# DYNAMIC CHARACTERISTIC OF CONNECTING ROD FOR FOUR STROKE ENGINE

Adi Firdaus[1], Sazali[1] & Muhammad Arieef[1]

[1]Politeknik Ungku Omar
adi@puo.edu.my
sazali@puo.edu.my
arieef@puo.edu.my

## ABSTRACT

This research 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 research 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 will increase the natural frequency of the connecting rod.

**Keywords** MODAL Analysis, Finite Element Method, mode shape, natural frequencies

## 1. Introduction

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. Therefore, computational methods 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. Furthermore, 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.

## 2. Method

### 2.1. 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 measurements of the connecting rod are accurate in order to obtain the 3D models that are closely fit scale.

Figure 2.1: 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 2.1. The overall dimensions of the connecting rod are shown in Figure 2.3.

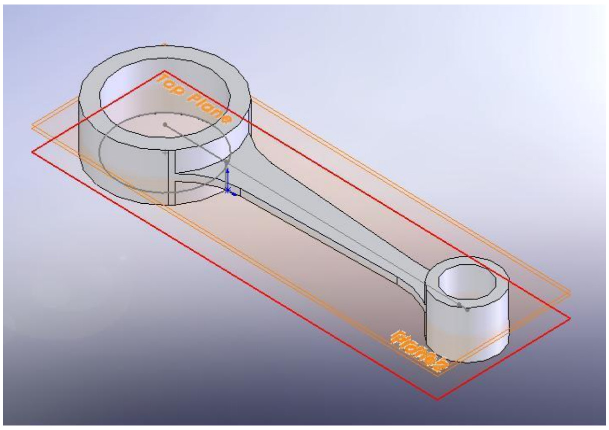

Figure 2.2. Structural model of the connecting rod

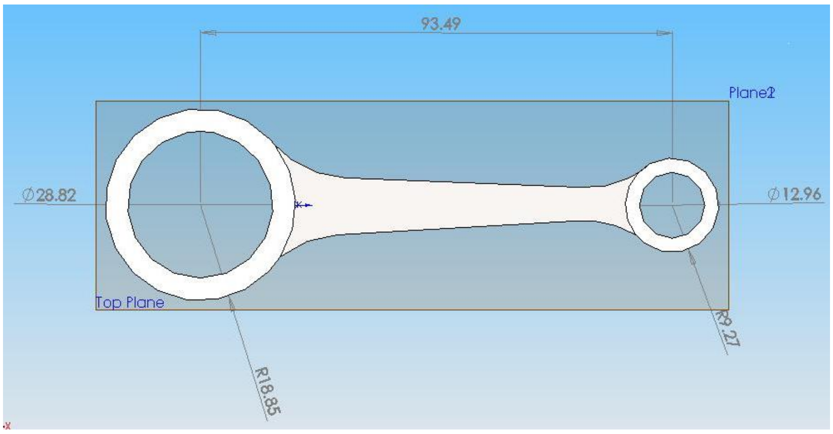


Figure 2.3: Overall dimension of connecting rod

## 2.2 Finite Element Modelling

The finite element is associated 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 edge, nodes located on faces or inside the element while the element vertices may be modes or not.

The smaller the size of the global length, higher accuracy of analysis result. Figure and 2.4 show models meshed with dimension of global edge length. The bigger the global edge length, the coarser the mesh it will have.

