# DYNAMIC CHARACTERISTIC OF CONNECTING ROD FOR FOUR STROKE ENGINE

ADI FIRDAUS B. HAT

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

## SUPERVISOR’S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering *with Automotive Engineering.

Signature

Name of Supervisor: ASSOC. PROF. DR. MD MUSTAFIZUR RAHMAN

Position: LECTURER

Date:

## STUDENT’S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature

Name: ADI FIRDAUS B. HAT

ID Number: MH06050

Date:

**Dedicated to my parents**

Mr Hat B. Malik

Mrs Zainab Bt Ahmad

# ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Assoc. Prof. Dr. Md Mustafizur Rahman for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Degree program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

My sincere thanks go to all my labmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to my classmates for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my brothers and sisters, who consistently encouraged me to carry on my higher studies. I am also grateful to my fellow colleuges for their sacrifice, patience, and understanding that were inevitable to make this work possible. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

**ABSTRACT**

This thesis deals with computational modal analysis of the connecting rod. The objective of this project is to investigate the effect of the modal updating to the dynamic characteristic of the connecting rod, and to develop a finite element model of structure. This project also studies the vibration of a connecting rod in order to determine its natural frequencies and mode shapes. The connecting rod of the Modenas Kriss 110 cc engine has been considered as the model of the analysis. Structural modeling of connecting rod has been developed using SOLIDWORK software. The structural model then imported to the MSC.PATRAN software for generating mesh and the numerical analysis was performed using MSC.NASTRAN software. Mesh sensitivity have been done in order to determine the suitable mesh for further analysis. Result of the modal analysis show that first mode of the connecting rod occur at 77.411 Hz with higher displacement equal to 16.7mm. Natural frequency of the connecting rod can be increase by improvement of the properties of the material. In modal updating analysis show that material with lower density and higher Modulus of Young’s will increase the natural frequency of the connecting rod.

.

## ABSTRAK

Tesis ini berkaitan pengkomputeran analisis getaran rod penghubung. Objektif dari projek ini adalah untuk meneliti kesan daripada prosess mengemaskini mod kepada ciri dinamik rod penghubung, dan untuk menghasilkan sebuah model elemen struktur. Projek ini juga mempelajari getaran rod penghubung dalam rangka untuk menentukan frekuensi semula jadi dan bentuk mod. Rod penghubung dari Modenas Kriss 110 cc enjin telah dianggap sebagai model analisis. Pemodelan struktur rod penghubung telah dibangunkan menggunakan perisian solidwork. Model struktural kemudian diimport ke perisian MSC.PATRAN untuk menghasilkan mesh dan analisis berangka dilakukan dengan menggunakan perisian MSC.NASTRAN. Mesh sensitiviti telah dilakukan dalam rangka untuk menentukan mesh berpadanan untuk analisa lebih lanjut. Keputusan analisis mod pertama menunjukkan bahawa mod daripada rod penghubung berlaku di 77,411 Hz dengan anjakan sebanyak 16.7mm. Frekuensi semulajadi rod penghubung boleh meningkat dengan peningkatan sifat-sifat bahan. Dalam pengemaskini modal analisis menunjukkan bahawa bahan dengan kepadatan yang lebih rendah dan lebih tinggi nilai Young’s Modulus akan meningkatkan frekuensi semula jadi batang penghubung.

**TABLE OF CONTENTS**

## LIST OF FIGURES

## LIST OF TABLES

# LIST OF SYMBOLS

| | |
|---|---|
| $E$ | Modulus of elasticity |
| $F$ | Force |
| $f$ | frequency |
| $I$ | Nodal displacement |
| $K$ | Material stiffness |
| M | Mass |
| $\varphi$ | Corresponding vector |
| $\rho$ | Density |
| $\omega$ | Natural frequency |

## LIST OF ABBREVIATIONS

| | |
|---|---|
| EMA | Experimental modal analysis |
| FEM | Finite element Method |
| FE | Finite element |
| FFT | Fast Fourier transform |
| FRF | Frequency response function |
| IC | Internal combustion |
| NVH | Noise and vibration harshness |
| SAE | Society of Automotive Engineers |

# CHAPTER 1

# INTRODUCTION

## 1.1 BACKGROUND

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. They can be produced either by casting, powder metallurgy or forging. There exist only few publications on the simulation of the forging process for the production of connecting rods. (Grass et al., 2005). Connecting rod transfer force from the piston after the combustion in engine to the crankshaft. (Bell, 1997). The connecting rod was invented sometime between 1174 and 1200 where a muslim inventor, engineer and craftsman named al-Jazari build five machine to pump water for the king of the Turkish Artaqid dynasty. The connecting rod are most made of steel for production engines, but can be made of aluminum (for lightness and the ability to absorb high impact at the expense of durability) or titanium (for combination of strength and lightness at the expense of affordability) for high performance engines, or of cast iron.

Automotive industry requirements for quality, productivity and cost efficiency are at a level where study of the design and manufacturing of a product must occur in the earliest stages of conception. The time spent in trial and error analysis in the design process needs to be eliminated for a manufacturer to remain competitive in a global market. There fore, computational method have been used in the early stage of the design. (Lauwagie et al., 2008)

Finite element method is applied during modal analysis of connecting rod. Modal analysis is the process of determining the inherent dynamic characteristics of a system in form of natural frequencies, damping factors and mode shapes, and using them to formulate a mathematical model for its dynamic behavior. Hence, mesh determination is too critical in order to ensure that the best mesh size is to be use in carry out the analysis for other parameter involves. As stability and convergence of various mesh processing applications depend on mesh quality, there is frequently a need to improve the quality of the mesh (Taubin, 1995). This improvement process is called mesh optimization (Hoppe et al., 1993).

For the improvement of the design, modal updating can be run in order to determine the effect of the material properties to the dynamic characteristic of the design. Further more, modal analysis can be used for the experimental modal analysis (EMA) for the experiment setup to locate the accelerometer and the result of the experimental modal analysis can be correlate with the result of the computational modal analysis.

## 1.2 PROBLEM STATEMENT

Any physical system can vibrate. The frequencies at which vibration naturally occurs, and the modal shapes which the vibrating system assumes are properties of the system can be determined analytically using modal analysis. Analysis of vibration modes is a critical component of a design, but is often overlooked. Inherent vibration modes in structural components can shorten connecting rod life, and causes premature or completely unanticipated failure, often resulting in hazardous situations. Detailed modal analysis determines the fundamental vibration modes shapes and corresponding frequencies. This can be relatively simple for basic components of a simple system, end extremely complicated when qualifying a complex mechanical device or a complicated structure exposed to periodic wind loading. These system required accurate determination of natural frequencies and mode shape using techniques such as Finite Element Analysis. Using Finite Element models to predict the dynamic properties of structures becomes more and more important in modern mechanical industries, such as the automobile industries. Whenever there is new design or modification of an existing design, the structural dynamic properties of the product must be examined to fulfill some criteria proposed either by the industry itself and/or external agencies before the product can be launched on the market. FE model predictions are used more and more to take the place of practical dynamic test data (Chen, 2001).

## 1.3 MOTIVATION

Automotive industry requirements for quality, productivity and cost efficiency are at a level where optimization of the design and manufacturing of a product must occur in the earliest stages of conception. The time spent in trial and error analysis in the design process needs to be eliminated for a manufacture to remain competitive in a global market. The Finite Element optimization approach allows an efficient evaluation of designs using advance mathematical tools. Methods must be developed, however, to create fully parameterized models for optimization which can be experimentally verified under pertinent loading conditions (Lauwagie et al., 2008).

## 1.4 OBJECTIVES

- To develop a finite element model of structure
- To study the vibration of a connecting rod in order to determine its natural frequencies and mode shapes.
- To investigate the effect of the modal updating to the dynamic characteristic of the connecting rod

## 1.5 SCOPE OF STUDY

- Solid work – The model of connecting rod are developed using Solid Work software after taking dimension.
- Finite Element Method – using for modal analysis

- Connecting rod are modeled as rigid bodies and are fixed in position
- Solid Work are used because it the easy way to develop the model compared to 3D scanner that hardly to run

## 1.6 OVERVIEW OF THE REPORT

Chapter 1 introduces the background of the project. The problem statement and the scopes of this study also included in this chapter. Chapter 2 presents the literature study about finite element method, modal analysis and modal updating. Chapter 3 discusses the development of finite element modeling and analysis, modal analysis and modal updating. Chapter 4 presents the results and analysis of the obtained results and discusses it elaborately. Chapter 5 presents the conclusion and recommendation of the future work.

# CHAPTER 2

# LITERTURE REVIEW

## 2.1 INTRODUCTION

This chapter reviews the literature of past research efforts related to finite element modeling. A review of other relevant research studies is also provided.

## 2.2 HISTORICAL DEVELOPEMENT OF MODAL ANALYSIS

There are two landmarks of such atomistic comprehension in the recent history of science which paved the way for the birth of modal analysis. Newton, from his observation of the spectrum of sunlight, confirmed its composition of color components. Fourier, based on earlier mathematical wisdom, claimed that an arbitrary periodic function with finite interval can always be represented by a summation of simple harmonic functions. The Fourier series and spectrum analysis laid on a solid foundation for the development of modal analysis.

Theoretical modal analysis can be closely identified with the wave equation which describes the dynamics of a vibrating string. From the solution, we can determine its natural frequencies, mode shapes and forced responses – constituents so accustomed by today's modal analysts. This stage of modal analysis, develop during the nineteenth century, was a largely dependent upon mathematics to solve partial differential equations which describe different continuous dynamic structures. The elegance of the solution is evident while the scope of solvable structures is limited.

The concept of discretization of an object and introduction of matrix analysis brought about a climax in theoretical modal analysis early in the last century. Theory was developed such that structural dynamic analysis of an arbitrary system can be carried out when knowing its mass and stiffness distribution in matrix forms. However, the theory could only be realized after computers became available. In that aspect, theoretical (or analytical) modal analysis is very much numerical modal analysis.

The invention of the fast Fourier transform (FFT) algorithm by Cooly and Tukey (1965) finally paved the way for raid and prevalent application of a experimental technique in structural dynamics. With FFT, frequency response of a structure cab is computed form the measurement of given inputs and resultant responses. The theory of modal analysis helps to establish the relationship between measured FRFs and the modal data of the tested specimen. Efforts were focused on deriving modal data from measured FRF data. Today, modal analysis has entered many fields of engineering and science. Application range from automotive engineering, aeronautical and astronautically engineering to bioengineering, medicine and science. Numerical (finite element) analysis have become two pillars in structural dynamics (He & Fu 2001).

