DESIGN OPTIMIZATION OF CONNECTING ROD FOR INTERNAL COMBUSTION ENGINE

MOHD SHAMIL BIN SHAARI

Report submitted in partial of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

DECEMBER 2010

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled "Design Optimization of Connecting Rod for Internal Combustion Engine" is written by Mohd Shamil Bin Shaari. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering with Automotive Engineering.

(ENGR. ZAMRI MOHAMED)

Examiner Signature

SUPERVISOR'S DECLARATION

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

Signature	
Name of Supervisor	: DR. MD. MUSTAFIZUR RAHMAN
Position	: ASSOCIATE PROFESSOR
Date	: 6 DECEMBER 2010

STUDENT'S DECLARATION

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

Signature

Name : MOHD SHAMIL BIN SHAARI

ID Number : MH08086

Date : 06 DECEMBER 2010

ACKNOWLEDGEMENT

In the name of Allah, the Most Benevolent, the Most Merciful. Alhamdulillah, all praises to Allah, the Almighty, on whom ultimately we depend for sustenance and guidance. All praises to Allah for the strengths and His blessing in completing this report.

Special appreciation goes to my supervisor, Associate Professor Dr. Md. Mustafizur Rahman, whose guidance, careful reading and constructive comments was valuable. His timely and efficient contribution helped me shape this into its final form and I express my sincerest appreciation for his assistance in any way that I may have asked. His invaluable help of constructive comments and suggestions throughout the report have contributed to the success of this project. My sincere thanks also go to Ir. Zamri Mohamed, I owe special thanks for his consultations especially on this report. Certainly, not forgetting the UMP Faculty of Mechanical Engineering for providing the support and equipment required in order to completing this study.

Sincere thanks to all my friends especially Alliff, Azlan, Raja Shahrul, Omar, Zulfadli, Ahmad Nazmi, Ezuwanizam, ‘Azim, Hishammudin, Azam and others for their kindness and moral support during my study. Thanks for the friendship and memories.

Last but not least, my deepest gratitude goes to my beloved parents; Mr. Shaari Bin Sudin and Mrs. Roshadah Binti Hamid and to my brothers for their endless love, prayers and encouragement. To those who indirectly contributed in this research, your kindness means a lot to me. Thank you very much.

Dedicated to my parents

ABSTRACT

This project describes the study of topology optimization for connecting rod of internal combustion engine. The objectives of this study are to develop structural modelling, finite element analysis and the optimization of the connecting rod for robust design using topology optimization technique. The structure of connecting rod was modelled using SOLIDWORKS software. Finite element modelling and analysis were performed using MSC.PATRAN and MSC.NASTRAN software. Linear static analysis was carried out to obtain the stress/strain state results. The mesh convergence analysis was performed to select the best mesh for the analysis. The topology optimization technique is used to reduce the weight of the connecting rod. From the FEA analysis results, TET10 predicted higher maximum stress than TET4 and maximum principal stress captured the maximum stress. From the topology optimization, the crank end was suggested to be redesign. The optimized connecting rod is 11.7% lighter and predicted lower maximum stress compare to initial design. For future research, the optimization should cover on material optimization to increase the strength of the connecting rod.

ABSTRAK

Projek ini menggambarkan kajian pengoptimuman topologi untuk batang penghubung enjin pembakaran dalaman. Tujuan kajian ini adalah untuk membangunkan model struktur, analisis unsur terhingga dan optimalisasi batang penghubung menggunakan teknik pengoptimuman topologi. Struktur batang penghubung dimodelkan menggunakan perisian SOLIDWORKS. Pemodelan dan analisis unsur terhingga dilakukan dengan menggunakan perisian MSC.PATRAN dan MSC.NASTRAN. Analisis linear statik dilakukan untuk mendapatkan hasil tekanan/regangan. Analisis konvergensi jejaring dilakukan untuk memilih jejaring terbaik untuk analisis. Teknik optimasi topologi digunakan untuk mencapai tujuan dari pengoptimuman bagi mengurangkan berat batang penghubung. Dari hasil analisis FEA, TET10 menganggarkan tekanan maksimum lebih tinggi dari TET4 dan tekanan utama maksimum dapat menangkap bacaan tertinggi. Berdasarkan hasil pengoptimuman topologi, akhir engkol disarankan untuk direka bentuk semula. Batang penghubung dapat dioptimumkan 11.7% lebih ringan dan dijangka tekanan maksimum yang rendah berbanding reka bentuk awal. Untuk kajian akan datang, pengoptimuman harus merangkumi pengoptimalan bahan untuk meningkatkan kekuatan batang penghubung.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

P	Normal pressure
P_o	Normal pressure constant
P_t	Total resultant load (Tensile)
P_c	Total resultant load (Compression)
E	Young’s modulus
$\mathbf{K}$	Stiffness matrix
$\mathbf{u}$	Vector of displacements
$\mathbf{f}$	Vector of applied forces
x	Relative density
ρ	Density
c	Compliance
x	Stiffness
g_e	Design variable

LIST OF ABBREVIATIONS

BC	Boundary Condition
CAD	Computer-aided design
CAE	Computer-aided engineering
CAO	Computer Aided Optimization
DOF	Degree of freedom
FE	Finite element
FEA	Finite Element Analysis
FEM	Finite element modelling
HEXA	Hexahedral
IC	Internal Combustion
PENTA	Pentahedral
RE	Reverse engineering
TET	Tetrahedral

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Connecting rod is a high volume production critical component that being use in automotive internal combustion engine. It connects the piston to rotating crankshaft and transmitting the power of combustion through piston then to the crankshaft. 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 (Grass et al., 2006). However, connecting rods could be produced by casting and the difference between others it usually have blow holes which are adverse from durability and fatigue points of view. The fact that forgings produce blow hole-free and better rods gives them an advantage over cast rods (Gupta, 1993). Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste. However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques (Repgen, 1998).

Reverse engineering (RE) is the process of taking something like a device, a mechanical component or even a software apart and analyzing its workings in detail, usually with the intention to construct a new device or program that does the same thing without actually copying anything from the original (Hashmi, 2005). In order to boost the quality and efficiency of the design, RE should be used. It is not just redesign the product but also to make sure that the new component can achieve more than original product.

1.2 PROBLEM STATEMENTS

Connecting rod is one of the most critical components internal combustion (IC) engines bearing the statically and dynamically fluctuating loads (Hashmi, 2005). The optimization of connecting rod had already started as early year (Webster, 1983). However, everyday consumers are looking for the best from the best. That's why the optimization is really important for automotive industry especially. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less total cost productions. The design and weight of the connecting rod influence on car performance. Hence, it effects on the car manufacture credibility. Change in the design and material results a significant increment in weight and also performance of the engine. The structural factors considered for weight reduction during the optimization include the buckling load factor, stresses under the loads, bending stiffness, and axial stiffness. Thus, the component can give the higher strength, efficient design and lighter that would create a major success in the automotive and manufacturing industry. The benefits of connecting rod optimization are eventually go back to consumer itself. Among the main objectives are to improves the engine performance and also to strengthen the product that is ensure the safety of human being. It undergoes high cyclic loads of the order of 10^8–10^9 cycles, which range from high compressive loads because of combustion, to high tensile loads because of inertia (Shenoy, 2004). Connecting rod failed due to insufficient strength to hold the load. By maximize the strength, automatically it will longer the life cycles of the connecting rod. Lots of knowledge will be apply and produced during the process. In this study, the design of the connecting rod will be improve and at the same time increase the strength. The study will be focus on the finite element modeling and analysis. From the analysis results, the decision whether connecting rod needs to be redesign or not will based on the topology optimization.

1.3 OBJECTIVES OF PROJECT

The objectives of the project are as follows

(i) To develop structural modeling of connecting rod

(ii) To perform finite element analysis of connecting rod

(iii) To develop structural optimization model of connecting rod

1.4 SCOPE OF PROJECT

Before performing the topology optimization, the structural modeling of the connecting rod needs to be developed by using computer-aided design (CAD) software. The structural modeling then imported into the computer-aided engineering (CAE) and began the meshing on the connecting rod. The finite element modeling (FEM) processes were performed by using MSC.PATRAN. The boundary condition (BC) and loading selected and place at the connecting rod. The finite element analysis (FEA) then carried out at the connecting rod. The MSC.NASTRAN used to solve the analysis equation from MSC.PATRAN. Thus, producing the result of stress, strain and displacement where it will be used to analyze the critical area of the connecting rod. Finally the topology optimization will take place and the result will be used to design new connecting rod.

1.5 OUTLINE OF REPORT

Chapter 1 introduces the background, problem statement and the scopes of this study. Chapter 2 presents the literature study about connecting rod, finite element method and optimization of the connecting rod. Chapter 3 discusses the development of structural modeling, finite element modeling and the optimization technique. Chapter 4 discusses the results and analysis of the finite element analysis, modal analysis and optimization of the connecting rod. Chapter 5 presents the conclusion and recommendation of the future work.

CHAPTER 2

LITERITURE REVIEW

2.1 INTRODUCTION

The purpose of this chapter is to provide information which related to the connecting rod, finite element analysis (FEA) and also about optimization of connecting rod.

2.2 CONNECTING ROD

In modern automotive internal combustion engines, the connecting rods are most usually made to absorb high impact stresses that occur onto it. Rasekh et al. (2009) explained about study of experimental equation that was performed for a Tractor MF-285 connecting rod and also using FEA. The maximum stresses in different parts of MF-285 connecting rod were determined. From the analysis, three parts were being considered of the stress distributions which are pin end, rod and crank end. Finally, a comparison between FEA results and experimental equation method were made.

Mirehei et al. (2008) investigated the connecting rod fatigue of universal tractor (U650) was through the ANSYS software application and its lifespan was estimated. The connecting rod behavior affected by fatigue phenomenon due to the cyclic loadings and to consider the results for more savings in time and costs, as two very significant parameters relevant to manufacturing. The results indicate that with fully reverse loading, one can estimate longevity of a connecting rod and also find the critical points that more possibly the crack growth initiate from. It is suggested that the results obtained can be useful for modifications in the process of connecting rod.

Afzal and Fatemi (2004) investigate and compare fatigue behavior of forged steel and powder metal connecting rods. The experiments included strain-controlled specimen testing, with specimens obtained from the connecting rods, as well as load-controlled connecting rod bench testing. Monotonic and cyclic deformation behaviors, as well as strain-controlled fatigue properties of the two materials are evaluated and compared. Experimental S-N curves of the two connecting rods from the bench tests obtained under R = -1.25 constant amplitude loading conditions are also evaluated and compared. Fatigue properties obtained from specimen testing are then used in life predictions of the connecting rods, using the S-N approach. The predicted lives are compared with bench test results and include the effects of stress concentration, surface finish, and mean stress. The stress concentration factors were obtained from FEA was used to account for the mean stress effect. Fractography of the connecting fracture surfaces were also conducted to investigate the failure mechanisms. A discussion of manufacturing cost comparison and recent developments in “crackable” forged steel connecting rods are also included.

2.3 FINITE ELEMENT MODELING AND ANALYSIS

The objective of FEA was to investigate stresses and hotspots experienced by the connecting rod. From the resulting stress contours, the state of stress as well as stress concentration factors can be obtained and consequently used for life predictions (Afzal and Fatemi, 2004).

Rahman (2009) discuss about FEA of the cylinder block of the free piston engine. The 4 nodes tetrahedral (TET4) element version of the cylinder block was used for the initial analysis. The comparison then are made between the TET4 and the 10 nodes tetrahedral (TET10) element mesh while using the same global mesh length for the highest loading conditions (7.0 MPa) in the combustion chamber. From the results, the TET10 mesh predicted higher von Mises stresses than that the TET4 mesh. The TET10 mesh is presumed to represent a more accurate than TET4 meshes. TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function. For the same element size, the TET10 is expected to be able to capture the high stress concentration associated with the bolt holes. A TET10 was then finally used for the solid mesh. Mesh study is performed on FE model to ensure sufficiently fine sizes are employed for accuracy of calculated results but at competitive cost (CPU time).

Bari et al. (2004) compare FEA of slab with others analytical solution. Slabs are most widely used structural elements that transmit load to the supporting walls and beams and sometimes directly to the columns by shear and torsion. Similarly with various classical mathematical procedures, simple beams were analyzed in which the concrete and the steel reinforcement were represented by two-dimensional triangular finite elements. Special bond link elements were used to connect the steel to the concrete. Linear analysis were performed on beams with predefined crack patterns to determine principal stresses in the concrete, stresses in the steel reinforcement and bond stresses. From the observations of the results, displacements and moments by using FEM were 20% more accurate than the analytical results. Accurate results for moments may be obtained if more refined mesh is taken into accounts.

Yang et al. (1992) were meshing finite element modeling. The connecting rod pin end as shown in Figure 2.1 contains 1012 linear HEXA elements, 4 linear PENTA elements, and 1569 grid points. The design goal is to minimize the material volume subject to a constraint on the von Mises stress. This constraint is imposed at each node in the finite element model of the connecting rod head except the nodes at the reentrant corner where the wrist pin leaves the rod. The singularity effects that occur here can be considered by imposing a stress concentration factor, but the interface between the pin end and crank end generally requires complex modeling techniques.

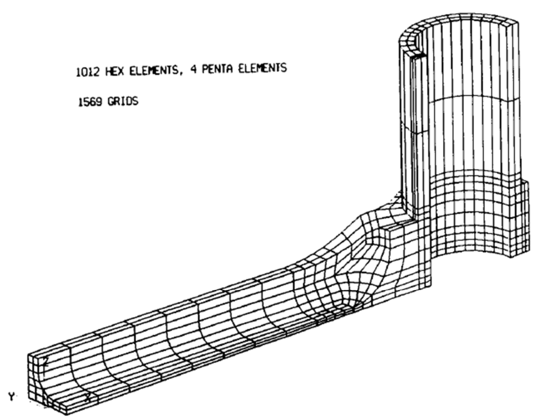


Figure 2.1: Finite element mesh of connecting rod pin end by (Yang et al., 1992).