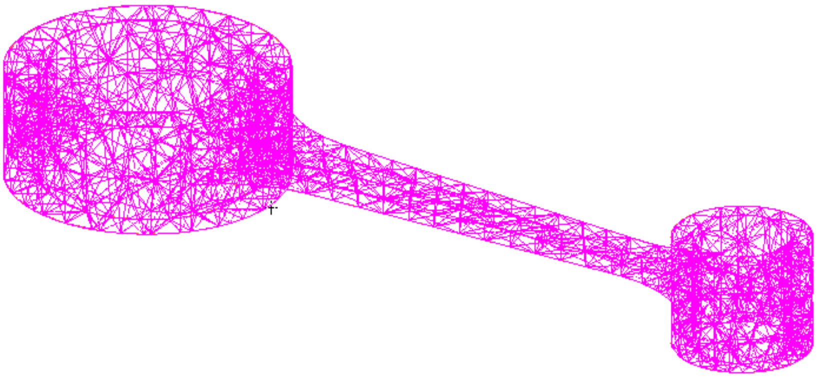

Figure 2.4: Tetrahedral with 10 nodes; Global Edge length = 4 mm

## 2.3 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 2.5 shows constrain of finite element model.

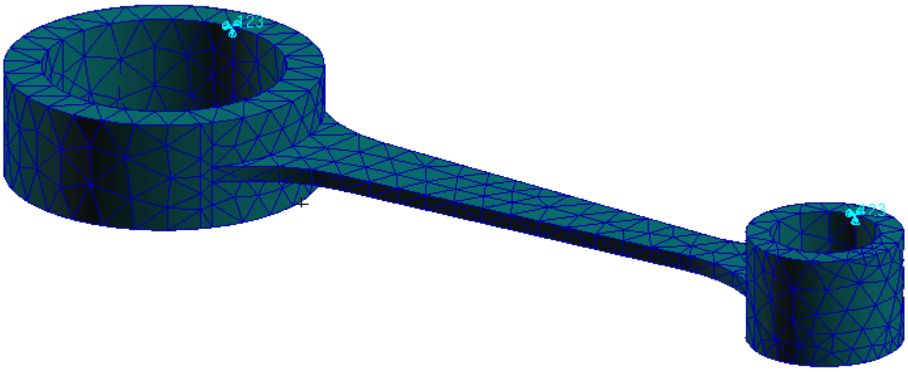

Figure 2.5: Constraints and boundary condition of finite element model

This study used real *eigenvalue* 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 2.1: 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.

## 2.4 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, only one parameter has been chose for the modal updating. The parameters are 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.

## 3. Results & Discussion

### 3.1. Modal Analysis

After determine the mesh that was used for the analyses, next processes 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 3.1 shows the value of the frequency of each modes of lumped and coupled mass.

Table 3.1: Frequency of each mode

| Mode | Frequency (Hz) |
|---|---|
| 1 | 77.411 |
| 2 | 210.96 |
| 3 | 318.61 |
| 4 | 336.86 |
| 5 | 408.82 |
| 6 | 668.16 |
| 7 | 692.59 |
| 8 | 760.25 |
| 9 | 987.21 |
| 10 | 1063.6 |

Analysis results are shown in Figure 3.1. It can be seen natural frequency is linearly increase as the mode number increase and mode shape are presented in Figure 3.2 to 3.11 for different 10 modes.

