International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 10 (2018) pp. 7298-7304

Structure Topology Optimization of Internal Combustion Engine Connecting Rode Using Finite Element Analysis

Mahmoud Helal[1a*] , Ageel Alogla[1], Kh. Abdel-Aziz [1b], N. Fouda[a] and Elsayed Fathallah[c]

[1] Taif University, Engineering College, Mechanical Engineering Department, Taif, Post Code 888, KSA.
[a] Production and Mechanical Design Dept., Faculty of Engineering, Mansoura University, Mansoura, Egypt.
[b] Zagazig University, Faculty of Engineering, Materials Engineering Department, Zagazig, Egypt.
[c] Civil Engineering Department, M.T.C. Kobry Elkobba, Cairo 11787, Egypt.

** Corresponding Author*

ABSTRACT

The connecting rods are widely used in variety of internal combustion (IC) engine to transmit the thrust of the piston to the crankshaft and results into conversion of the reciprocating motion of piston to the rotational motion of crankshaft. The connecting rod has a complex geometry. Also, it faces a lot of tensile and compressive loads generated by mass and fuel combustion, during its life time. These loads results in axial and bending stresses. Bending stresses appear due to eccentricities, crankshaft, case wall deformation, and rotational mass force; therefore, a connecting rod must be capable of transmitting axial tension/compression and bending stresses caused by the thrust and pull on the piston and by the centrifugal force. The concept of structural optimization has been more and more widely accepted in many engineering fields. The structural optimization can result in a much more reasonable and economical structure design. Topology optimization is the technique that finds the optimal layout of the structure within a specified design domain. Topology optimization techniques are relatively inexpensive, easy to implement, and can serve as a good starting point for further design improvements and better material utilization. This paper presents the structure topology optimization of internal combustion engine connecting rode using finite element analysis. The optimization algorithm is implemented using interface between Matlab and ANSYS. The FEA results are used to estimate the deflection stresses under subjected loads. The design is showed that significant potential improvement in the design configuration of connecting rod. The final optimal topology configuration of theconnecting rode is shown and discussed.

Keywords: Finite element analysis, Connecting rod, Topology optimization, Matlab, ANSYS.

INTRODUCTION

Topology optimization has proven to be a powerful method for the conceptual design of structures and mechanisms. The topology optimization methods search for an ideal material distribution of a structure. The goal of the topology optimization is to determine the best distribution of a given sum of material in the design space until the extreme measure for a given objective is achieved. In other words, topology optimization can help in determining the general layout of a structure within a prescribed region for given design specifications. Topology optimization techniques are relatively inexpensive, easy to implement, and can serve as a good starting point for further design improvements and optimum material utilization. A significant weight reduction of the structure can often be achieved without greatly compromising the strength below an acceptable limit. The topology optimization technique is used to achieve the objectives of optimization which is to reduce the weight moving parts. There are also efforts to develop bio-integrated prosthetic devices to restore functions to individuals who lost a limb to improve functionality. A critical step in advancing this technology will be to securely and efficiently attach the device to remnant bones [1]. Recent advances in manufacturing have provided unprecedented opportunities for producing complex structures to meet the increasing demands for implants with customized mechanical performance. At the same time, topology optimization techniques have been developed to enable the internal architecture to be designed to achieve specified mechanical properties [2, 3].

Over the last three decades, many mathematical and heuristic optimization methods have been developed [4]. Topology optimization has recently experienced considerable development indicated by several different kinds of methods such as the material distribution approach [5], the homogenization method [4, 6], the level set method [7, 8], the genetic algorithm (GA) method [9-14] and so on. In particular, the solid isotropic materials with penalizations method (SIMP) has been very popular due to its concept and implementation simplicity. The key concept of the SIMP is to indicate the dependence of material properties upon discrete pseudo-densities using a nonlinear interpolation scheme [4, 13].

Connecting rod is one of the most important components of the whole engine assembly as it acts as a mediator between crankshaft and piston assembly. The Connecting rods are widely used in variety of internal combustion (ICEs). In the automobile industry connecting rod has high volume production. The Connecting rods are used to transmit the thrust of the piston to the crankshaft and results into conversion of the reciprocating motion of piston to the rotational motion of crankshaft.