2.4 OPTIMIZATION OF CONNECTING ROD

A connecting rod is subjected to many millions of repetitive cyclic loadings. So, typically designed for infinite-life and the primary design criterion is endurance limit. Therefore, the durability of this component is of critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance simulation, fatigue and a lot more. It is necessary to investigate finite element modeling techniques, optimization techniques and new design in reducing the weight and at the same time increase the strength of the connecting rod itself. Shenoy (2004) performed a study to explore weight and cost reduction opportunities for a production forged steel connecting rod. The study has dealt with two parts, firstly dynamic load and quasi-dynamic stress analysis of the connecting rod and secondly to optimize the weight and cost. From the results, the existing connecting rod can be replaced with a new connecting rod made of C-70 steel that is 10% lighter and 25% less expensive due to the steel's fracture crackability. Yet, the same performance can be expected in terms of component durability without additional machining of the mating surfaces. The optimize design is show in Figure 2.2.

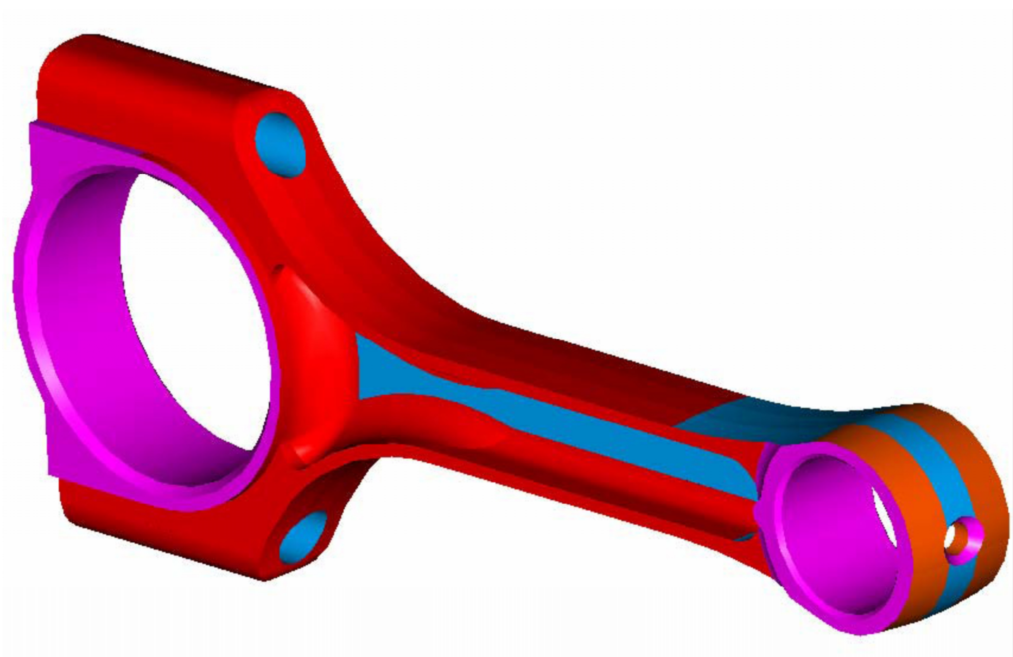


Figure 2.2: Optimized connecting rod by (Shenoy, 2004).

Zoroufi (2004) explained about exploring the design criteria and optimization potentials which is vital to the automotive industry. The study was aimed to developing general procedures for fatigue analysis and optimization of safety critical automotive components with manufacturing considerations. From the results, in terms of structural performance and durability based on both material testing and component evaluation, forged steel was found superior to cast iron which in turn was found superior to cast aluminum. In terms of overall weight and cost reductions of at least 12% and 5%, respectively, are estimated for the example part following the optimization task. The cost of the saved material is additional reduction, though not very considerable due to small portion of material cost within the total production cost.

Ulatowska (2008) explained about shape optimization of a connecting rod for a steel forged connecting rod by using computer aided optimization (CAO). The task purpose was reduction of large notch stresses. In initial model, there are stresses less than 100 MPa and more than 320 MPa, in during the optimization procedure has been try to keep Min and Max von Mises stress between 270 MPa and 300 MPa. When there is stress less than 270 MPa, CAO program remove material in order to increase the stress in this point. When there is more stress than 300 MPa, CAO add material in order to decrease the stress in this point.

Shenoy and Fatemi (2005) explained about optimization study was performed on a steel forged connecting rod with a consideration for improvement in weight and production cost. Weight reduction was achieved by using an iterative procedure. In this study weight optimization is performed under a cyclic load comprising dynamic tensile load and static compressive load as the two extreme loads. The study results in an optimized connecting rod that is 10% lighter and 25% less expensive, as compared to the existing connecting rod.

Yang et al. (1992) describes a successful process for performing component shape optimization should be focused on design modeling issues. A modular software system is described and some of the modules are widely available commercial programs such as PDA/PATRAN and MSC/NASTRAN. The upper end (pin end) of an automotive connecting rod is optimized under a variety of initial assumptions to illustrate the use of the system.

2.5 CONCLUSION

This chapter has been the summary of previous works that related to this project. The works were discussed are about connecting rod, FEA and optimization of connecting rod. The next chapter will be discussed about the methodology of this project.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter presents the overall methodology of the optimization based on finite element analysis. The optimization is the most critical process in the automotive industry. It is very important that any production company invested millions of their profits into Research and Development (R&D). The aim of this chapter is to develop a methodology to improve the process of optimizing a certain product.

3.2 THEORETICAL BASIS OF CONNECTING ROD

Shenoy (2004) performed the optimization on the connecting rod for static FEA loading from Webster et al. (1983) experimental results. Figure 3.1 shows the coordinate system of connecting rod at crank end.

The normal pressure (P) on the contact surface is given by:

$$\mathrm{P} = \mathrm{P_o}\cos\theta \tag{3.1}$$

The load is distributed over an angle of 180$^{\mathrm{o}}$. The total resultant load ($\mathrm{P_t}$) is given by:

$$\mathrm{P_t} = \int_{-\frac{\pi}{2}}^{\frac{\pi}{2}} \mathrm{P_o}\left(\cos\theta\right)^2 rtd\theta = \mathrm{P_o}rt\left(\frac{\pi}{2}\right) \tag{3.2}$$

The normal pressure constant (P_o) is calculated from Eq. (3.2):

$$P_o = P_t / rt\left(\frac{\pi}{2}\right) \tag{3.3}$$

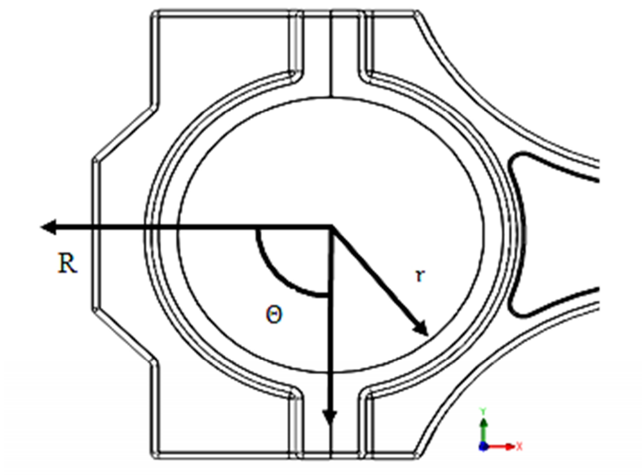


Figure 3.1: Coordinate system of connecting rod.

For compressive loading of the connecting rod, the crank and the piston pin ends are assumed to have a uniformly distributed loading through 180^o contact surface. The normal pressure is given by Eq. (3.4):

$$P = P_o \tag{3.4}$$

The total resultant load (P_c) is expressed in Eq. (3.5):

$$P_c = \int_{-\frac{\pi}{3}}^{\frac{\pi}{3}} P_o(\cos\theta) rt d\theta = P_o rt\sqrt{3} \tag{3.5}$$

The normal pressure constant (P_o) is then given by Eq. (3.6):

$$P_o = P_c / rt\sqrt{3} \tag{3.6}$$

In this study four Finite Element (FE) models were analyzed. FEA for both tensile and compressive loads were conducted. Two cases were analyzed for each case, Firstly, load applied at the crank end and restrained at the piston pin end, and secondly, load applied at the piston pin end and restrained at the crank end and the axial load was 26.7 kN in both tension and compression (Shenoy, 2004).

3.3 PROJECT FLOWCHART

An optimization model is required for an optimization in the same way as an analysis model is required for a finite element analysis. For topology optimization this also covers the statement of allowable topology changes. To perform the optimization the structural model of connecting rod need to be design as an initial design. Then finite element modeling of the connecting rod which consist of material, mesh types, sizes and boundary condition will be choose. The FEA will analyze the model and from the result of stress, strain and displacement, the model need to be validated to make sure the analysis done is correct. Flowchart of optimization based finite element analysis is shown in Figure 3.2.

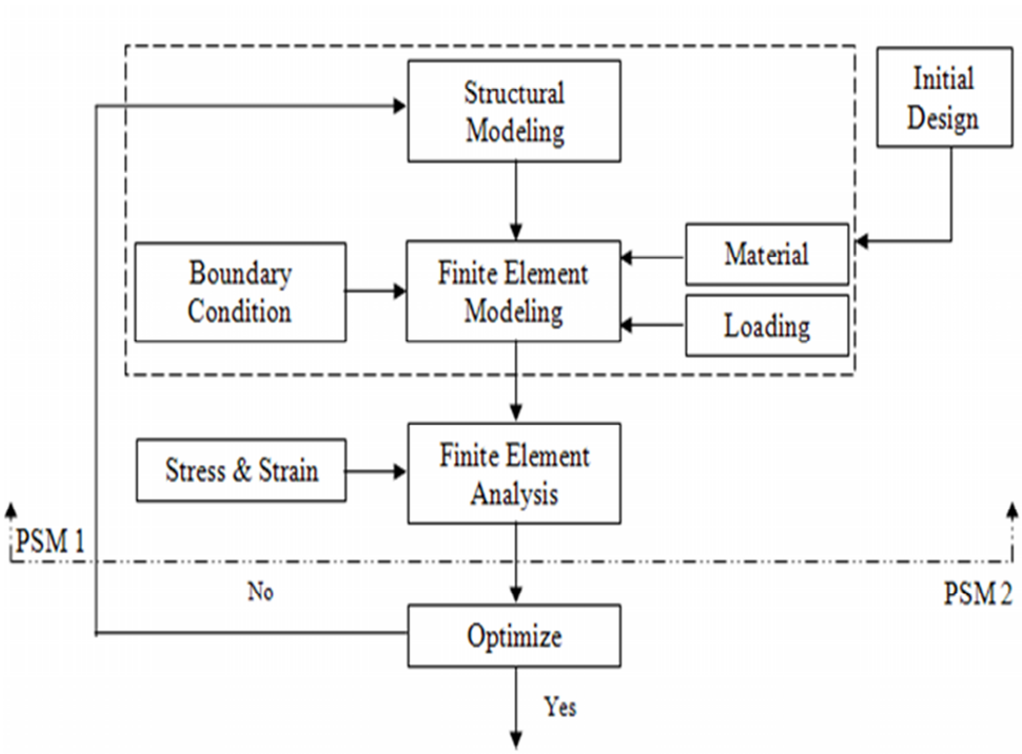


Figure 3.2: Optimization based on finite element analysis.

3.4 MODELING OF THE CONNECTING ROD

The connecting rod is one of the most important components in the internal combustion engine. Any unnecessary parts or shape will be removed during modeling the optimized design. A simple three-dimensional model of connecting rod with mass of 0.577 kg was developed using SOLIDWORKS as shown in Figure 3.3. The dimension of the connecting rod as in Figure 3.4 inspired from Shenoy (2004).

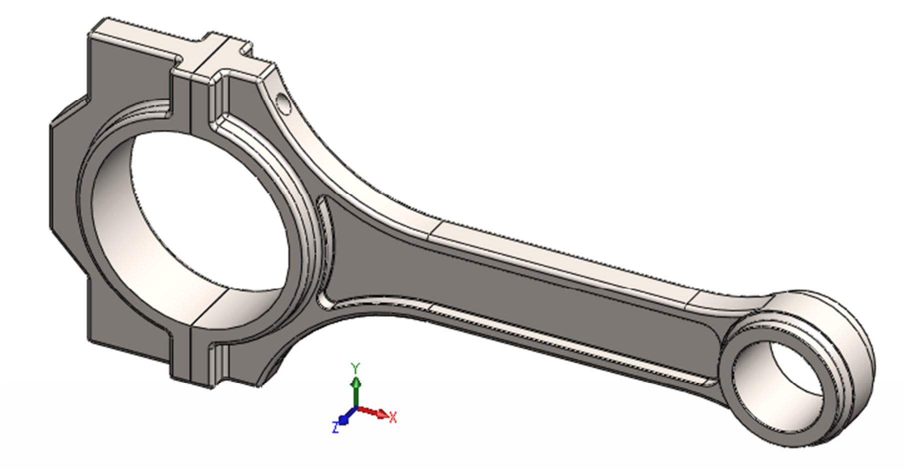


Figure 3.3: Isometric view of 3D modeling of connecting rod

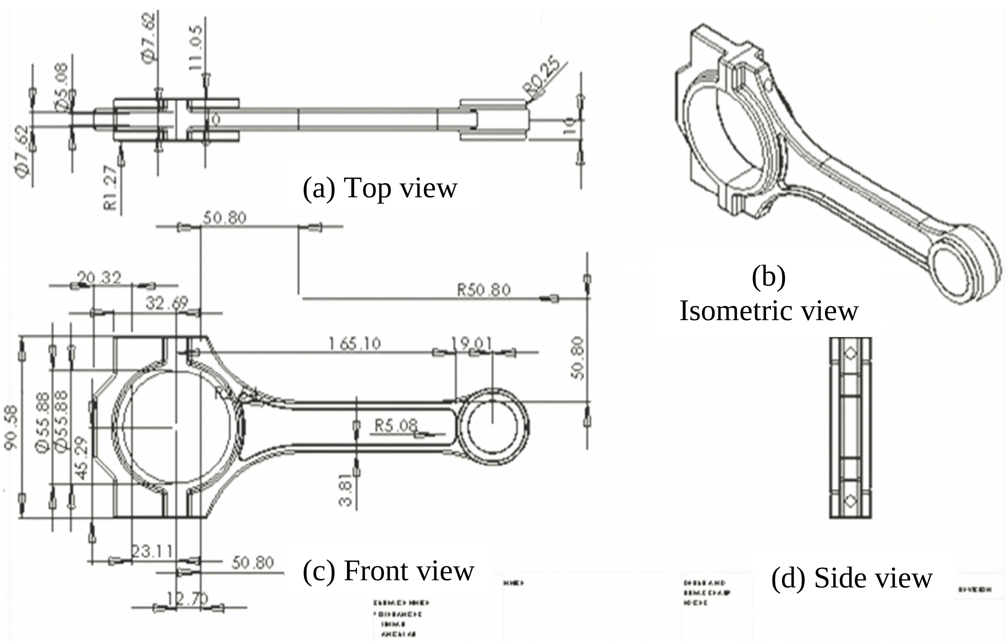


Figure 3.4: 3D modeling of initial connecting rod.