## 2.3 MODAL ANALYSIS

Modal analysis as the procedure by witch orthogonality properties of the modes of vibration are utilized to transform the equations of motion from the physical coordinate system to the principal coordinates system and thereby decuople the equation of motion. Modal analysis also can be define as the process of determining the inherent dynamic characteristics of a system in form of natural frequencies, damping factors and mode shapes, and using them to formulate a mathematical model for its dynamic behavior.

Modal analysis is based upon the fact that the variation response of a linear time-invariant dynamic system that can be expressed as the linear combination of a set of simple harmonic motions called the natural modes of vibration. This concept is akin to the use of a Fourier combination of sine and cosine waves to represent a complicated waveform. The natural modes of vibration are inherent to a dynamic system and are determined completely by its physical properties (mass, stiffness, damping) and their spatial distributions. Each mode is described in terms of its modal parameters: natural frequency, the modal damping factor and characteristic displacement pattern, namely mode shape. The mode shape may be real or complex. Each corresponds to a natural frequency. The degree of participation of each natural mode in the overall vibration is determined both by properties of the excitation source(s) and by the mode shapes of the system. The theoretical modal analysis anchors on a physical model of a dynamic system comprising its mass, stiffness, and damping properties. These properties may be given in forms of partial differential equations. A more realistic physical model usually comprise the mass, stiffness and damping properties in terms of their spatial distributions, namely the mass, stiffness and damping matrices. These matrices are incorporated into a set of normal differential equations of motion. The superposition principle of a linear dynamic system enables us to transform these equations into a typical eigenvalue problem. Its solution provides the modal data of the system.

Modern finite element analysis empowers the discretization of almost any linear dynamic structure and hence has greatly enhanced the capacity and scope of theoretical modal analysis. On the other hand, the rapid development over the last two decades of data acquisition and processing capabilities has given rise to major advances in the experimental realm of the analysis, which has become known as modal testing (He & Fu, 2001).

## 2.4 DIFFERENCE BETWEEN MODAL AND STATIC ANALYSIS

The fundamental FEM equation that is applicable to static analysis is expressed as Equation (2.1).

$$[F] = [K][d] \tag{2.1}$$

To consider dynamic effect, the equation is extended to account for inertial and damping effects and for the fact that load can be a function of time which is expressed as Equation (2.2).

$$[M]\ddot{\imath} + [C]\dot{\imath} + [K]\,\mathrm{I} = F(t) \tag{2.2}$$

where

[M] = Known mass matrix

[C] = Known damping matrix

[K] = Known stiffness matrix

[1] = unknown vector of nodal displacement

Modal analysis deals with free and undamped vibrations where F(t) = 0 (no excitation force) and [C] = 0 (no damping). There Equation (2.2) can be rewritten as Equation (2.3)

$$[M]\ddot{\imath} + [K]\mathrm{I} = 0 \tag{2.3}$$

Non-zero solution of Equation (2.3) presents an eigenvalue problem and provide with modal frequencies and associated modal shape of vibration:

$$[K]\{\varphi\}i \;=\; \omega\, i\,^2[M]\,\{\varphi\}\,i \tag{2.4}$$

Equation (2.4) has *n* solution, where $\omega i^2$ is called the eigenvalue, and the corresponding vector {φ}I is called the eigenvector. The relation between the eigenvalue and the frequency expressed in Hertz [Hz] in Equation (2.5)

$$fi \;=\; \frac{\omega i^2}{2л} \tag{2.5}$$

## 2.5 INTERPOLATION OF DISPLACEMENT AND STRESS RESULTS IN MODAL ANALYSIS

Modal analysis cannot provide any quantitative information about displacements or stresses. Displacement calculated in modal analysis may be used only to compare the relative displacement between different portions of the analyzed structure and only within the same mode of vibration (Kurowski, 2004).

## 2.6 IMPORTANCE OF SUPPORTS IN MODAL ANALYSIS

The natural frequencies depend on the applied supports. Natural frequencies will increase with the addition of supports because the added support will increase the structural stiffness (Kurowski, 2004).

Applications of modal analysis are closely related to utilizing the derived modal model in design, problem solving and analysis. Theoretical modal analysis relies on the description of physical properties of a system to derive the modal model. Such a description usually contains the mass, stiffness and damping matrices of the system thus, it is a path from spatial data to modal model (He & Fu, 2001).

## 2.7 PRACTICAL APPLICATIONS OF MODAL ANALYSIS

The majority of practical application has been those from aeronautical engineering, automotive engineering and mathematical engineering in particular. This is not to discredit the fact that the application of modal analysis is becoming more strongly interdisciplinary. In automotive engineering, the enormous commercial and safety aspect associated with redesigning a vehicle oblige the best possible understanding of dynamic properties of vehicular structures and the repercussion of any design changes. Modern vehicle structure must be light in weight and high in strength. A study on analytical modal analysis enables improved design of automotive components and enhancement of dynamic properties of a vehicle.

Analytical modal analysis as a troubleshooting tool also plays a crucial role in the study of vehicle noise and vibration harshness (NVH). A simple modal analysis of a body-in-white or a sub frame structure is a typical application. However, more sophisticated applications have been achieved such as those involved in modal sensitivities of vehicle floor panels, structural optimization for vehicle comfort, vehicle fatigue life estimation, vehicle suspension with active vibration control mechanism and condition monitoring and diagnostic system for the vehicle engine. Another prominent application area for modal analysis is in the study of vehicle noise. Modal analysis is used as a tool to understand structure-borne noise from vehicle components or airborne sound transmission into vehicle cabins through door-like structure and the path noise takes to transmit in vehicles.

A more advanced application of modal analysis involves interior noise reduction through structural optimization or redesign. As a whole, modal analysis has been an effective technique for automotive engineering, in its quest to improve a vehicle's NVH. The rapid development in the aeronautical and astronautically industries has challenged many disciplines of engineering with diverse technological difficulties. The structural dynamics of both aircraft and spacecraft structures have been a significant catalyst to the development of modal analysis. Aircraft and spacecraft structures impose stringent requirements on structural integrity and dynamic behavior which are shadowed by rigorous endeavors' to reduce weight. The large dimension of spacecraft structures also asserts new stimulus for structural analysts.

In some multi-disciplinary topics such as fluid induced vibration and flutter analysis, modal analysis has also been found to be a useful tool. Light and large-scale on-orbit space structures impose a control problem for structural engineers. The research in this area has led to notable advances in recent years on blending modal analysis and active control of space structures.

Modal analysis also found increasing acceptance in the civil engineer's community where structural analysis has always been a critical area. Civil structures are usually much larger than those mechanical and aeronautical structures for which modal testing were originally developed. A large number of applications are concerned with prediction of responses of a civil construction due to ambient vibration or external loadings. This response prediction endeavor relies on an accurate mathematical model which can be derived by modal analysis.

Examples of such applications range from tall buildings, soil-structure interaction to a dam-foundation system. The real life force inputs involved in civil structures are earthquake waves, wind, ambient vibration, traffic load, etc. Recent years have seen an upsurge of modal testing on bridges to complement traditional visual bridge inspection and static testing. Acoustic modal analysis has been used to analyze the dynamic characteristics of speaker cabinets. This analysis provides crucial information if the design of new speakers with improved sound quality (He & Fu, 2001).

## 2.8 MODAL ANALYSIS COMPARED TO DIRECT METHOD

The direct method has the advantages that a solution for the response does not require a special form for the damping, as doe's modal analysis. However, in general, the solution for the roots of the characteristic determinant becomes computationally intensive for system with more than a few degrees of freedom. In fact, there are no exact solutions for the roots of a characteristic polynomial of order greater than four. In these instances, numerical techniques for determining the eigenvalues are used. The most suitable method for a specific eigenvalue problem depends to some extend on the size of the matrix (number of vibration degrees of freedom) and how many eigenvalue and eigenvectors are needed to model the response. Matrix iteration techniques are the basic for many eigenproblem solution techniques used in practice. It is because the numerical methods for estimating frequencies and vectors are iterative, that is the estimates are sequential, that the modal approach permits the analyst to retain only as many degrees of freedom as needed for an accurate result.

In modal analysis, the symmetry of the stiffness and mess matrices are key to the orthogonality of the natural modes. The orthoganality relations are also useful as a check on the accuracy of a numerical computation of the modes. The modal analysis approach proceeds with decouple oscillator equations that are solved independently in modal space. Modal analysis is also useful as an approximate method for systems with a large number of degrees freedom, for example, hundreds to thousands of degrees of freedom. If the forcing has only lower frequencies components, modal analysis with only a small number of lower modes can be effectively used. It is numerically possible to selectively pick particular eigenvalue ranges within the response is needed. Generally, this is due to loading that has a particular frequency band. In this way, the critical modes are solved for without wasting computational time on those modes that have little or no contribution (Benaroya, 2004).

## 2.9 FINITE ELEMENT METHOD

The finite element method is a powerful numerical method that is used to provide approximations to solution of static and dynamic problems for continuous system. The disciplines in which the finite element method can be applied include stress analysis, heat transfer, electromagnetic, fluid flow, and vibrations. Application of the finite element method to a continuous system requires the system be divided into a finite number of discrete elements. Interpolations for the dependent variables are assumed across each element and are chosen to assure appropriate interelement continuity.

The interpolating functions are developed in terms of the unknown value of the dependent variables at discrete points, called nodes. The nodes for a one-dimensional system are located at element boundaries. A variation principle is applied to derive equations whose solution leads to approximations to the dependent variables at the nodes. The defined interpolations are used to provide approximations to the dependent variables across the system. Lagrange's equations, derived by the calculus of variations, are applied for variations problems, resulting in a set of differential equations for the dependent variables at the nodes. (Graham, 2000). The finite element method for the vibration problem could be derived by applying the Rayleigh-Ritz method with the interpolating function chosen to be defined piecewise over each element.

Boundary conditions for continuous system are classified as two types: (i) geometric boundary conditions are those that must be satisfied according to geometric constraints. (ii) natural-boundary conditions that must be satisfied as a result of force and moment balances. Since the natural boundary conditions are incorporated into the energy scalar products, the chosen interpolating functions for a Rayleigh-Ritz approximation must satisfy only the geometric boundary conditions. The FEM is an extension of Rayleigh-Ritz method. The classical Rayleigh-Ritz method represents a variation approach whereby a distributed system is approximated by a discrete one by assuming a solution of the differential eigenvalue problem as a finite series of admissible functions. The basic difference between the classical Rayleigh-Ritz methods and the finite element method lies in fact that in the latter an approximate solution is constructed using local admissible functions defined over small sub domains of the structure. The concept of local functions defined over small sub domain carries enormous implications and is key to the success of the finite element method. As the sub domains are small, good approximations can be realized with local admissible functions in the form of low-degree polynomials, which are referred to as interpolation functions.

