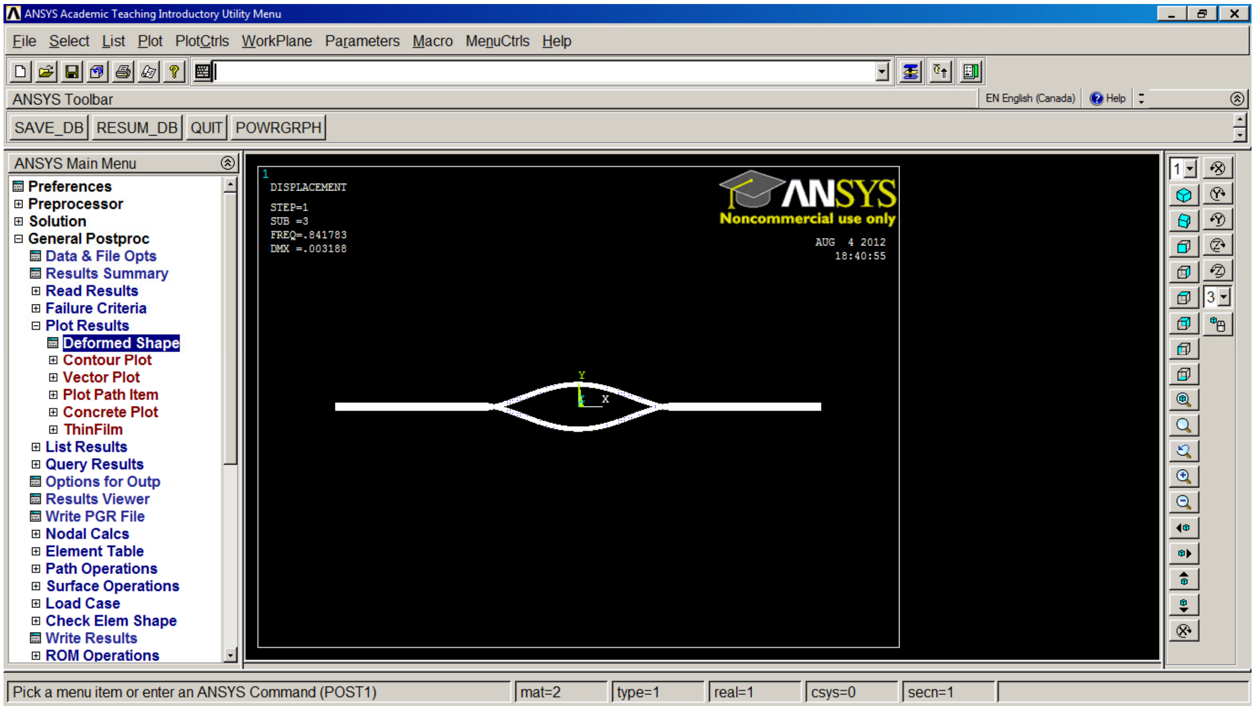
```
/STATUS Command
File

                  S O L U T I O N   O P T I O N S

   PROBLEM DIMENSIONALITY. . . . . . . . . . . . .2-D
   DEGREES OF FREEDOM. . . . . . UX   UY
   ANALYSIS TYPE . . . . . . . . . . . . . . . . .MODAL
      EXTRACTION METHOD. . . . . . . . . . . . . .BLOCK LANCZOS
   EQUATION SOLVER OPTION. . . . . . . . . . . . .SPARSE
   NUMBER OF MODES TO EXTRACT. . . . . . . . . . .    10
   GLOBALLY ASSEMBLED MATRIX . . . . . . . . . . .SYMMETRIC
   NUMBER OF MODES TO EXPAND . . . . . . . . . . .    10
   ELEMENT RESULTS CALCULATION . . . . . . . . . .OFF

                  L O A D   S T E P   O P T I O N S

   LOAD STEP NUMBER. . . . . . . . . . . . . . . .     1
   PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
   DATABASE OUTPUT CONTROLS. . . . . . . . . . . .ALL DATA WRITTEN
```

