National Conference in Mechanical Engineering Research and Postgraduate Studies (2nd NCMER 2010)
3-4 December 2010, Faculty of Mechanical Engineering, UMP Pekan, Kuantan, Pahang, Malaysia; pp. 155-166
ISBN: 978-967-0120-04-1; Editors: M.M. Rahman, M.Y. Taib, A.R. Ismail, A.R. Yusoff, and M.A.M. Romlay

DESIGN OF CONNECTING ROD OF INTERNAL COMBUSTION ENGINE: A TOPOLOGY OPTIMIZATION APPROACH

M.S. Shaari[1], M.M. Rahman[1,2], M.M. Noor[1], K. Kadirgama[1] and A.K. Amirruddin[1]

[1]Faculty of Mechanical Engineering, Universiti Malaysia Pahang
26600 UMP, Pekan, Kuantan, Pahang, Malaysia
Phone: +6094242246, Fax: +609-4242202
E-mail: shamilshaari@yahoo.com; mustafizur@ump.edu.my
[2]Automotive Engineering Centre, Universiti Malaysia Pahang

ABSTRACT

This paper presents the design connecting rod of internal combustion engine using the topology optimization. The objectives of this paper are to develop structural modeling, finite element analyze and the optimization of the connecting rod for robust design. The structure of connecting rod was modeled utilized SOLIDWORKS software. Finite element modeling 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 achieve the objectives of optimization which is 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. The crank end is suggested to be redesign based on the topology optimization results. The optimized connecting rod is 11.7% lighter and predicted low maximum stress compare to initial design. For future research, the optimization should cover on material optimization to increase the strength of the connecting rod.

Keywords: Topology optimization, connecting rod, finite element analysis, minimum weight, linear static method.

INTRODUCTION

Connecting rods are highly dynamically loaded components used for power transmission in combustion engines. The optimization of connecting rod had already started as early year 1983 by Webster and his team. However, each day consumers are looking for the best from the best. That's why the optimization is really important especially in automotive industry. Optimization of the component is to make the less time to produce the product that is stronger, lighter and less cost. The design and weight of the connecting rod influence on car performance. Hence, it is effect on the car manufacture credibility. Change in the structural design and also material will be significant increments in weight and performance of the engine. Mirehei et al. (2008) were performed the study regarding the fatigue of connecting rod on universal tractor (U650) by using ANSYS software application and the lifespan was estimated. The authors also investigated that the stresses and hotspots experienced by the connecting rod and the state of stress as well as stress concentration factors can be obtained and consequently used for life predictions. Rahman et al. (2008a, 2009a) 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 TET4 and 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.

A connecting rod is subjected to many millions of repetitive cyclic loadings. Therefore, durability of this component is of critical importance. It is necessary to investigate finite element modeling techniques, optimization techniques and new design to reduce the weight at the same time increase the strength of the connecting rod itself. Shenoy (2004) was explored the weight and cost reduction opportunities for a production forged steel connecting rod. The study has dealt with two parts which are dynamic load and quasi-dynamic stress analysis of the connecting rod, and second to optimize the weight and cost. Shenoy and Fatemi (2005) were 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. 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 MSC/PATRAN and MSC NASTRAN. The upper end (pin end) of a connecting rod is optimized under a variety of initial assumptions to illustrate the use of the system. The objectives of the study are to develop structural modeling of connecting rod and perform finite element analysis of connecting rod. The main objective is to develop topology optimization model of connecting rod.

OPTIMIZATION APPROACH

The objective of optimization technique is to minimize the mass of the connecting rod and reduce the cost of production. The connecting rod subjected to tensile load at crank end, while using factor of safety 3 as recommended by Shenoy (2004). The maximum stress of the connecting rod monitored and make sure it is not over the allowable stress. The load of the connecting rod optimized is comprised of the tensile load of 26.7 kN at crank end. Linear buckling analysis was performed on the connecting is 26.7 kN. The buckling load factor is considered also 3. The optimization technique methodology flowchart is shown in Figure 1.