The procedure for determining the element stiffness, mass matrices and assembling them into global stiffness and mass-matrices is demonstrated by means of second-order systems such as the string in transverse vibration using interpolation functions. The real power of FEM becomes evident in 2-dimensional system such as membranes and plates, where finite elements of various shapes like triangular, rectangular and quadrilateral elements as well as elements with boundaries are encountered (Dukkipati, 2006).

The FEM is a technique for the spatial discretization of distributed parameter systems. It consists of dividing the domain of system into a set of sub domains and describing the motion over each of these sub domains by means of a linear combination of trial functions. The sub domains are called finite elements, the set of finite elements is known as the mesh of the trial functions are referred to as interpolation functions. The accuracy of finite element models can be improve by reducing the sized of h of finite elements in a process known as refining the mesh, which is the same as increasing the number of elements. Accuracy can also be improved by increasing the degree of interpolation functions, but this approach Is less commonly used than refining the mesh method. Both of these approaches result in an increase in the number of degrees of freedom of the finite element models.

The element stiffness, and mass matrices, and force vectors are derived from basic principle foe a bar element, a torsion element, and a beam element. The transformation of element matrices and vectors from the local to the global coordinate system is presented. The equations of motion of the complete system of finite element together with the boundary conditions are described. The concepts of consistent and lumped mass matrices are discussed. Only the used of one-dimensional elements in considered in the numerical treatment (Dukkipati, 2006).

## 2.10 CONCLUSION

This chapter is about the summary of previous works that related to this project. The works were discussed about finite element method, modal analysis and comparison of modal analysis to the static analysis. The next chapter will be discussed methodology of this project.

# CHAPTER 3

# METHODOLOGY

## 3.1 INTRODUCTION

This chapter presents the overall methodology of the finite element based modal analysis. The aim of this chapter is to develop a methodology for finite element modeling, modal analysis and modal updating.

## 3.2 STRUCTURAL MODELLING

Before creating structural model using SolidWork software, the typical measurements of the connecting rod have to be done. The connecting rod used for measurement is Modenas Kriss 100cc engine. The model is constructed in software using 1:1 scale. The measurement of the connecting rod are accurate in order to obtain the 3D model that are closely fit scale.

**Figure 3.1:** Typical connecting rod

The model created in the structural design is developed carefully to make sure that in the meshing step, the model can be mesh. Sharp edges, open lines and hidden bodies are the common things that will interrupt the meshing process of the structural model. The accuracy of the further result is influenced by meshing criteria of the structural model. The structural model of connecting rod is shown in Figure 3.1. The overall dimensions of the connecting rod are shown in Figure 3.3.

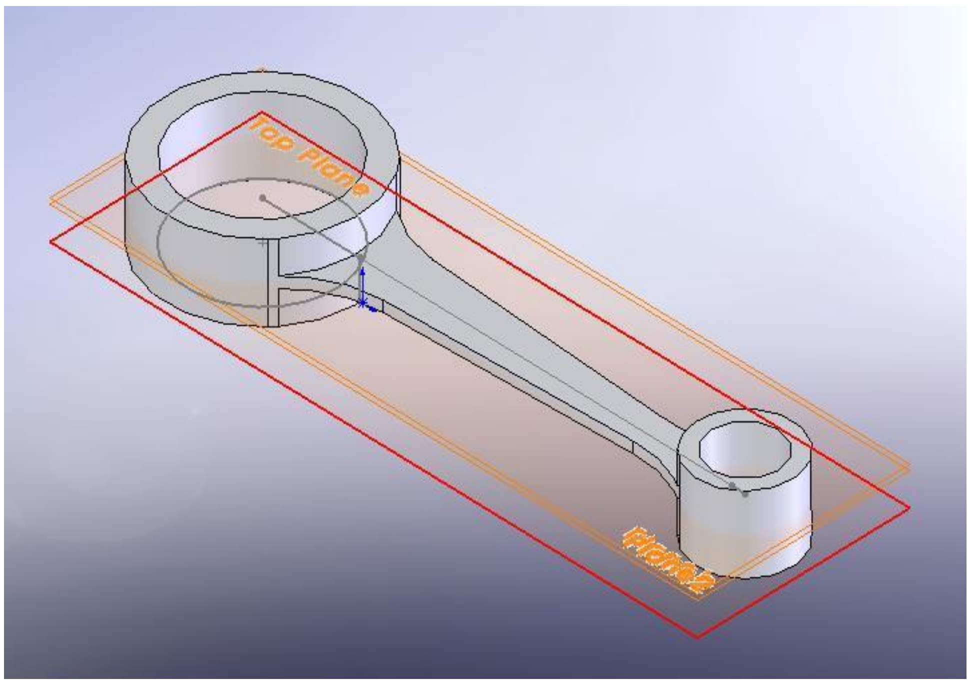


**Figure 3.2:** Structural model of the connecting rod

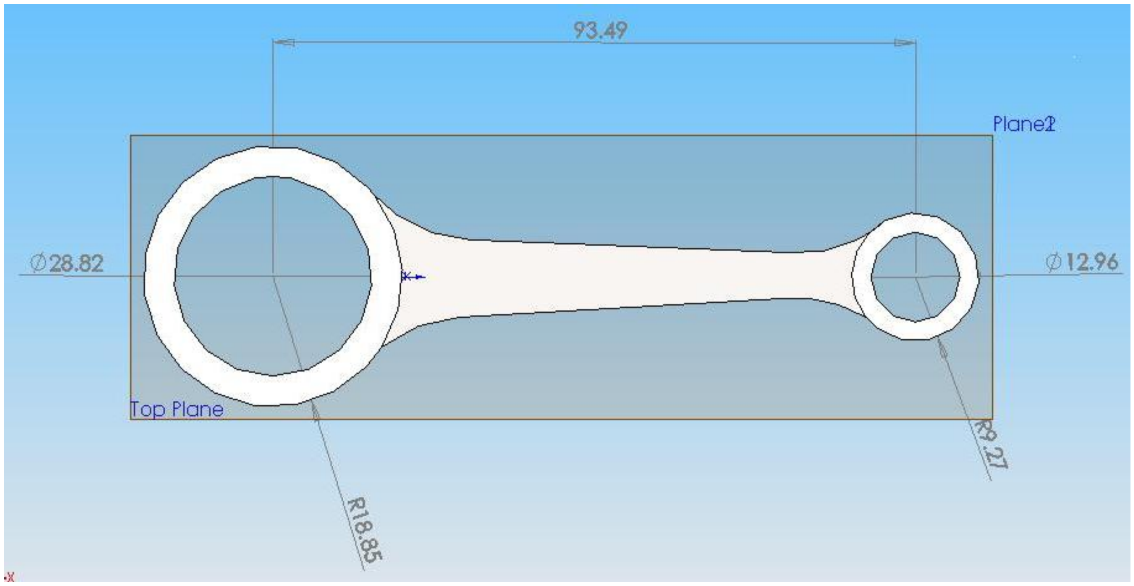


**Figure 3.3:** Overall dimension of connecting rod

## 3.3 FINITE ELEMENT MODELLING

The finite element is associate with the mesh elements at the computational step. For the moment, finite element is a geometric element supplied with a list of nodes. A node is a point supporting degrees of freedom. The nodes are defined according to the interpolation used in the computation. For a given geometric element, several finite elements may be exhibited as a function of the interpolation step. The simplest finite element is the Lagrange $P^1$ finite element whose nodes are the element vertices. A Lagrange $P^2$ finite element includes as nodes the elements vertices and a point on each of its edge. Other finite element may involve several nodes for each edges, nodes located on faces or inside the element while the element vertices may be modes or not.

The variability of the global length is also be tested to determine the optimum size of mesh. The smaller the size of the global length, higher accuracy of analysis result. The global edge length with the best accuracy and consistency of analysis will be selected throughout the whole analysis process. Figure 3.4 and 3.5 show that difference between two structural models meshed with different dimension of global edge length. The bigger the global edge length, the coarser the mesh it will have.

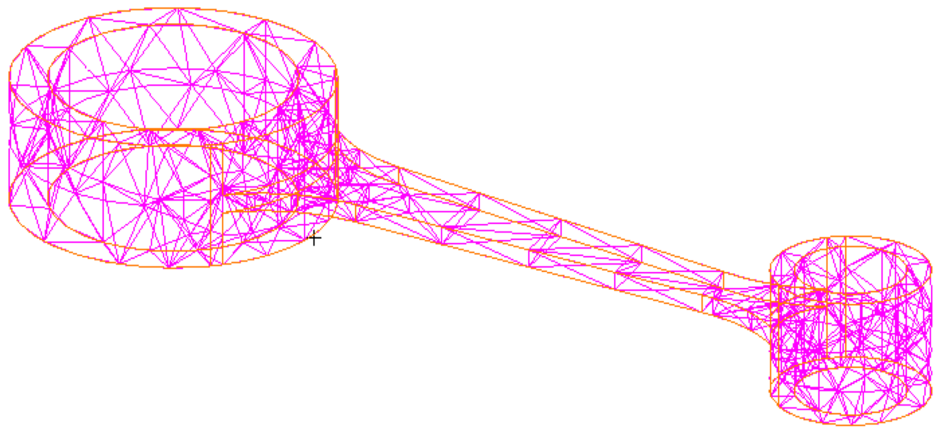

**Figure 3.4:** Tetrahedral with 10 nodes; Global Edge Length = 10mm

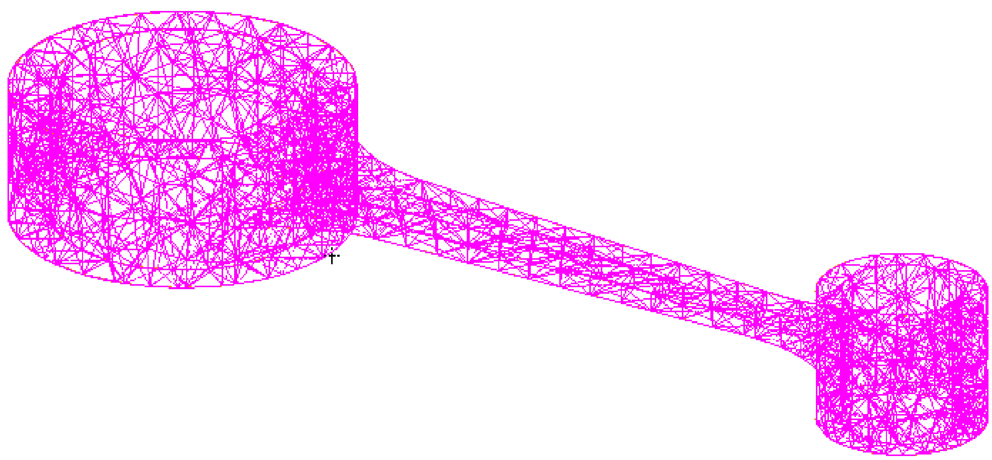

**Figure 3.5:** Tetrahedral with 10 nodes; Global Edge length = 4 mm

## 3.4 SENSITIVITY ANALYSIS

Sensitivity coefficients quantify the variation of a response value as a result of modifying parameter value. Sensitivity is not only used by the updating algorithm to find the necessary parameter changes, they are also helpful to gain insight in the structure, determine important parameters, and thus refine selection of updating parameters. Methods to compute sensitivity should be available for every element of the model, the number of gradients that must computed can become very large. An array of methods based on finite differences and differential approaches should be available. Fast approximate methods that produce color-coded displays with sensitivity distribution as required encouraging analysis to use this type of investigations.