with applying B.C. ,the following result is obtained

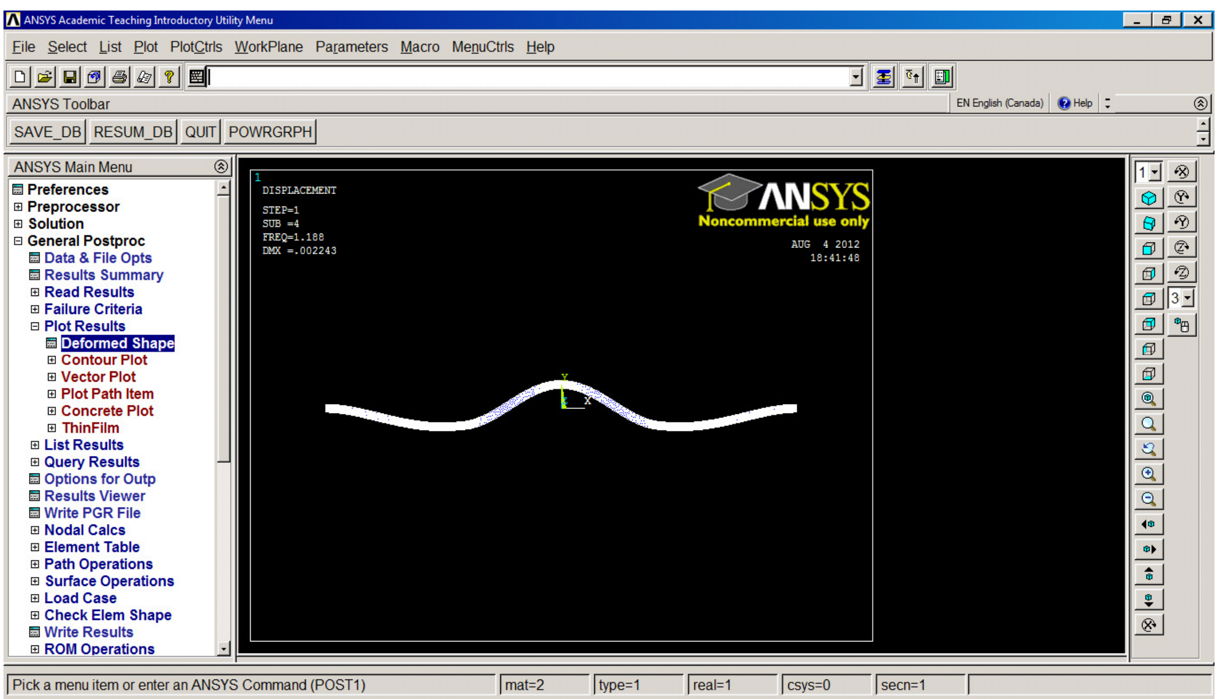
```
SET,LIST Command
File

 *****  INDEX OF DATA SETS ON RESULTS FILE  *****

  SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
    1 0.27859            1          1         1
    2 0.56021            1          2         2
    3 0.84178            1          3         3
    4  1.1877            1          4         4
    5  1.9390            1          5         5
    6  2.3162            1          6         6
    7  2.8192            1          7         7
    8  3.7773            1          8         8
    9  4.5279            1          9         9
   10  5.2618            1         10        10
```

Deformed shape of First 3 modes of this examination are illustrated as follow:

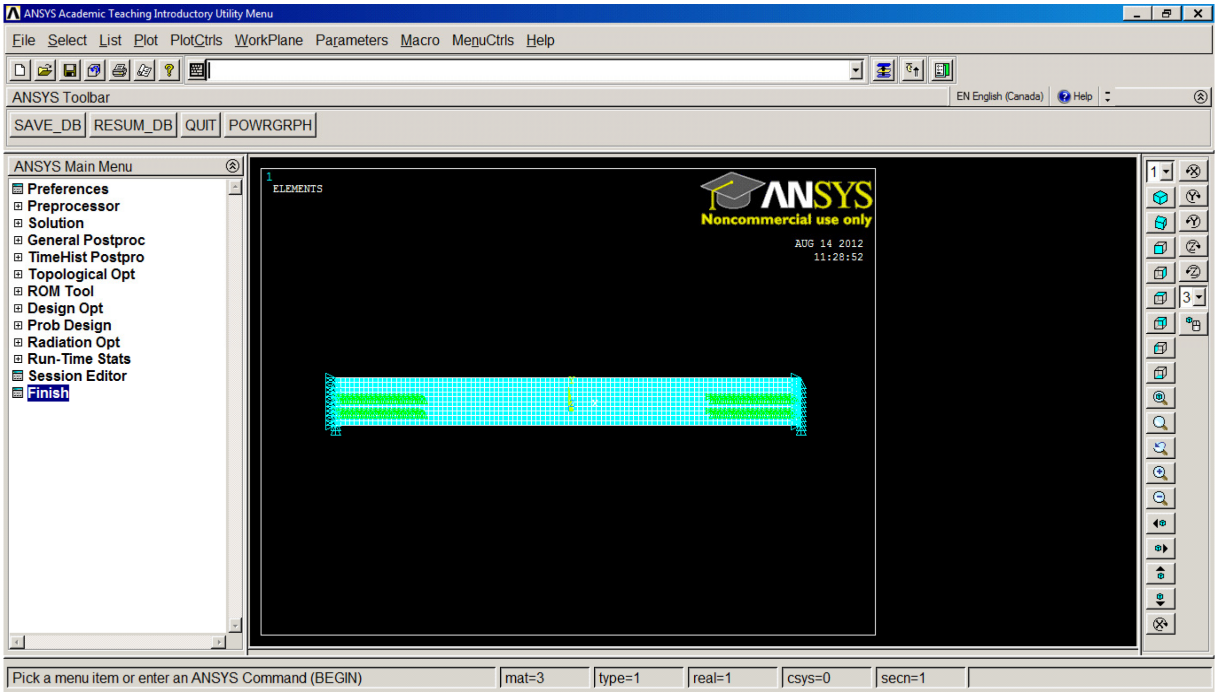
Third shape mode



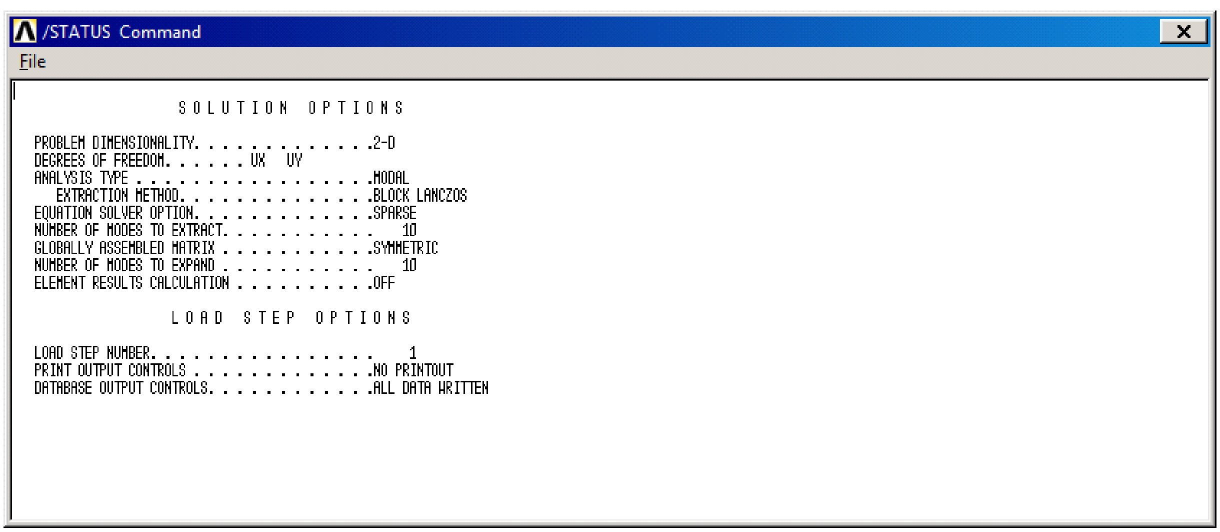
Forth mode



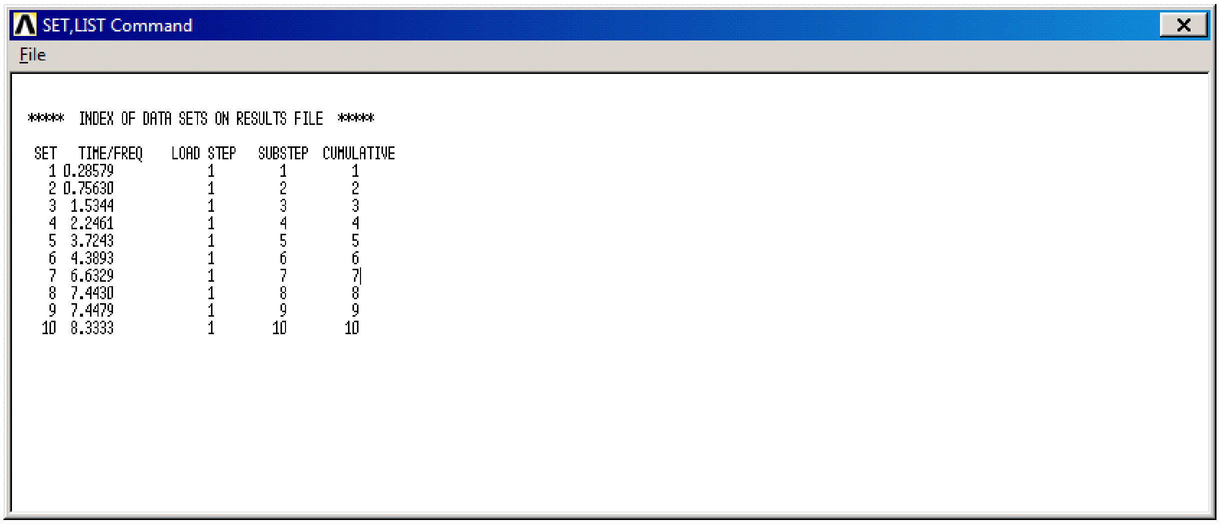
2D Double centered delamination clamped-clamped



Modal test is applied on the above beam and 10 first modes is extracted from this test.



Modal test result for double centered delamination on clamped-clamped beam is summarized in the following file:

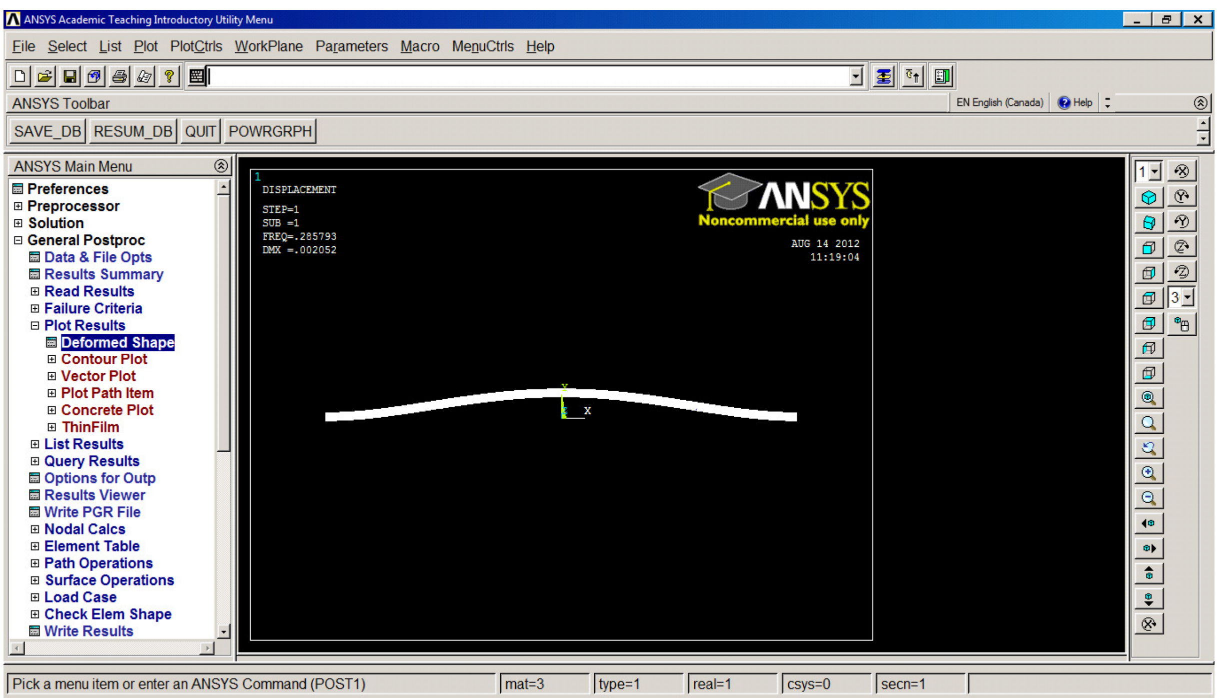

SET,LIST Command

File

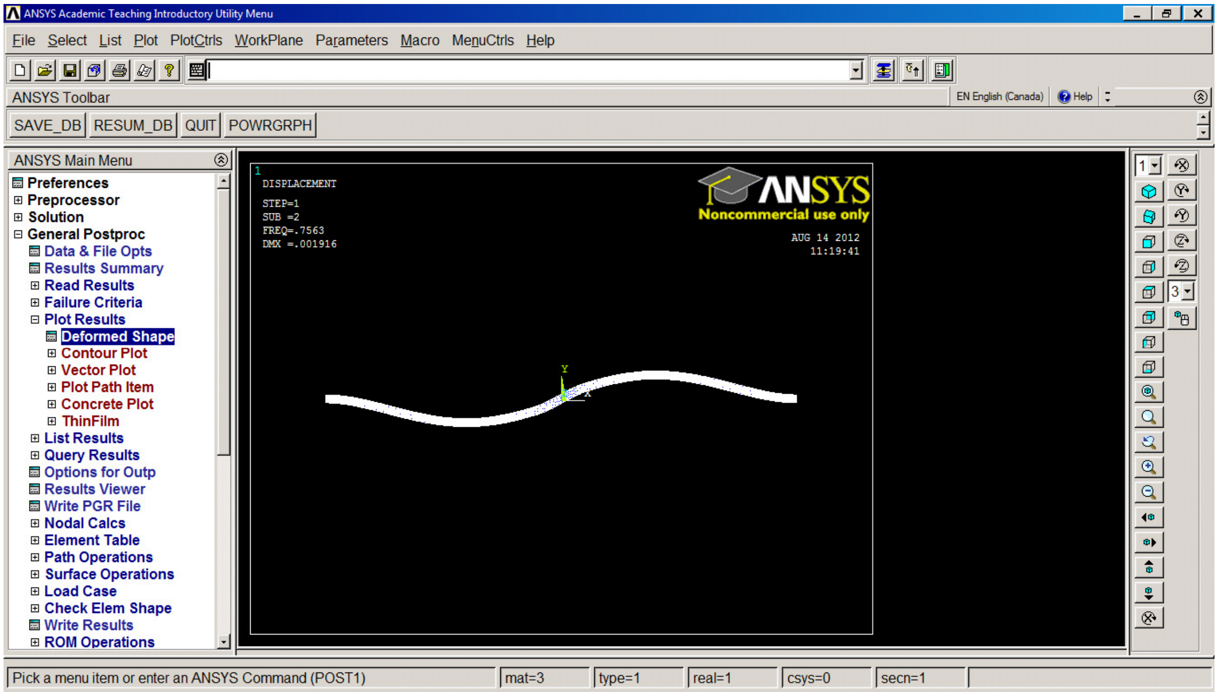
***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.28579	1	1	1
2	0.75630	1	2	2
3	1.5344	1	3	3
4	2.2461	1	4	4
5	3.7243	1	5	5
6	4.3893	1	6	6
7	6.6329	1	7	7
8	7.4430	1	8	8
9	7.4479	1	9	9
10	8.3333	1	10	10