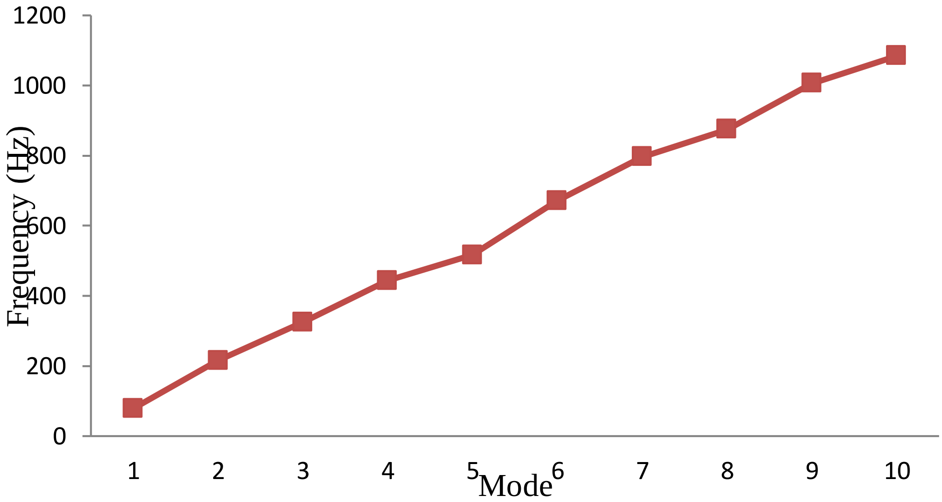


Figure 3.1: Frequency of each mode

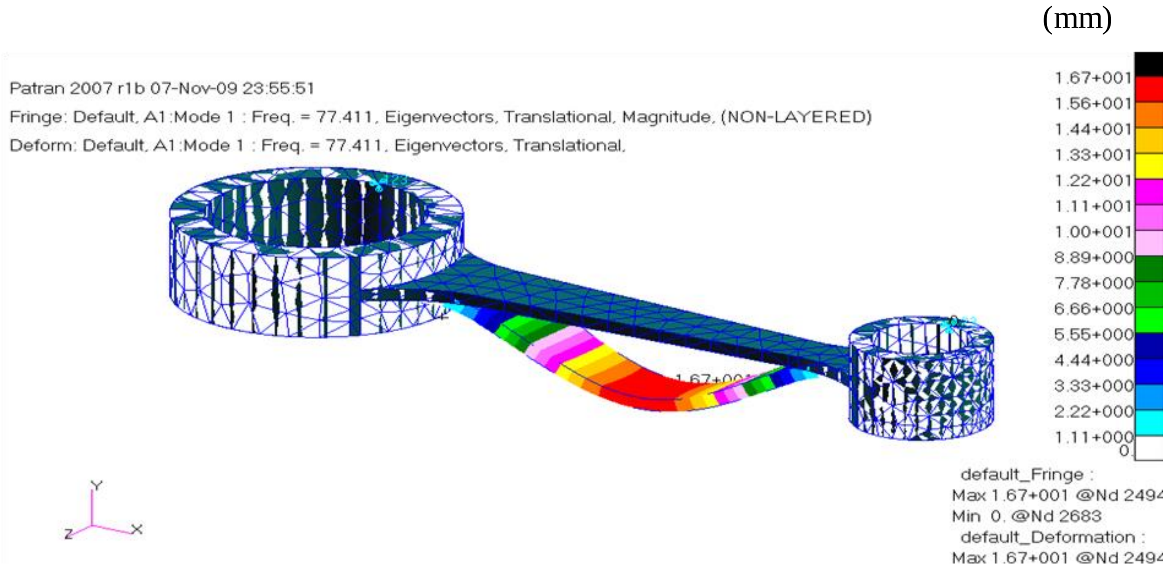


Figure 3.2: Natural frequency and mode shape for mode 1

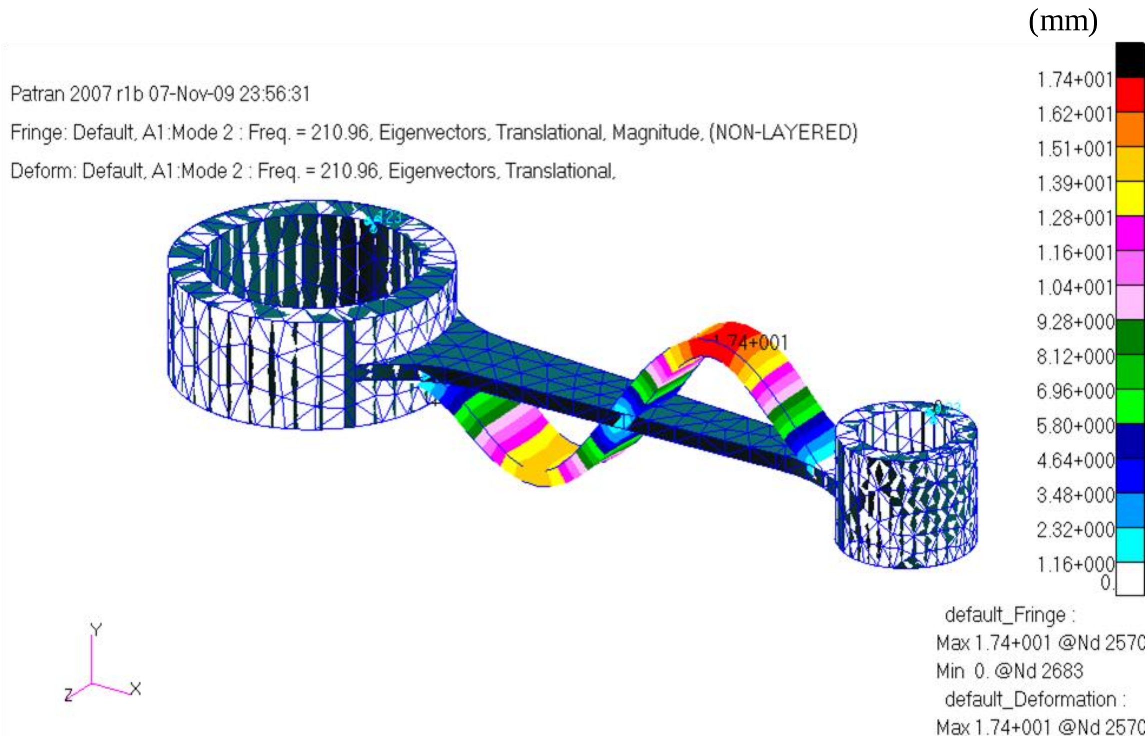


Figure 3.3: Natural frequency and mode shape for mode 2

(mm)