3.5 LINEAR STATIC ANALYSIS

The FEA can be used to calculate the stress distribution for an entire component or structure. The linear static analysis is considered in this project. Amin (2008) was stated that the linear static analysis is one of the most common engineering analyses and represents the most basic of analysis. The term linear means that the displacement or stress is proportion to the applied force. The term static means that the forces do not vary with time. Figure 3.5 shows the relationship between load and displacement.

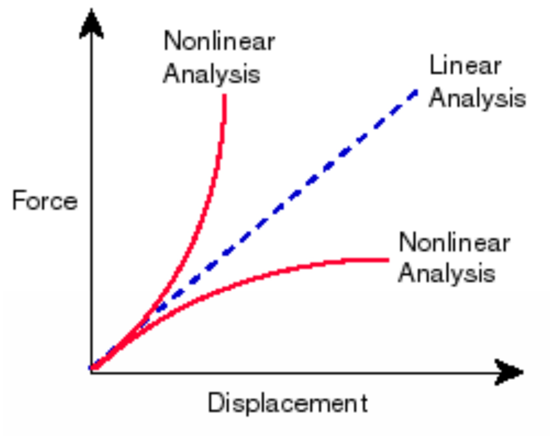


Figure 3.5: Relationship between load and displacement

Rahman (2007) was stated that, due to the load does not change according to the time, the equation can be written in simple equation which neglecting the inertia forces, damping forces and nonlinearity. Outputs or results of the linear static analysis are displacements, strains, stresses, and reaction forces under the effects of applied loads. Figure 3.6 shows the flowchart to perform the linear static analysis.

Static analyses derive in Eq. (3.7):

$$[K]\{u\} = \{f\} \tag{3.7}$$

Where,

$[K]$ = Stiffness matrix (based on the geometry and properties)

$\{u\}$ = Vector of displacements

$\{f\}$ = Vector of applied forces

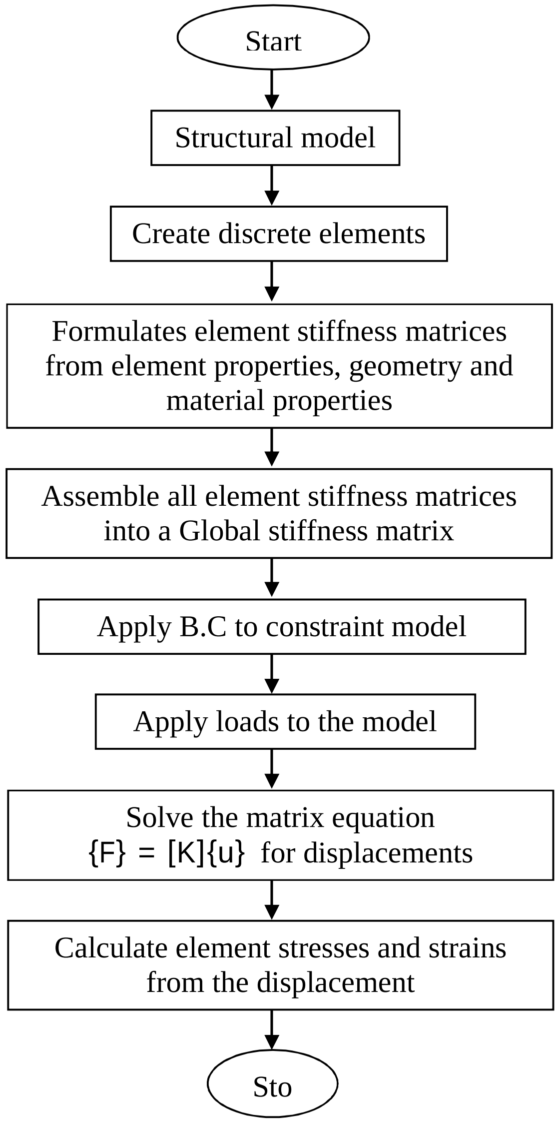


Figure 3.6: Flowchart to perform the linear static analysis by (Rahman, 2007).

3.6 OPTIMIZATION TECHNIQUE

Figure 3.7 shows the flowchart of optimization approach. The optimization carried out is typically an optimum solution for the minimum or maximum possible value of the objective function which defined as set of constraints. The weight of the optimized connecting rod is definitely lower than the initial connecting rod. But this may not be the minimum possible weight under the set of constraints which already defined. However the optimization which done here is an effort to reduce the weight and increase the strength. Rather than using numerical optimization techniques for weight reduction, judgment has been used. The quantitative results were examined qualitatively, and the structure modified. Since this optimization task was performed manually, considering manufacturing feasibility and cost, it cannot be guaranteed that the weight of the 'optimized part' is the minimum weight.

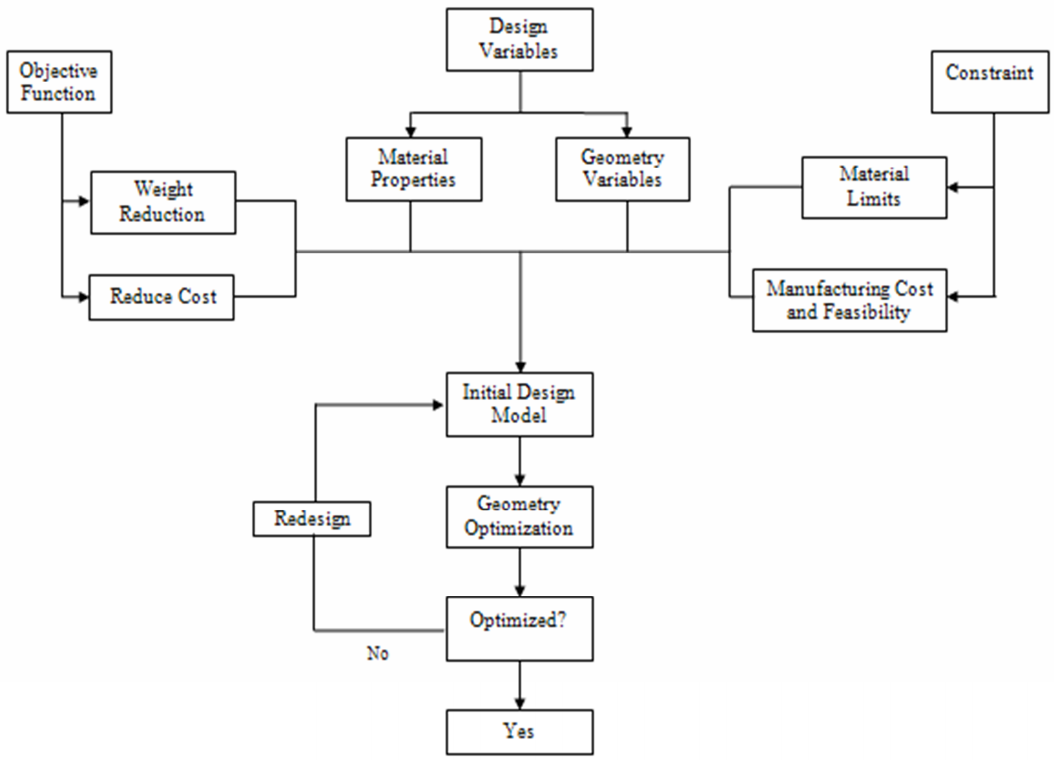


Figure 3.7: Flowchart of optimization approach

Objective of the optimization was to minimize the mass of the connecting rod under tensile and compressive load at crank end and pin end. The shape of the connecting rod will be redesign and automatically, the production cost of the connecting rod will also minimize. Furthermore, the factor of safety is considered based on Shenoy (2004) assumption, which is 3 to 6.

The optimization statement would appear as follows:

(i) Objective: Minimize mass and cost.
(ii) Design Variable: Shape and dimension of connecting rod.

(iii) Subject to:

a) Tensile load = Crank and Pin End

b) Compressive Load = Crank and pin end

(iv) Constraint ≤ Allowable stress.

The load range under the connecting rod was optimized is comprised of the tensile load and compressive load of 26.7 kN, as suggested from Shenoy (2004). The compressive load of 26.7 kN is independent of the geometry of the connecting rod.

3.6.1 The Topology Optimization Formulation

Niemann (2008) using the approach of topology optimization which used in MSC.NASTRAN and based on the power-law approach as in Figure 3.8. Each element in FE model is given an additional property of relative density (x) and density (ρ) in Eq. (3.8) which alters the stiffness properties of the elements.

$$x_e = \rho_e / \rho_o \tag{3.8}$$

The density fraction is expressed as Eq. (3.9):

$$E_e = x_e^p E_o \tag{3.9}$$

where, E is the young's modulus.

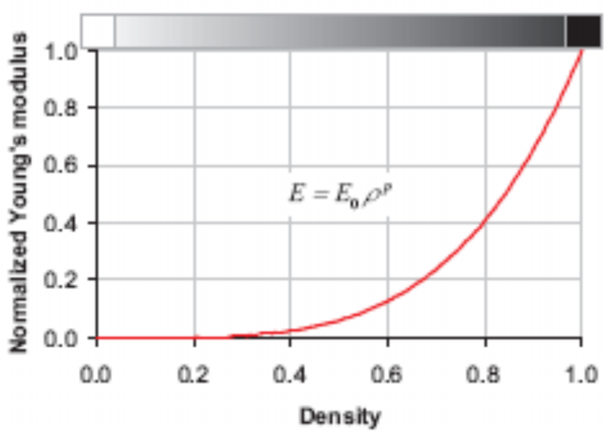


Figure 3.8: Power-law approach where x is declared as ρ by (Niemann, 2008).

The objective of most common topology optimization problems is to find the minimum compliance (*c*) of a structure by a change in the distribution of mass or, in a fixed geometry (volume), the distribution of densities. The objective function can therefore be defined as Eq. (3.10).

$$c\,(\mathbf{x}) = \mathbf{f}^{\mathrm{T}}\mathbf{u} \tag{3.10}$$

where,

f = Vector of applied force

c = Compliance

u = Vector of displacement

This compliance is the scalar product of the two vectors and resembles the work done by the force vector along the calculated displacements. Thus the given expression is actually a potential similar to common formulations for energy equilibrium. Thereby the force vector (**f**) is equal to the resulting displacements of the finite elements analysis multiplied by the stiffness matrix, (**K**) with the current density distribution derive in Eq. (3.11).

$$\mathbf{K}\,(\mathbf{x})\,\mathbf{u} = \mathbf{f} \tag{3.11}$$

where,

K = Stiffness matrix

x = Stiffness

With some further transformation the objective function can be written as Eq. (3.12). The compliance here is a linear combination of the compliances of each element.

Min compliance:

$$c\,(\boldsymbol{x}) = \sum_{\mathrm{e}=1}^{\mathrm{N}} \mathbf{u_e}^{\mathrm{T}}\mathbf{k_e}\,(\mathrm{x_e})\mathbf{u_e} \tag{3.12}$$

Since it is a normalized value, the design variable, (g_e) can only range between the values 0 (void) and 1 (solid) and therefore has to be restricted. For prevention of possible singularities in the system's matrices the densities are not restricted by zero but by a lower bound Eq. (3.13).

$$g_e^{low} = x_{min} - x_e \leq 0$$
$$g_e^{up} = x_e - 1 \leq 0 \tag{3.13}$$

where, design variable is (g_e).

Since this optimization method is basically a redistribution of material, the mass has to be constrained Eq. (3.14).

$$h(\boldsymbol{x}) = \sum_{e=1}^{N} V_e \rho_e - M_o = 0 \leftrightarrow \frac{V(\mathbf{x})}{V_o} = \text{const} \tag{3.14}$$

The complete topology optimization problem statement for minimizing compliance therefore reads as Eq. (3.15).

$$\frac{min}{x \in \mathbf{R}^n} \{c(\mathbf{x}) | h(\mathbf{x}) = 0, \mathbf{g}^{low} \leq 0, \mathbf{g}^{up} \leq 0\} \tag{3.15}$$

Although there are many optimization problems that can be solved with MSC.NASTRAN, this problem statement has been customized in MSC.PATRAN and can be easily used on a given geometry.

3.7 CONCLUSION

This chapter discussed about the method of finite element analysis and the implementation of boundary condition. The theoretical basis of connecting rod has been presented. The technique of optimization also discussed along with the optimization objectives and constraint. It is an importance major input in the optimization process. The next chapter will be discuss about the result of finite element based on optimization of the connecting rod.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 INTRODUCTION

This chapter presents the details of FE modeling, selection of the mesh type and the influence of mesh type, identification of mesh convergence, linear static stress analysis, modal analysis and optimization of the connecting rod.