First mode shape of this test is shown in this picture:

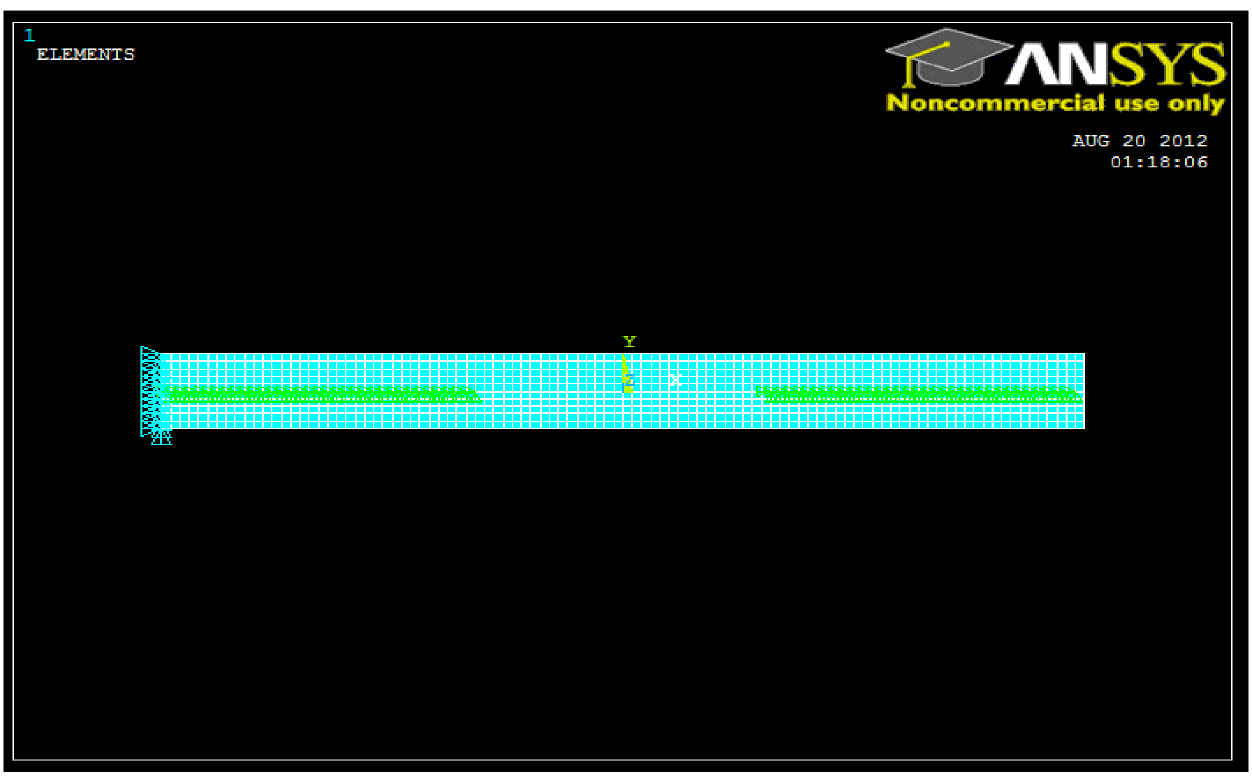


Second mode shape

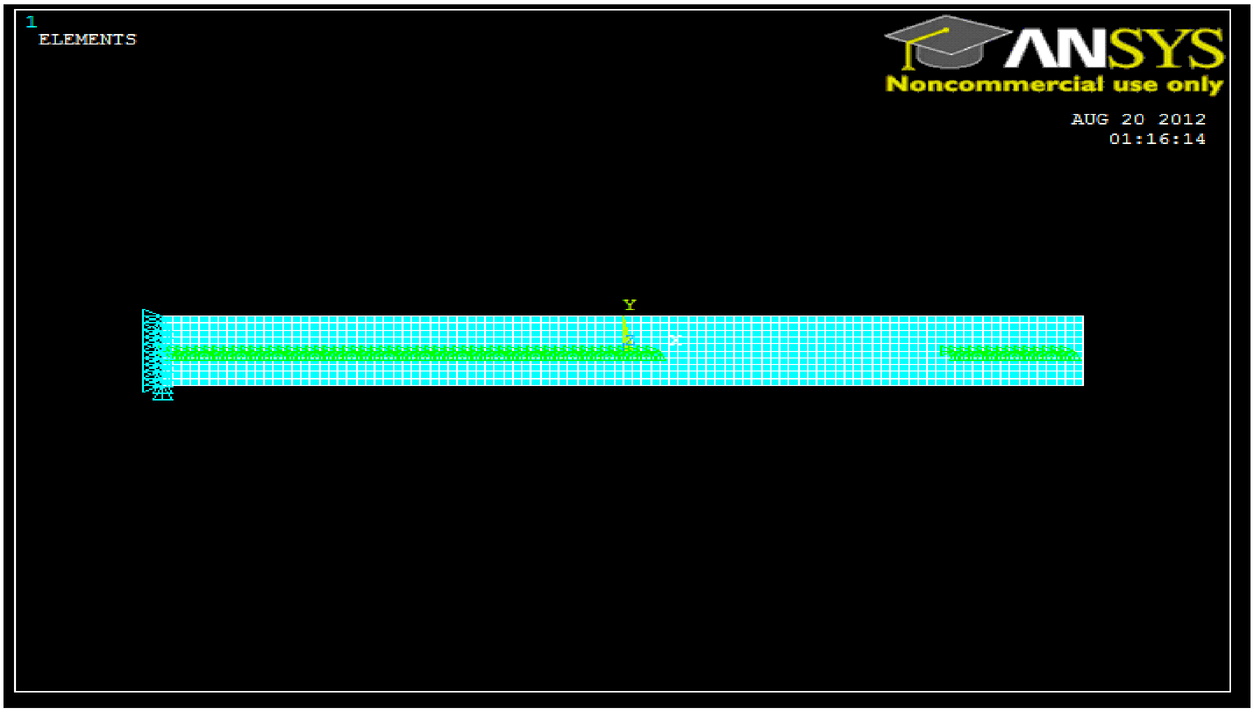


The following pictures are providing some of the