RESULTS AND DISCUSSION

This section presents the details results of FE Analysis, selection of the mesh type and influence of mesh type, identification of mesh convergence and optimization of the connecting rod.

Finite Element Modeling and Analysis

The connecting rod is one of the most important components in the internal combustion engine (Rasekh et al., 2009). Therefore, the initial design is compared to other design before performing the optimization. A simple three-dimensional model of connecting rod was developed using SOLIDWORKS software and finite element model was created using TET10 as shown in Figure 2. Mesh study was performed on the FE model to ensure sufficiently fines sizes are employed for accuracy of the calculated result depends on the CPU time Rahman et al. (2009b,c). 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.

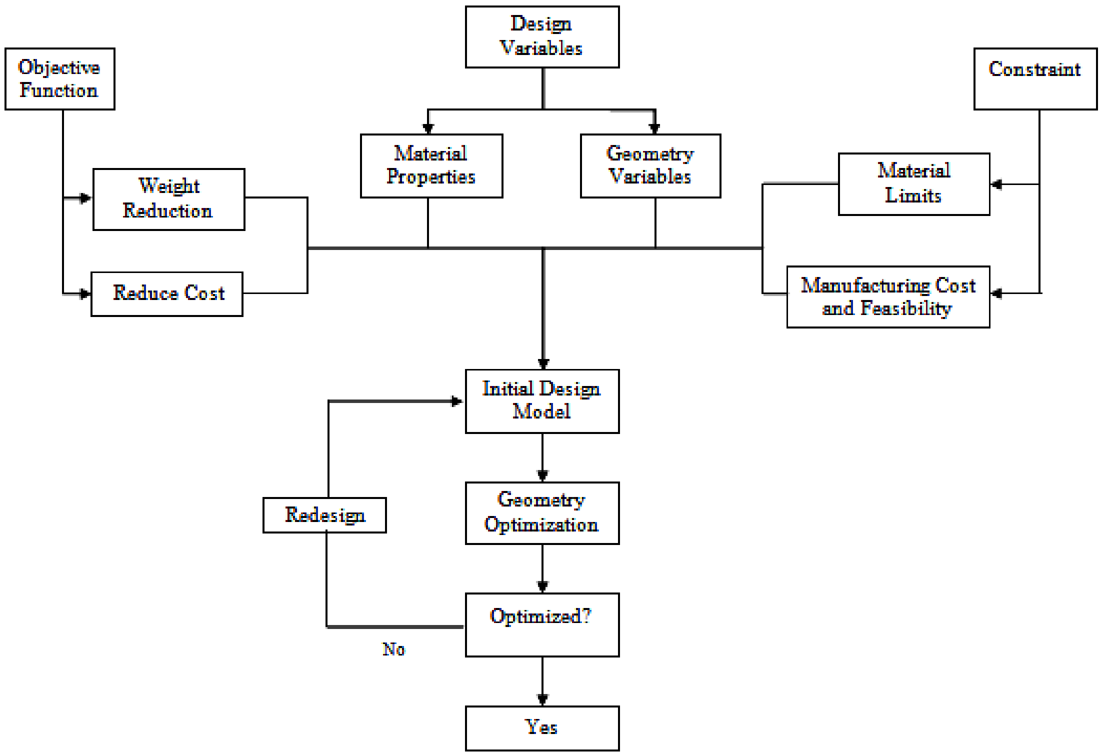
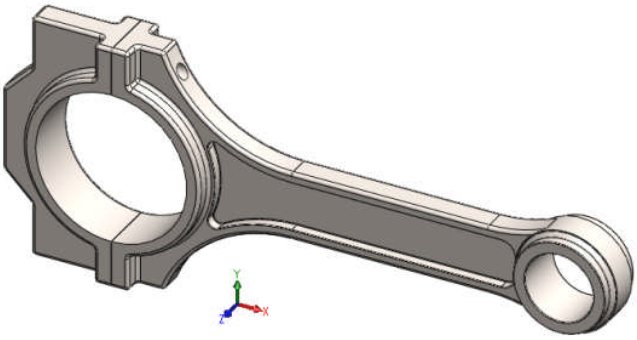
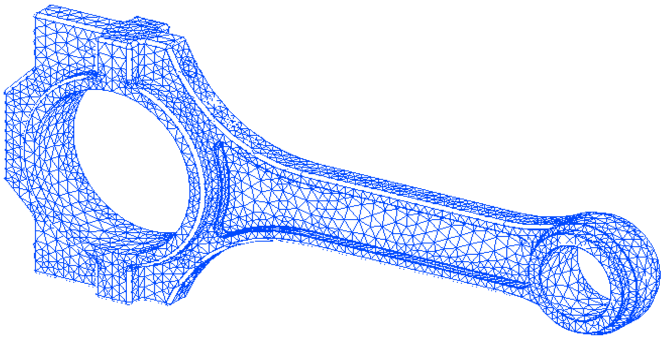


