# VIBRATION ANALYSIS OF A RECTANGULAR PLATE

A thesis submitted to the department of Mechanical and Chemical Engineering (MCE), Islamic University of Technology (IUT), in the partial fulfillment of the requirement for the degree of Bachelor of Science in Mechanical Engineering.

**Prepared by:**

Syed Shaheer Uddin Ahmed (081418)

Nagib Mehfuz (081424)

Ebadur Rahman (081430)

**Supervised by:**

Dr. Md. Zahid Hossian

Dr. Mir Md. Maruf Morshed

Department of Mechanical and Chemical Engineering

**Islamic University of Technology**

**The Organization of Islamic Cooperation (OIC)**

**Declaration:** This is to certify that the work presented in this thesis is an outcome of the experiment and research carried out by the authors under the supervision of Dr. Zahid Hossain and Dr. Mir Md. Maruf Morshed.


_____          _____          _____

Syed Shaheer Uddin Ahmed                Nagib Mehfuz                    Ebadur Rahman

(Author)                          (Author)                          (Author)

Date:                          Date:                          Date:


_____                                    _____

Dr. Md. Zahid Hossain                              Dr. Mir Md. Maruf Morshed

(Associate Professor)                              (Assistant Professor)

(Project Supervisor)                              (Project Supervisor)

Date:                                          Date:

# ABSTRACT

Vibration is an inevitable occurrence in any structure. The effect of vibration however can be devastating. It can cause stark premature failure incurring huge cost for replacement or repair. Nevertheless if the vibration characteristics can be accurately predicted such incident can be avoided.    This work presents an investigation on vibration characteristics of a rectangular plate. The Finite Element Method (FEM) is employed to obtain the eigenvalues of natural frequencies of vibration for two opposite end welded condition and also for four ends bolted condition of an immovable rectangular plate. Numerical results obtained by this method are compared with those obtained by theoretical and experimental method.

It has been found that there is a strong conformation within the results obtained by FEM, theoretical and empirical analysis.

**Key word:** Vibration, Plate, Modal analysis, Harmonic Analysis, ANSYS, FEM,

# **Acknowledgement**

All praises are for Almighty Who has given us the opportunity to work with such a relevant project. Our gratitude and respect to our supervisors Associate Prof. Dr. Md. Zahid Hossian and Assistant Prof. Dr. Mir Md. Maruf Morshed, MCE Department, Islamic University of Technology(IUT) for their encouragement, erudite advices and extreme patience throughout the course for establishing such a creative project.

We also thank Lecturer Enayat Ghany Ovy for providing us with useful ideas and careful assistance. We thank the IUT workshop instructor and carpenter for aiding us in our construction of experimental setup. Finally we thank our families whose love has kept us going through the project and our friends in IUT whose support encouraged us to handle this project.

**Syed Shaheer Uddin Ahmed**

Student No: 081418

**Nagib Mehfuz**

Student No: 081424

**Ebadur Rahman**

Student No: 081430

## List of Content:

## List of Figures:

## List of Tables:

# **OBJECTIVE**

The main Objective of this work is to study the vibration characteristics of plate structure with different end conditions.

The objectives are outlined below:-

1. Numerical Modeling

2. Theoretical Modeling

3. Experimental verification

4. Comparison between the results

5. Providing foundation for further research on the topic

# INTRODUCTION

## Basics of Vibration

Vibration is the mechanical oscillation of a particle, member, or a body from its position of equilibrium [1]. It is the study that relates the motion of physical bodies to the forces acting on them. The basic concepts in the mechanics of vibration are space, time, and mass (or forces). When a body is disturbed from its position, then by the elastic property of the material of the body, it tries to come back to its initial position. In general, we may see and feel that nearly everything vibrates in nature; vibrations may be sometimes very weak for identification. On the other hand, there may be large devastating vibrations that occur because of manmade disasters or natural disasters such as earthquakes, winds, and tsunamis.
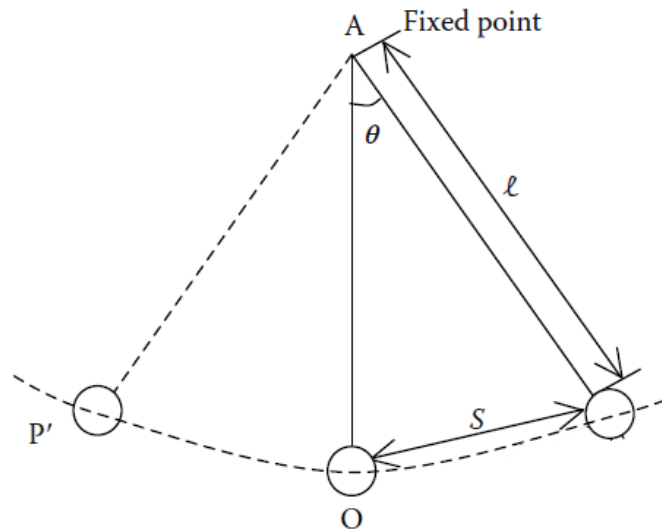


**Figure 1: vibration of a pendulum**

Natural and human activities always involve vibration in one form or the other. Recently, many investigations have been    motivated by the engineering applications

of vibration, such as the design of machines, foundations, structures, engines, turbines, and many control systems. Vibration is also used in pile-driving, vibratory testing of materials, and electronic units to filter out unwanted frequencies. It is also employed to simulate the complex earthquake phenomenon and to conduct studies in the design of nuclear reactors.

On the one hand, vibrations are of great help, while on the other, there are many cases of devastating effects of excessive vibration on engineering structures. Therefore, one of the important purposes of vibration study is to reduce vibration through proper and comparatively accurate design of machines and structures. In this connection, the mechanical, structural, and

aerospace engineers need the information regarding the vibration characteristics of the systems before finalizing the design of the structures.

In a dynamics problem, the applied loadings (and hence the structural response such as deflection, internal forces, and stress) vary with time. Thus, unlike a statics problem, a dynamics problem requires a separate solution at every instant of time. The structure may be considered as subjected to two loadings, namely the applied load and the inertia forces. The inertia forces are the essential characteristics of a structural dynamics problem. The magnitude of the inertia forces depends on

(1) The rate of loading,

(2) The stiffness of the structure

(3) The mass of the structure.

Generally, structural systems are continuous and their physical properties or characteristics are distributed. However, in many instances, it is possible to simplify the analysis by replacing the distributed characteristics by discrete characteristics, using the technique of lumping. Thus, mathematical models of structural dynamics problems may be divided into two major types:

1. Discrete systems with finite degrees of freedom (DOFs)

2. Continuous systems with infinite DOFs

The motion of discrete system is governed by only time variable, whereas continuous systems are governed by variables that depend not only on time but also on the space coordinates.

## Causes of Vibration [1]

The main causes of vibration are as follows:

- Unequal distribution of forces in a moving or rotating machinery
- External forces like wind, tides, blasts, or earthquakes
- Friction between two bodies
- Change of magnetic or electric fields
- Movement of vehicles, etc.

## Requirements for Vibration [1]

The main requirements for the vibration are as follows:
- There should be a restoring force.
- The mean position of the body should be in equilibrium.
- There must be inertia (i.e., we must have mass).

## Classification of Vibration [2]

Vibration can be classified in several ways. Some of the important classifications are as follows.

**Free Vibration:**

The ensuing vibration after a system, following an initial disturbance, is left to vibrate on its own frequency.

**Forced Vibration:**

If a system is subjected to an external force, the resulting vibration is known as forced vibration. The Oscillation that arises in machines such as diesel engines is an example of forced vibration.

If the frequency of actuation coincides with one of the natural frequencies of the system the system vibrates with dangerously high amplitudes, this phenomenon in forced vibration is known as resonance.

**Undamped Vibration:**

If no energy is lost or dissipated in friction or other resistance during oscillation, the vibration is known as Undamped vibration.

**Damped Vibration:**

If any energy is lost or dissipated in friction or other resistance during oscillation, the vibration is known as Damped vibration

**Boundary Condition**

**Clamped boundary condition**: In this case, both the displacement and slope are considered to be zero.

**Simply supported boundary condition**: Here the displacement and bending moment must be zero.

**Free boundary condition**: In this boundary condition, the bending moment and shear force must be zero.

## Modal analysis

**Modal analysis** is a process of obtaining innate dynamic characteristics of a system in the forms of natural frequencies, damping factors and mode shapes and using them to formulate mathematical model for its dynamic behavior.[4] The mathematical model is referred to as the modal mode of the system and the information for the characteristics is known as the modal data.

Modal analysis is based upon the fact that the vibration response of the linear time-invariant dynamic system can be expressed as the linear combination of a set of simple harmonic motions called the natural modes of vibration. The natural modes of vibration are inherent to a dynamic system and are determined completely by it physical properties like mass, stiffness, damping and their spatial distribution. Each mode is described by the modal parameters: natural frequency, damping factor and characteristic displacement pattern, which is mode shape. The mode shape corresponds to a natural frequency.

## Harmonic analysis

**Harmonic analysis** is a branch of mathematics concerned with the representation of functions or signals as the superposition of basic waves, and the study of and generalization of the notions of Fourier series and Fourier transforms[7]. Externally force is applied in time variant manner and the amplitude response of the system is calculated. The frequency of the actuation is varied and the change in amplitude of the system on which force is applied. Thus the variation of amplitude with the change in frequency of actuation can be obtained.

# PLATE VIBRATION

Study of vibration of plates is an extremely important area owing to its wide variety of engineering applications such as in aeronautical, civil, and mechanical engineering. Since the members, viz., beams, plates, and shells, form integral parts of structures, it is essential for a design engineer to have a prior knowledge of the first few modes of vibration characteristics before finalizing the design of a given structure. In particular, plates with different shapes, boundary conditions at the edges, and various complicating effects have often found applications in different structures such as aerospace, machine design, telephone industry, nuclear reactor technology, naval structures, and earthquake-resistant structures. A plate may be defined as a solid body bounded by two parallel, flat surfaces having two dimensions far greater than the third.

The vibration of plates is an old topic in which a lot of work has already been done in the past decades. In earlier periods, results were computed for simple cases only where the analytical solution could be found. The lack of good computational facilities made it almost impossible to get reasonably accurate results even in these simple cases. This may be the cause for why in spite of a lot of theoretical developments; numerical results were available only for a few cases.

By the aid of fast and efficient algorithms complex plate vibration problems can be solved in a very short time and give comparatively accurate results. Methods like finite element methods, boundary integral equation methods, finite difference methods, and the methods of weighted residuals have made handling any shape and any type of boundary conditions possible.


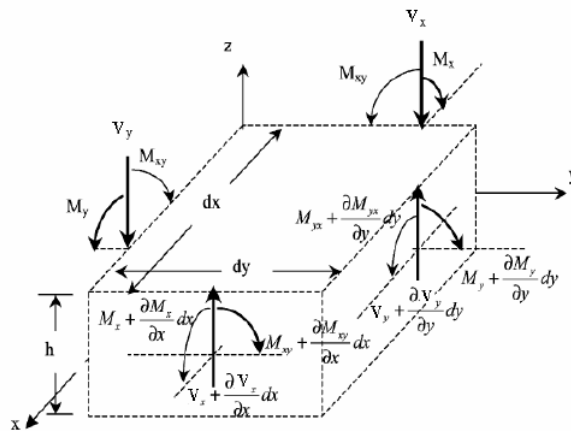
**Figure 2: Free body diagram of a body located on the plate [3]**

In this work some of the problems with varying boundary condition are dealt with in Finite Element Theoretical and Experimental realm to determine the veracity of each method and

whether the solutions of each approach to solving the problems regarding plate structure conduce each other.