## 3.5 MODAL ANALYSIS

In modal analysis, the parameters needed is, Young's Modulus (E), density (ρ), and poison ratio. Mesh used for the analysis is Tetrahedral 10 with global edge length = 4 mm. After the meshing process completed, the next procedure will be setting constrains for the finite element model of the connecting rod. The surface that connect with the bearing on the connecting rod are set to be fix in X, Y, and Z direction translational and rotational. Figure 3.6 shows constrain of finite element model.

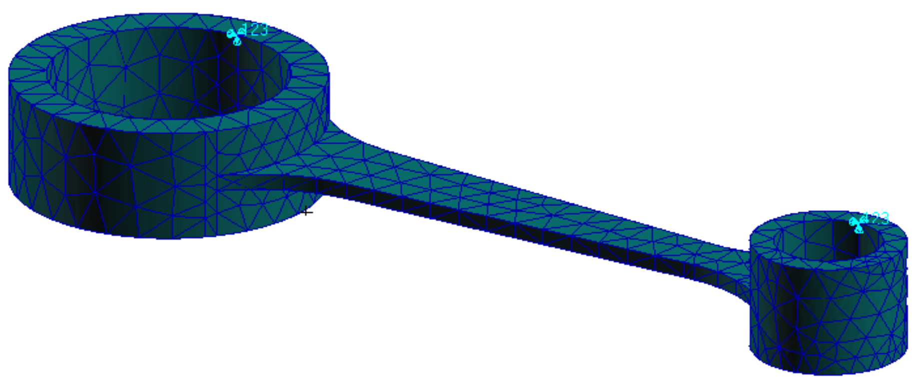

**Figure 3.6:** Constraints and boundary condition of finite element model

This study used real eiginvalue analysis to determine basic dynamic characteristics of a model. For this analysis, there are no forces applied to the model. The material properties have to be defined. Table 3.1 show the properties of forged 34CrMo4 TQ+T ISO 683-1 steel (Rabb, 1995).

**Table 3.1:** Forged 34CrMo4 TQ+T ISO 683-1 Steel Properties

| Properties | Value |
|---|---|
| Modulus of Elasticity | 200 GPa |
| Density | 8.05 x $10^{-8}$ Kg/$m^3$ |
| Poisson ratio | 0.3 |

Next step is to determine the mass input type that is be used. Mass is formulated either lumped mass or coupled mass. Technically lumped mass matrices contain uncoupled, translational component of mass. Coupled mass matrices contain translational components of mass with coupling between the components. Coupled mass has been chose for this analysis because it result higher value compare to lumped mass.

## 3.6 MODAL UPDATING

After done modal analysis, analysis can be proceeding to modal updating. The purpose of modal updating is to analyze the effect of the parameter updating to the natural frequencies. For this study, two types of parameters has been chose for the modal updating. The parameters are Modulus of Elasticity (E) and Density (ρ) of the material. The use of multiple targets to be matched by a simulation model i.e. resonant frequencies, mode shapes, frequency response, etc. may require local changes of stiffness and mass. With sensitivity analysis and modal updating of local parameters it is possible to visualize the areas in a model that need changes.

**CHAPTER 4**

**RESULT AND DISCUSSION**

## 4.1 INTRODUCTION

This chapter represents the details results of FE modeling, selection of the mesh type, result of modal analysis and effect of modal updating.

## 4.2 VON MISES STESS

Von mises stress is another active variable have been study in linear static stress analysis of connecting rod. The analysis between two different types of tetrahedral mesh is done using variable global edge length. Figure 4.1 and 4.2 show the result of von mises stress for different nodes.

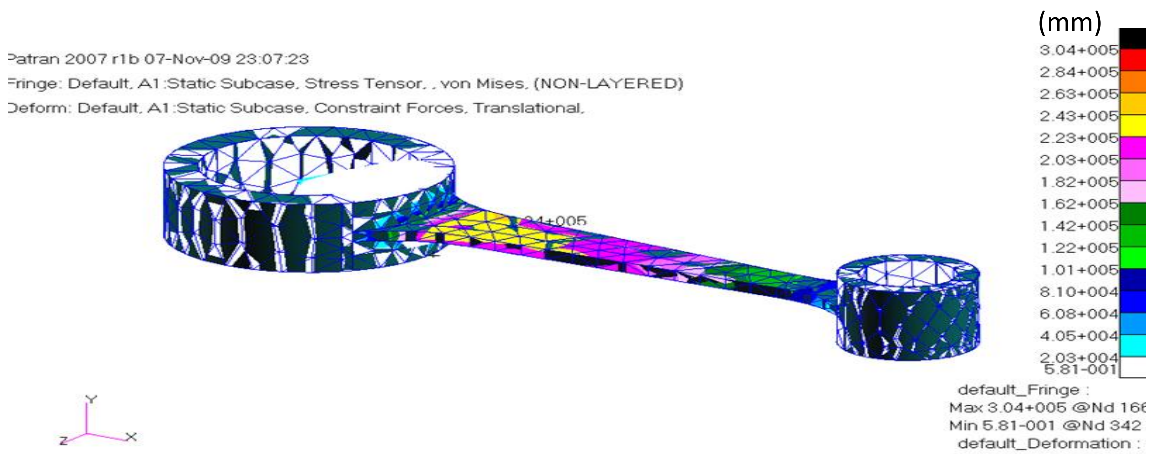


**Figure 4.1:** Von Mises Stress for tetrahedral with four nodes

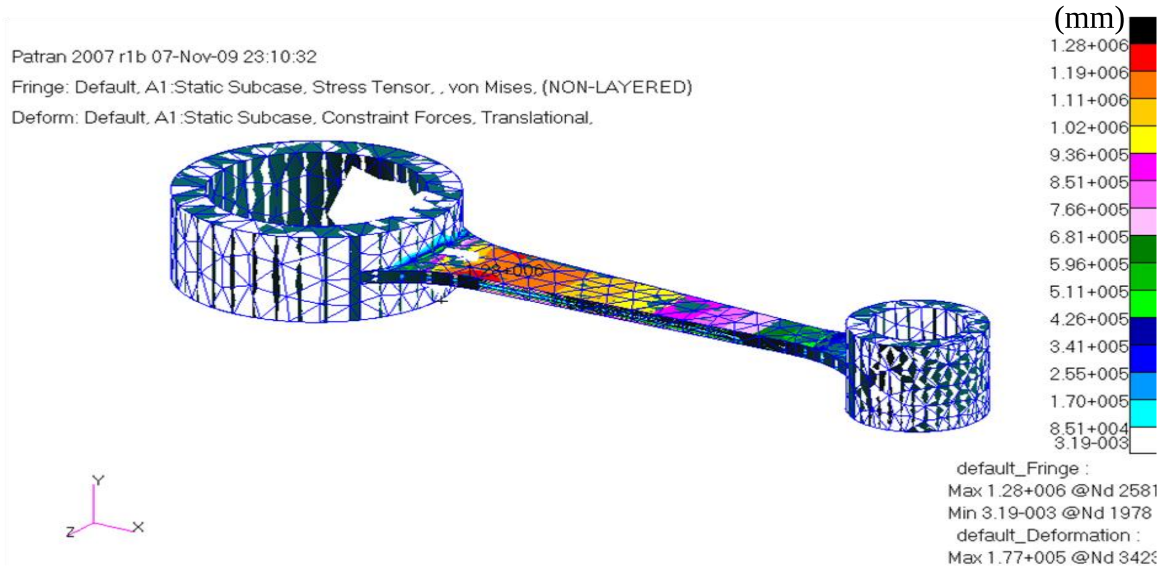


**Figure 4.2:** Von Mises Stress for tetrahedlral with ten nodes

Different contour of color shows the difference between lower and the higher von mises stress of the finite element model of connecting rod. The data for the vom mises stress then is tabulated to plot the graph of von mises stress versus global edge length for each types of tetrahedral mesh. Table 4.1 show the difference of von mises stress with variable global edge length.

**Table 4.1:** Von Mises Stress with Variable Global Edge Length

| **Global Edge Length (mm)** | **Von Mises Stress (Pa)** | |
|---|---|---|
| | **Tetrahedral 4 (x $10^5$)** | **Tetrahedral 10 (x $10^5$)** |
| 4 | 3.04 | 12.8 |
| 5 | 2.52 | 12.6 |
| 6 | 2.26 | 12.4 |
| 7 | 2.10 | 12.2 |
| 8 | 1.95 | 12.0 |
| 9 | 1.80 | 11.8 |
| 10 | 1.72 | 11.7 |
| 11 | 1.70 | 11.5 |
| 12 | 1.68 | 11.4 |
| 13 | 1.58 | 11.3 |

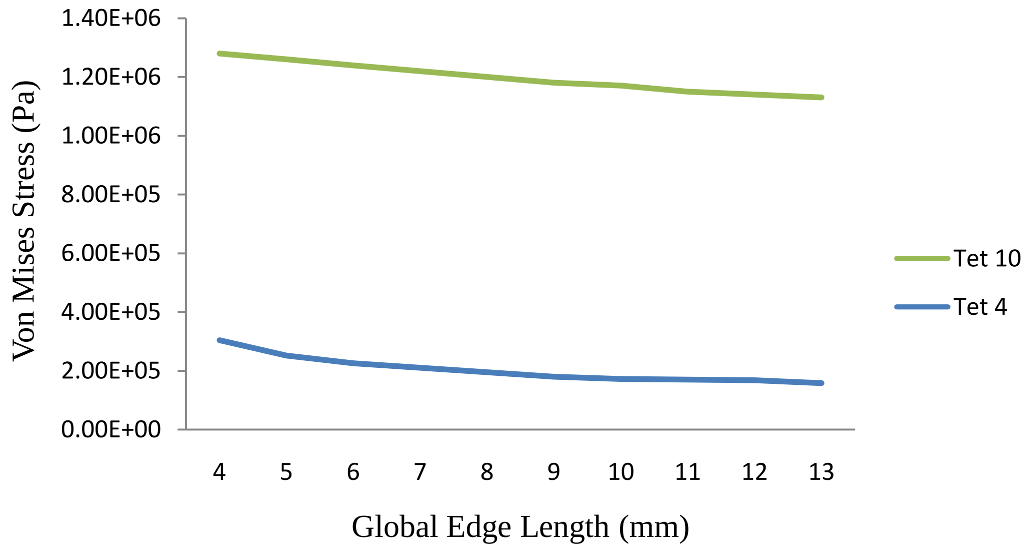


**Figure 4.3:** Von Mises Stress with variable global edge length

The figure 4.3 is plotted to exhibit the trend and study the correlation between von mises stress and global edge length. From the graph, its shows that tetrahedral with ten nodes gives higher value of von mises stress compare to the tetrahedral with four nodes mesh. It also show that lower value of global edge length gives higher value of von mises stress. This is because tetrahedral with ten nodes with lower value of global edge length are more sensitive mesh.