Figure 1: Flowchart of optimization approach

The uniformly distributed tensile load 180^{o} on the inner surfaces of the crank end while the other part, pin end is restrain as in Figure 3. 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 2, shows the boundary condition of the connecting rod in three-dimensional FE model with load and constraints. In this study four finite element 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).

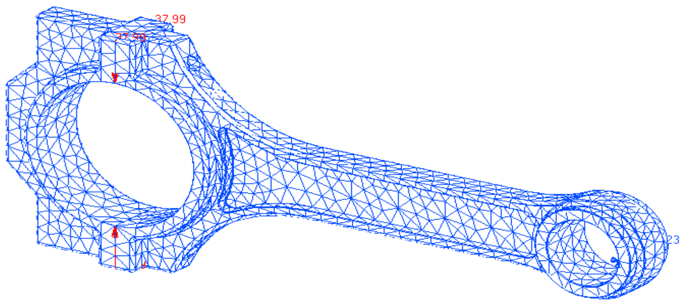


(a) Isometric 3D view

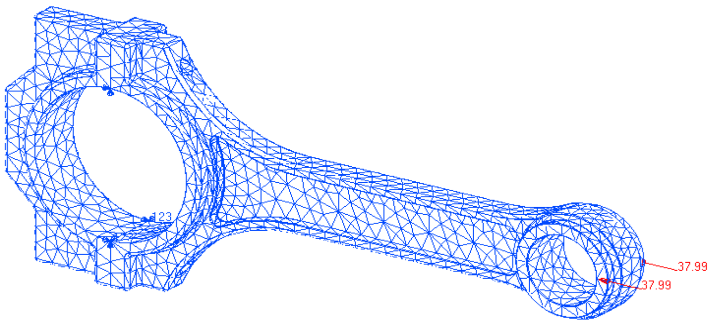


(b) TET10 (nodes 65547, element 39987)

Figure 2: (a) Structural model of initial design. (b) Connecting rod by using



(a)



(b)

Figure 3: (a) Tensile load at crank end and fixed at pin end; (b) Tensile load at pin end and fixed at crank end.

Influence of Mesh Type

The tetrahedral meshing approach is employed for the meshing of the solid region geometry. Tetrahedral meshing produces high quality meshing for boundary representation of solid structural model. Since the tetrahedral is found to be the best meshing technique, TET4 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 (26.7 kN). Material properties play an important role in the result of the FE analysis. The material properties are one of the major inputs to perform the FEA and optimization. C-70 steel is The mechanical properties of C-70 steel are shown in Table 1. From the results, it can be found that the TET10 mesh predicted higher von Mises stresses than that TET4 meshes are known to be dreadfully stiff (Rahman et al., 2007, 2008b). TET4 employed a linear order interpolation function while TET10 used quadratic order interpolation function (Wang et al., 2004). For the same mesh size, 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 computational time (CPU time). Figure 4 shows the von-Mises stresses contour for TET10 meshes element at a high load level. Table 2 shows that TET10 mesh predicts higher von-Mises stresses than TET4 mesh. Specifically, TET10 mesh predicts the highest von-Mises stresses is 301 MPa at mesh size 4 mm. The comparison was made by using different length of mesh. Figure 5 and 6 show the variation of stress and displacement over the global edge length and it can be seen that the TET10 is always captured the higher values.