The connecting rod has a complex geometry Also, it faces a lot of tensile and compressive loads generated by mass and fuel combustion, during its life time. These loads results in axial and bending stresses. Bending stresses appear due to eccentricities, crankshaft, case wall deformation, and rotational mass force; therefore, a connecting rod must be capable of transmitting axial tension/compression and bending stresses caused by the thrust and pull on the piston and by the centrifugal force. For quick, better and accurate analysis CAD and FEA have proved very useful. Two major problems that arise in last many years are pollution and fuel crisis. These two problems can be reduce by reduce the weight of the moving part of ICEs. Lighter connecting rods help to decrease lead caused by forces of inertia in engine as it does not require big balancing weight on crankshaft.

Shenoy and Fatemi [15], presented the finite element analysis procedure for connecting rod optimization for weight and cost reduction. An optimization study was performed on a steel forged connecting rod with a consideration for improvement in weight and production cost. Mirehei et al. [16] carried out the fatigue analysis of connecting rod. The connecting rod fatigue of universal tractor (U650) was investigated through the ANSYS commercial software application. Gangwani and Metkar [17], explained the computer aided modeling and finite element analysis connecting rod. Yang Kun et al. [18], carried out the computer aided numerical analysis upon the connecting rod bearing cave corroding fault. Xianjun et al. [19] carried out the Sensitivity Analysis and Optimization for Connecting rod of LJ276M Electronic Gasoline engine.

Mansour et al. [20], obtained the maximum stresses in different parts of tractor (Mf-285) connecting rods using finite element method. Pranav et al. [21], presented the Analysis & Optimization of Connecting Rod The main objective of this study was to explore weight reduction opportunities for production forged steel Connecting rod. Vivek et al. [22], dealt with the stress analysis of connecting rod by finite element method. The stress induced in the small end of the connecting rod are greater than the stresses induced at the bigger end, therefore, the chances of failure of the connecting rod may be at the fillet section of both end.

Sharma et al. [23], performed the static finite element method of the connecting rod using the software and the optimization was performed to reduce weight. Weight can be reduced by changing the material of the current forged steel connecting rod to crack able forged steel (C70). Kumar et al. [24], described the modeling and the analysis of connecting rod. The carbon steel connecting rod is replaced by aluminum boron carbide connecting rod. Aluminum boron carbide is found to have working factor of safety is nearly to the theoretical factor of safety, also, increases the stiffness by 48.55% and to reduce stress by 10.35%. Ramani et al. [25], investigated the stress developed at different parts of connecting rod using CAE software. The maximum stress developed was between pin end and rod linkages and between bearing cup and connecting rod linkage. The maximum tensile stress developed in lower half of pin end and between pin end and rod linkage. It was suggested that the results obtained can be useful to bring about modification in design of connecting rod.

Bansal et al. [26], the dynamic simulation was conducted on a connecting rod made of aluminum alloy using FEA. The analysis of connecting rod was performed under dynamic load for stress analysis and optimization. Dynamic load analysis was performed to determine the in service loading of the connecting rod and FEA was conducted to find the stress at critical locations. Leela and Venu [27], demonstrated that the factor of safety (from Soderberg's), stiffness of forged steel is more than the existing carbon steel found and the weight of the forged steel material is less than the existing carbon steel.

Kuldeep and Faheem [28], described that the weight can be reduced by changing the material of the current Al360 connecting rod to hybrid AlFASi Composite. The aluminum composite connecting rod is 43.48% lighter than the Al360 connecting rod and much stiffer. Gupta and Nawajish [29], compared three materials used for manufacturing of connecting rod these are Al360, magnesium alloy and beryllium alloy. The modeling and analysis of connecting rod was done. Comparing the different results obtained from the analysis, it is concluded that the stress induced in the beryllium alloy is less than the aluminum and magnesium alloy. Marthanapalli [30], a connecting rod for a 150cc engine has been modeled in 3D modeling software Pro/Engineer. Chauhan et al. [31], have concluded that the basic study and research work is done in improving in material of connecting rod.