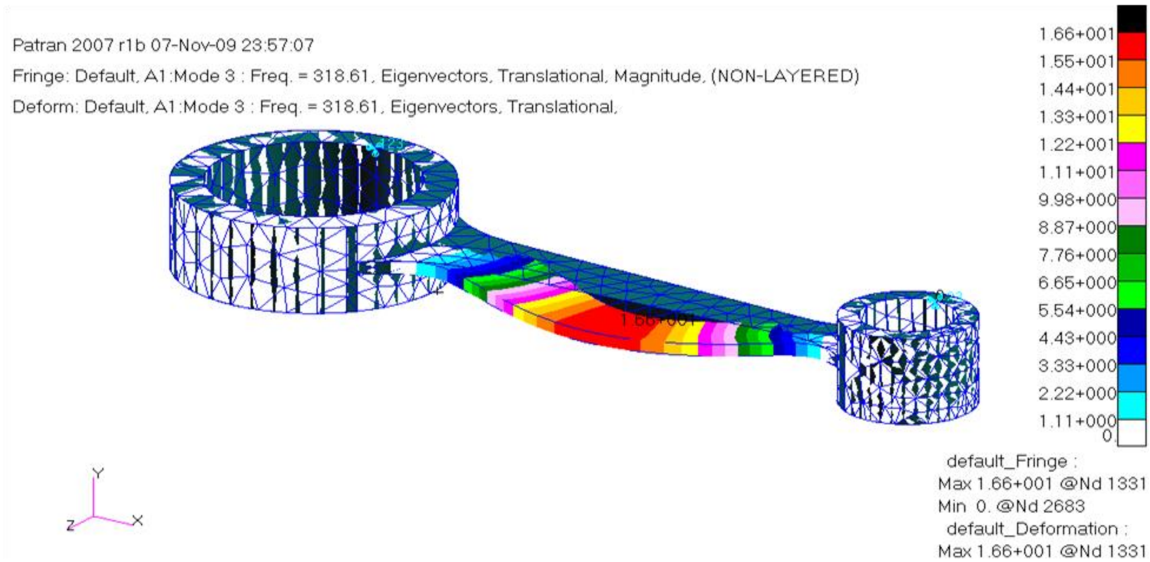


Figure 3.4: Natural frequency and mode shape for mode 3

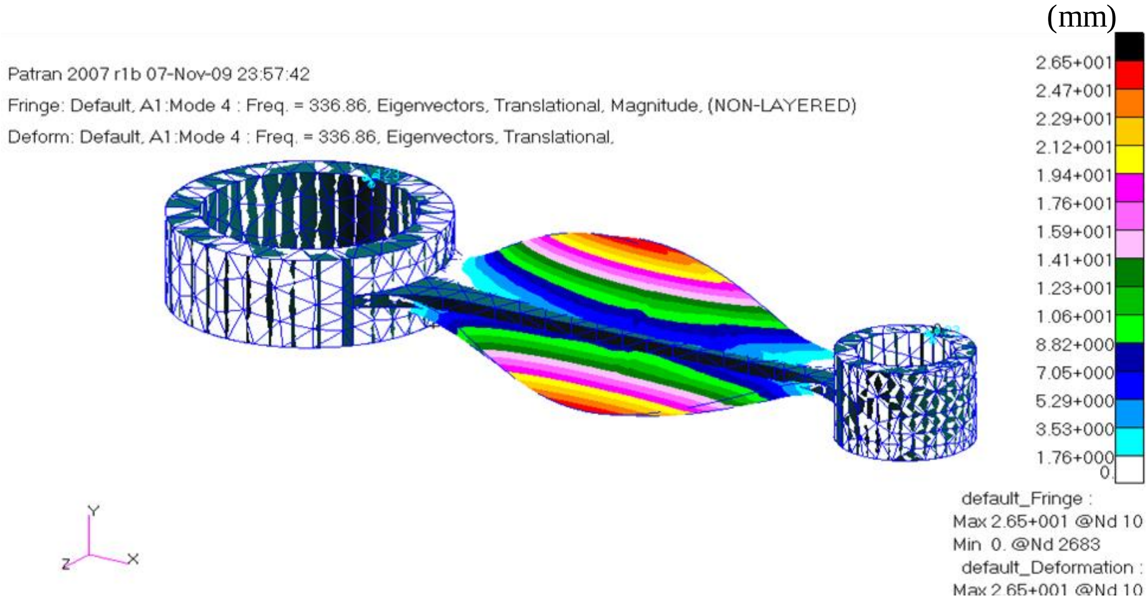


Figure 3.5: Natural frequency and mode shape for mode 4