Table 1: Mechanical properties of C-70 Steel

Properties	**Value**
Tensile strength, σ_{UTS} (MPa)	965.8
Yield strength(0.2% offset), σ_{YS} (MPa)	573.7
Young's modulus, E (GPa)	211.5
Percent elongation, %EL (%)	27%
Percent reduction in area, %RA (%)	25%

Table 2: Comparison between TET4 and TET10 with stress and displacement.

	von Mises (MPa)		**Displacement (mm)**	
Mesh Size (mm)	**TET4**	**TET10**	**TET4**	**TET10**
16	22.4	107.0	0.060	0.351
12	32.7	142.0	0.111	0.475
8	49.5	261.0	0.145	0.604
4	67.6	301.0	0.201	0.850

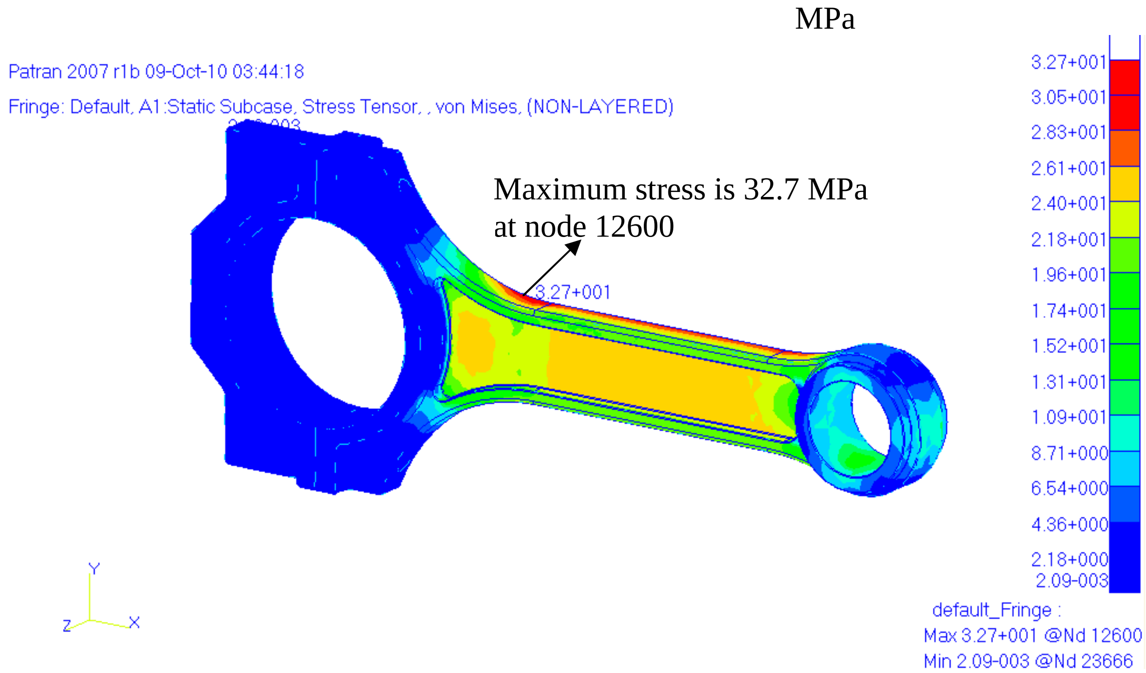


Figure 4: Von-Mises stresses contour for TET4 tensile load at pin end.