## **IMPORTANCE OF PLATE VIBRATION STUDY**

Most human activities involve vibration in one form or other. Most prime movers have vibration problems due inherent unbalance in the engines. In turbines, vibration cause drastic mechanical failures. Plates are the most commonly used element in mechanical structures and machines such as aircrafts, ships and submarines. In designing a structure, plates are usually specified only to withstand applied static loads. However, this is inadequate for more accurate applications. Dynamic forces and random cyclic loads also threaten the stability of a system. There exist a large number of discrete frequencies at which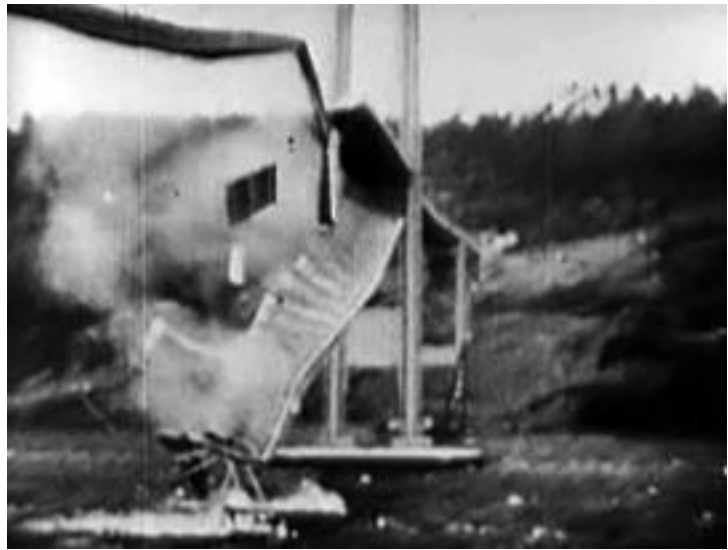 a rectangular plate will undergo large amplitude vibration by sustained time varying forces of matching frequencies. Thus, the possibility of large displacement and stresses due to this recent type of excitation must be taken into account.



**Figure 3: Destruction of Tacoma narrows bridge due to resonance**

By predicting the salient frequencies in which the vibration is maximum for specific plate structure active and appropriate measures, like passive and active methods can be installed adequately in response to the working range of the structure. When resonance occur the deflection of the structure is excessive and it can lead to devastating failures which can incur excessive cost to repair or replace. So it's of great concern to study the structure before hand and avoid such havoc. Hence vibration analysis of plate is imperative in today's structure designs

# PROBLEM SPECIFICATION

There are two cases considered in this thesis to compare and extract a level of relation with different mode. A third analysis is done solely in Finite Element Interface (ANSYS) to observe analyze harmonic response when external force is applied in a sinusoidal fashion.

**Case 1:** A square plate with 0.6 m side length and 0.004m thickness is bounded by two ends clamped is taken as the study model. Both modal and harmonic analysis are performed

**Case 2:** A square plate with 0.6 m side length and 0.004m thickness is bounded by two ends clamped and two other ends simply supported is taken as study model.

Mathematical and Finite Element Analysis is performed and the end results are compared to see whether or not the models are compatible and confer each other.

**Case 3:** A Plate with same dimension is bolted at 4 different positions, both FEM analysis and experimental verification is performed and its conclusions are analyzed to see how well one confers with other.
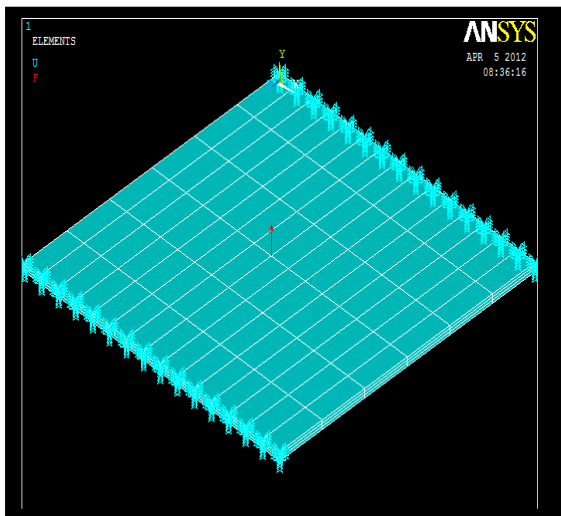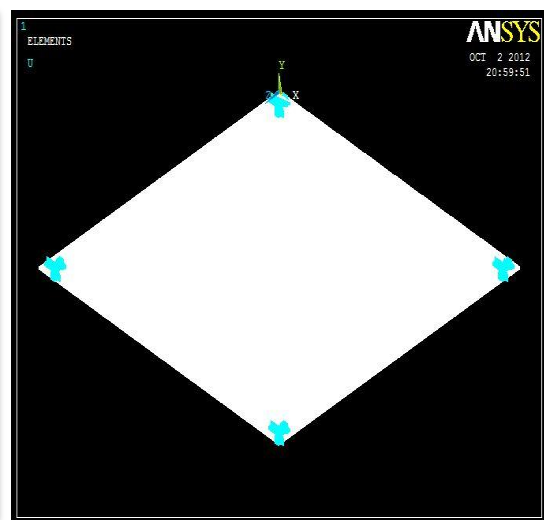


**Fig:** Welded in opposite ends          **Fig:** Bolted in four Ends

# NUMERICAL MODELING

## Analysis using FEM

**Finite Element Method (FEM)** is a numerical approach to obtain approximate solutions to partial differential equations and their systems [5]. A complicated problem is divided into finite number of small and simple elements; these are then solved in relations to each other, hence the overall solution for the whole problem can be approximated. Thus this method provides a very good means to approximate the modal shape and frequency of a system.

The same technique can be applied for harmonic analysis and the amplitude variation due to external actuation can be approximated and it can be obtained whether or not the maximum amplitude is concurrent with the modal frequency.

## ANSYS

**ANSYS** is engineering simulation software uses FEM to predict system response. Structural mechanics solutions from ANSYS provide the ability to simulate every structural aspect of a product, including linear static analyses that simply provide stresses or deformations, modal analysis that determines vibration characteristics; through to advanced transient nonlinear phenomena involving dynamic effects and complex behaviors[6].

The ANSYS Mechanical (APDL) software suite is used for both modal and harmonic analysis.

# MODAL ANALYSIS USING ANSYS

The modal analysis is carried out in all of the above structure and their modal shape and natural frequency is obtained.

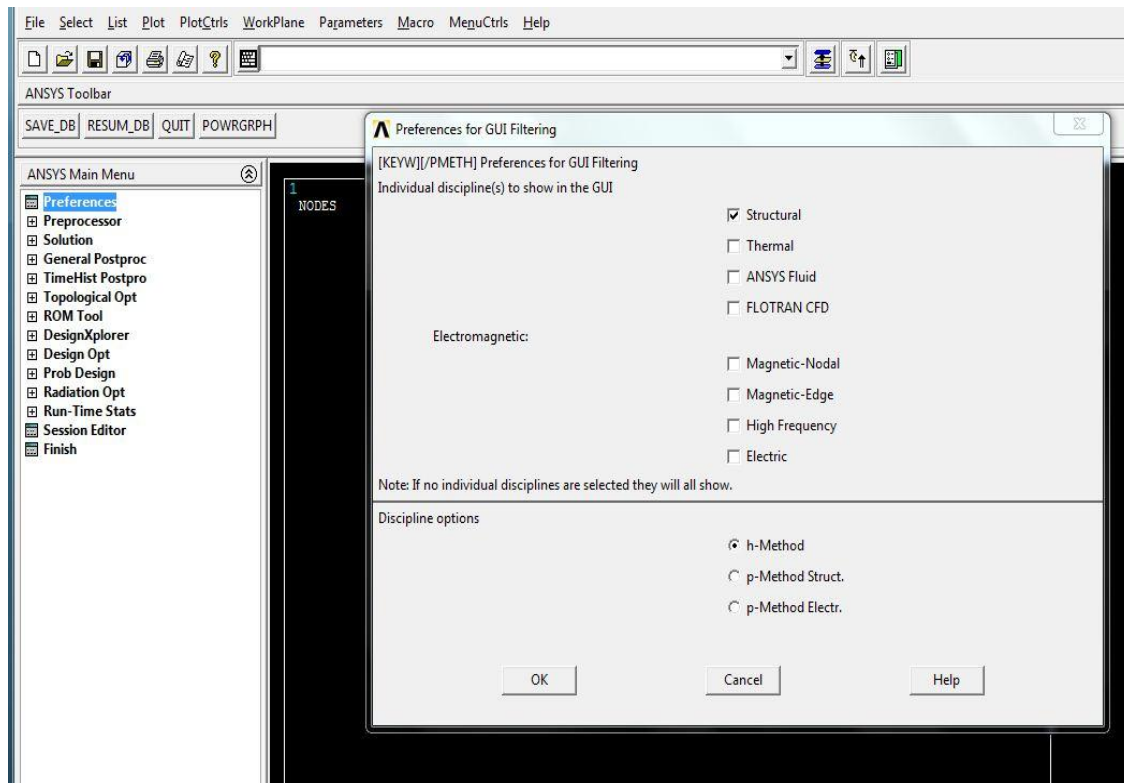At first the preference is specified to structural analysis



**Figure 4: ANSYS modeling, Preference specification**

After that the geometry is drawn in the ANSYS CAD interface:
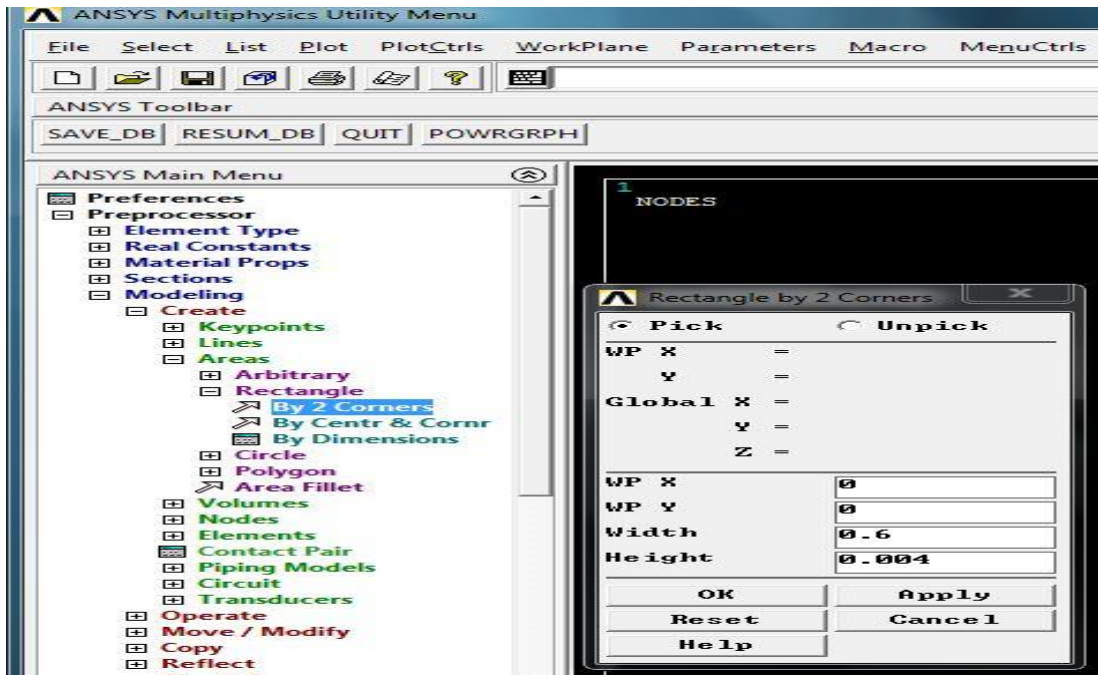