## 4.3 MODAL ANALYSIS

After determine the mesh that was used for the analyses, next process are continued by carried the modal analysis of finite element model of connecting rod. In this process, two types of mass are used including lumped and coupled mass. Table 4.2 shows the value of the frequency of each modes of lumped and coupled mass.

**Table 4.2:** Frequency of each mode

| Mode | Frequency (Hz) | |
|---|---|---|
| | Lumped | Coupled |
| 1 | 76.874 | 77.411 |
| 2 | 209.82 | 210.96 |
| 3 | 317.64 | 318.61 |
| 4 | 334.85 | 336.86 |
| 5 | 407.88 | 408.82 |
| 6 | 667.07 | 668.16 |
| 7 | 692.23 | 692.59 |
| 8 | 759.02 | 760.25 |
| 9 | 986.3 | 987.21 |
| 10 | 1060.1 | 1063.6 |

Comparison between Lumped and Coupled mass Techniques is shown in Figure 4.4. it can be seen that both Technique are linearly increase the natural frequency and mode shape are presented in Figure 4.5 to 4.14 for different 10 modes.

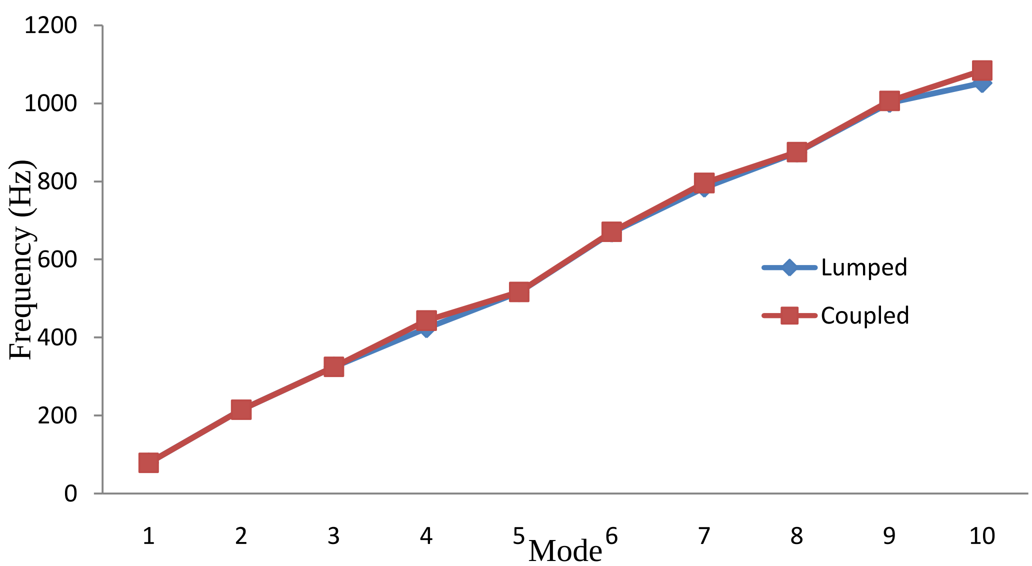


**Figure 4.4:** Frequency of each mode

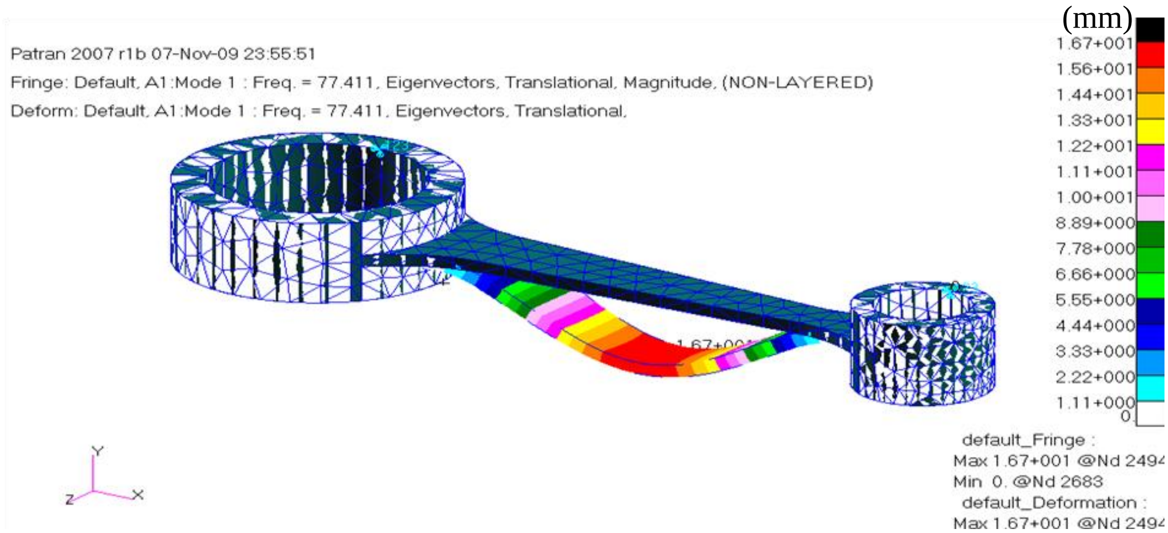


**Figure 4.5:** Natural frequency and mode shape for mode 1

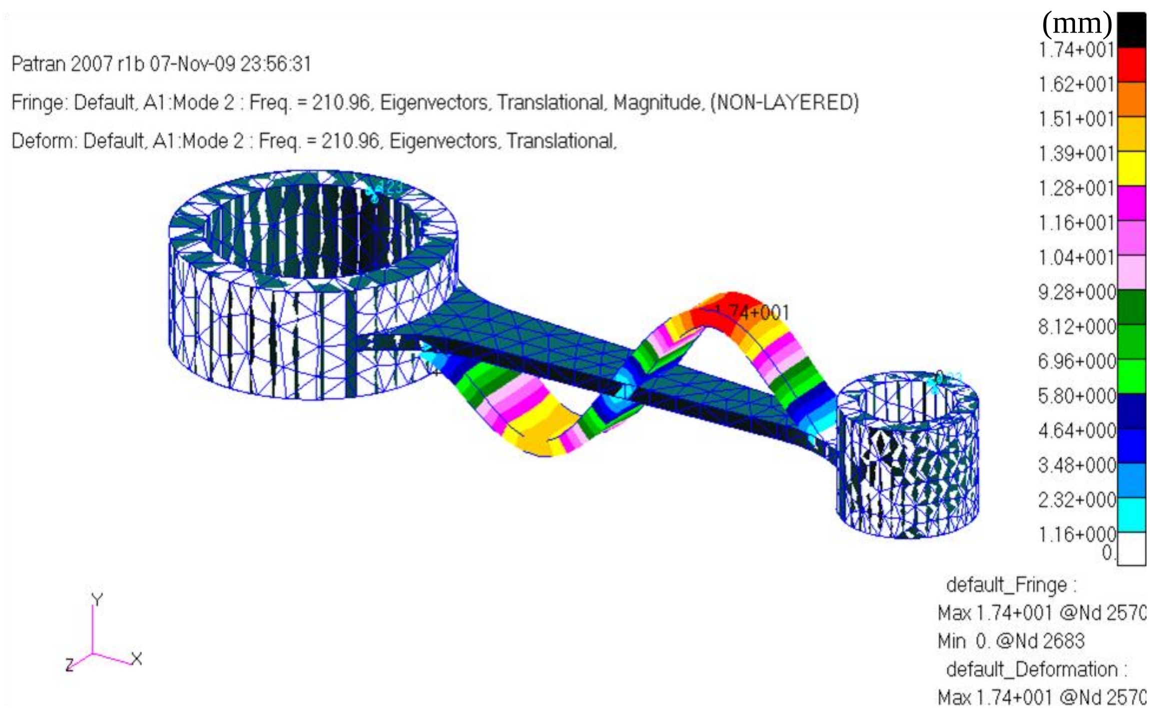


**Figure 4.6:** Natural frequency and mode shape for mode 2

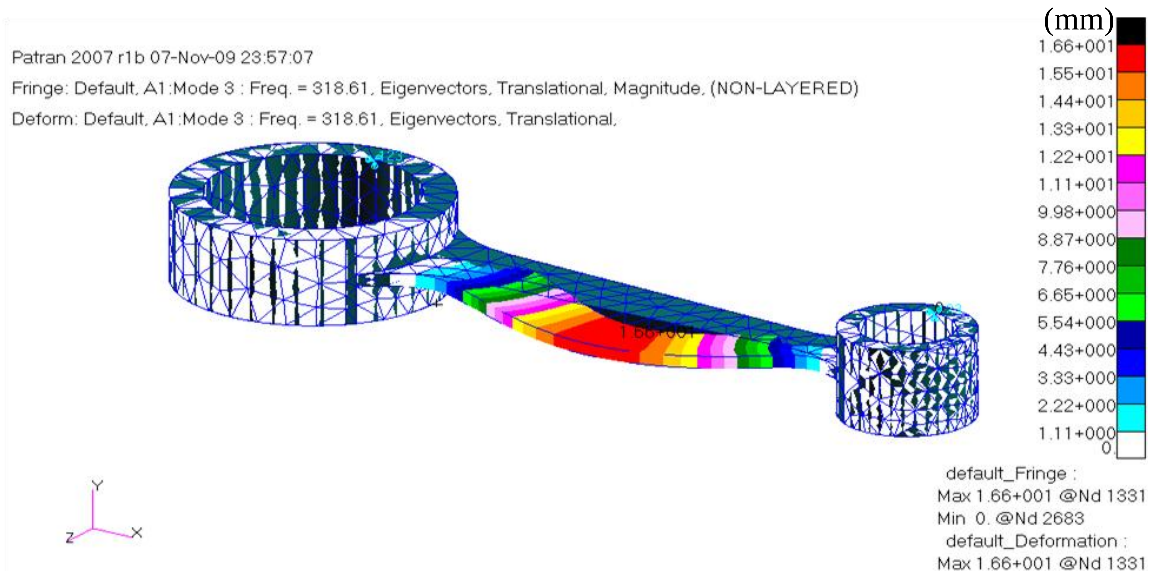


**Figure 4.7:** Natural frequency and mode shape for mode 3

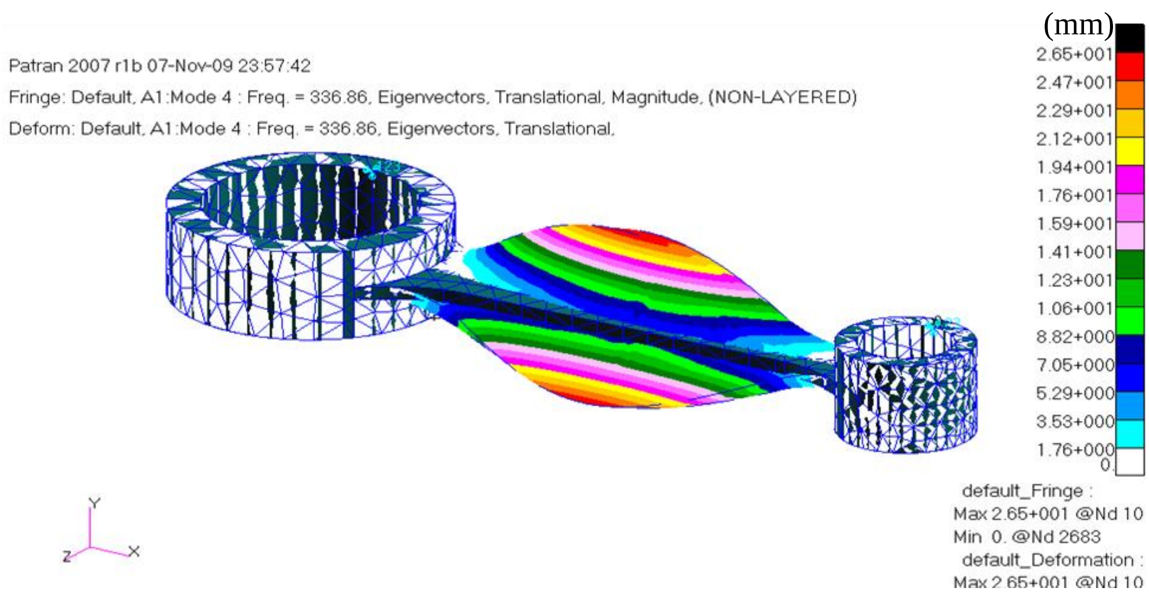


**Figure 4.8:** Natural frequency and mode shape for mode 4

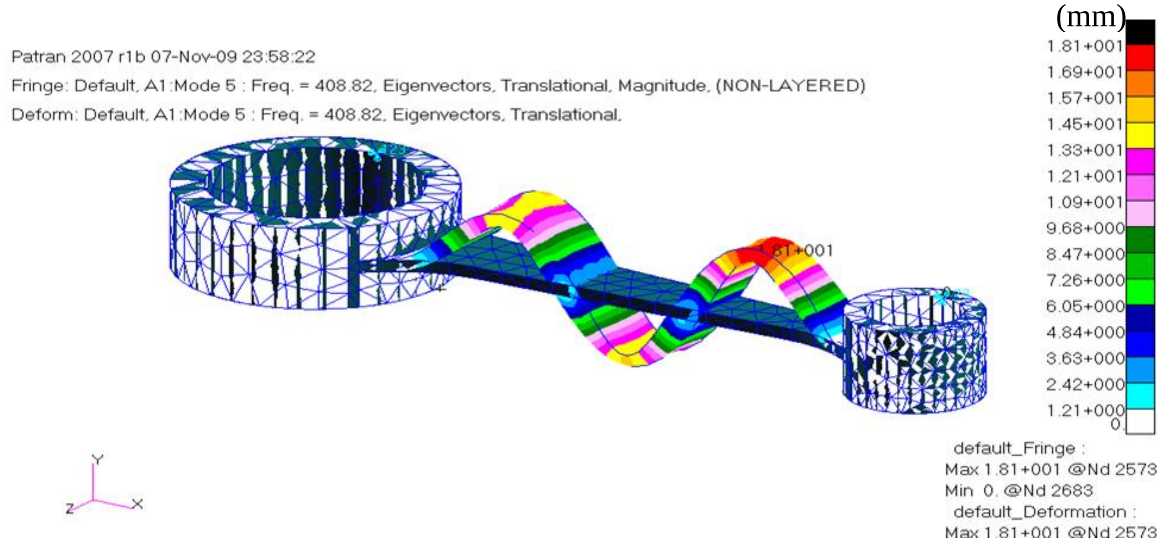


**Figure 4.9:** Natural frequency and mode shape for mode 5

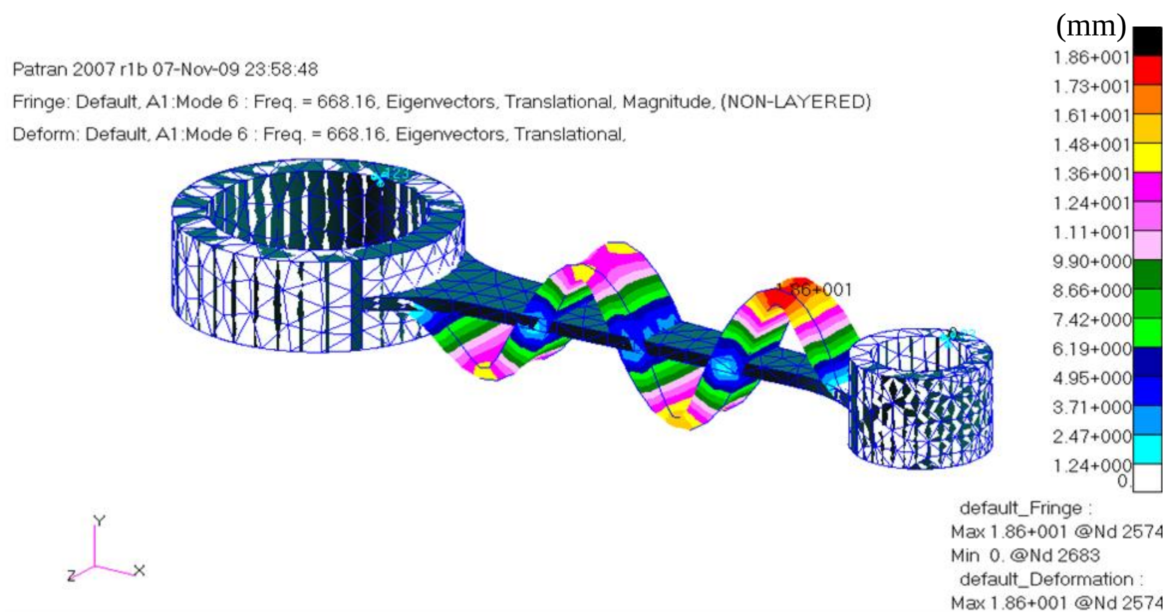