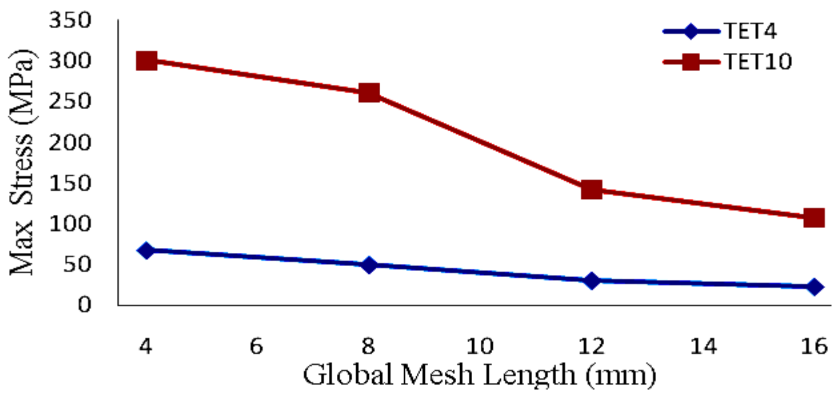


Figure 5: Comparison between TET4 and TET10 on von-Mises stress

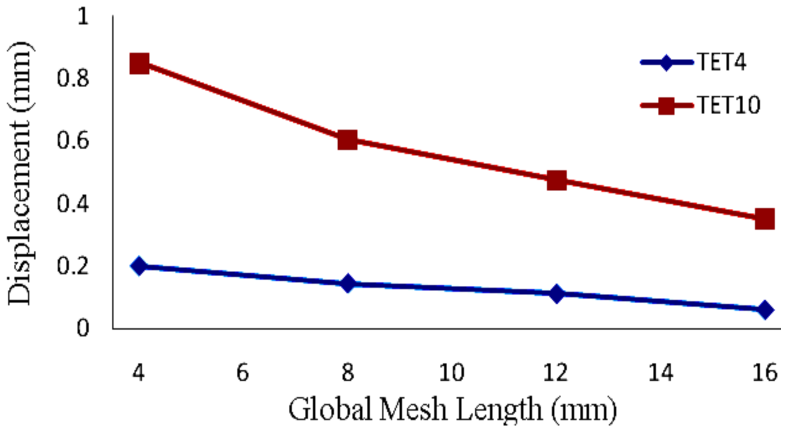


Figure 6: Comparison between TET4 and TET10 displacement

Identification of 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. It can be seen from Figure 7 that mesh size of 4 mm is obtained largest stresses. The smaller size less than 4 mm do not implemented due to the limitation of 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, Tresca and maximum principal stresses.

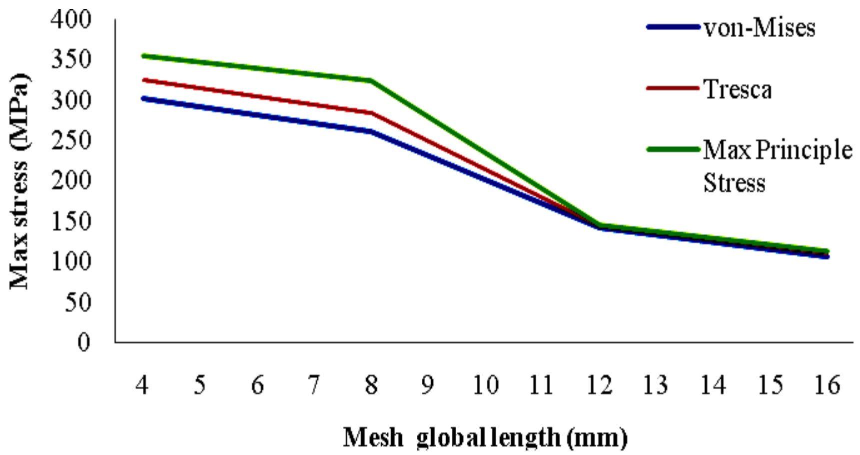


Figure 7: Stresses versus mesh size at critical location of for mesh convergence.

Optimization of Connecting Rod

The optimization of the connecting rod carried out using topology optimization technique. The optimization focused on the uncritical sections which need to be reduced. From the topology optimization, it is suggest the unnecessary shape and design of the connecting rod. The optimization results are tabulated in Table 3 and the objective function history are shown in Figure 8. The results of topology optimization of the connecting rod are shown in Figure 9. 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 8 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. Table 4 shows the comparison between initial and optimize designs on max principles stress and mass of the 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 10 shows the new design of the connecting rod and mass of the connecting rod is 0.464 kg compare to initial design 0.577 kg which is 11.7% lighter.