**Figure 5: ANSYS modeling, creating 2D plate**

The elements for vibration simulations are specified from the Element type drop down in ANSYS main menu
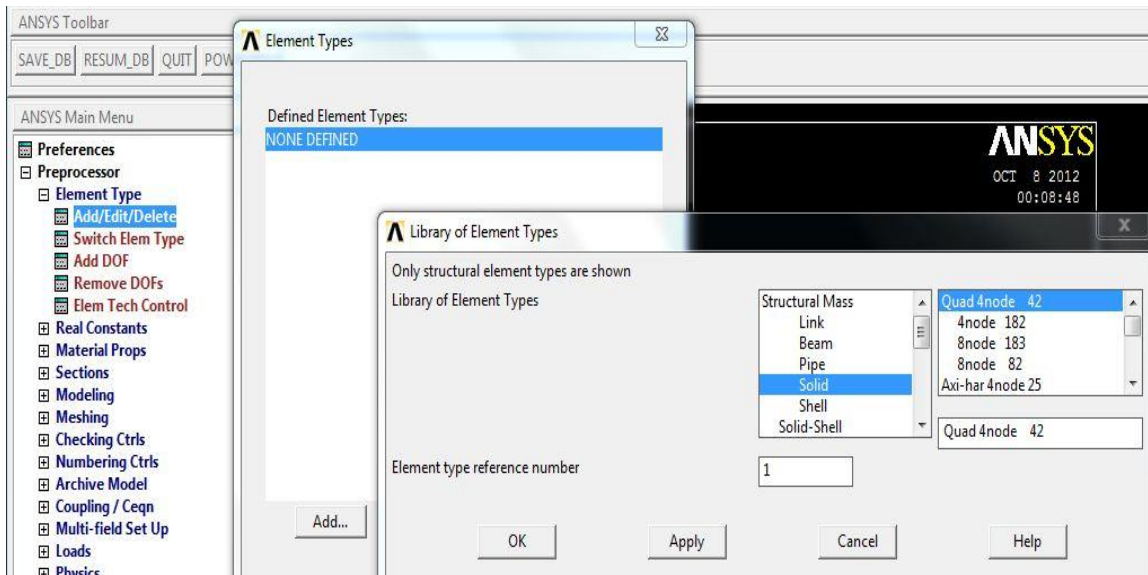


**Figure 6: ANSYS modeling, selecting simulation elements**

Both Quad 4 Node and Solid 45 are selected for the analysis.

The Material properties are to be specified following the element selection
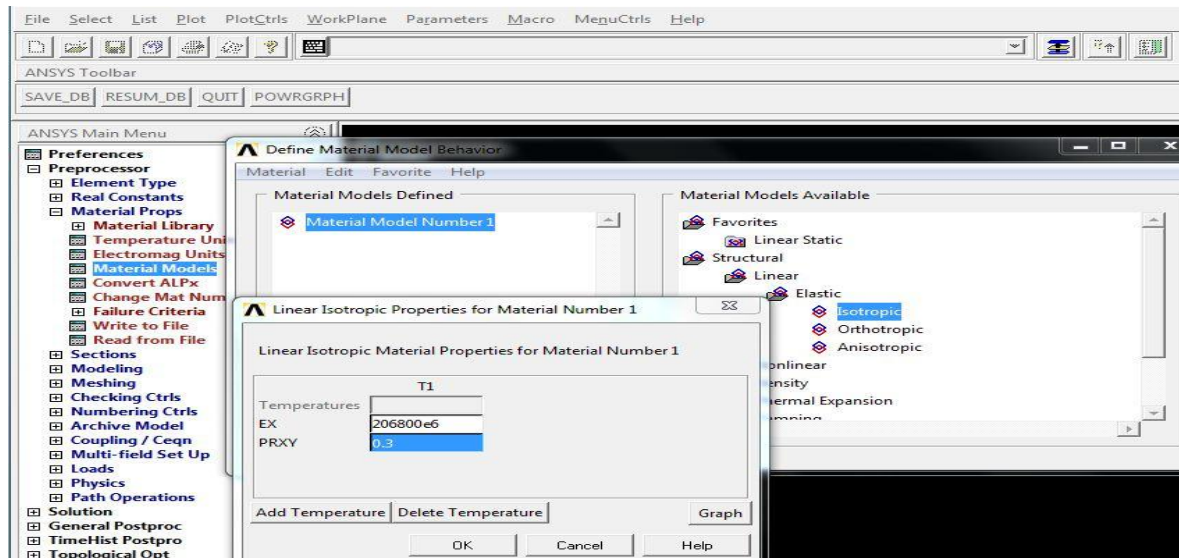


**Figure 7: Specification of material property**

After specifying the Young's modulus and Poisons ratio the density is to be specified. Following the feeding of material property into the system meshing is to be done with the aid of mesh tool from the Main Menu.

`
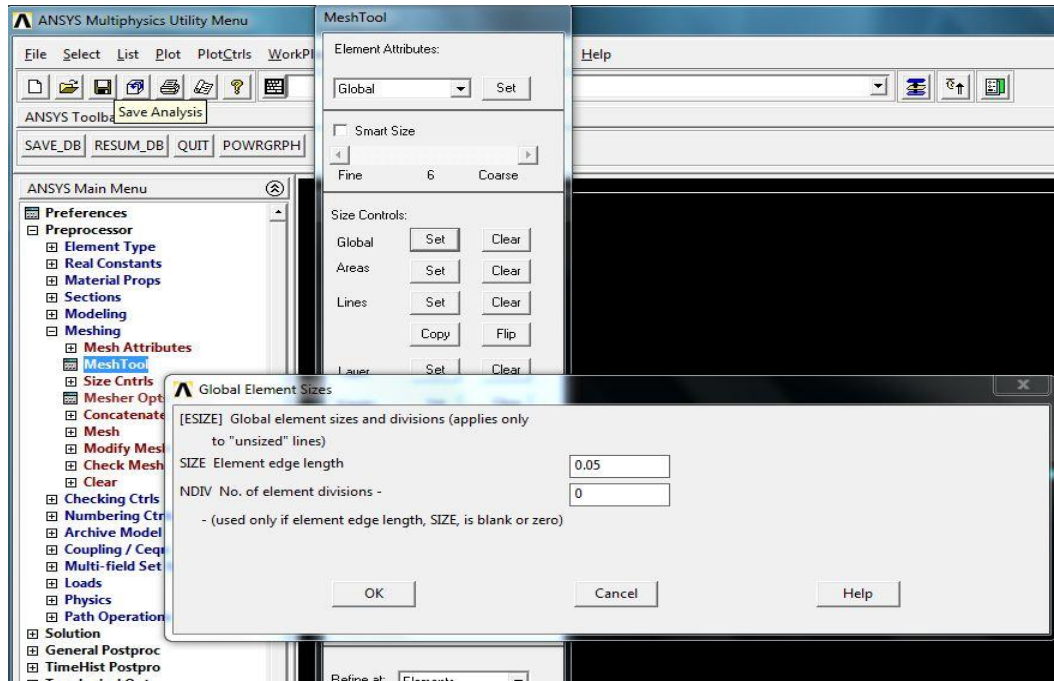
**Figure 8: setting Global Mesh size**

The Mesh size can be varied by varying the element edge length. When meshing command is prompted a window to select the meshing area will appear. In order to mesh the structure completely 'pick all' command is to be selected.

Once meshing is complete extruding the 2D plate section to a 3D plate structure is to be done. For that the 'extrude element option' is used. The no of mesh division in the extruded zone is to be specified
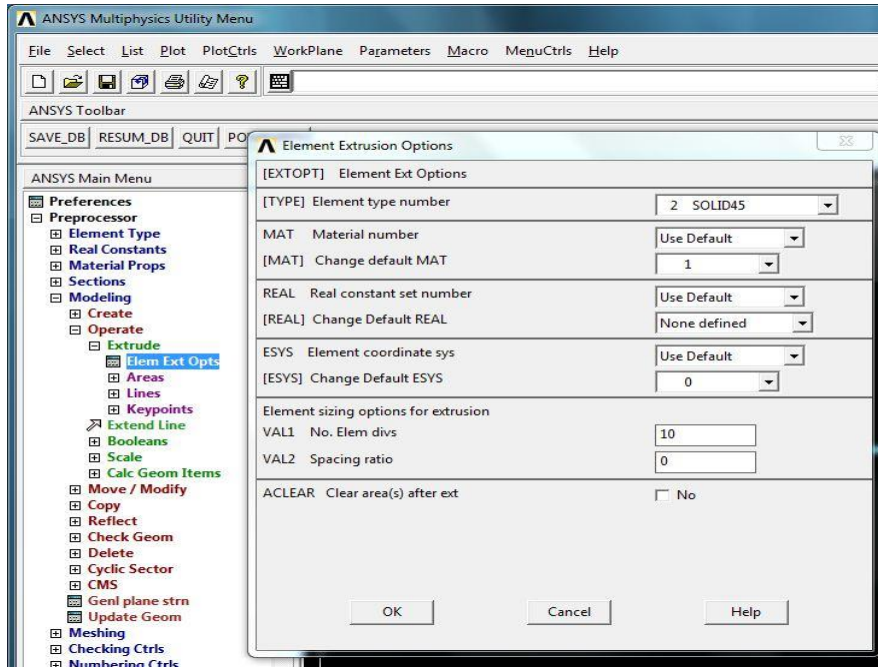
**Figure 9: Fig: Extruding the 2D section**

The final step for extrusion is to offset the shape by required dimension



Once extruded the view of the shape should be changed to see it in a 3 D scale.
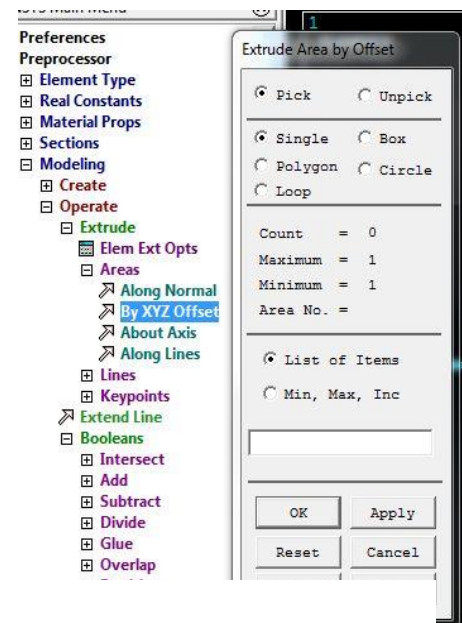
**Figure 10: extruding the 2 D section**