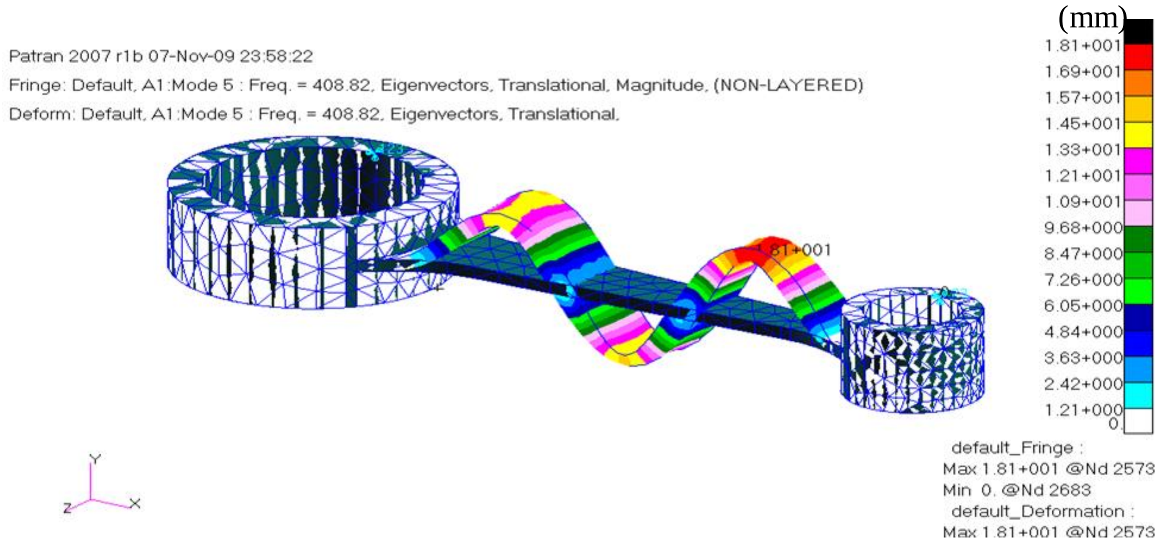


Figure 3.6: Natural frequency and mode shape for mode 5

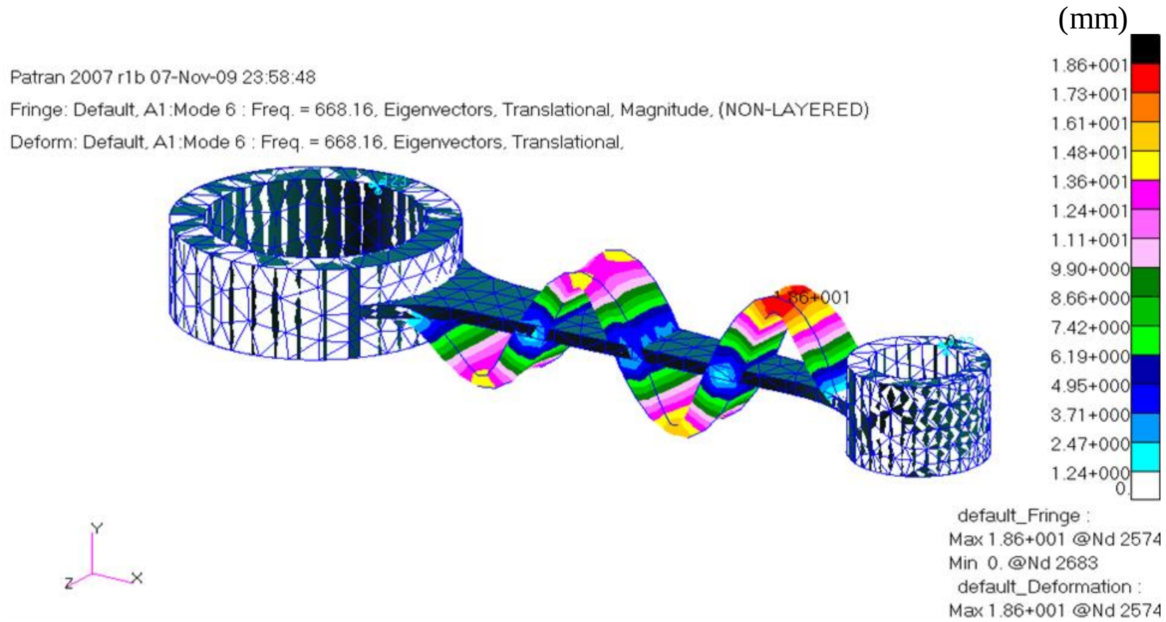


Figure 3.7: Natural frequency and mode shape for mode 6