This paper presents the optimal topology of the connecting rod using the interface between Matlab and Ansys to find the optimal material distribution in order to improve the performance internal combustion engine. The structure of connecting rod will be modeled utilized finite element analysis software. Finite element modeling and analysis were performed using ANSYS finite element commercial software. The mesh convergence analysis will be considered to select the best mesh for the analysis. The topology optimization technique will be used to achieve the objectives of optimization which is to reduce the weight of the connecting rod. Linear static analysis will be carried out to obtain the von Misses stress; von Misses strain and displacement.

STRUCTURE TOPOLOGY OPTIMIZATION

Structure topology optimization methods enable designers to find the best structural layout for a required structural performance. Generally, the structure topology optimization methods [4] search for an ideal material distribution of a structure, such that the objective function is optimized. The optimization formulation is as follows:

A generic optimization problem can be written in the form [9]:

Objective function:

Minimize $f(x),\ x \in \Omega$ (1)

Limitation:

$c(x) \geq 0$ (2)

$0 \leq x_i \leq 1$, i=1 to N (3)

where x_i represent the i^th normalized density variable, N is the number of finite elements in the structure. Ω is the design space, f (x) is the objective function, and c(x) are the constraints of the optimization. The functions f(x) and c(x) are usually computed via a suitable numerical technique while the optimization algorithm iteratively looks for the best configuration. The density variable, *x*, was varied between 0 and 1, where x_i close to 0 represents the material to be removed; x_i close to 1 represents the material that should be retained.

In topology optimization, since gradient information is readily available from the Finite Elements analyses, a large number of variables can be easily handled, accepting that a gradient-based optimization algorithm is adopted.

The most popular methods for topology optimization are the homogenization methods. In particular, in this paper the solid isotropic material with penalization methods, SIMP, is employed. The theory of this method is described extensively by Bendsøe and Sigmund [5] and Bendsøe and Kikuchi [6]. The main scope of the method is to find the optimum material distribution in a structure. Finite Element analyses are performed assuming as a parameters vector the element-by-element relative material density, which is allowed to vary with continuity:

The density of the i^{th} element is given by:

$\rho_i(x_i) = x_i \rho^*$ (4)

where ρ* is the full density of the material. The material density and the material stiffness are correlated. According to the homogenization method, the intermediate density material is treated as a homogeneous porous material with micro scale voids, usually assumed to be rectangular. The local stiffness tensor of such a material is computed rigorously as a function of the holes shape, size, and orientation. This can involve a rather complicated mathematical procedure.

The SIMP method, which is adopted in this paper, is simpler, and assumes that the stiffness of the i^{th} element is given by:

$E_i(x_i) = x_i^p E^*$ (5)

where E^* is the full stiffness of the isotropic material.

Two parameters control the behaviour of the algorithm: the penalty factor p, and the sensitivity filter r. The penalty factor $p \geq 1$ appears in equation (5). Its role is to make intermediate densities unfavorable in the optimized solution. Setting the filter $r \geq 1$ the sensitivity of each element is averaged with the sensitivities of its surrounding elements within a radius equal r times the average mesh size, thus preventing the phenomenon of checkerboarding.The sequential linear programming (SLP) [33] has been applied to this study.

INTERFACES BETWEEN MATLAB AND ANSYS

The basic requirement for integrating an application is that can be run in the batch mode and has accessible model and result files, which ANSYS all complies with .ANSYS Parametric Design Language (APDL) makes the modifying parameters especially easy. There are two possible approaches to change parameters in an ANSYS input file. One is the use of a separate parameter file, which contains only the design variables. This file is rewritten each time the design variables are changed and then read in during the preprocessing of the model input file. Another approach is to directly modify the parameters in the ANSYS input file by search and replace commands provided by an editor. The execution of ANSYS inside MATLAB is simply done by system level commands.

MATLAB CALLING ANSYS

Matlab call Ansys statement is as:
system('C:\Ansys\v181\ANSYS\bin\intel\ansys1231 -b -p ansys-roduct-feature -i input file -o out file.bat')

Where -b = batch mode, -p = product: ANSYS/Mechanical for example, -i = your input file: file.inp for example, -o = the output file of ansys: file.out for example, ansys-product-feature Ansys, -j = jobname, -m = memory size, -db = database size -g = graphic mode, -l = language, ...etc.