4.2 MATERIAL INFORMATION

Material properties play an important role in the FEA. The material properties are one of the major inputs to perform the optimization. The materials parameters required depend on the analysis methodology being used. The mechanical properties of C-70 steel are listed in Table 4.1.

Table 4.1: Mechanical properties of C-70 Steel.

Properties	C-70 Steel
Tensile strength, σ_{UTS} (MPa)	965.8
Yield strength(0.2% offset), σ_{YS} (MPa)	573.7
Young's modulus, E (GPa)	211.5

4.3 FINITE ELEMENT MODELING

Structural analysis performed to create high and low stress region from the input of material, loads, boundary condition and geometry. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry and boundary conditions (Lee et al., 2010). The structural analyses allow stresses and strains to be calculated in FEA, by using the structural model. From the analysis results, the critical areas at the structural model can be analyze and improve it in the optimize design.

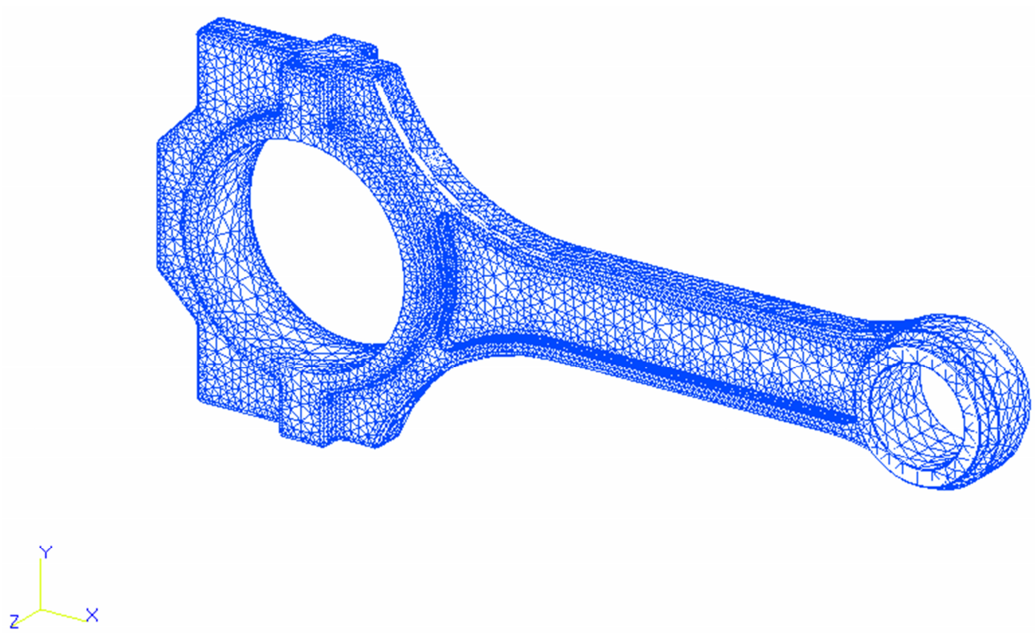
Mesh study was performed on the FE model to define the best mesh for the analysis. It is to increase the accuracy result at the same time not high on the cost (CPU processing time). Rahman et al. (2009) b, the mesh convergences were monitored and evaluated during the analysis. The mesh convergence are based on the geometry, model topology and analysis objectives. Due to tetrahedral mesh produce high quality meshing for solid model imported from CAD systems. FEM of connecting rod is developed utilizing the MSC.PATRAN and create the mesh by using the 10 node tetrahedral elements (TET10) and 4 node tetrahedral elements (TET4).

Mesh study was performed on the FE model to ensure sufficiently fines sizes are employed for accuracy of the calculated result depends on the competitive cost (CPU time). Mesh convergence analysis was performed to determine the optimum element size. It was preformed iteratively at different mesh global length until the appropriate accuracy obtained. Convergence of the stresses was recorded as the mesh global length refined. During the analysis, the specific variable and the mesh convergence was monitored and evaluated. The mesh convergence is based on the geometry, model topology and analysis objectives. For this analysis, the auto tetrahedral meshing approach is employed for the meshing of the solid region geometry. Tetrahedral (TET) meshing produce high quality meshing for boundary representation most of solids model imported from CAD systems. Since the TET is found to be the best meshing technique, the TET 4 version of the connecting rod was then used for the initial analysis. Figure 4.1 shows the comparison of TET4 with difference global edge length of 4 mm and 8 mm. Figure 4.2 shows the results of nodes and elements by using TET4 and TET10 by using the same edge length of 8 mm.

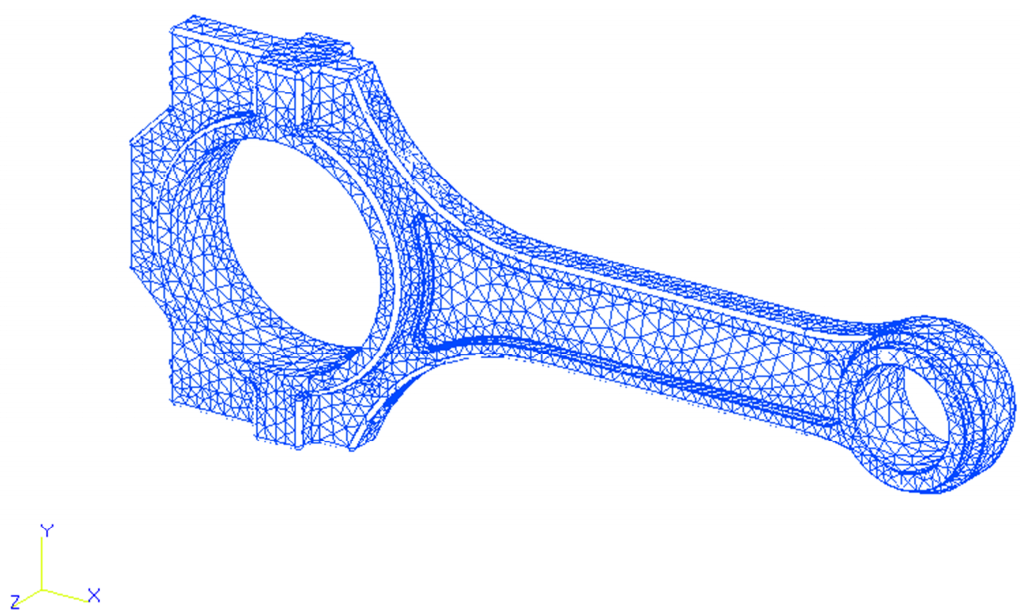
As already discussed before this, a TET10 was used for the solid mesh and tensile loads will be used to analyze the FEM. However, it will be 2 sub cases which is one part are loaded and the other part restrained. The FEA will analyze on the 2 cases and produce with the results. From Shenoy (2004), the uniformly distributed tensile load 180^oon the inner surfaces of the crank end while pin end will restrain as in Figure 4.3 (a). It is just same when load uniformly distributed on pin end surfaces, the crank end will restrain in all direction. This both cases also work exactly in compressive load. In Figure 4.3 (a) and (b), shows the boundary condition of the connecting rod in three-dimensional FE model with load and constraints.

Figure 4.4 shows the von-Mises stresses contour for TET4 and TET10 meshes element at a high load level respectively. The TET10 mesh is presumed to represent a more accurate solution since TET4 meshes which are known to be dreadfully stiff (Rahman et al., 2009). TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function. The TET10 is expected to be able to capture the high stress concentration occurs on connecting rod. Both of the meshes have some distorted elements cause error to the modeling in areas of elevated stress. It can be seen that the TET10 meshes predict higher von-Mises stresses than the TET4. From the Figure 4.4, TET4 predicted 49.5 MPa while TET10 predicted 261 MPa.

Variation of stresses in term of von-Mises, Tresca, maximum principal stress and displacement for TET4 and TET10 are tabulated in Table 4.2 and 4.3 respectively. The comparison was made between these two elements based on stresses concentration and displacement which are in Figure 4.5 and 4.6 respectively. It can be seen that, TET10 is always captured the higher stresses and displacement. Thus, support the conclusion of Wang et al. (2004) which, quadratic tetrahedral elements (TET10) are very good and can always be used. For better and accurate results, TET10 was used instead of TET4. This is because, the number degree of freedom (DOF). TET10 have 30 DOF more than TET4 which only have 12 DOF.

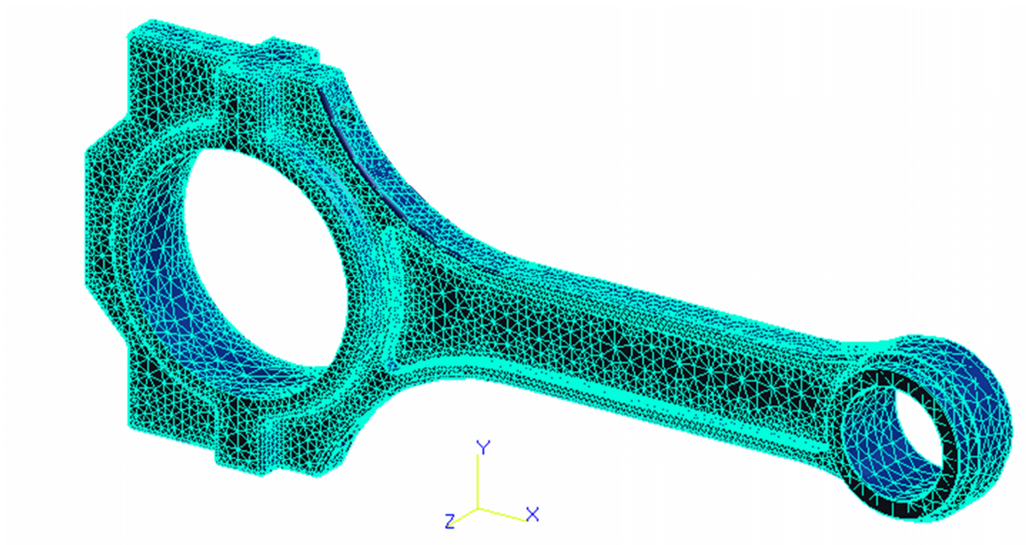


(a) 4 mm (25185 nodes and 109950 elements).

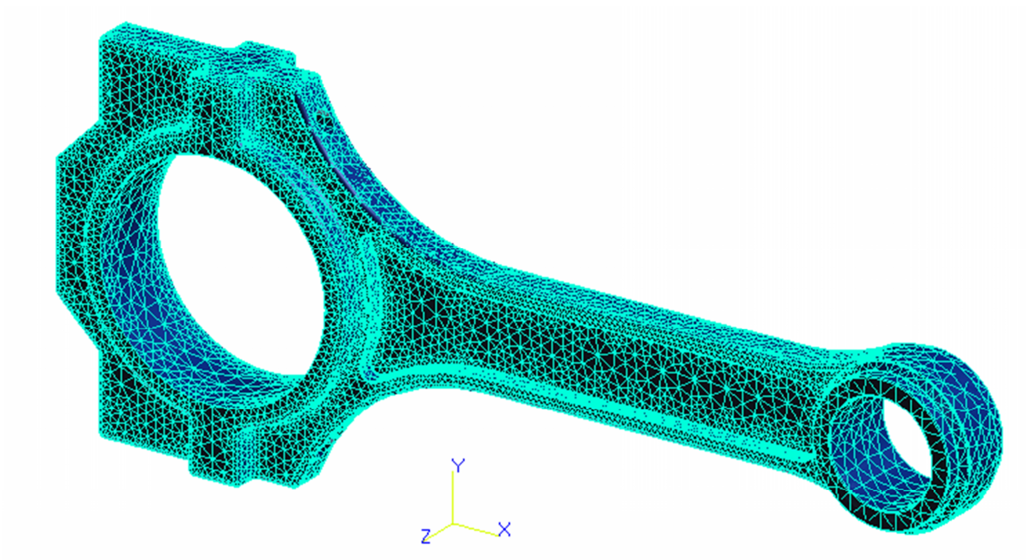


(b) 8 mm, (9717 nodes and 39520 elements).

Figure 4.1: Finite element model with difference mesh size.

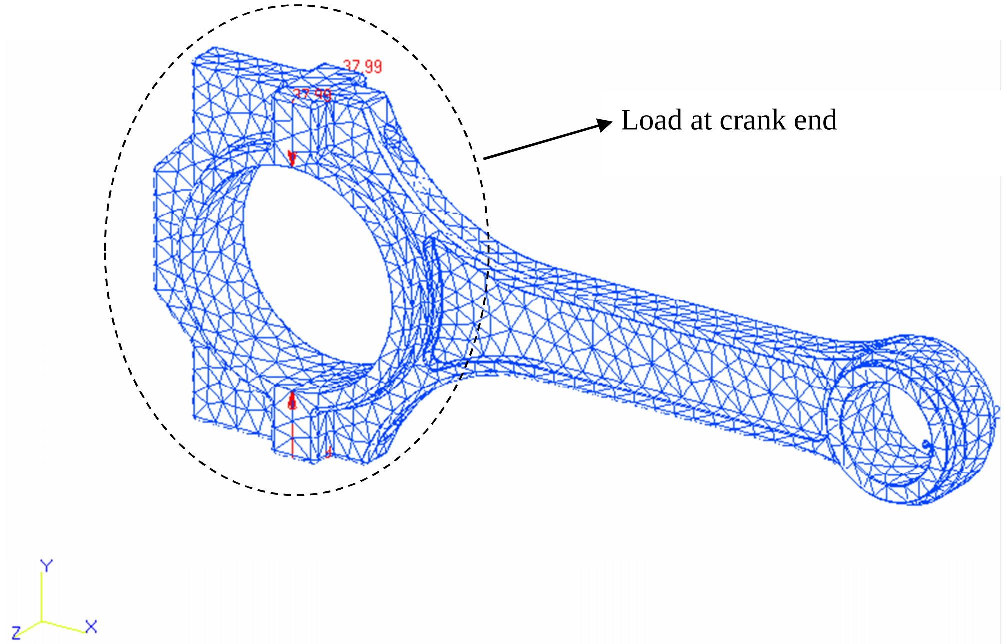


(a) TET4 (109950 nodes and 25185 elements).