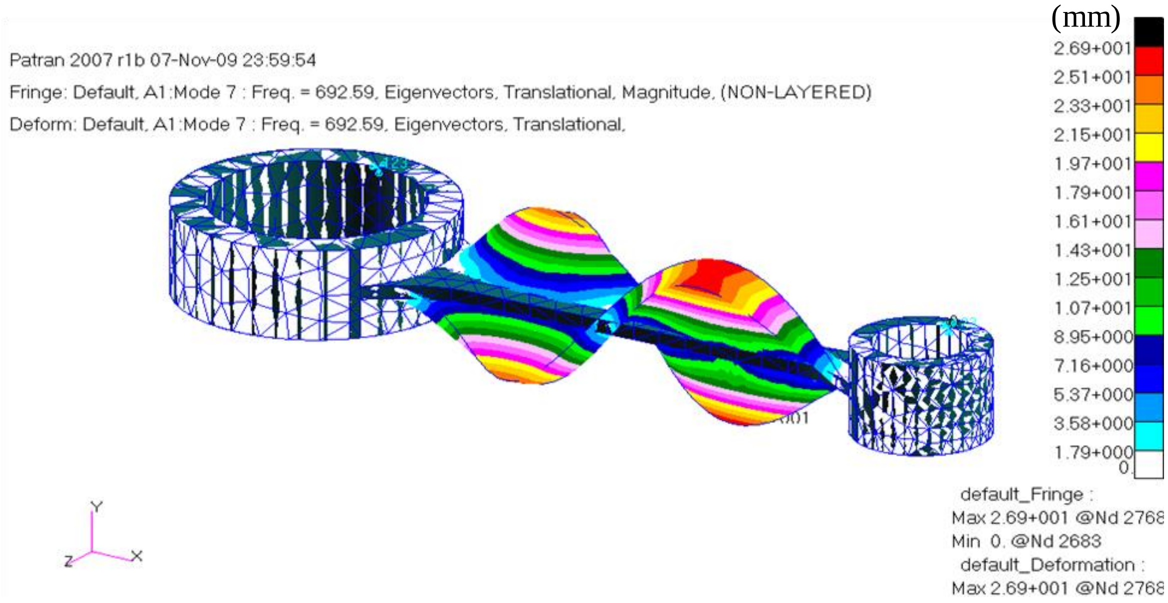


Figure 3.8: Natural frequency and mode shape for mode 7

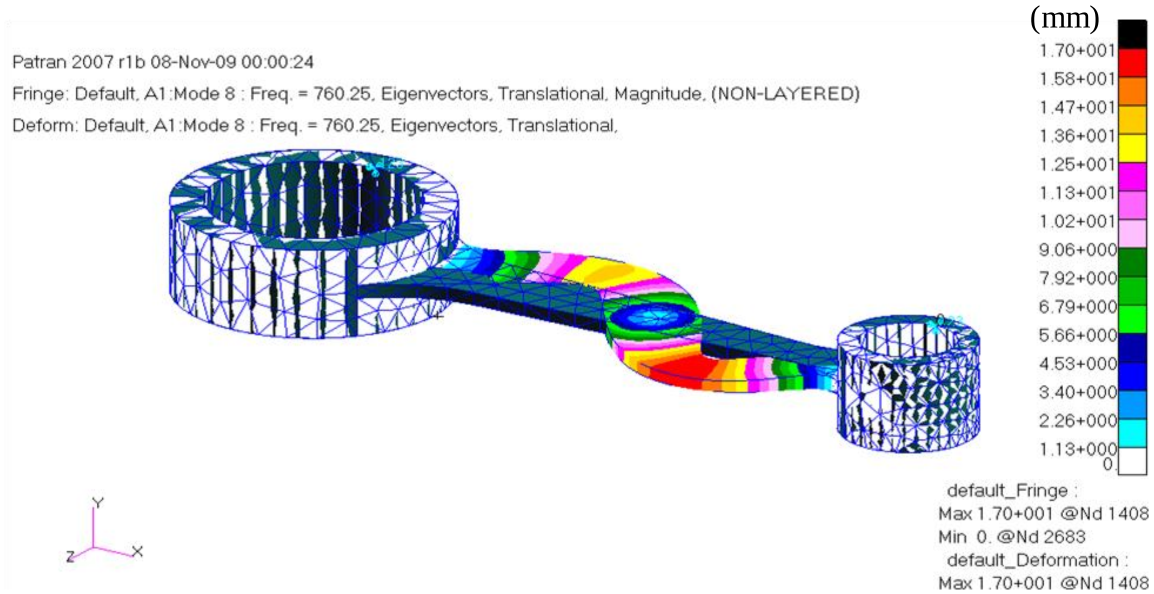


Figure 3.9: Natural frequency and mode shape for mode 8

(mm)