This can be done from the Pan Zoom Rotate option in Plot Control tab in the utility menu. The Isometric view will show the complete 3D shape of the plate.
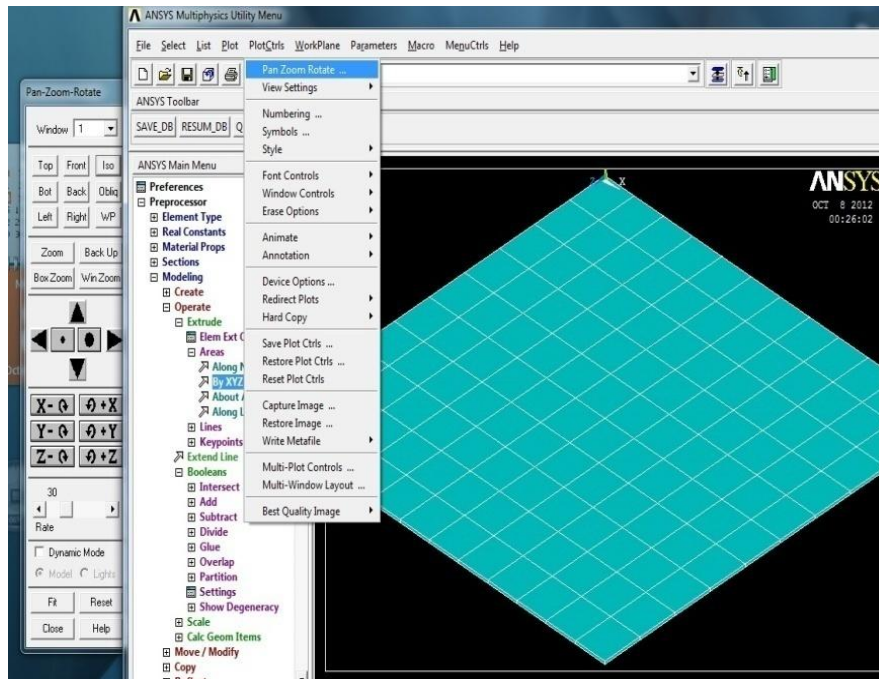


**Figure 11: shifting the view of the CAD model**

As the whole analysis will be done in 3D model the 2D plane 42 must be unselected which can be done by the option 'select entities' found in Utility menu

**Figure 12: selecting nodes using XYZ coordinates**
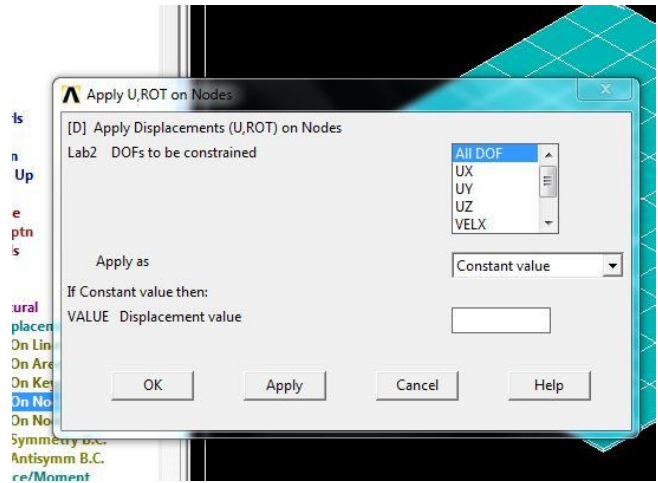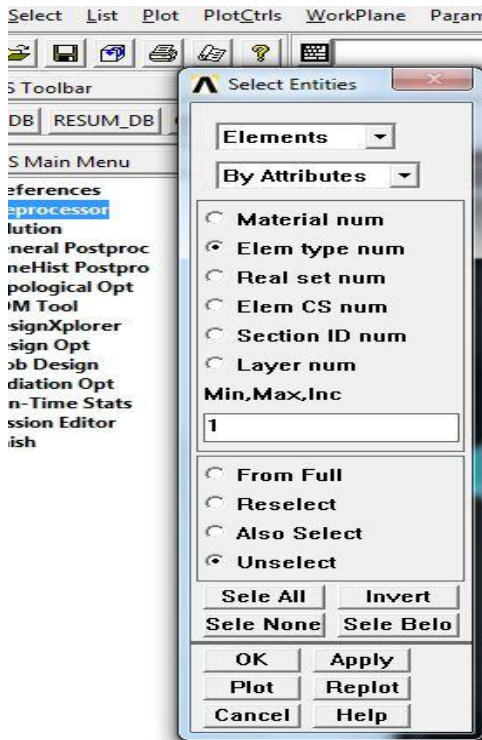
**Figure 13: restricting the DOF of the selected nodes**

**Figure 14: Plane 42 Unselecting**

After unselecting Plane 42 the DOFs must be applied, if the boundary condition is such that two ends are to be clamped all of the nodes of one end is to be selected by the help of the select entity option. All of the nodes in one plane can be selected by specifying the coordinates of the nodes to be selected. The method is shown in Fig:
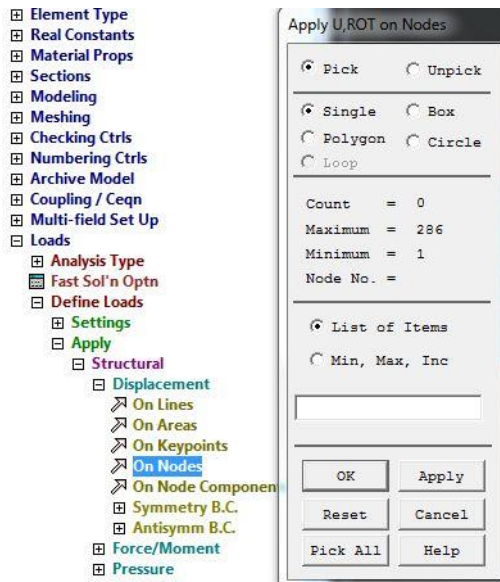


**Figure 15: applying Structural, Displacement load**

**Figure 16: restricting DOF**

After selecting the nodes the loads must be applied, this will give rise to the desired boundary condition. If fully clamped boundary condition is required then the loads must be applied such that all the degrees of freedom are restricted.

`



**Figure 17: selecting all nodes**

If the DOF s are fully restricted the resulting representation in ANSYS will take the shape shown in Figure
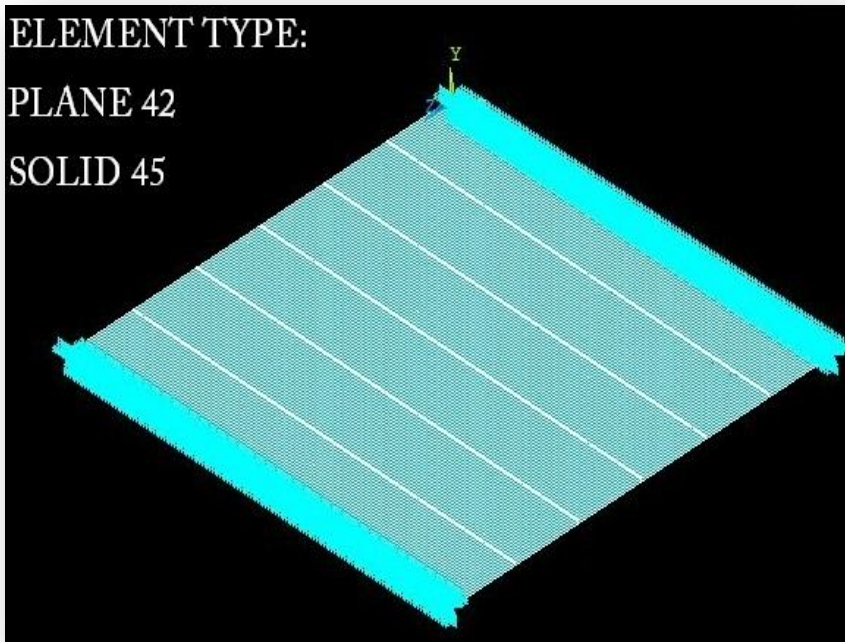
**Figure 18: Two end clamped plate**

As only one set of nodes in a particular coordinate plane is selected therefore in order to analyze total number of nodes are to be reselected.

This can be done by the help of Num/pick option in the 'Select entities' windows. In the Num/Pick option the whole nodes can be selected using the option 'Sele all'. The process is shown in the Figure 20.

When all of the needed boundary conditions are applied then the setup is ready for Modal analysis

Modal analysis is done by at first specifying the new analysis option found in the Main Menu.
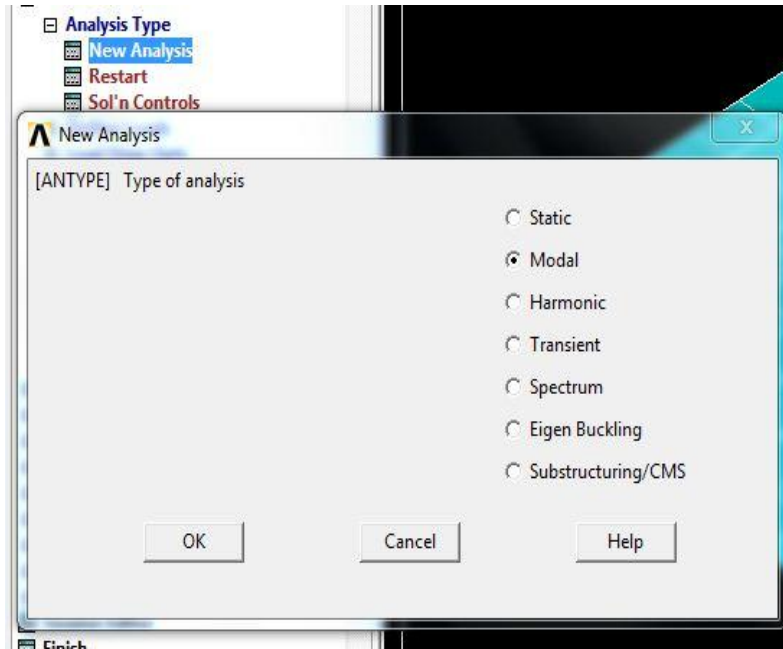
**Figure 19: selecting Modal analysis**

The analysis option is then fixed accordingly by selecting Block Lanczos solving method and the number of modes to extract is then specified.
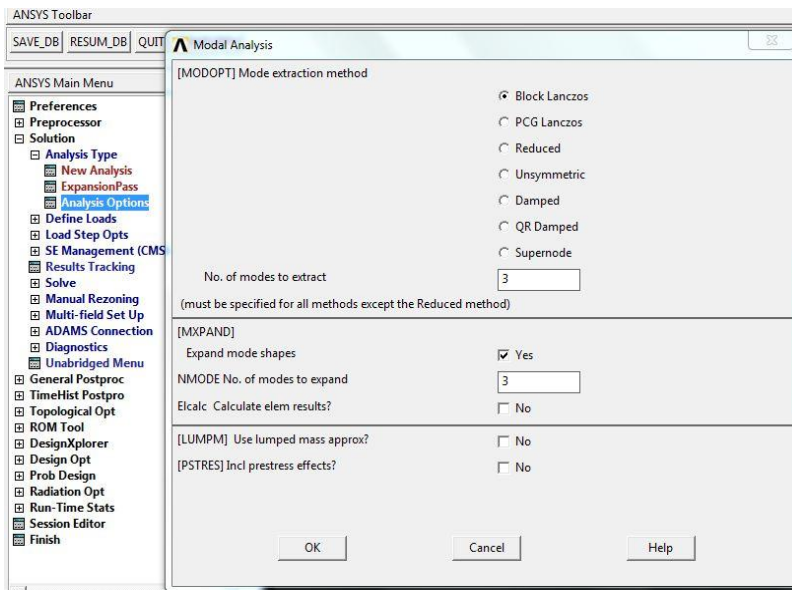


**Figure 20: Modal analysis specifications**

Solving for the whole set up then yields the modal shapes characteristic to the structure



**Figure 21: solving to obtain modal shapes**

# Harmonic Analysis using ANSYS [7]

The modeling in harmonic analysis is same as that of Modal analysis (described above). There is a change in setting the analysis type. The process for harmonic analysis is as follows:

1.**Define Analysis Type (Harmonic)**

Solution > Analysis Type > New Analysis > Harmonic

2. **Set options for analysis type:**
Select: **Solution > Analysis Type > Analysis Options..**
The following window will appear
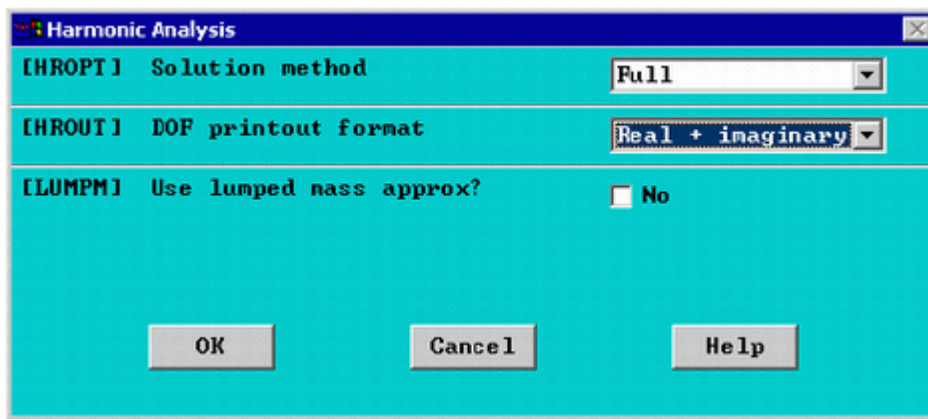


**Figure 22: Harmonic analysis option window**

The following window will appear
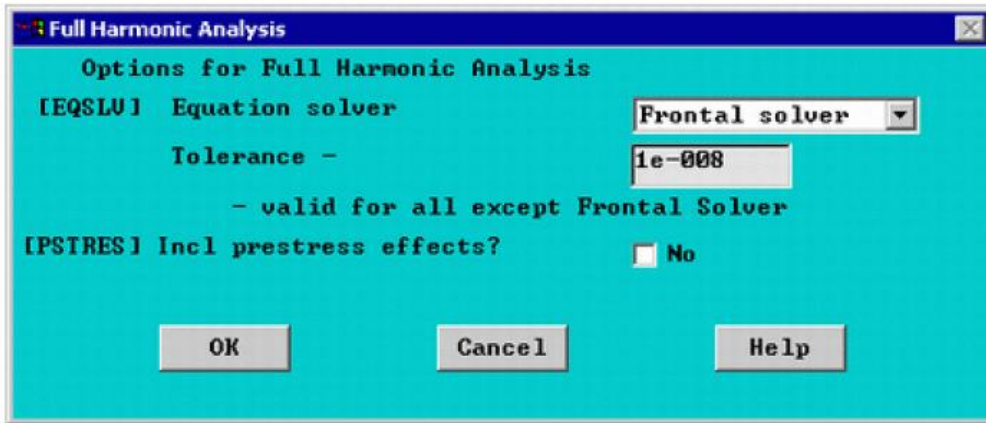


**Figure 23: selecting solver**

If the frontal solver is not present in the available option the default solver known as sparse solver can be used.

3**. Apply Loads:**

Select **Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes**
select the node manually and then the following windows should be filled according to Figure 24
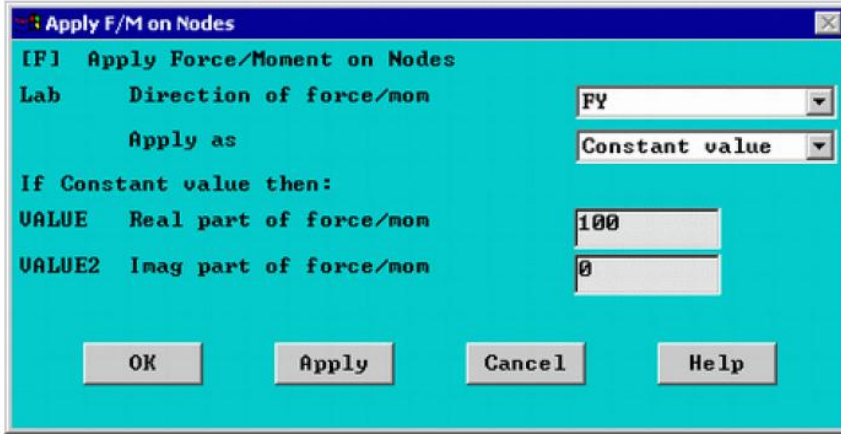


**Figure 24: load application**

5. **Set the frequency range**
Select **Solution > Load Step Opts > Time/Frequency > Freq and Substps**
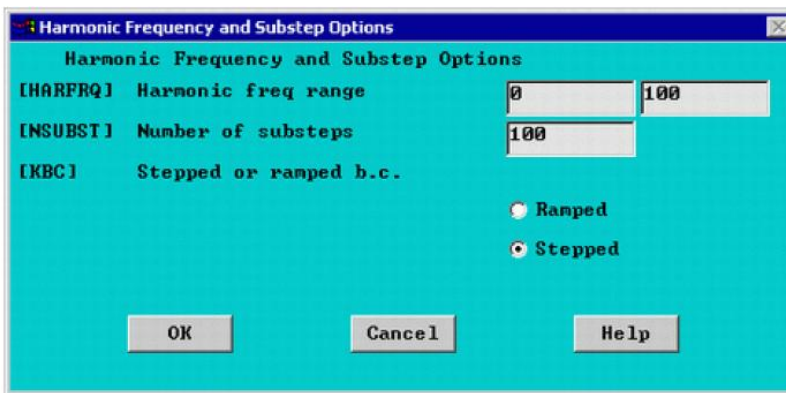


**Figure 25: setting the frequency range**

6. **Solve the System**
Solution > Solve > Current LS

In order to view the results the Variable Viewer from the TimeHist Post Processing menu is to be opened which can be found in the Main menu section.

The variable viewer should look like:
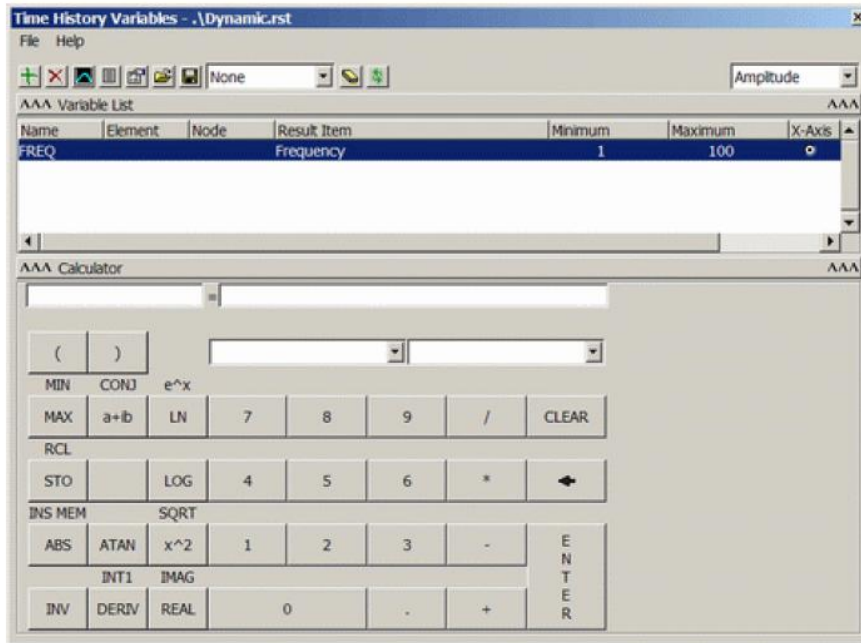


**Figure26: variable viewer**

Selecting Add (the green '+' sign in the upper left corner) from this window should make the following window appear.
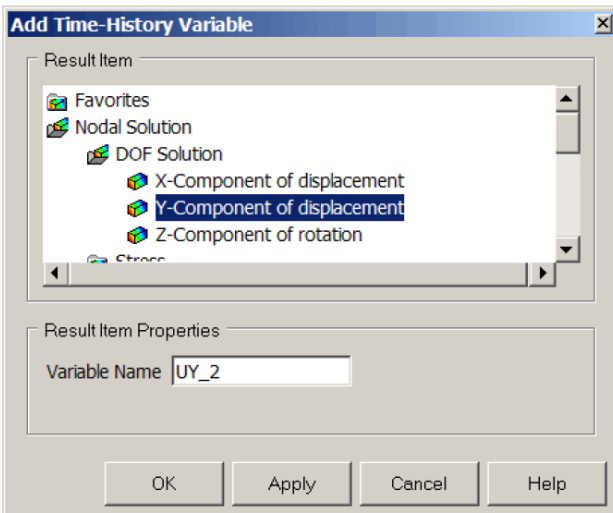


**Figure 27: selecting the coordinate for which data for harmonic response is to be taken**

The coordinate for which data is to be obtained must be selected.

This will create a new row in the variable viewer, clicking the plot option in the variable viewer (two buttons to the left of Add) would plot the data in a graph of Amplitude vs Frequency/

This is the desired Harmonic response.

## Plate modeling

A plate with the dimension of 0.6 m 0.6 m and 0.004 m is designed as a specimen, following the above specifications.

For case 1, two sides are fully restricted in movement and its eigenvalues are obtained.
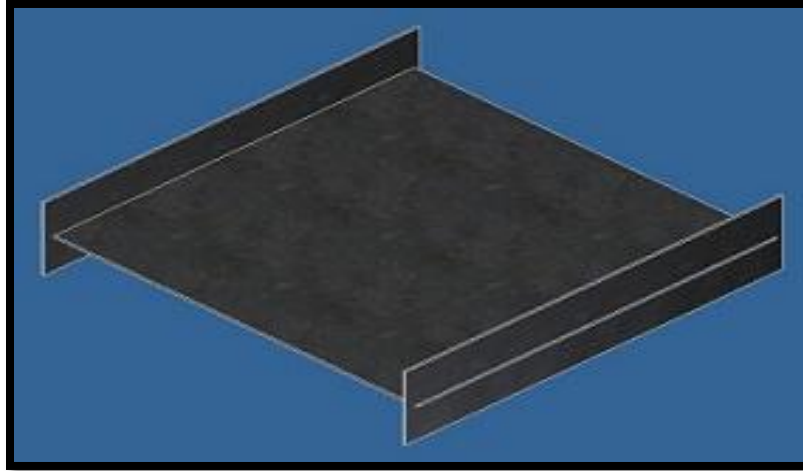