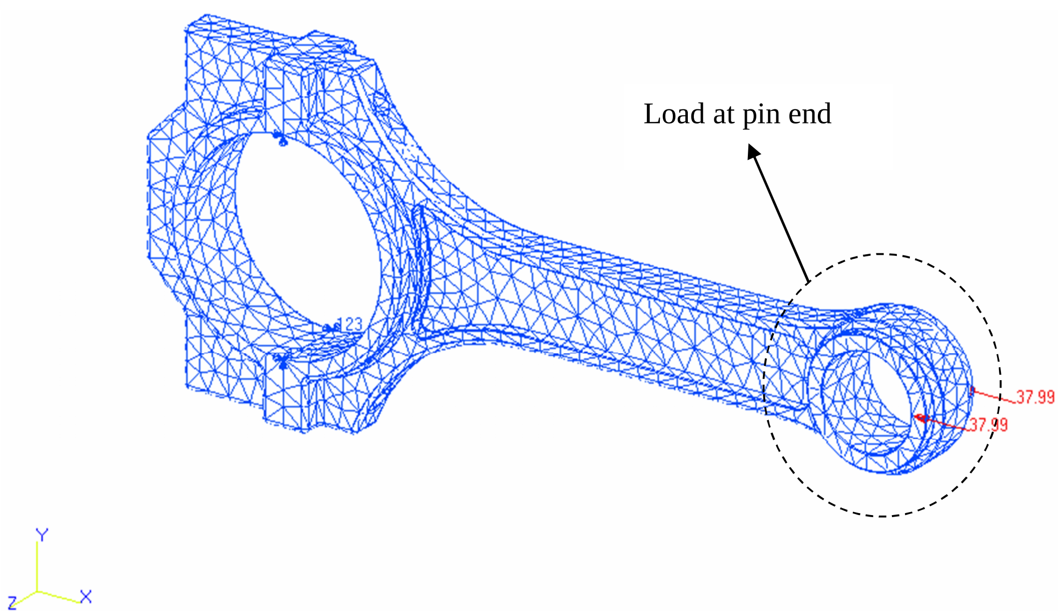


(b) TET10 (110703 nodes and 174743 elements).

Figure 4.2: FEM using 8 mm mesh size with nodes and element.

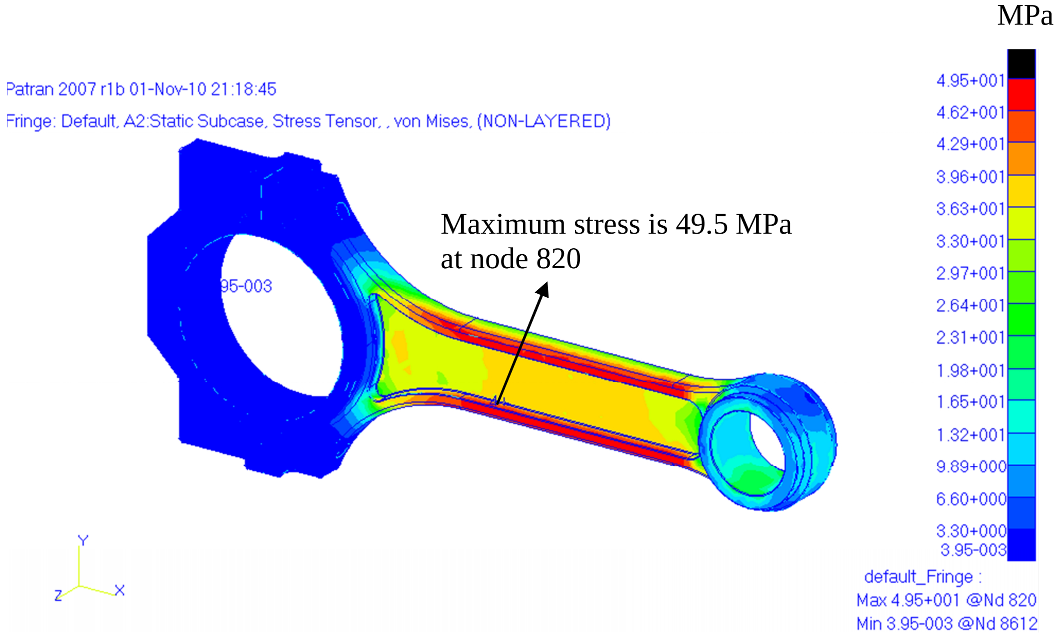


(a) Tensile load at crank end and fixed at pin end.

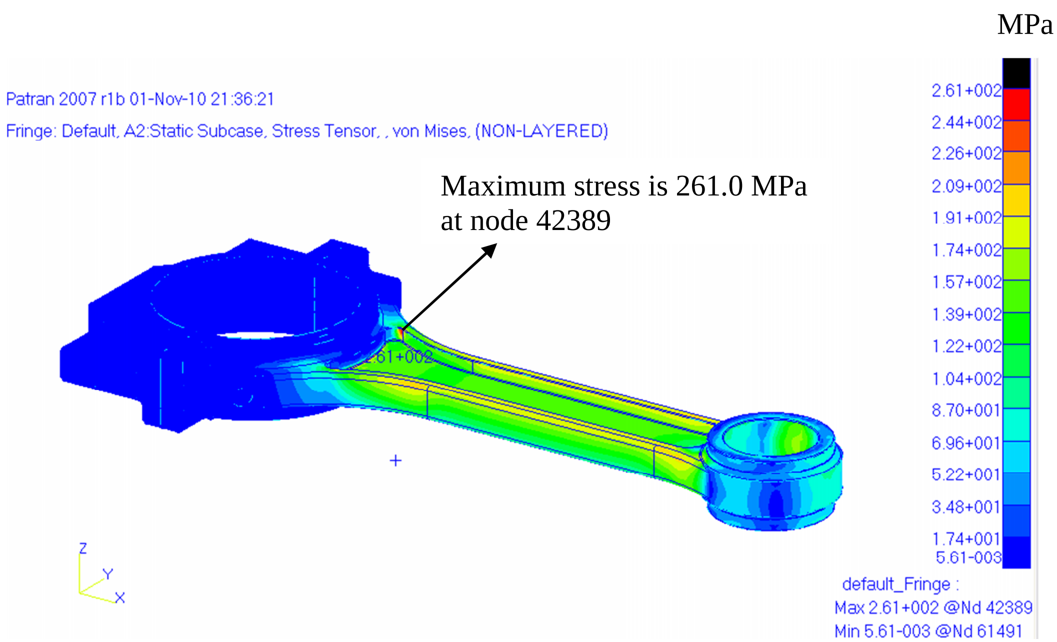


(b) Tensile load at pin end and fixed at crank end.

Figure 4.3: Boundary condition with tensile load.



(a) TET4 (maximum stress is 49.5 MPa at node 820).



(b) TET10 (maximum stress is 261.0 MPa at node 42389).

Figure 4.4: Von-Mises stresses contour.

Table 4.2: Variation of stresses concentration at the critical location of the connecting rod for TET4 mesh.

Mesh Size (mm)	Number Of Node	Number Of Element	Von Mises (MPa)	Tresca (MPa)	Max principal stress (MPa)	Displacement (mm)
16	3012	10575	22.4	22.7	22.5	0.060
12	4596	16882	30.1	30.6	30.5	0.111
8	9717	39520	49.5	50.1	49.6	0.145
4	25185	109950	67.6	68.8	67.8	0.201

Table 4.3: Variation of stresses concentration at the critical location of the connecting rod for TET10 mesh.

Mesh Size (mm)	Number of Node	Number Of Element	von Mises (MPa)	Tresca (MPa)	max principal stress (MPa)	Displacement (mm)
16	19410	10823	107.0	111.0	113.0	0.351
12	29698	17018	142.0	145.0	146.0	0.475
8	65200	39752	261.0	284.0	325.0	0.604
4	174743	110703	301.0	324.0	355.0	0.850

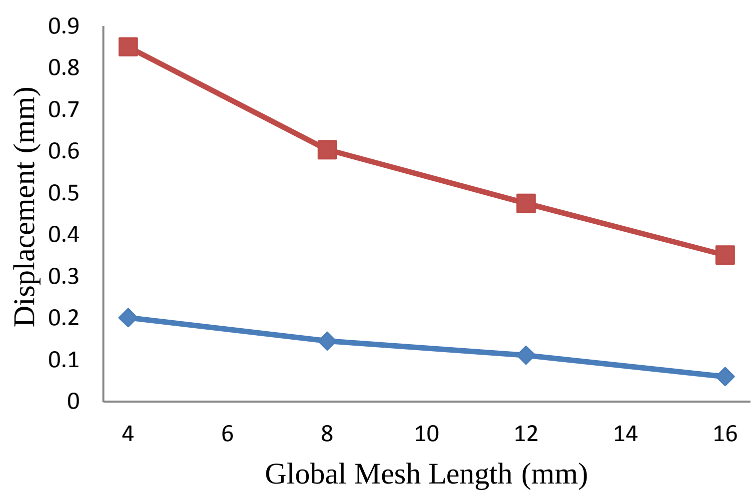
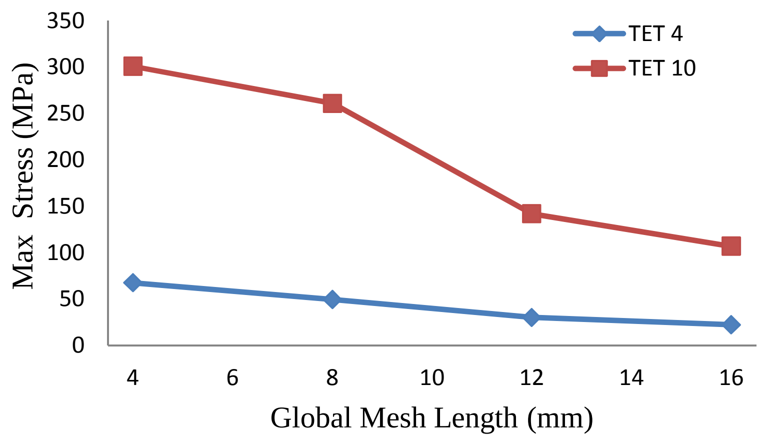
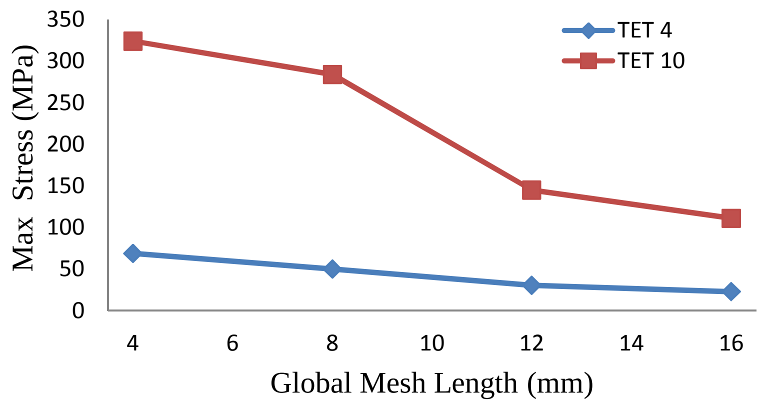


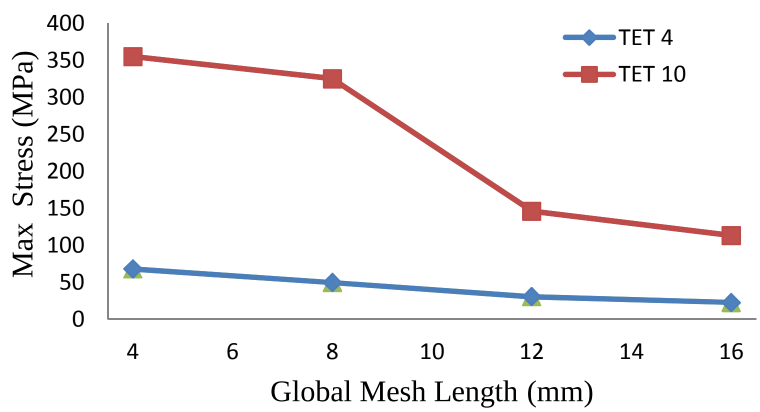
Figure 4.5: Comparison between TET4 and TET10 on maximum displacement.



(a) Von-Mises principal stress.



(b) Tresca principal stress.



(c) Maximum principal stress.

Figure 4.6: Comparison of stresses betweenTET4 and TET10.

4.4 IDENTIFICATION OF THE MESH CONVERGENCE

The convergence of the stress was considered as the main criteria to select the mesh type. The finite element mesh was generated using the TET10 for various mesh global length as shown in Table 4.4. Stresses concentration at critical location for TET10 is presented in Figure 4.7. It can be seen that mesh size of 4 mm obtained largest stresses. The smaller size less than 4 mm do not implemented due the limitation of the CPU time and storage capacity of the computer. Hence, the maximum principal stress based on TET10 at 4 mm mesh size is used in the analysis since the stress is higher compared to Von Mises and Tresca principal stress.

Table 4.4: Variation of mesh size related to number of element and node for TET10.