ANSYS PARAMETRIC DESIGN LANGUAGE (APDL)

The finite element method has been largely used to analyze various mechanical systems. The finite element analyses were implemented using the ANSYS Parametric Design Language (APDL) [34, 35].

The objective of the present work is the design of internal combustion engine connecting rod. For setting up a topology optimization problem, both the domain of the structure (design space) and the governing parameters, i.e. the objective and the constraints of the optimization, have to be defined. The domain of the structure should be as wide as possible, according to the project restriction, in order to guarantee the maximum freedom to the optimization process in choosing the optimal material distribution. **Fig. 1** shows the flow chart of the structural optimization processes, whereas **Fig. 2** shows the schematic diagram of the general topological structure design model.

The adoption of a geometry as simple as possible is very important to achieve a regular high quality mesh. Sigmund and Petersson [36] and Zuo et al. [37], studied the mesh dependency of topology optimization results noting that different structural solutions can be obtained if different mesh size and quality are considered. Wankhade And Zolekar [38], used a more complex geometry of the piston, but in this case the ring belt belonged to the non-design space, so that this feature did not aggravate the analysis.

Fig. 3 shows full domain of the connecting rod, where R_1 and R_2 are the radii of pin end and the crank end, respectively. L and W and the length and the width of the connect rod, respectively. C is the center distance of the connecting rod.

According to the symmetry only half model is constructed as shown in **Fig. 4**.

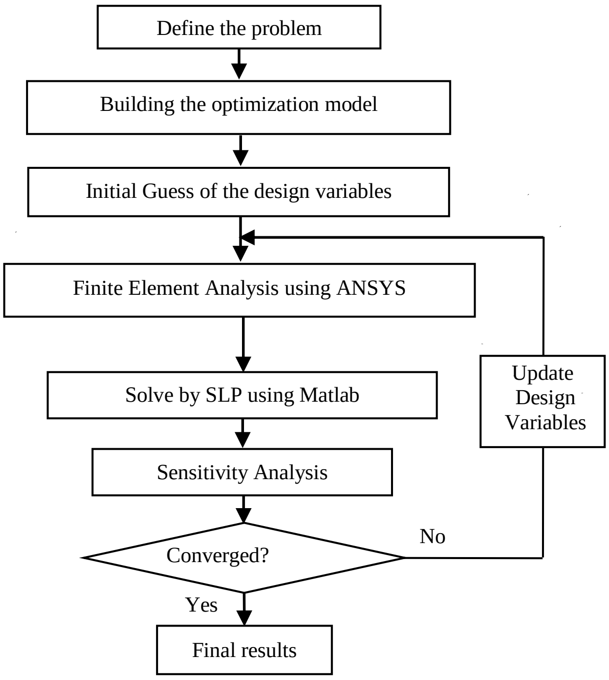

Figure 1: Structural topology optimization flow chart.

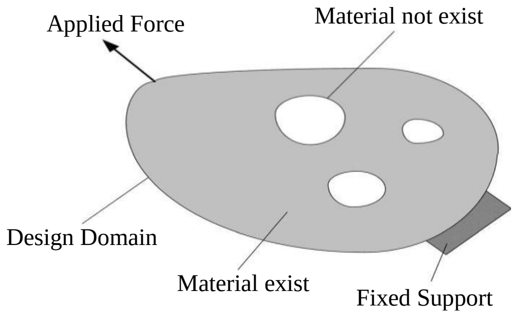

Figure 2: General schematic diagram of the topological structure design.

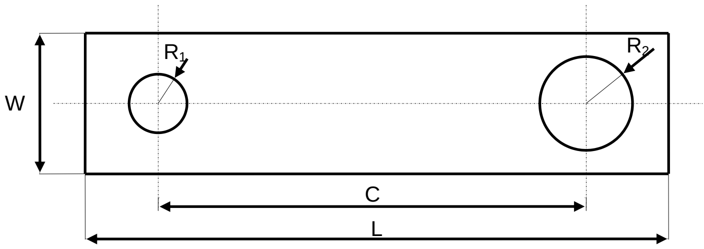

Figure 3: Full model of the connecting rod.