Table 3: Objective and maximum constraint from optimization of connecting rod

Cycle Number	Objective Function		Fractional Error	Maximum Constraint
	Approximate Optimization	Exact Analysis	Approximation	
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}

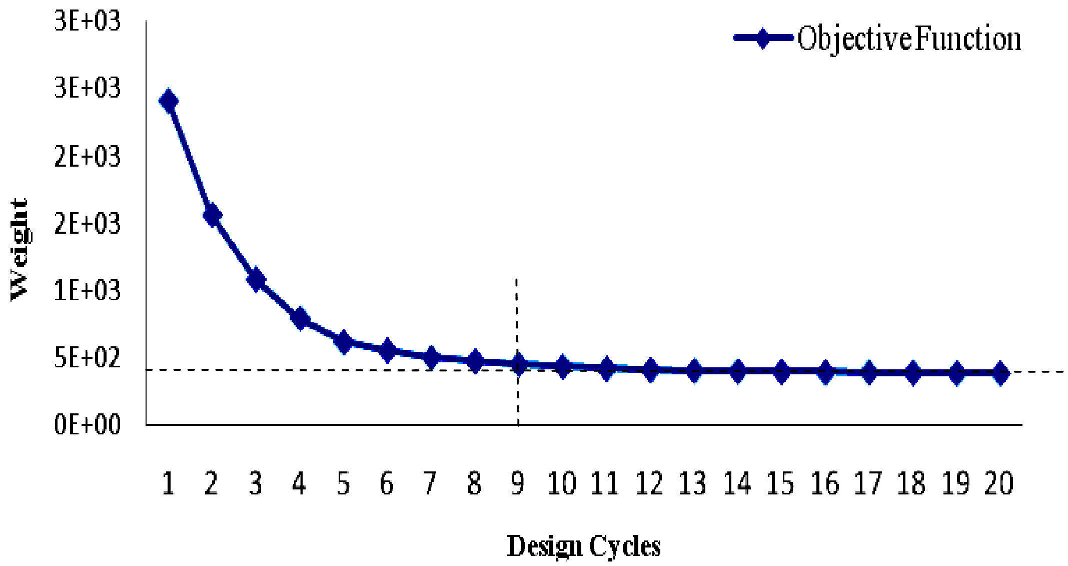
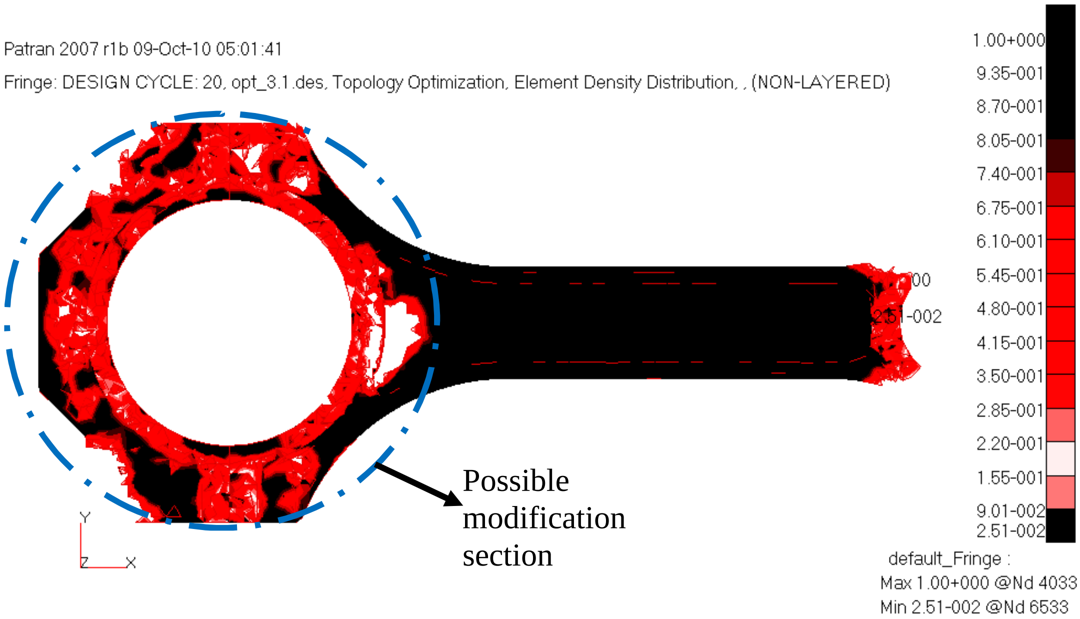
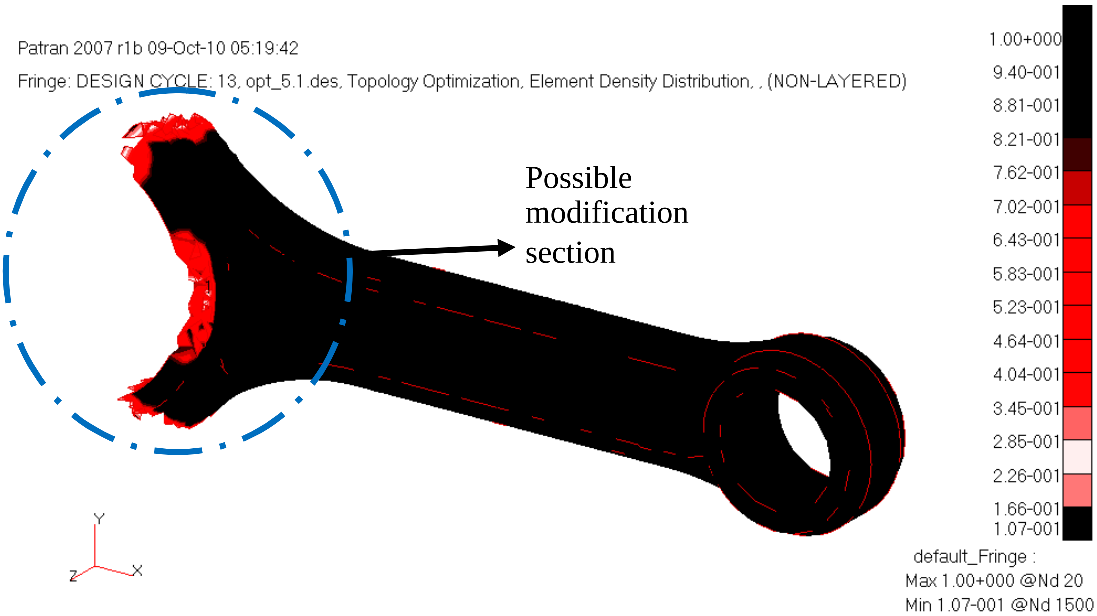