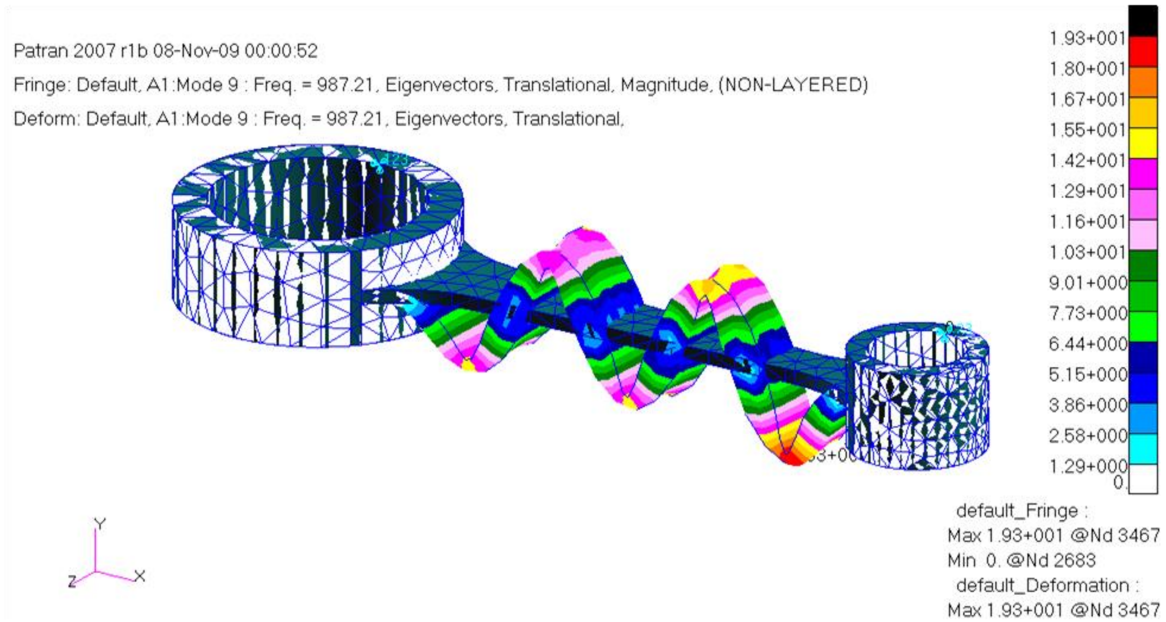


Figure 3.10: Natural frequency and mode shape for mode 9

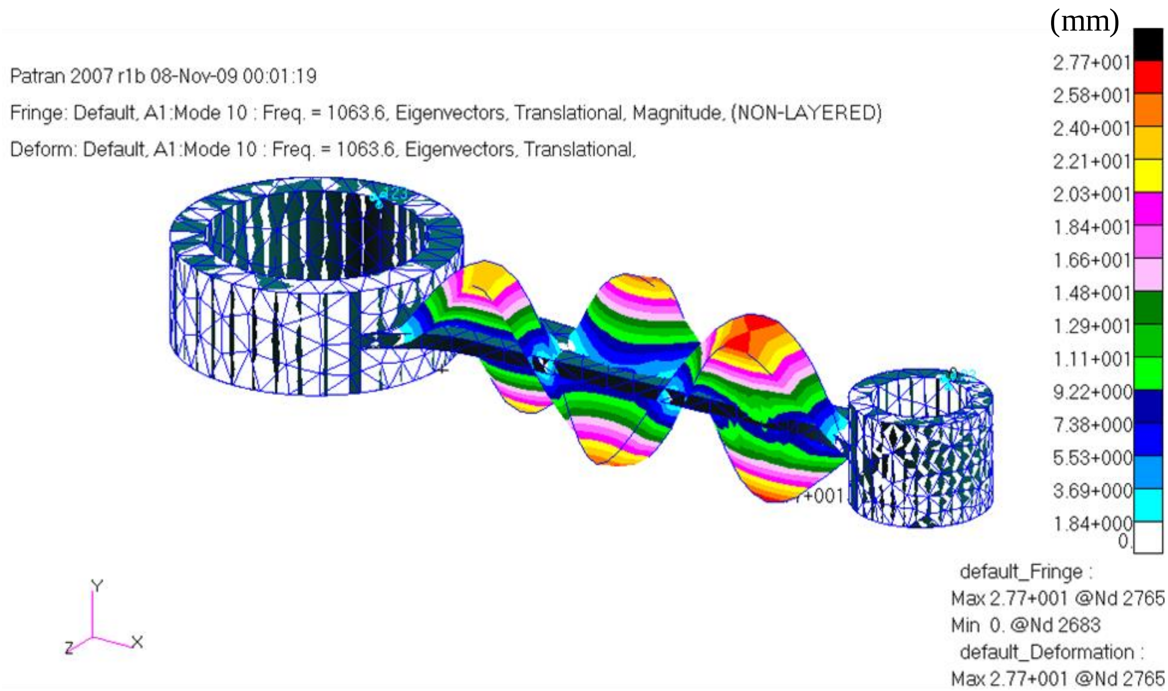


Figure 3.11: Natural frequency and mode shape for mode 10

## 3.2. 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, parameter that has been chosen that is density of the steel.

The results of modal updating are tabulated in Table 3.2 The effect of the variable parameter to the natural frequency of the finite element modeling of connecting rod are shown in Figure 3.12 