**Figure 28: CAD representation of the Plate**
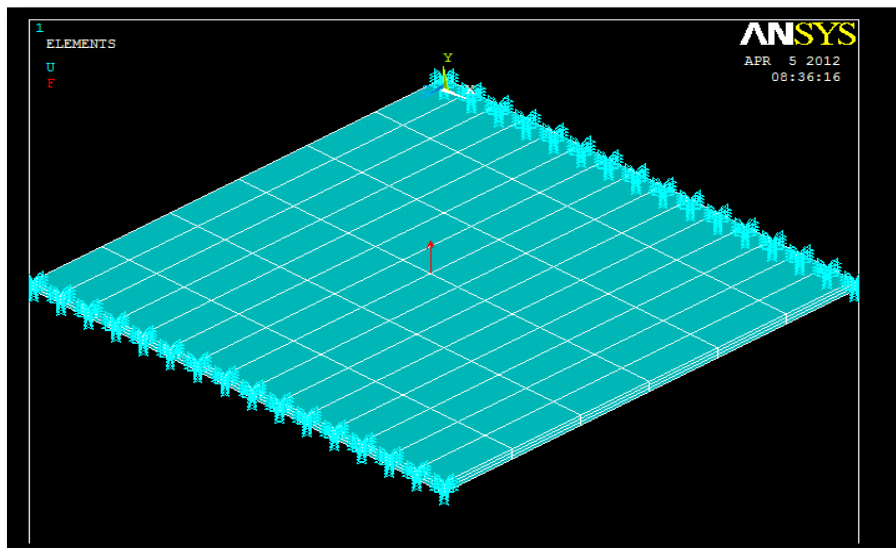


**Figure 29: ANSYS Model of the Plate**

For case 2 analysis the end conditions are: i. two ends restricted

ii. Other two ends simply supported

An ANSYS model for these end condition is developed and modal analysis is performed the element edge length of the mesh size is taken as 0.01.
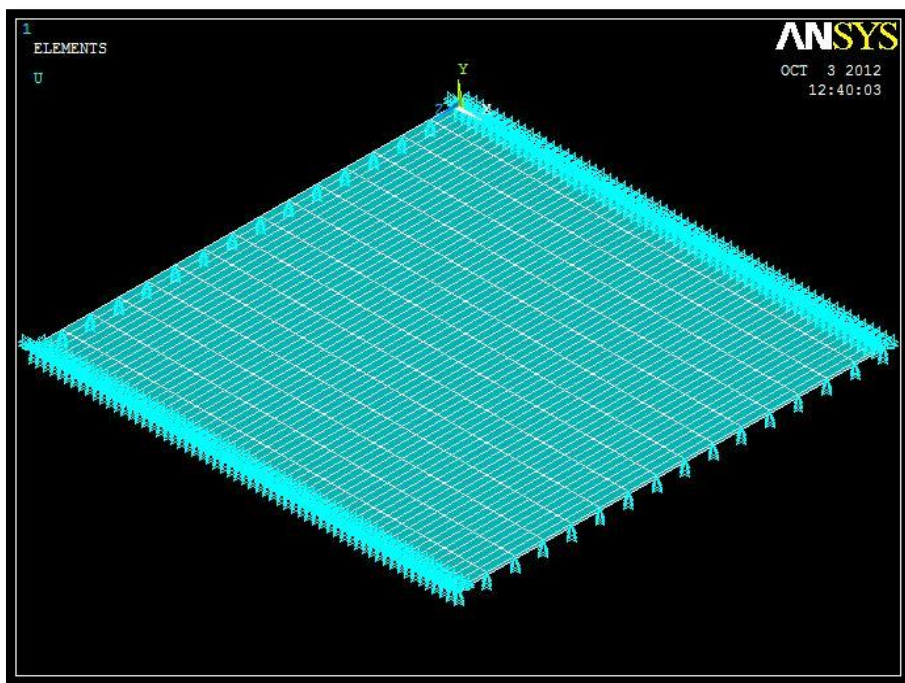


**Figure 30: ANSYS Model with 2 ends restricted and 2 ends simply supported**

For case 3 a model with restricted degree of freedom on four different coordinates on the plate is used to simulate the plate bolted and fixed at four places. The element edge length is taken 0.001.
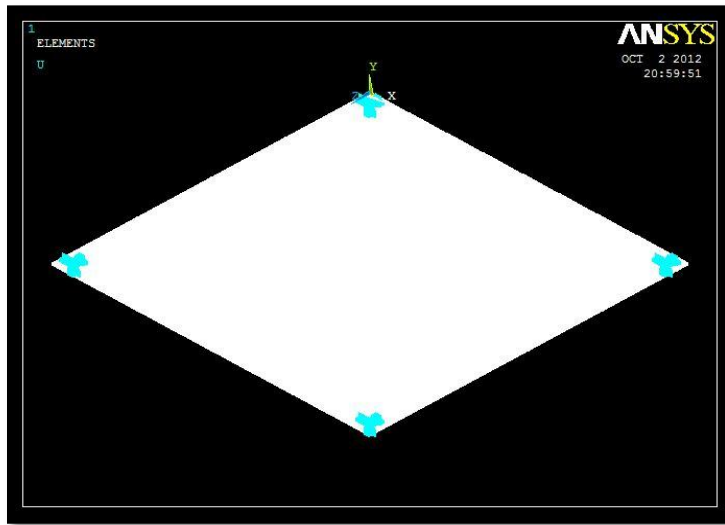


**Figure 31: ANSYS Model with 4 coordinates restricted**

## Modal analysis Results

For the first model the obtained mode has the modal frequency 66 Hz, second mode has the modal frequency of 78 Hz and the third mode has the modal frequency of 124 Hz.
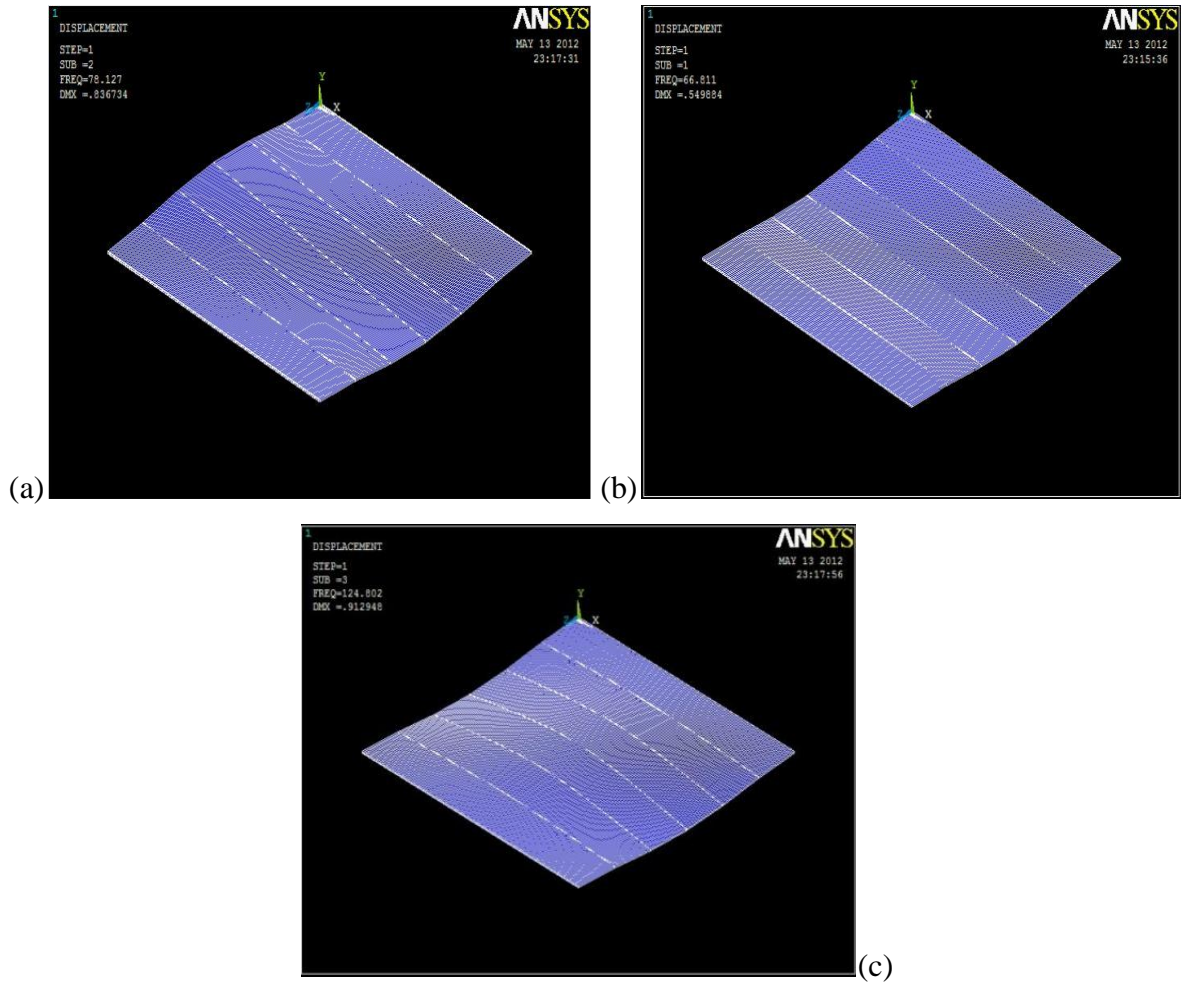
**Figure 32: mode shape for case 1 (a) first mode (b) second mode (c) third mode**

For the second model that is for **case 2** modal analysis is conducted and the obtained mode shapes are 80.523 Hz for first mode and 151.87 Hz for the second mode.
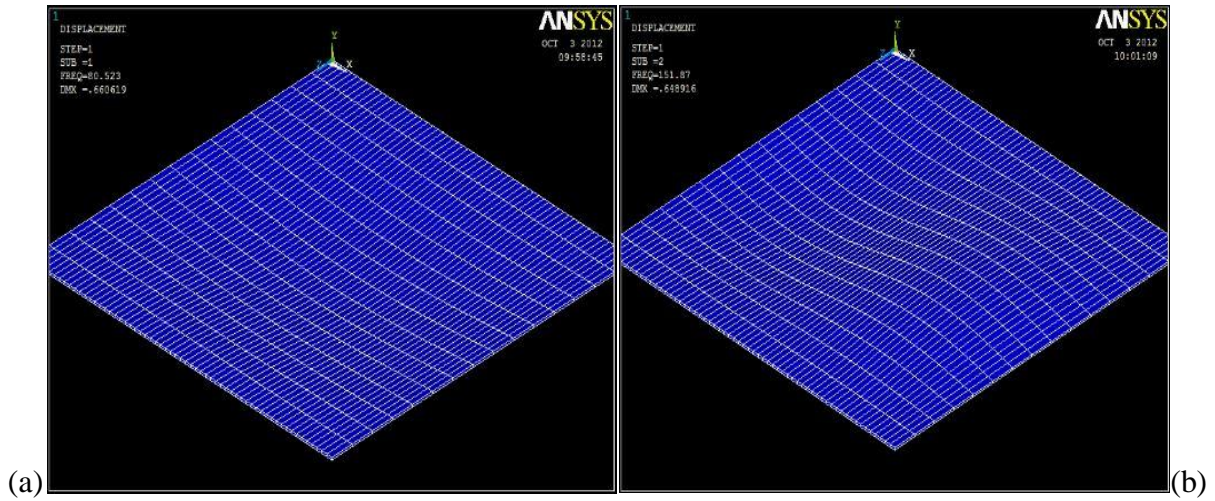
**Figure 33: Mode shape for case 2, (a) first mode (b) second mode**

Two modes for **case 3** are extracted using ANSYS. The obtained modes are 36.044 Hz, 66.257 Hz.



**Figure 34: Mode shape for case 3, (a) first mode (b) second mode**

## Harmonic Analysis Results

**Harmonic analysis** is conducted for the first case. ANSYS offers two types of linear equation solvers: direct and iterative. The sparse solver in ANSYS is still the default solver for virtually all analyses. It is the most robust solver in ANSYS, but it is also compute- and I/O-intensive. The sparse solver used in ANSYS is designed to run in different modes of operation, depending on the amount of memory available. It is important to understand that the solver's mode of operation can have a significant impact on runtime.