**Figure 4.10:** Natural frequency and mode shape for mode 6

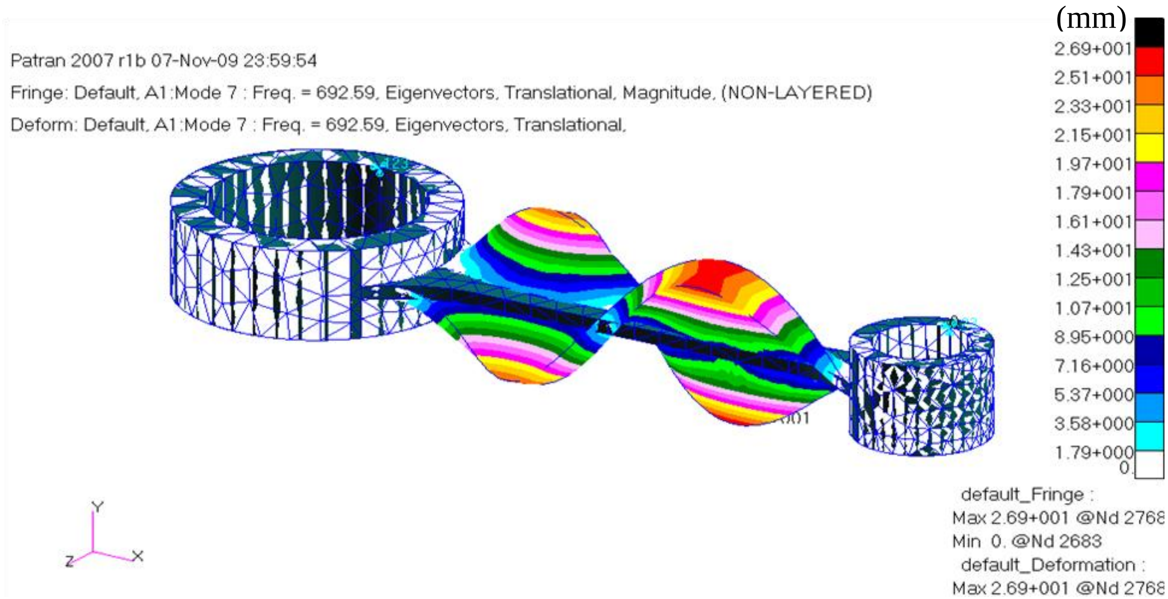


**Figure 4.11:** Natural frequency and mode shape for mode 7

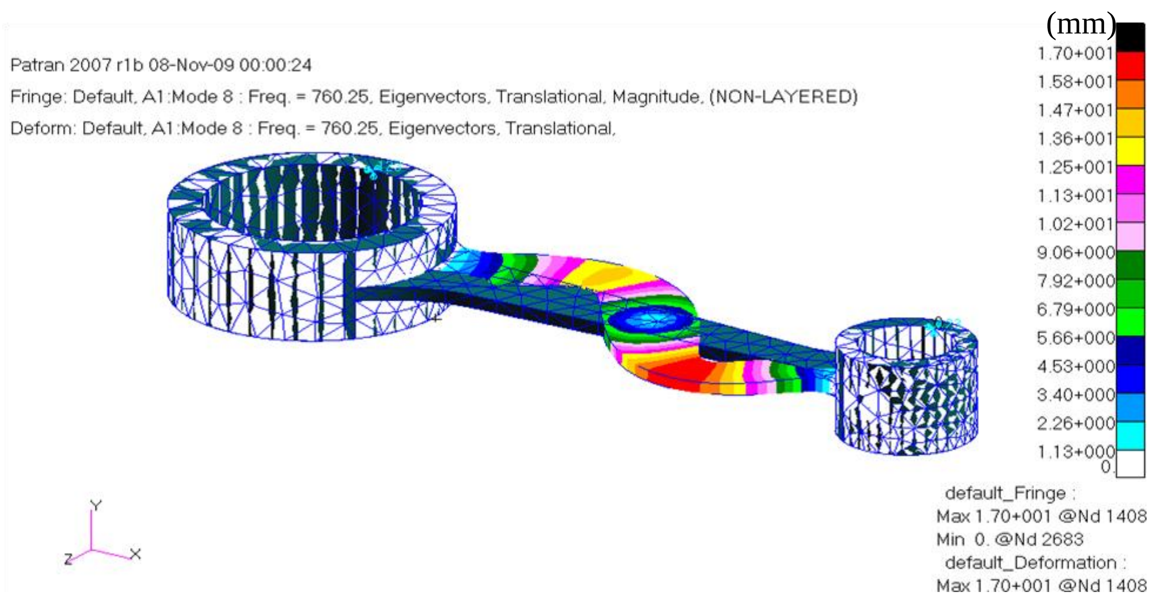


**Figure 4.12:** Natural frequency and mode shape for mode 8

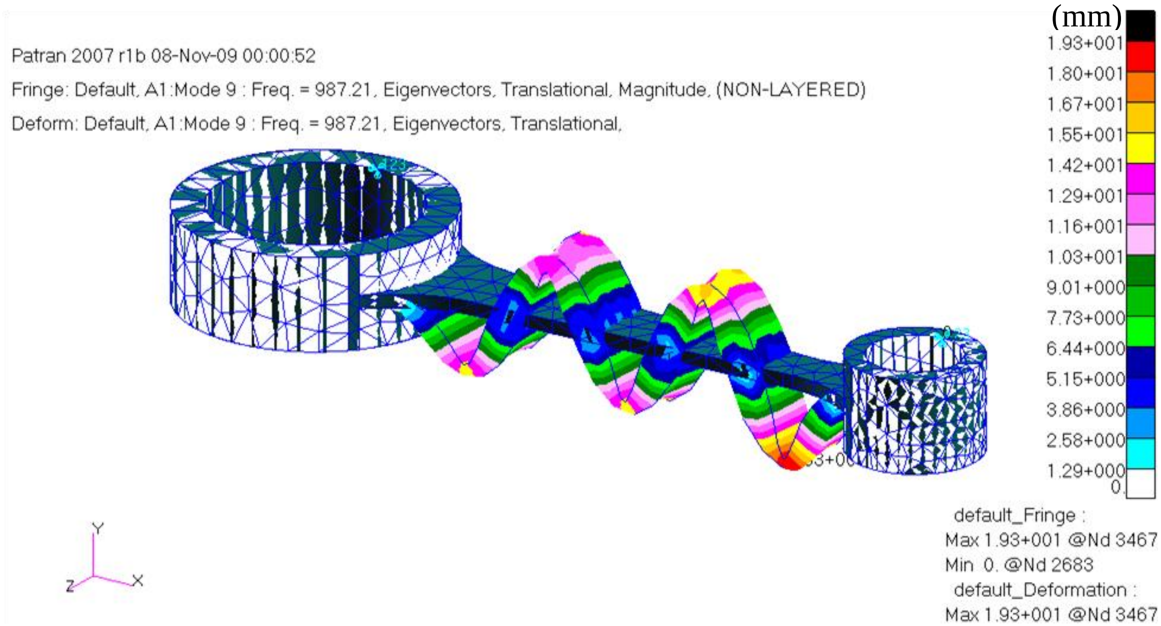


**Figure 4.13:** Natural frequency and mode shape for mode 9

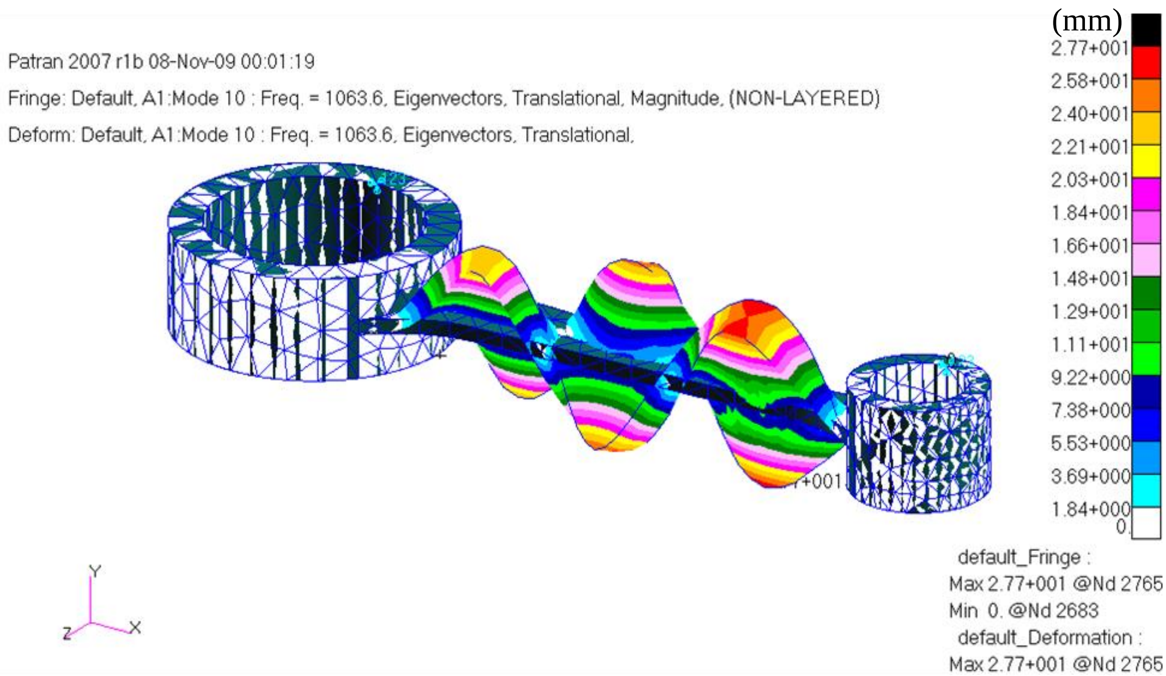


**Figure 4.14:** Natural frequency and mode shape for mode 10

## 4.4 MODAL UPDATING

The modal updating is one of the main techniques to evaluate the dynamic characteristics. After done with modal analysis, the study continued with modal updating. In modal updating the parameter of the material properties was changed in order to determine the effect of the variable parameter to the natural frequency. For the modal updating analysis, two parameter have been chosen that is Modulus of Young's and density of the steel.

The results of modal updating are tabulated in Table 4.3 and 4.4. The effect of the variable parameter to the natural frequency of the finite element modeling of connecting rod are shown in Figure 4.15 and Figure 4.16 for different density and modulus of elasticity respectively. It can be seen from both figure that the frequency increases with decrease of density and increase of modulus of elasticity.

**Table 4.3:** Frequency with variable density of each mode

| **Mode** | **Frequency (Hz) for different density in g/cm³** | | | | | | |
|---|---|---|---|---|---|---|---|
| | **7.15** | **7.45** | **7.75** | **8.05** | **8.35** | **8.65** | **8.95** |