Figure 8: Objective function history.



(a)



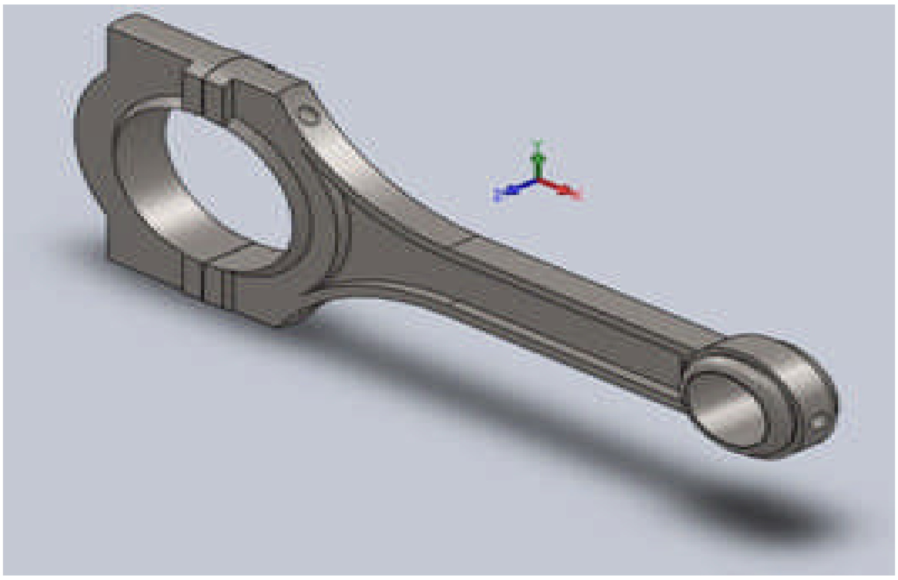
(b)