Memory usage for a direct sparse solver is determined by several steps. In ANSYS, the matrix that is input to the sparse solver is assembled entirely in memory before being written to the FULL file. The sparse solver then reads the FULL file, processes the matrix, factors the matrix, and computes the solution. Direct method solvers factor the input matrix into the product of a lower and upper triangular matrix in order to solve the system of equations. For symmetric input matrices (most of ANSYS matrices are symmetric), only the lower triangular factor is required since it is equivalent to the transpose of the upper triangular factor. Still, the process of factorization produces matrix factors which are 10 to 20 times larger than the input matrix. The calculation of this factor is computationally intensive. In contrast, the solution of the triangular systems is I/O or memory access-dominated with few computations required. [8]

For the analysis in this thesis uses Sparse Solver to obtain the results. Force is varied from 0 to 20 N and the frequency of actuation is varied from 0 to 130 N. The graphical representations are given below.

**Figure 35: Amplitude (m) vs Frequency (Hz)**



**Figure 36: Bode Plot**

The data from which the graphs are drawn is tabulated below

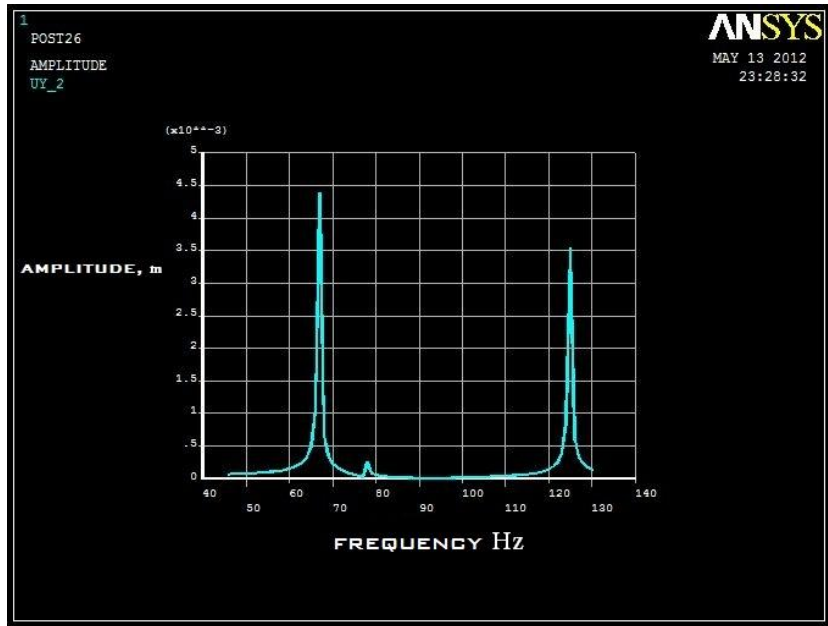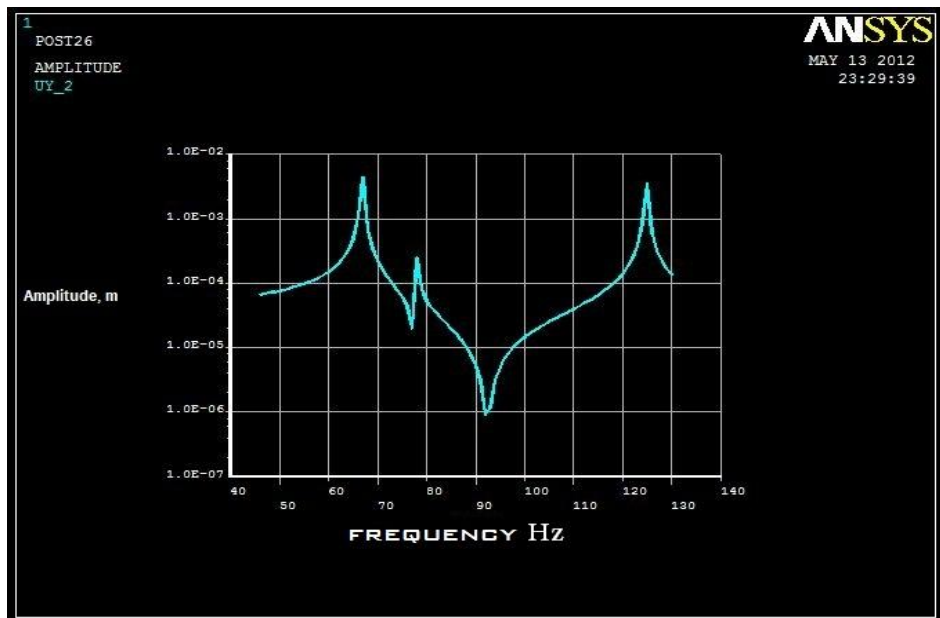| TIME | 321 UY | | TIME | 321 UY | |
|---|---|---|---|---|---|
| | UY_2 | | | UY_2 | |
| | AMPLITUDE | PHASE | | AMPLITUDE | PHASE |
| 261.00 | 0.235362E-04 | 0.00000 | 280.00 | 0.641849E-04 | 0.00000 |
| 262.00 | 0.242634E-04 | 0.00000 | 281.00 | 0.715151E-04 | 0.00000 |
| 263.00 | 0.250458E-04 | 0.00000 | 282.00 | 0.808723E-04 | 0.00000 |
| 264.00 | 0.258901E-04 | 0.00000 | 283.00 | 0.932327E-04 | 0.00000 |
| 265.00 | 0.268038E-04 | 0.00000 | 284.00 | 0.110320E-03 | 0.00000 |
| 266.00 | 0.277959E-04 | 0.00000 | 285.00 | 0.135486E-03 | 0.00000 |
| 267.00 | 0.288771E-04 | 0.00000 | 286.00 | 0.176242E-03 | 0.00000 |
| 268.00 | 0.300598E-04 | 0.00000 | 287.00 | 0.253574E-03 | 0.00000 |
| 269.00 | 0.313591E-04 | 0.00000 | 288.00 | 0.456827E-03 | 0.00000 |
| 270.00 | 0.327932E-04 | 0.00000 | 289.00 | 0.244147E-02 | 0.00000 |
| 271.00 | 0.343843E-04 | 0.00000 | 290.00 | 0.716690E-03 | 180.000 |
| 272.00 | 0.361597E-04 | 0.00000 | 291.00 | 0.309949E-03 | 180.000 |
| 273.00 | 0.381535E-04 | 0.00000 | 292.00 | 0.196688E-03 | 180.000 |
| 274.00 | 0.404086E-04 | 0.00000 | 293.00 | 0.143479E-03 | 180.000 |
| 275.00 | 0.429802E-04 | 0.00000 | 294.00 | 0.112569E-03 | 180.000 |
| 276.00 | 0.459400E-04 | 0.00000 | 295.00 | 0.923676E-04 | 180.000 |
| 277.00 | 0.493833E-04 | 0.00000 | 296.00 | 0.781301E-04 | 180.000 |
| 278.00 | 0.534391E-04 | 0.00000 | 297.00 | 0.675545E-04 | 180.000 |
| 279.00 | 0.582871E-04 | 0.00000 | 298.00 | 0.593884E-04 | 180.000 |

| Time | UY_2 | | TIME | 321 UY | |
|---|---|---|---|---|---|
| | AMPLITUDE | PHASE | | UY_2 | |
| 299.00 | 0.528920E-04 | 180.000 | | AMPLITUDE | PHASE |
| 300.00 | 0.476002E-04 | 180.000 | 318.00 | 0.151101E-04 | 180.000 |
| 301.00 | 0.432060E-04 | 180.000 | 319.00 | 0.144461E-04 | 180.000 |
| 302.00 | 0.394985E-04 | 180.000 | 320.00 | 0.138236E-04 | 180.000 |
| 303.00 | 0.363280E-04 | 180.000 | 321.00 | 0.132385E-04 | 180.000 |
| 304.00 | 0.335854E-04 | 180.000 | 322.00 | 0.126872E-04 | 180.000 |
| 305.00 | 0.311892E-04 | 180.000 | 323.00 | 0.121665E-04 | 180.000 |
| 306.00 | 0.290776E-04 | 180.000 | 324.00 | 0.116736E-04 | 180.000 |
| 307.00 | 0.272022E-04 | 180.000 | 325.00 | 0.112059E-04 | 180.000 |
| 308.00 | 0.255254E-04 | 180.000 | 326.00 | 0.107610E-04 | 180.000 |
| 309.00 | 0.240169E-04 | 180.000 | 327.00 | 0.103369E-04 | 180.000 |
| 310.00 | 0.226524E-04 | 180.000 | 328.00 | 0.993145E-05 | 180.000 |
| 311.00 | 0.214120E-04 | 180.000 | 329.00 | 0.954291E-05 | 180.000 |
| 312.00 | 0.202792E-04 | 180.000 | 330.00 | 0.916945E-05 | 180.000 |
| 313.00 | 0.192403E-04 | 180.000 | 331.00 | 0.880935E-05 | 180.000 |
| 314.00 | 0.182840E-04 | 180.000 | 332.00 | 0.846084E-05 | 180.000 |
| 315.00 | 0.174005E-04 | 180.000 | 333.00 | 0.812211E-05 | 180.000 |
| 316.00 | 0.165816E-04 | 180.000 | 334.00 | 0.779123E-05 | 180.000 |
| 317.00 | 0.158201E-04 | 180.000 | 335.00 | 0.746598E-05 | 180.000 |
| | | | 336.00 | 0.714377E-05 | 180.000 |

| TIME | 321 UY | TIME | 321 UY |
|---|---|---|---|

| | UY_2 | | | UY_2 | |
|---|---|---|---|---|---|
| | AMPLITUDE | PHASE | | AMPLITUDE | PHASE |
| 337.00 | 0.682137E-05 | 180.000 | 356.00 | 0.495532E-05 | 180.000 |
| 338.00 | 0.649449E-05 | 180.000 | 357.00 | 0.473777E-05 | 180.000 |
| 339.00 | 0.615713E-05 | 180.000 | 358.00 | 0.453953E-05 | 180.000 |
| 340.00 | 0.580031E-05 | 180.000 | 359.00 | 0.435645E-05 | 180.000 |
| 341.00 | 0.540962E-05 | 180.000 | 360.00 | 0.418562E-05 | 180.000 |
| 342.00 | 0.495986E-05 | 180.000 | | | |
| 343.00 | 0.440198E-05 | 180.000 | | | |
| 344.00 | 0.362489E-05 | 180.000 | | | |
| 345.00 | 0.231018E-05 | 180.000 | | | |
| 346.00 | 0.982013E-06 | 0.00000 | | | |
| 347.00 | 0.505353E-04 | 0.00000 | | | |
| 348.00 | 0.140015E-04 | 180.000 | | | |
| 349.00 | 0.917395E-05 | 180.000 | | | |
| 350.00 | 0.760488E-05 | 180.000 | | | |
| 351.00 | 0.676918E-05 | 180.000 | | | |
| 352.00 | 0.621743E-05 | 180.000 | | | |
| 353.00 | 0.580656E-05 | 180.000 | | | |
| 354.00 | 0.547671E-05 | 180.000 | | | |
| 355.00 | 0.519838E-05 | 180.000 | | | |

**Table1**: Variation of Amplitude and Phase with Time

# THEORETICAL MODELING

## General formula derivation

The equation of motion for the transverse vibration of a plate is[9]:

$$D\nabla^4 u + \rho h \frac{\partial^2 u}{\partial t^2} = 0$$

Where

u is the transverse displacement of the plate

$\rho$ is the density of the material of the plate

h is the thickness of the plate

D=$\frac{Eh^3}{12(1-v)^2}$

$v$ is the Poisson's ratio

E is the Young's modulus of elasticity

$\nabla^4$ is the biharmonic operator

The Biharmonic operator can be expanded as

$$\nabla^4 u = \frac{\partial^4 u}{\partial x^4} + 2\frac{\partial^4 u}{\partial x^2 \, \partial y^2} + \frac{\partial^4 u}{\partial y^4}$$