simulation of all possibilities that exist with relationship between delamination, frequency and different type of beams.

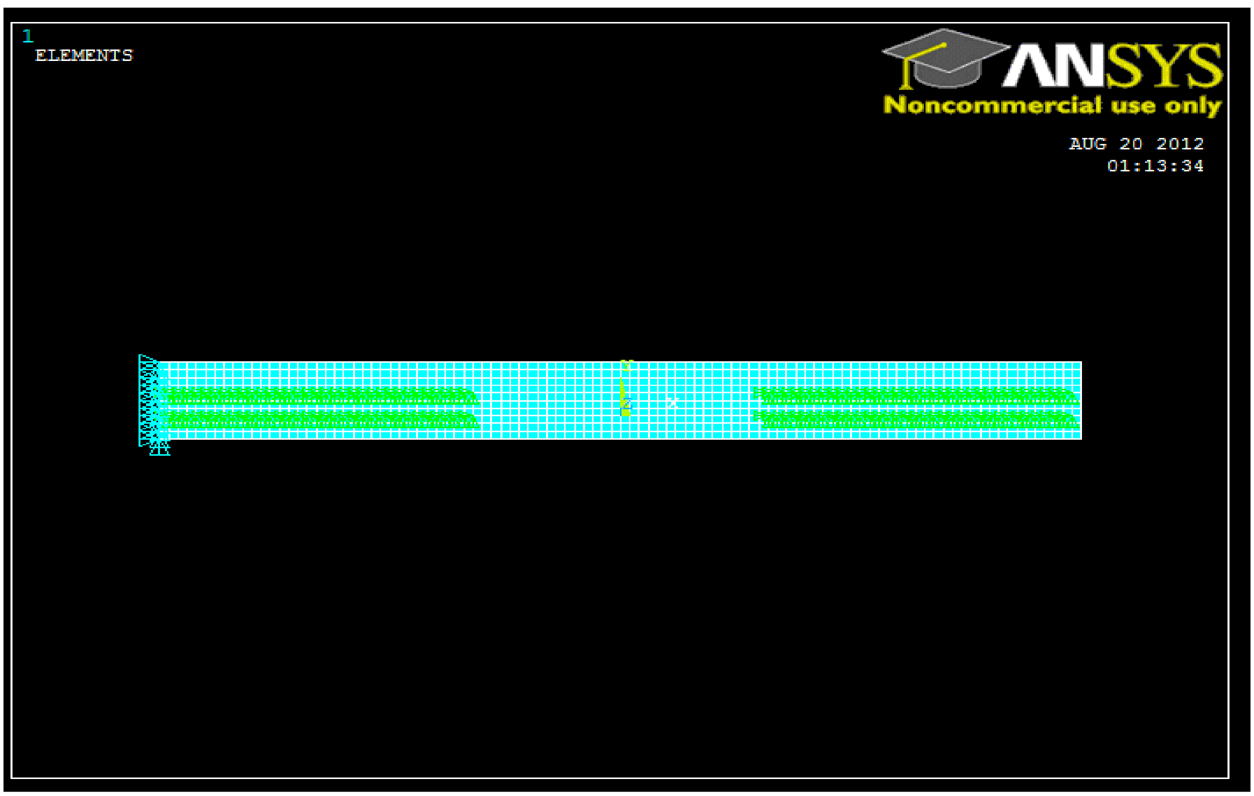
1-2D single centered delamination cantilever beam