Figure 9: (a) Optimized connecting rod when force at crank end. (b) Optimized connecting rod when force at pin end.

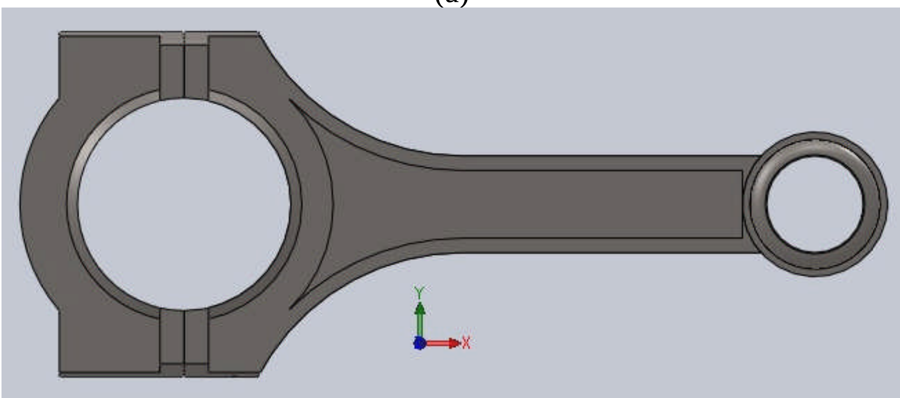
Table 4: Comparison between initial and optimize designs on stress and mass.

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)



(b)

Figure 10: (a) Isometric 3D view of optimized design. (b) Front 3D view of new design.

CONCLUSION

The modeling of connecting rod and FE Analysis has been presented. Topology optimization were analyzed to the connecting rod and according to the results, it can be concluded that the weight of optimized design is 11.7% lighter and maximum stress also predicted lower than the initial design of connecting rod. The results clearly indicate that the new design much lighter and has more strength than initial design of connecting rod. Material optimization approach will be considered for future research.

ACKNOWLEDGEMENT

The authors would like to thank the University Malaysia Pahang for provides laboratory facilities.

REFERENCES

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.

Rahman, M.M., Ariffin, A.K., Abdullah, S., Noor, M.M. and Bakar, R.A. 2009b. 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., Ariffin, A.K., Abdullah, S., Noor, M.M., Bakar, R.A. and Maleque, M.A. 2008b. Finite element based fatigue life prediction of cylinder head for two-stroke linear engine using stress-life approach. Journal of Applied Sciences, 8(19):3316-3327.

Rahman, M.M., Ariffin, A.K., Jamaludin, N. and Abdullah, S. 2007. Effect of nitriding treatment on fatigue life for free piston linear engine component using frequency response method: a finite element approach. Structural Durability and Health Monitoring, 3(4): 197-209.

Rahman, M.M., Ariffin, A.K., Jamaludin, N., Abdullah, S. and Noor, M.M. 2008a. Finite element based fatigue life prediction of a new free piston engine mounting. Journal of Applied Sciences, 8(9):1612-1621.

Rahman, M.M., Ariffin, A.K., Rejab, M.R.M., Kadirgama, K., Noor, M.M. 2009c. Multiaxial fatigue behaviour of cylinder head for a free piston linear engine" Journal of Applied Sciences, 9(15): 2725 - 2734.

Rahman, M.M., Kadirgama, K., Noor, M.M., Rejab, M.R.M. and Kesulai, S.A., 2009a, 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.

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

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

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.