Mesh size (mm)	Number of node	Number element
4	174743	110703
8	65200	39752
12	29698	17018
16	19410	10823

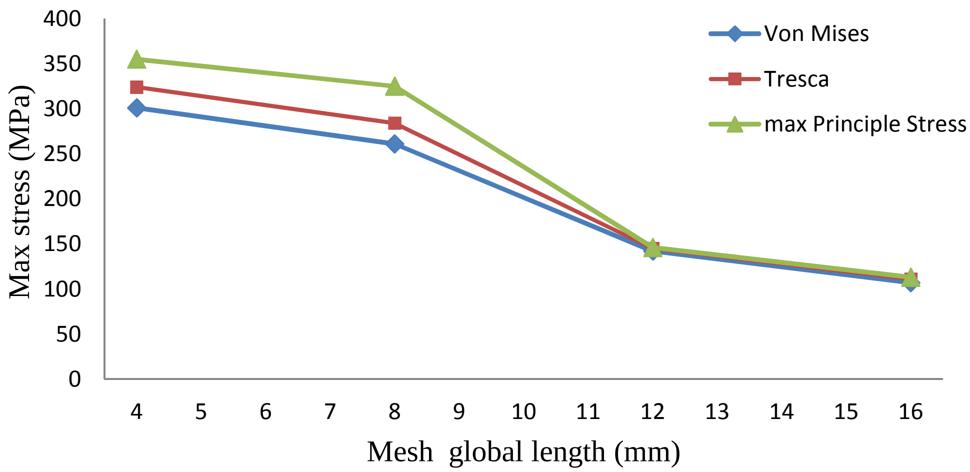


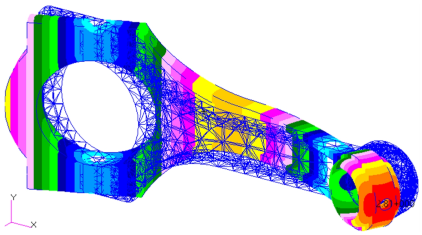
Figure 4.7: Stresses concentration versus mesh global length for TET10 to check mesh convergence.

4.5 MODAL ANALYSIS

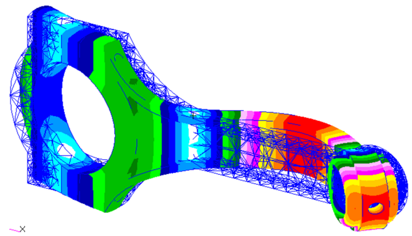
The modal analysis is usually used to determine the natural frequencies and mode shapes of a component. It can also be used as the starting point for the frequency response, the transient and random vibration analysis (Rahman, 2007). The finite element analysis codes usually used several mode extraction methods. The Lanczos mode extraction method is used in this study due to it is the recommended method for the medium to large models. In addition to its reliability and efficiency, the Lanczos method supports sparse matrix methods that significantly increase computational speed and reduce the storage space. This method also computes precisely the eigenvalues and eigenvectors. The number of modes was extracted and used to obtain the connecting rod stress histories, which is the most important factor in this analysis. Using this method to obtain the first 10 modes of the connecting rod, which are presented in Table 4.5 and the shape of the mode are shown in Figure 4.8.

Table 4.5: The result of modal analysis.

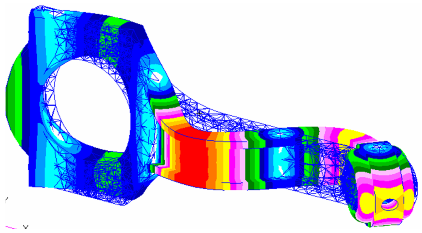
Mode	Frequency (Hz)
1	19.11
2	61.21
3	142.31
4	210.24
5	221.30
6	250.69
7	320.85
8	466.34
9	466.82
10	483.95



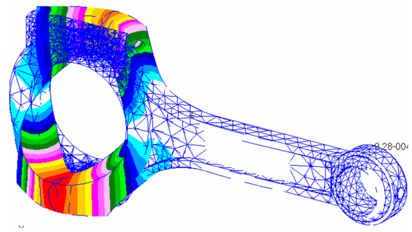
Mode 1, 19.11 Hz



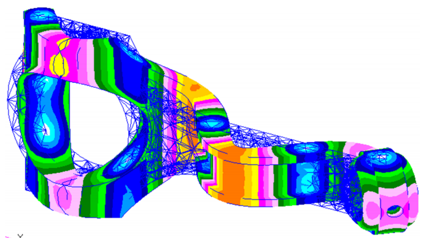
Mode 2, 61.21 Hz



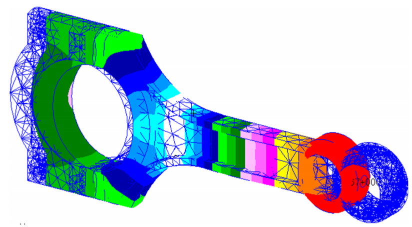
Mode 3, 142.31 Hz



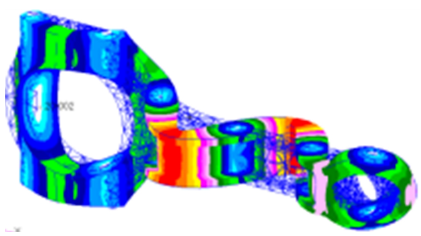
Mode 4, 210.24 Hz



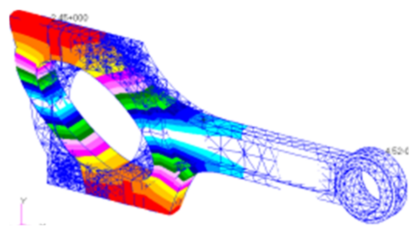
Mode 5, 221.30 Hz



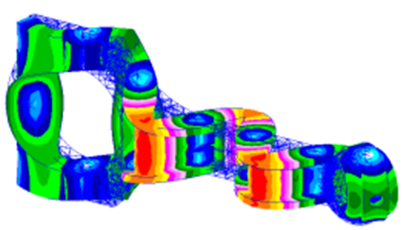
Mode 6, 250.69 Hz



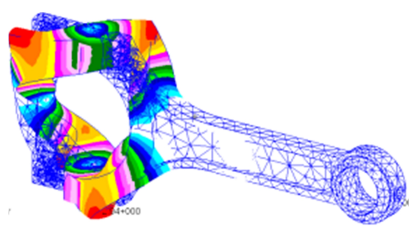
Mode 7, 320.85 Hz



Mode 8, 466.34 Hz



Mode 9, 466.82 Hz



Mode 10, 483.95 Hz

Figure 4.8: The mode shapes of the connecting rod.

4.6 OPTIMIZATION OF CONNECTING ROD

The optimization of the connecting rod carried out by using topology optimization technique. The optimization focused on the uncritical sections which need to be reduced. From the topology optimization, it suggests the unnecessary shape and design of the connecting rod. The results of topology optimization of the connecting rod for crank end and pin end are shown in Figure 4.9.

The optimization results are tabulated in Table 4.6. The objective function and maximum constraint history are shown in Figure 4.10 and 4.11. In finite element model of connecting rod, there are several contact areas concerning multi-point constraint. Therefore, constraints are employed the following purposes:

(i) Specify the prescribe enforce displacements
(ii) To simulate the continuous behavior of displacement in the interface area
(iii) To enforce rest condition in the specified directions at grid points of reaction

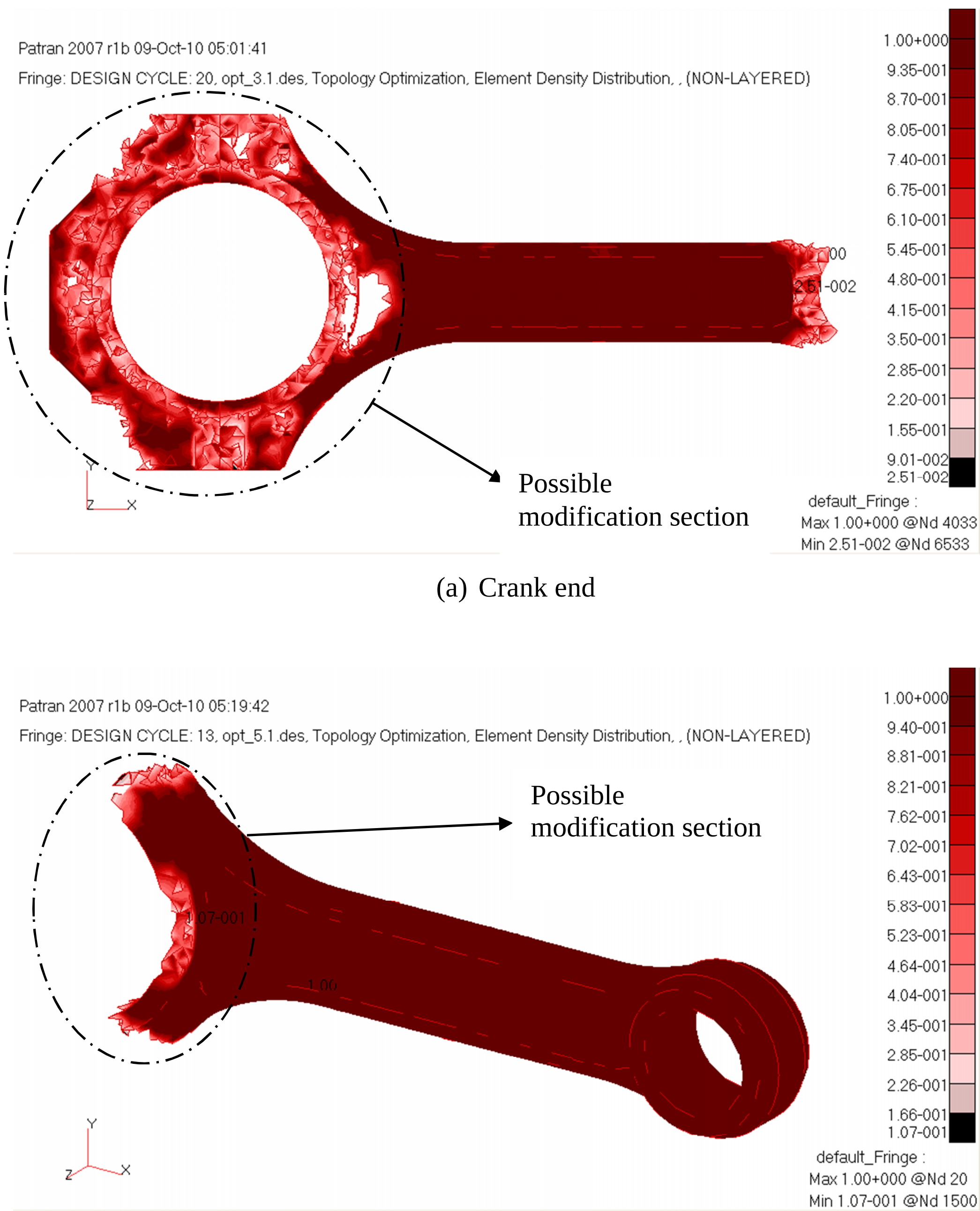
The main objective is to minimize the weight of the connecting rod as well as the total production cost. It can be seen that the optimized model is reduce the weight from initial design until the value converges. Figure 4.10 shows the objective function history of the optimization. The convergence of the design is immediately after cycle no. 9.

The implementation of these optimizations is to find out the best design and topology of the connecting rod to improve the performance and the strength especially at the critical location. The possible modification section of the optimized connecting rod is indicated in the figure. The section with lower value than initial value considered as the suggestion to be optimized in the new design.

Figure 4.11 shows the maximum constraint history during optimization on the connecting rod. The implementation of these optimizations is to find out the best design and topology of the connecting rod to improve the performance and the strength especially at the critical location. The uncritical section colored with light color while the critical section valued as 1.00 as the initial value. The section with lower value than initial value considered as the suggestion to be optimized in the new design.

Table 4.6: Objective function and maximum constraint history from optimization of connecting rod.

Cycle Number	Objective From Approximate Optimization	Objective From Exact Analysis	Fractional Error of Approximation	Maximum Value of Constraint
Initial		2.41×10^{3}		0.00
1	8.91×10^{2}	1.56×10^{3}	-4.29×10^{-1}	1.21×10^{-4}
2	6.99×10^{2}	1.08×10^{3}	-3.54×10^{-1}	1.31×10^{-5}
3	5.73×10^{2}	7.94×10^{2}	-2.78×10^{-1}	-2.20×10^{-6}
4	5.05×10^{2}	6.15×10^{2}	-1.77×10^{-1}	1.10×10^{-5}
5	5.15×10^{2}	5.53×10^{2}	-6.86×10^{-2}	-1.98×10^{-5}
6	4.78×10^{2}	5.00×10^{2}	-4.45×10^{-2}	6.79×10^{-6}
7	4.61×10^{2}	4.72×10^{2}	-2.32×10^{-2}	1.09×10^{-4}
8	4.42×10^{2}	4.49×10^{2}	-1.50×10^{-2}	-3.91×10^{-5}
9	4.27×10^{2}	4.31×10^{2}	-9.32×10^{-3}	4.60×10^{-5}
10	4.15×10^{2}	4.18×10^{2}	-7.51×10^{-3}	1.72×10^{-4}
11	4.07×10^{2}	4.09×10^{2}	-4.61×10^{-3}	-2.82×10^{-5}
12	4.01×10^{2}	4.02×10^{2}	-3.14×10^{-3}	-3.17×10^{-5}
13	3.96×10^{2}	3.97×10^{2}	-2.26×10^{-3}	-2.77×10^{-5}
14	3.93×10^{2}	3.94×10^{2}	-1.79×10^{-3}	7.73×10^{-5}
15	3.90×10^{2}	3.91×10^{2}	-1.51×10^{-3}	5.71×10^{-5}
16	3.88×10^{2}	3.89×10^{2}	-1.04×10^{-3}	-9.83×10^{-6}
17	3.86×10^{2}	3.86×10^{2}	-6.33×10^{-4}	1.77×10^{-4}
18	3.85×10^{2}	3.85×10^{2}	-8.03×10^{-4}	-1.34×10^{-4}
19	3.83×10^{2}	3.83×10^{2}	-7.98×10^{-4}	7.08×10^{-5}
20	3.82×10^{2}	3.82×10^{2}	-5.96×10^{-4}	6.80×10^{-5}



(a) Crank end

(b) Pin end

Figure 4.9: Optimized connecting rod.