For free vibration with circular (natural) frequency v, we can write the motion of the plate in polar coordinates as

u (r, θ, t) = U(r, θ) e$^{iwt}$

and in Cartesian coordinates

u (x, y, t) = U(x, y) e$^{iwt}$

In general, this will be written in the form

u = U e$^{iwt}$

By substituting the general equation in the equation of motion we get:

$$(\nabla^4 - \beta^4)U = 0$$

Where $\beta^4 = \frac{\rho h u^2}{D}$

If we substitute the above equations we get the final form

$$D \left( \frac{\partial^4 u}{\partial x^4} + 2\frac{\partial^4 u}{\partial x^2 \, \partial y^2} + \frac{\partial^4 u}{\partial y^4} \right) + \rho h \frac{\partial^2 u}{\partial t^2} = 0$$

## Modification using Boundary Condition

Let us suppose that the simply supported edges occur always along the x= 0 and x=a edges as shown in Fig . The boundary conditions on these edges are therefore

u (0,y,t)=0

u (a,y,t)=0

Substituting the strain–displacement relations

U (0, y) =0
U (a, y) =0

$\frac{\partial^2 U}{\partial x^2}(0,\ y)=0$

And $\frac{\partial^2 U}{\partial x^2}(a,\ y) =0$
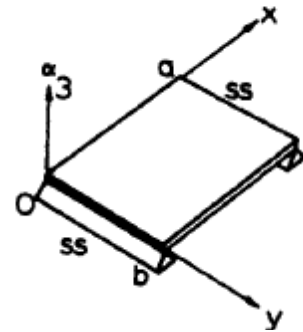


**Figure 37: Rectangular plate simply supported**

Variables become separated and the boundary conditions are satisfied if we assume as solution of the form:

U(x,y)=Y(y) sin$\frac{m\pi x}{a}$

We obtain the general solution in the form: $\lambda = \pm\frac{b}{a}m\pi\sqrt{1 \pm k}$ [9]

Where k= $\frac{\omega}{(m^2\pi^2/a^2)\sqrt{D/\rho h}}$

If we use the apply clamping on other two sides the additional boundary conditions are

u (x, 0, t )= 0

u (x, b, t) = 0

$\frac{\partial^2 U}{\partial x^2}$(x, 0, t)=0

And $\frac{\partial^2 U}{\partial x^2}$(x, b, t)=0

Applying the above boundary conditions in the main equation along with the boundary conditions for two sides supported as given before we arrive at a final form

$$\omega_{mn}\, a^2 \sqrt{\frac{\rho h}{D}} = m^2\pi^2 k_n$$

Where m= 1,2… and n=1,2….

For a square plate for $m^2\pi^2 k_n$ we get the values [9]

| m | N | | |
|---|---|---|---|
|   | 1 | 2 | 3 |
| 1 | 28.9 | 69.2 | 129.1 |
| 2 | 54.8 | 94.6 | 154.8 |
| 3 | 102.2 | 140.2 | 199.9 |

**Table 2**: Solution of $m^2\pi^2 k_n$

## Calculation of Theoretical Natural Frequency

Using the above data we can find the first mode of the plate with two ends clamped and other two ends simply supported

The value of the constants in the equations used is $\rho$= 7803Kg/$m^3$, E= 206800×$10^6$ Pa, $v$= 0.3, h= 0.004m, a= 0.6m

Where the symbols have their usual meanings as discussed earlier

If we put the values of the constants and input the value of the left hand side from the table with m=1 and n=1 we obtain the angular frequency, $\omega_{mn}$.

We know linear frequency $\lambda = \frac{\omega}{2\pi}$

Using the above relation we obtain the first natural frequency of the system to be **79.4 Hz**.

## EMPIRICAL ANALYSIS

## Experimental Rig

The experimental verification involves enacting the boundary conditions in real structure. An actual mild steel plate with the dimension of 0.6m by 0.6m by 0.004m is drilled at four corners and is bolted at a fixed support to ensure that the vibration is restricted to the plate only.

The actuation for the plate to vibrate at the first mode is supplied by simply hitting the plate with a hammer. The vibration is sensed by a proximity sensor. The sensor is connected to an oscilloscope and the response is recorded in terms of frequency only.



**Figure 38: The experimental rig**

Hence from the experimental rig we can obtain the frequency of the plate only bolted at four corners.

# Sensor

Sensor is an element in a mechatronics or measurement system that acquires physical parameters and changes it into a signal that can be processed by the system.

A Proximity sensor consists of an element that changes its state or an analog signal when it's close to, but often not actually touching, an object [10]. A proximity sensor sometimes uses an electromagnetic field or beam and look for changes in the field. The object being sensed is often referred to as the proximity sensor's target. Different proximity sensor targets demand different sensors. For example, a capacitive or photoelectric sensor might be suitable for a plastic target; an inductive proximity sensor requires a metal target. [11]

In this study Lion Precision ECL202 Eddy-Current Displacement Sensor is used. It provides high-resolution, noncontact measurement of position changes of a conductive target. The system consists of driver electronics and a probe calibrated for a specific material and range. [12]

The ECL202 provides a linear analog voltage proportional to changes in the target position and a



**Figure 4: the sensor setup**

digital switched (set point) output with a user programmed switching set point.
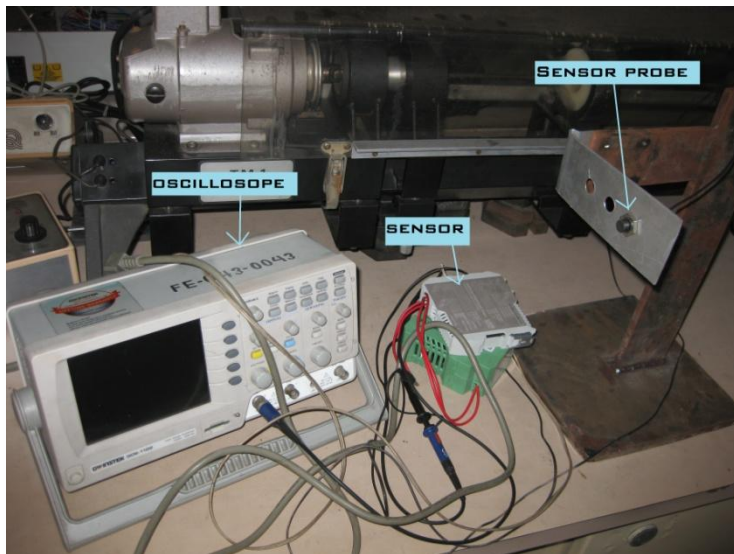
The output signal is an analog voltage of 0-10 VDC. The output voltage is proportional to the probe-target gap. As the probe-target gap increases, the voltage becomes more positive. The outputs can be viewed by means of an oscilloscope.

## Empirical results

In order for a sound reception of the vibration a thin metal foil (Aluminium) is glued on the surface where the probe is pointed. This is done because the sensor is keen in sensing Aluminium and there might be uneven surface factors or rust induced noise that might tamper the result.

The Vibration is induced crudely by hitting the plate moderately using a hammer. This process is repeated three times and we obtain the following frequency: **36 Hz, 37 Hz, and 36 Hz.**

The mean obtained frequency for the given boundary condition is thus **36.33 Hz**

# CASE COMPARISON AND CONCLUSION

## Case 1

### Modal

For first case we had FEM analysis of 0.6 m square plate with 4 mm thickness. The Modal values we obtained from ANSYS (APDL) were **66 Hz, 78 Hz** and **124 Hz**.

### Harmonic

 The plate was analyzed with external actuation the harmonic peaks observed were at **66 Hz, 78 Hz** and **124 Hz** respectively.

### Comparison

We can see that when the actuation frequency coincides with the modal frequency resonance occurs and hence Modal and Harmonic analysis confers with each other.

### Conclusion

Resonance occurs in the modal frequency obtained.

## Case 2

### Modal

For second case FEM was used again to analyze the modal frequency with the aforementioned boundary conditions. The obtained modes are **80.523 Hz** and **151.87 Hz**.

**Theoretical**

By the aid of Table 2 the first natural frequency for the plate was found to be **79.4 Hz.**

**Comparison**

It is evident that the modes obtained using FEM and Theoretical method confers well with each other the difference between the obtained modes from the two different method is 1.123 Hz. The percentage deviation is only 1.41% hence the two methods are compatible.

**Conclusion**

The mathematical model and the numerical analysis are complimentary to each other hence we can use either method to infer the vibration characteristics.

## Case 3

**Modal**

For the third case 2 modes were extracted. First mode one had the frequency **36.044 Hz** and second mode had the frequency **66.257 Hz.**

**Empirical**

After hammering the bolted plate the frequency was obtained by reading from the oscilloscope coupled with proximity sensor. In order to ensure accurate results three readings were recorder and the mean of the three readings were taken as the natural frequency of the structure. The obtained mean frequency was **36.333 Hz.**

**Comparison**

From the above comparison we can affirm that the modal frequencies obtained by FEM analysis are very much close to the frequency obtained by empirical method. The difference between the results is **0.289 Hz** the percentage difference is only 0.8%

**Conclusion**

The experimental and Finite element method are complimentary to each other thus we can predict the vibration behavior of any structure and avoid any possible damage due to adverse effects of vibration.

## **FURTHER WORK**

The above study provides comprehensive analysis between the numerical theoretical and empirical modeling. Natural frequencies were obtained by individual method and compared. This study hence provides a platform to take the FEM study a step further by verifying its veracity and applicability in harmonic analysis.

- Mathematical modeling for harmonic analysis can be performed to compare the harmonic analysis response as obtained from the FEM analysis. Empirical data can be obtained by designing or using a standard actuator to provide sinusoidal force on to the plate and obtaining the amplitude rating against frequency of actuation.
- Mathematical expression for bolted condition can be derived to connect the FEM, Theoretical and empirical methods.
- Boundary conditions could be varied to obtain vibration characteristics for multiple circumstances. Dampers can be incorporated to examine the change in natural frequency and amplitude of vibration.
- A plate can be deformed to various shapes namely airplane wing or turbine blades. Thus CFD can be coupled with vibration analysis to obtain the dynamic variation of vibration characteristics with the change in fluid flow over it.

Thus exploiting the superficial approach, while keeping the basic same, plethora of real life structural aspects can be evaluated. Hence a lot of destructive effects of vibrations can be avoided.

# References:

[1] *SK Chakraverty, Vibration of Plates*

[2] *Singiresu S. Rao, Mechanical Vibration*

[3] *Numerical Simulation of a Flexible Plate System for Vibration Control by A.R. TAVAKOLPOUR, I.Z. MAT DARUS, M. MAILAH*

[4] *Modal analysis by Zhi Fang Fu and Jimin He*

[5] ^ Vibration Type Approval Guide Book (PDF)

[6] *ANSYS website ( http://ansys.com/Products/Simulation+Technology/Structural+Mechanics)*

[7] *Dynamic Analysis of Cantilever Beam, University of Alberta*

[8] *Linear Equation Solver Memory Requirements (https://www.sharcnet.ca/Software/Fluent13/help/ans_per/perlineqsolv.html)*

[9] *Werner Soedel, Vibration of Shells and Plates*

[10] *Introduction of Mechatronics and Measurement Systems By David G. Alciaore and Michael B. Histand*

[11] *http://www.sensors-transducers.machinedesign.com/guiEdits/Content/bdeee4/bdeee4_7.aspx*

[12] *INSTRUCTION MANUAL, ECL202/ECL202e Eddy-Current Displacement Sensor. Measurement Systems from LION PRECISION.*