| 1 | 92.138 | 90.467 | 98.895 | 97.41 | 86.007 | 84.678 | 83.415 |
| 2 | 213.84 | 209.29 | 205 | 200.96 | 197.13 | 196.51 | 194.07 |
| 3 | 310.07 | 300.19 | 298.72 | 288.61 | 283.13 | 278.36 | 270.16 |
| 4 | 410.43 | 405.16 | 403.3 | 392.86 | 380.75 | 374.96 | 369.47 |
| 5 | 508.79 | 499.97 | 491.66 | 480.82 | 466.41 | 455.39 | 447.72 |
| 6 | 608.97 | 594.55 | 580.97 | 568.16 | 556.05 | 544.57 | 533.68 |
| 7 | 734.89 | 719.94 | 705.84 | 692.59 | 680.04 | 668.14 | 656.85 |
| 8 | 856.68 | 840.27 | 824.83 | 809.25 | 796.47 | 783.41 | 771.02 |
| 9 | 1007.5 | 986.2 | 966.1 | 947.21 | 929.31 | 912.35 | 896.25 |
| 10 | 1128.5 | 1105.6 | 1084 | 1063.6 | 1044.3 | 1026.1 | 1008.7 |

**Table 4.4:** Frequency with variable Modulus of Young's of each mode

| Mode | Frequency (Hz) for different Modulus of Young's in GPa | | | | | | |
|---|---|---|---|---|---|---|---|
| | **170** | **180** | **190** | **200** | **210** | **220** | **230** |
| 1 | 72.737 | 74.846 | 76.897 | 78.895 | 80.843 | 82.745 | 84.605 |
| 2 | 178.22 | 183.97 | 186.56 | 195 | 200.31 | 205.5 | 210.56 |
| 3 | 279.37 | 288.05 | 296.49 | 304.72 | 312.74 | 320.57 | 328.22 |
| 4 | 366.52 | 375.7 | 384.62 | 393.31 | 401.79 | 410.07 | 418.16 |
| 5 | 454.14 | 465.28 | 476.11 | 486.66 | 496.95 | 510.85 | 521.82 |
| 6 | 547.82 | 561.03 | 573.73 | 585.97 | 597.79 | 614.21 | 630.26 |
| 7 | 650.78 | 669.65 | 688 | 705.87 | 723.3 | 740.32 | 756.96 |
| 8 | 764.36 | 785.07 | 805.21 | 824.83 | 843.96 | 862.65 | 880.91 |
| 9 | 877.61 | 904.5 | 930.66 | 956.1 | 981 | 1005.2 | 1029 |
| 10 | 999.41 | 1028.4 | 1050.6 | 1084 | 1110.8 | 1136.9 | 1162.5 |