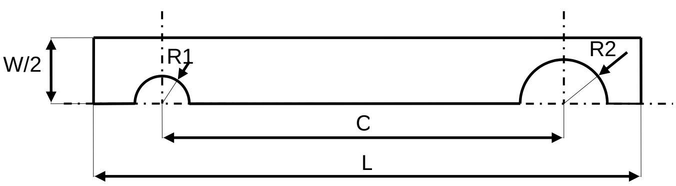

Figure 4: Half model of the connecting rod.

A two-dimensional finite element analysis model using ANSYS program is constructed for the propose domain. **Table 1** provides the complete material property of the high strength steel which is used in the analysis of the connecting rod.

Table 1: Material properties for high strength steel.

Properties	Values
Young's Modulus	210GPa
Poisson Ratio	0.3
Density	7890Kg/m^3
Yield Strength	680MPa
Ultimate Tensile Strength	760MPa

The recommended element type for the two-dimensional is PLANE82 (an 8-node element and is defined by eight nodes having two degree of freedom at each node) [34, 35], as shown in **Fig. 5**. Two load cases are defined, the first is the compressive force and the second is the tensile force. The forces are applied to the small pin hole (piston pin) considering a uniformly distributed load along 120º region of the pin hole, as in **Fig. 6.**

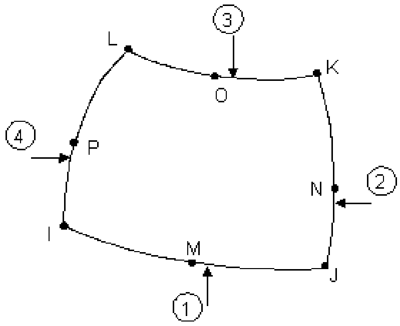

Figure 5: PLANE82 (8-node element Structural Solid).

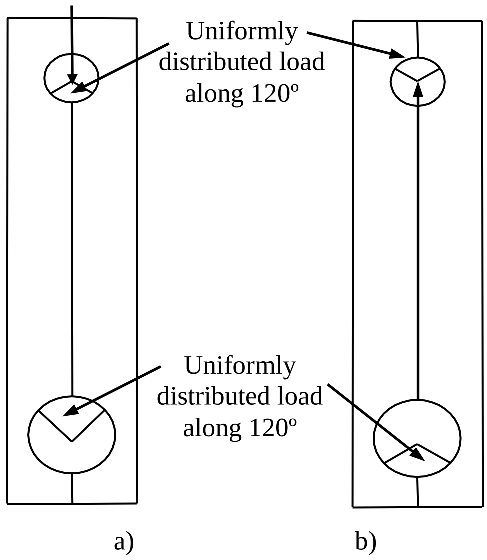

Figure 6: The Loading and boundary conditions: a) compressive load, b) tensile load.

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 10 (2018) pp. 7298-7304

For each load case a displacement restraint is applied to the big pin hole (crank pin) to the nodes along 120º region. For compression the forces are applied to the lower part of the small pin hole and restraints to the upper part of the big pin hole. For tension the opposite is done. **Fig. 6** shows the boundary conditions of the connecting rod applied to the design domain the two dimensional FE model.

RESULTS

The connecting rod is one of the most important components in the internal combustion engine. Therefore, the initial design is compared to other design before performing the optimization. A simple two-dimensional model of connecting rod was developed using ANSYS software and finite element model was created using PLANE82 as shown in **Fig. 7** (15912 nodes and 5139 elements).

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. 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.

Fig. 8 and **Fig. 9** show the final optimal topological optimization configuration for connecting rod under compassion and tension loads, respectively. **Fig. 10** and **Fig. 11** illustrate full two-dimensional optimal topological optimization for connecting rod under compassion and tension loads, respectively. **Fig. 12** shows the final three-dimensional model for the connecting rod.

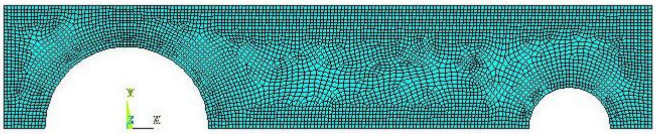

Figure 7: Initial two-dimensional mesh for connecting rod.