for different density respectively. It can be seen that the frequency increases with decrease of density

Table 3.2: 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 |

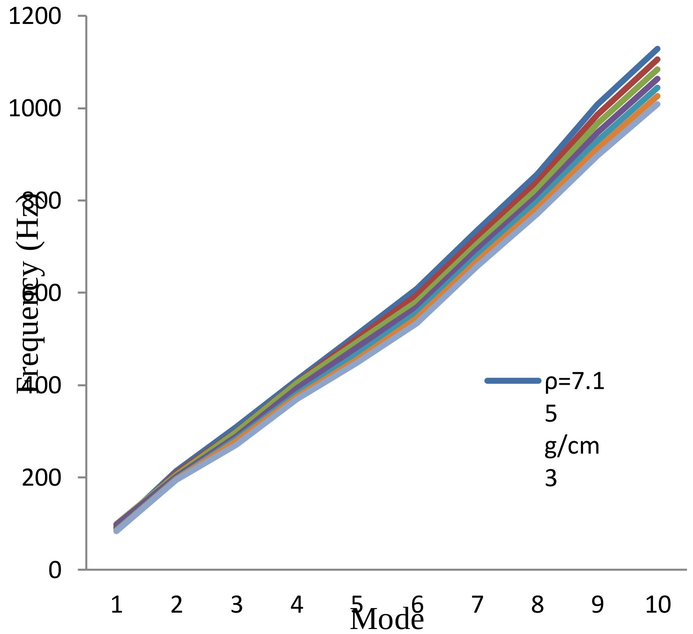


Figure 3.12: Frequency with variable density of each mode

## 4. Conclusion

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 operated in higher working frequency.

.

## References

Bell A. G. 1997. Modern Engine Tuning. Haynes, 2nd Edition.

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

.