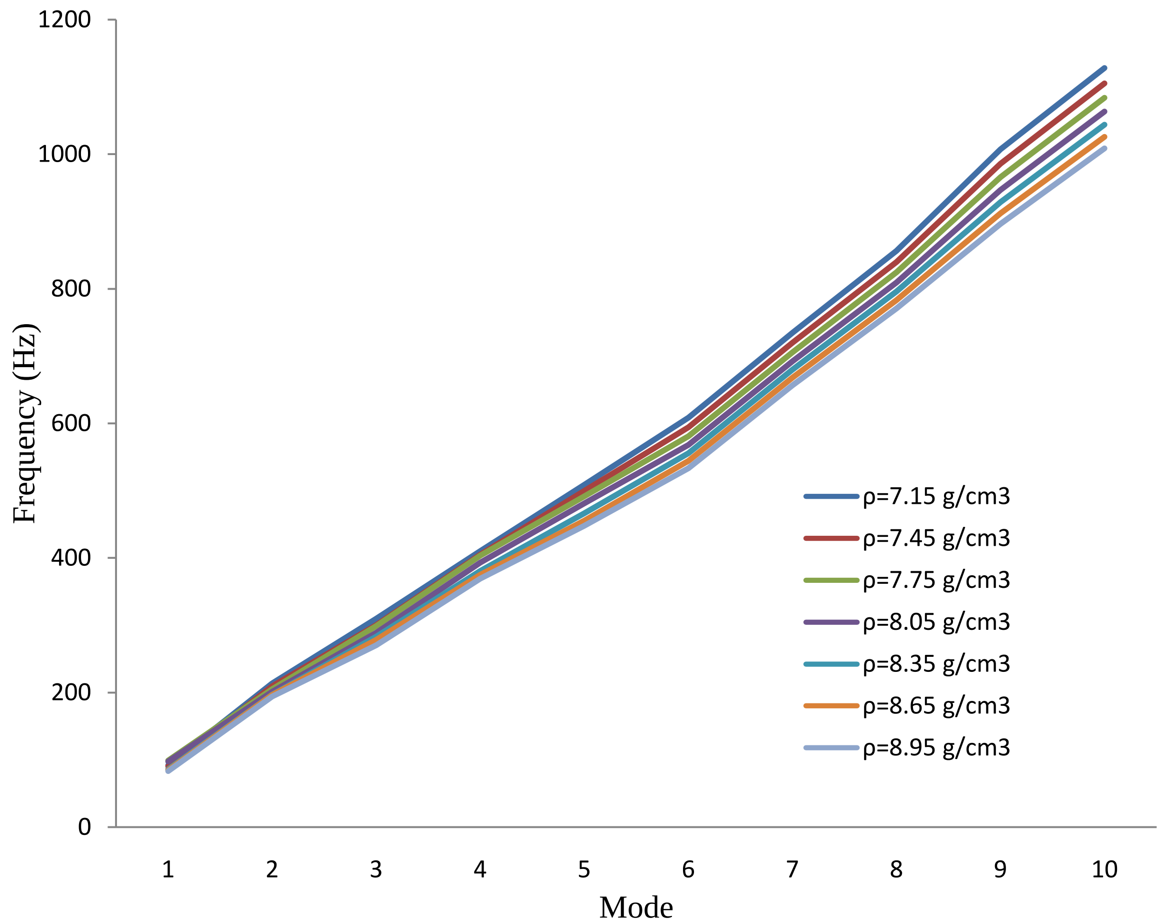


**Figure 4.15:** Frequency with variable density of each mode

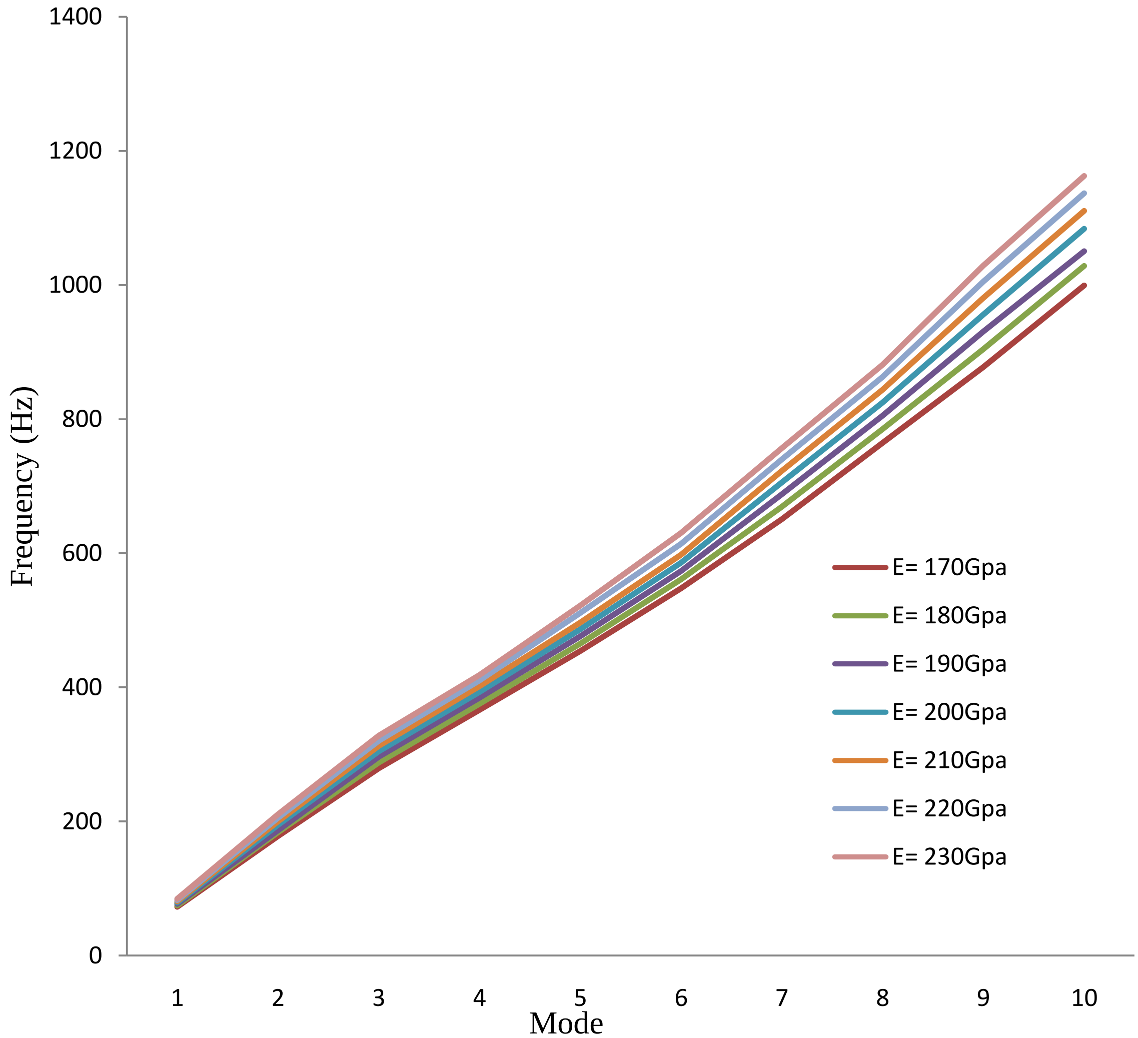


**Figure 4.16:** Frequency with variable Modulus of Young's of each mode

# CHAPTER 5

# CONCLUSION AND RECOMMENDATION

## 5.1 INTRODUCTION

This chapter summarized the conclusion and recommendations for the overall objective of the project based on finite element analysis.

## 5.2 CONCLUSIONS

Finite element modeling has been developed and mesh of tetrahedral with 10 nodes with global edge length of 4 mm has been chosen for the analysis. Modal analysis of connecting rod has been successful run and the results of the natural frequency for first until tenth node have been obtained. Result show that first mode occur at 77.411 Hz.

Material with low density and higher Modulus of elasticity can be use for the improvement of the connecting rod. It is shown in the modal updating process with variable density and Modulus of elasticity that low density and higher Modulus of elasticity will increase the natural frequency of the connecting rod. With the increasing of the natural frequency, connecting rod can be operate in higher working frequency.

## 5.2 RECOMMENDATION

For future study, the experimental modal analysis can be proceed and the result of the experimental modal analysis can be validate and correlate with the computational modal analysis. Computational modal analysis can be used as a guide for the experimental analysis by determining the critical point at the connecting rod for the experiment setup.

**REFERENCES**


Bell A. G. 1997. Modern Engine Tuning. Haynes, 2 edition.

Benaroya, H. 2004. Mechanical Vibration, Analysis, Uncertainties, and Control. Marcel Dekker, 2 edition.

Chen, G. 2001. FE Model Validation for Structural Dynamics. Thesis for the degree of Doctor of Philosophy. University of London: 15

Dascotte, E. 2006. Model Updating For Structural Dynamics : Past, Present and Future Outlook. *Dynamic Design* Solutions, *64,* B-3001 Leuven.

Dukkipati, R. V. 2006. Advanced Mecahanical Vibrations. Alpha Science.

Frey, P. J. & George, P-L. 2008. Mesh Generation. Application To Finite Element. Wiley, 2 edition.

Grass, H., Krempaszky, C. & Werner, E. 2006. 3-D FEM-simulation of hot forming processes for the production of a connecting rod. *Computational Materials Science, 36*, 480–489.

He, J. & Fu, Z.-F. (2001) Modal Analysis. Butteworth Heinemann, 2 edition.

Hoppe, H., DeRose, T., Duchamp, T., McDonald, J., Stuetzle, W. 1993. Mesh optimization. *In: Proceedings of SIGGRAPH*, 19–26.

Kelly, S. G. (2000). Fundamentals Of Mechanical Vibrations. Mc Graw Hill, 2 edition.

Kurowski, P. M. 2004. Finite Element Analysis for Design Engineers. *SAE International. Warrendale, Pa.*

Lauwagie, T., Strobbe, J., Dascotte, Clavier J. and monteogudo, M. 2008. Optimization of the Dynamic Response of a Complete Exhaust System. *Proceeding of ISMA*. 1881-1894.

Lee J. & Kim D. 1995. Experimental Modal Analysis And Vibration Monitoring Of Cutting Tool Support Structure. *Int. J. Mech. Sci. 37,* 1133-1146.

Lin R.M. & Zhu J. 2006. Finite Element Model Updating Using Vibration Test Data Under Base Excitation. *Journal of Sound and Vibration. 303,* 596–613.

MSC Software Corporation, 2006.

Rabb, R. 1995. Fatigue Failure of a Connecting Rod. *Engineering Failure Analysis, 3,* 13-28.

Taubin, G. 1995. A signal processing approach to fair surface design. *In: Proceedings of SIGGRAPH*, 08, 351.