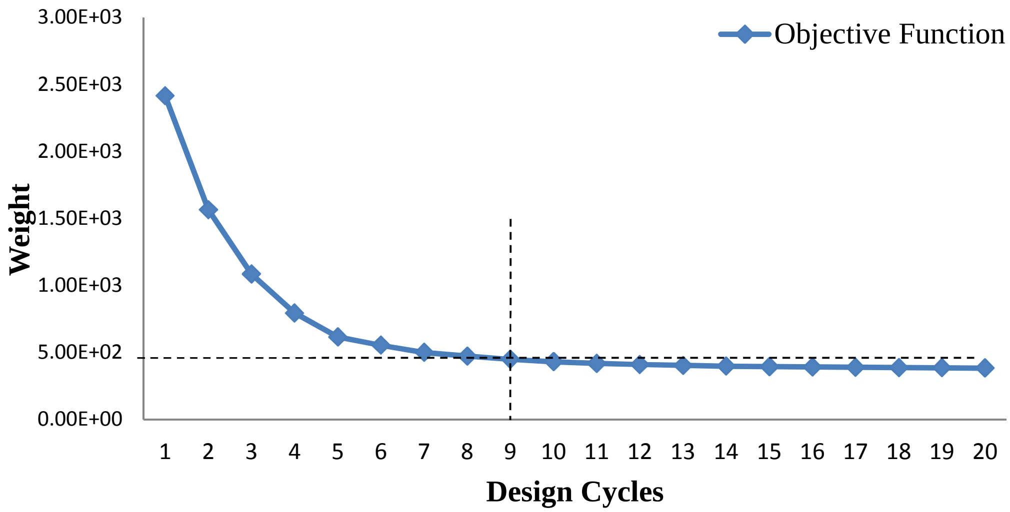


Figure 4.10: Objective function history.

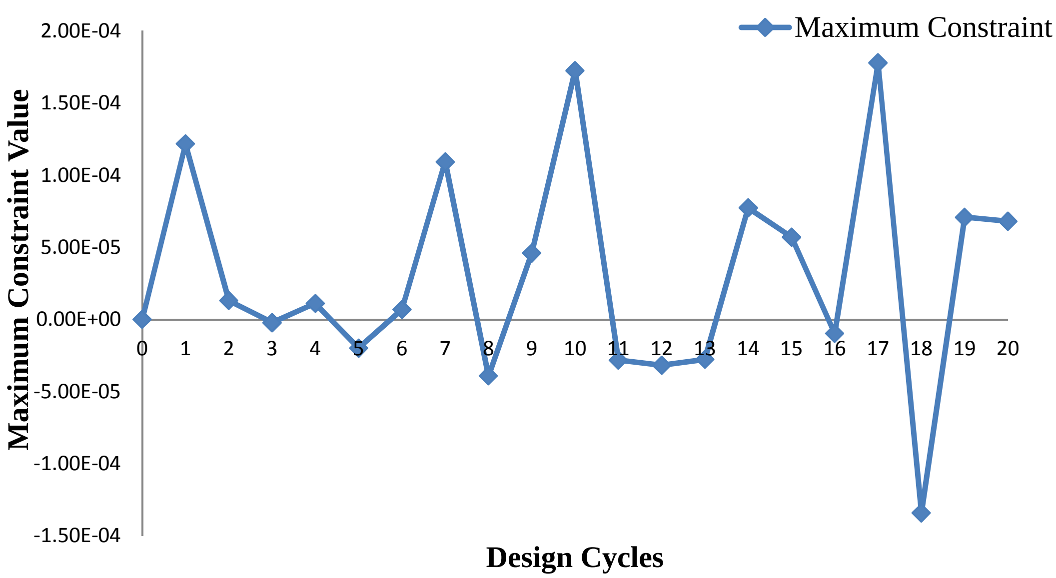


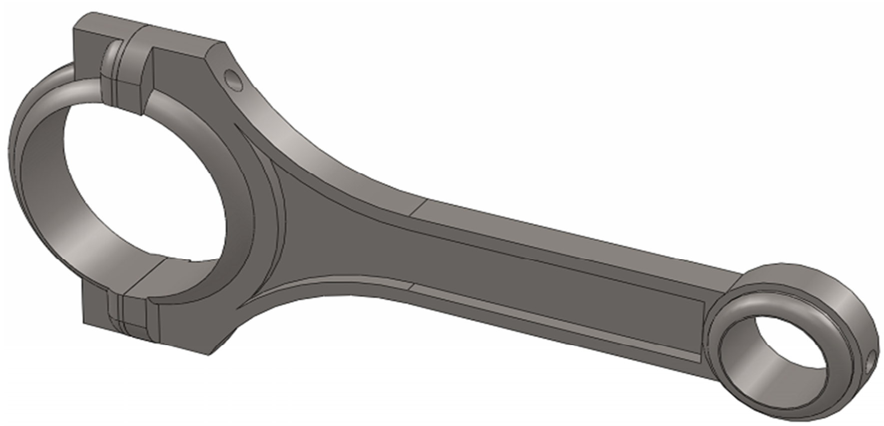
Figure 4.11: Maximum constraint history.

4.7 PROPOSED NEW DESIGN

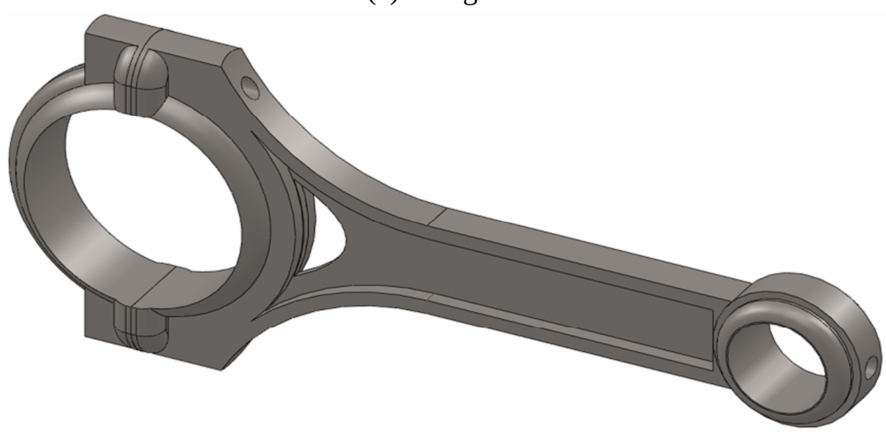
Based from the optimization of the connecting rod, the new designs were developed by using SOLIDWORKS. Figure 4.12 and Figure 4.13 shows the choices of new connecting rod. The optimize connecting rod no 4 was choose as the best optimize design due to the lowest occurred stress and mass. Even though the mass of the optimize connecting rod is not the lowest, but the decision was also based on the maximum stress which is 320 MPa. Figure 4.13 shows the new design of the connecting rod. Table 4.7 shows the comparison between initial and optimize designs on max principles stress and mass of the connecting rod.

Mass of the connecting rod is 0.464 kg compare to initial design 0.577 kg which is 11.7% lighter. Figure 4.14 shows the existing connecting rod superimposed with optimized connecting rod with dimension. Major changes of the shape were made at the crank end of the connecting rod. Figure 4.15 shows the top view of the connecting rod where, the thickness of the connecting rod is reduced especially at the crank end as can be seen at the dimension. More fillets were used on the new connecting rod especially at top edge of crank end and pin end. Figure 4.16 shows the full front view of the connecting rod compare with existing design.

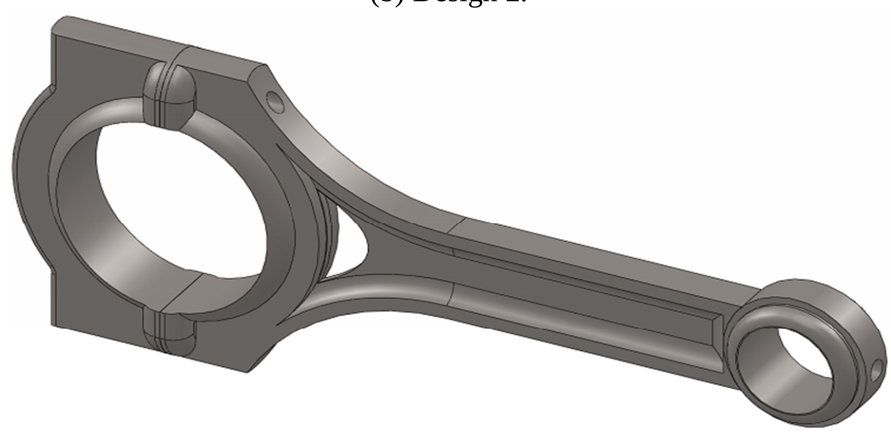
The new designs were validated to check on max principle stress and weight. From the results of the analysis, the best design with lower max principle stress and weight were choose. Figure 4.17 shows the validation of the optimization on the connecting rod. The maximum principle stress occur on the connecting rod is 320 MPa when the tensile load at pin end and 325 MPa at crank end which is lower than yield strength of the material of connecting rod.



(a) Design 1.

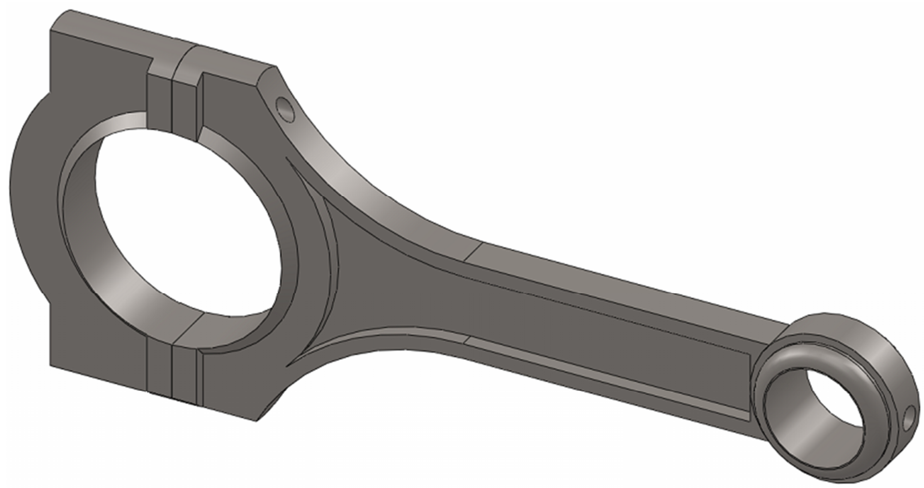


(b) Design 2.

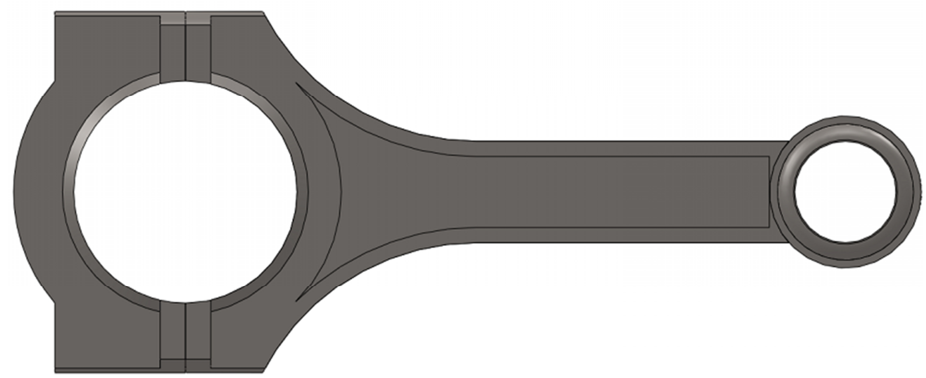


(c) Design 3.

Figure 4.12: Isometric 3D view of new design.



(a) Isometric 3D view



(b) Front view.

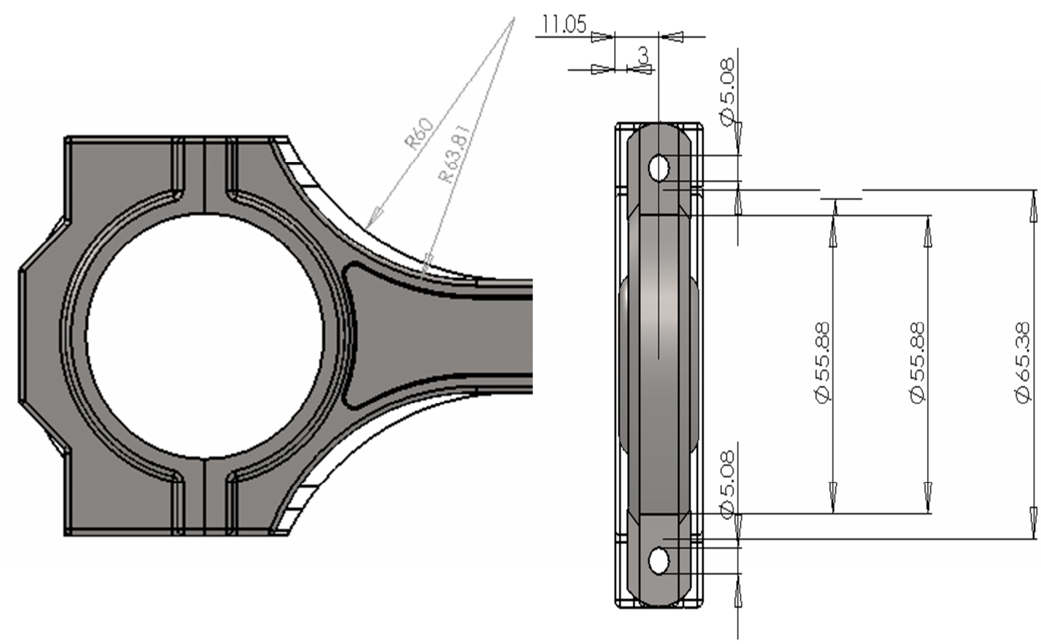
(c) Top view.

(d) Side view.

Figure 4.13: Different view of optimized design.

Table 4.7: Comparison between initial and optimized designs on stress and mass.