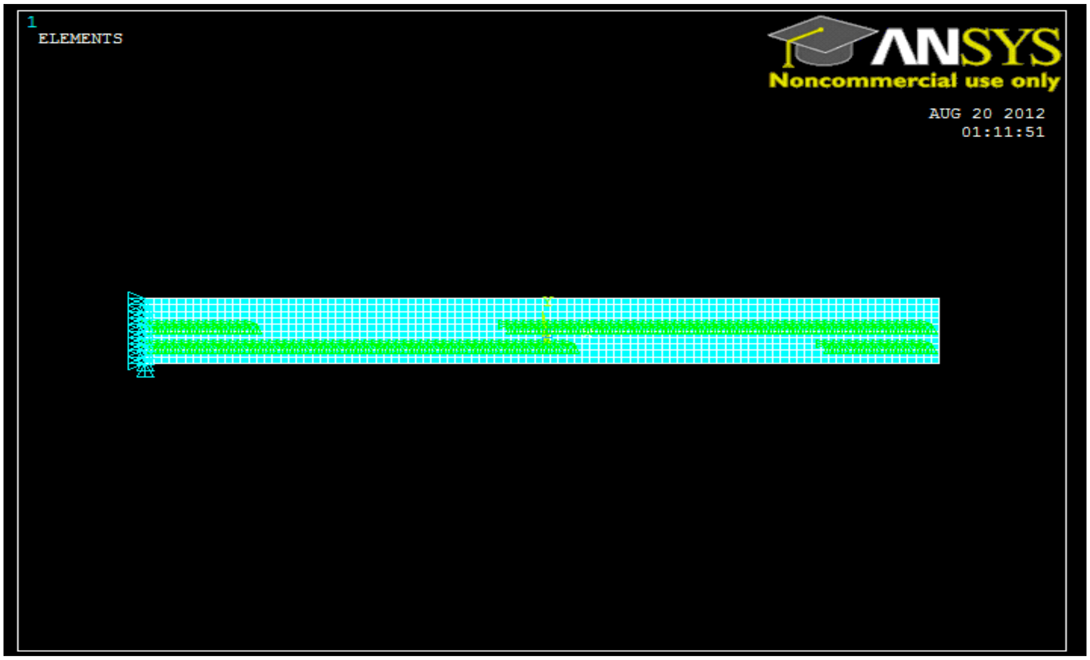
2-2D single off-centered delamination Cantilever



3-2D double centered delamination Cantilever



4-2D double off-centered delamination Cantilever



5-2D Double non-enveloped delamination clamped-clamped beam