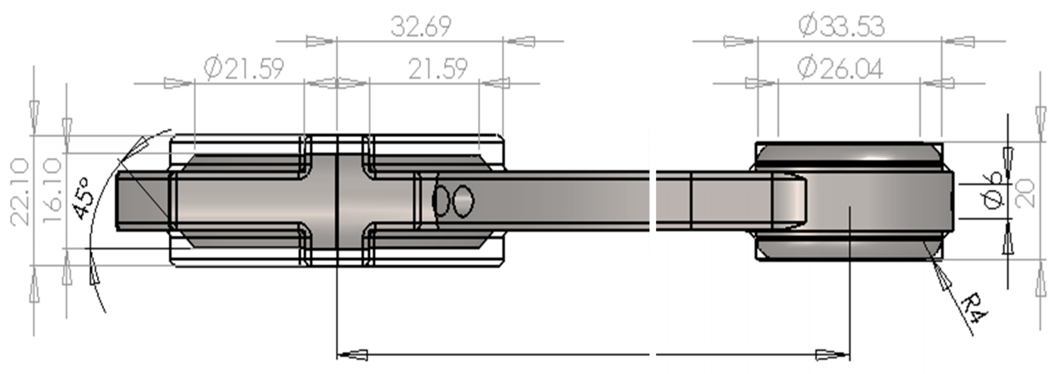
Design	Maximum principles stress (MPa)	Mass (kg)
Initial	1230	0.577
Optimize No 1	1870	0.436
Optimize No 2	1340	0.429
Optimize No 3	906	0.513
Optimize No 4	320	0.464



(a) Crank end.

(b) Side view.

Figure 4.14: 3D view of optimized design superimposed with initial design.



(a) Crank end. (b) Pin end.

Figure 4.15: Top 3D view of optimized design superimposed with initial design.

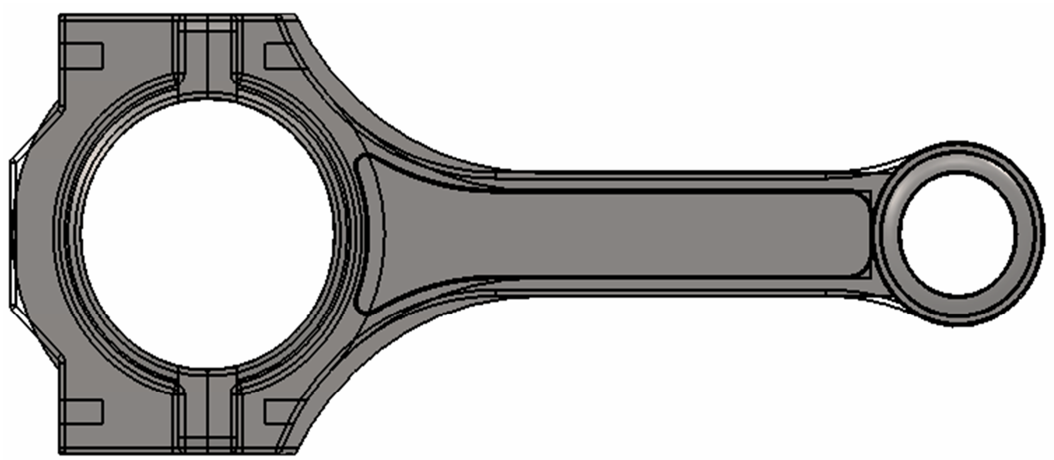
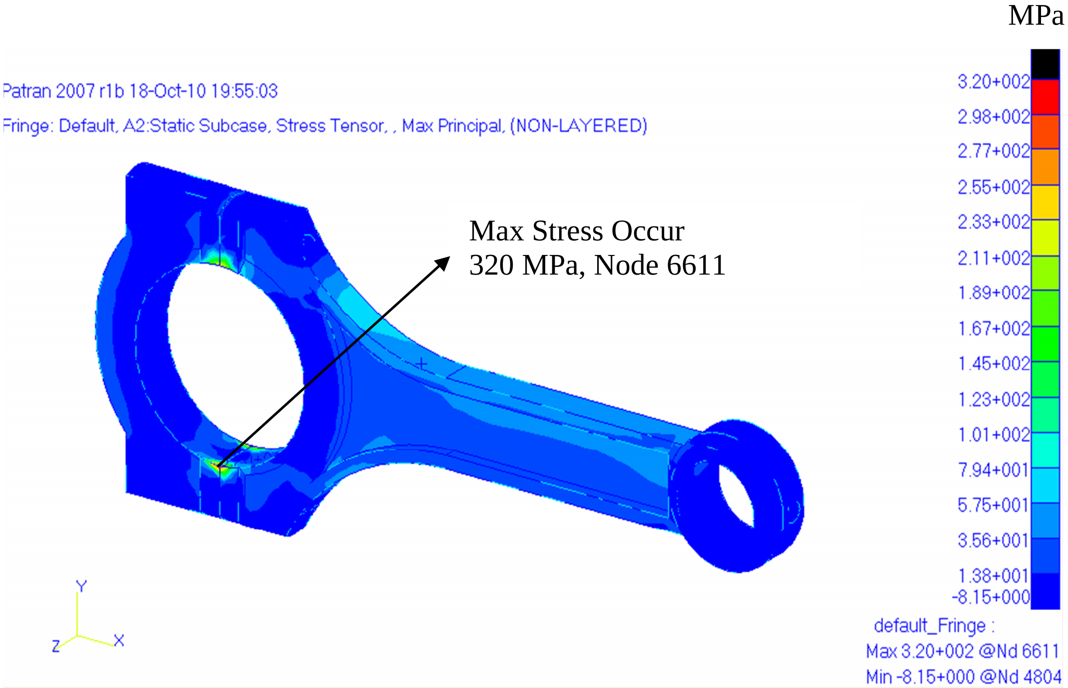
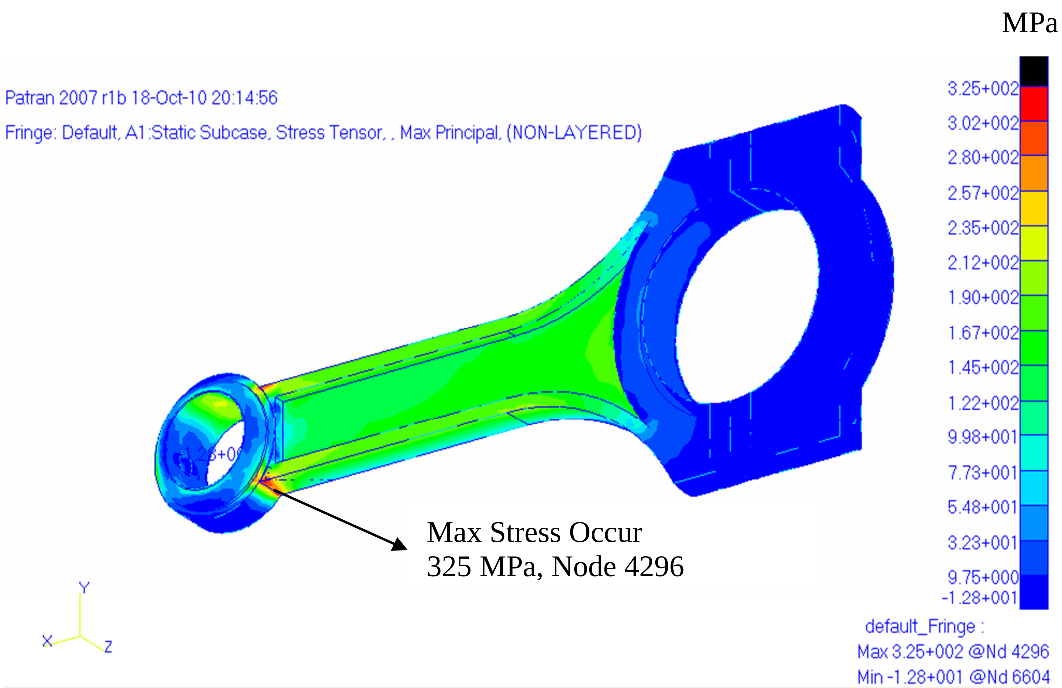


Figure 4.16: Front 3D view of optimized design superimposed with initial design.



(a) Tensile load at crank end (maximum stress is 320 MPa at node 6611).



(b) Tensile load at pin end (maximum stress is 325 MPa at node 4296).

Figure 4.17: Maximum principle stresses contour.

4.8 CONCLUSION

The finite element modeling and analysis of connecting rod has been presented. The mesh convergence and stress analysis was performed to determine the best parameters to be used in FEA. The modal analysis was done to get the mode shapes. The optimization of the connecting rod was successfully performed and the weight of the connecting rod reduced 11.7% compare to initial design. The summary of the finding will be present in the next chapter.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 INTRODUCTION

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

5.2 CONCLUSION

The modeling of connecting rod and FEA has been presented. From the study, the structural modeling is one of important factors which need to be carefully design before performing the optimization to the connecting rod. Thus, becoming the initial design of the connecting rod and were compared with the optimized design based on the objective of optimization.

The FEM of the connecting rod is best to be modeled by using TET10 due to higher prediction of stress and displacement compare with TET4. The TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function. The TET10 is able to capture the high stress concentration occurs on connecting rod. To achieve the accurate result from the FEA, the best stress choose to be the maximum principle stress which gave more accurate result compare to Von Mises and Tresca principal stress. Smaller mesh global length is used to give more accurate result on stress, strain, displacement and optimization compare with bigger sizes of mesh global length. However, it should count limitation of CPU time and storage capacity of the computer during analysis.

The optimization shows that the objective functions start to converge after design cycle no 9, which means that the weight of the connecting rod were reduced and the process of optimizing the connecting rod cycles back until it optimum level. From the topology optimization, suggestion on design modification is made on the crank end and no design modification is made on pin end section of connecting rod. The result of optimization shows that optimized design is 11.7% lighter than the initial design of the connecting rod.

5.3 RECOMMENDATION

There is still scope for further study to improve the optimization analysis. The recommendations are as follows:

(i) The structural analysis can be perform by using different materials and loading condition. It can be done by performing the dynamic load analysis where the loading condition varies at different crank angle.

(ii) The dimension of the new connecting rod can be modified to get the significant result during the optimization. The best optimization should be perform on the existing connecting rod where it is available on today's market. From there, the design optimization can be validate and contribute for a better product design.

(iii) Different variables, objective functions and method of optimization can be modified in order to varying the optimization results. However, the optimization still needs to focus on the main improvements which are the strength and the performance of the product itself.

REFERENCES

Afzal, A and Fatemi, A. 2004. A comparative study of fatigue behavior and life predictions of forged steel and PM connecting rods, *SAE Technical Paper*, Paper No. 2004-01-1529.

Amin, H.A. 2008. *A comparative study of 2 CAD integrated FE-programs using the linear static analysis*. Master's Thesis. Halmstad University, Sweden.

Bari, M.W., Rahman, M.M., Ali, M.M.Y. and Awall, M.R. 2004. Finite Element Analysis of Slab and A Comparative Study With Others Analytical Solution. *Seventh International Conference on Concrete Technology in Developing Countries (7th ICCT)*, pp. 265-272, Kuala Lumpur.

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

Gupta, R.K. 1993. Recent developments in materials and processes for automotive connecting rods. *SAE Technical Paper*, Paper No. 930491.

Hashmi, S.I.A. 2005. *Designing 3D Modeling, Static and Dynamic Analysis of Connecting Rod and Machining of its Die on Cnc Machine*. Bachelor's Thesis. University of Engineering and Technology, Pakistan.

Lee, M.K., Lee, H., Lee, T.S. and Jang, H. 2010. Buckling sensitivity of a connecting rod to the shank sectional area reduction, *Materials and Design*, **31:** 2796–2803.

Mirehei, A., Zadeh, H.M., Jafari, A. and Omid. M. 2008. Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS). *Journal of Agricultural Technology*. **4**(2): 21-27.

Niemann, H. 2008. A simple topology optimization example with MD R2 Patran. Dépatement Mécanique des Structures et Matériaux. Université de Touluse.

Rahman, M.M. 2007. *Finite element based durability assessment for a new free piston linear engine.* PhD Thesis. Universiti Kebangsaan Malaysia, Malaysia.

Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M. and Bakar, R.A. 2009. Durability assessment of cylinder block for two stroke free piston linear engine using random loading. American Journal of Applied Sciences, 6(**4**): 726-735.

Rahman, M. M., Kadirgama, K., Noor, M. M., Rejab, M. R. M. and Kesulai, S. A. 2009b. Fatigue life prediction of lower suspension arm using strain-life approach, *European Journal of Scientific Research.* **30**(3): 437-450.

Rasekh, M., Asadi, M.R., Jafari, A. and Kheiralipour, K. 2009. Obtaining maximum stresses in different parts of tractor (MF-285) connecting rods using finite element method. *Journal of Basic and Applied Sciences*. **3**(2): 1438-1449.

Repgen, B. 1998. Optimized connecting rods to enable higher engine performance and cost reduction. *SAE Technical Paper*, Paper No. 980882.

Shenoy, P.S. 2004. *Dynamic load analysis and optimization of connecting rod*. Master's Thesis. University of Toledo, USA.

Shenoy, P.S and Fatemi, A. 2005. Connecting rod optimization for weight and cost reduction, *SAE Technical Paper*, Paper No. 2005-01-0987.

Ulatowska, A. 2008. *Shape optimization of the connecting rod*. Master's Thesis. Politechnika Krakowska, Thailand.

Wang, E., Nelson, T. and Rauch, R. 2004. Back to Elements - Tetrahedra vs. Hexahedra, CAD-FEM GmbH, Germany.

Yang, R.J., Dewhirst, D.L., Allison, J.E. and Lee, A. 1992. Shape optimization of connecting rod pin end using a generic model. *Finite Elements in Analysis and Design.* **11**: 257-264.

Zoroufi, M. 2004. *Manufacturing process effects on fatigue design and optimization of automotive components – an analytical and experimental study*. PhD Thesis. University of Toledo, USA.