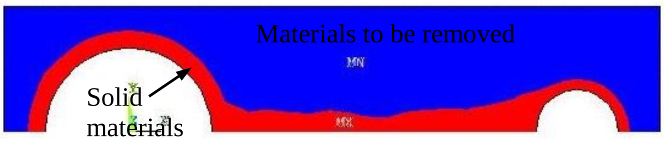

Figure 8: The final optimal topological configuration for connecting rod under compression load.

Figure 9: The final optimal topological configuration for connecting rod under tension load.

Fig. 13 and **Fig. 14** show von-Mises stress distribution of the three-dimensional optimal topological optimization of connecting rod under compassion and tension loads, respectively. **Table 2** shows the von-Mises stress and displacement performance for the connecting rod. The results clearly indicate that the new design much lighter and has more strength.

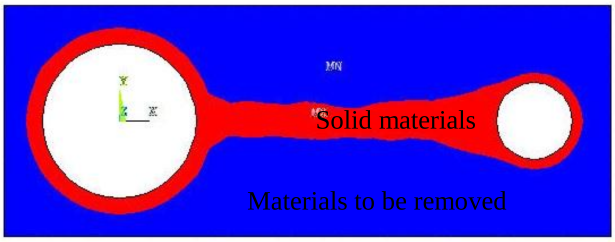

Figure 10: The full two-dimensional optimal topological optimization for connecting rod under compassion load.

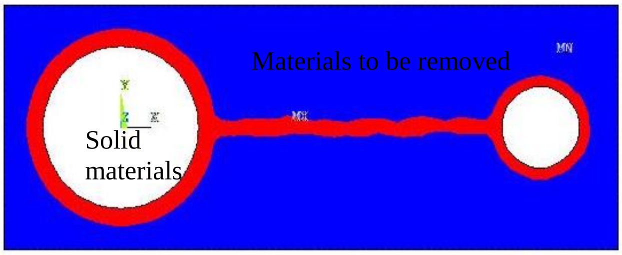

Figure 11: The full two-dimensional optimal topological optimization for connecting rod under tension load.

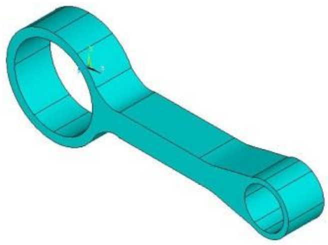

Figure 12: the final three dimensional model for the connecting rod.

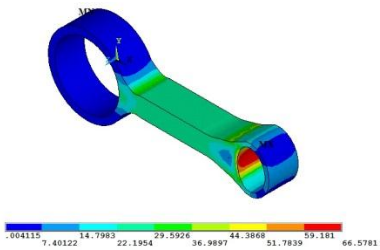

Figure 13: von-Mises stress distribution of the three-dimensional optimal topological optimization of connecting rod under compassion.

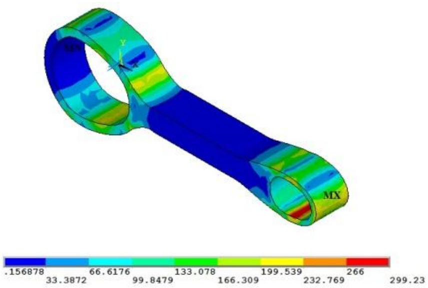

Figure 14: von-Mises stress distribution of the three-dimensional optimal topological optimization of connecting rod under tension load.

Table 2: Maximum von-Mises stress and maximum displacement.

Loads	Maximum von-Mises stress, MPa	Maximum displacement, mm
Compression Load	66.58	0.16
Compression Load	299.23	0.13

CONCLUSIONS

Structure topology optimization methods enable designers to find the best structural layout for a required structural performance. In this work, the structure topology optimization formulation for designing the connecting rod is done in order to improve the performance internal combustion engine. The topology optimization procedure is implemented using the interface between Matlab and the commercial software ANSYS. The problem is parameterized with the SIMP model, and the optimization is solved by the optimality criteria method. In particular, the presented methodology aimed at finding more efficient layout solutions for the piston framework feasible with Additive Manufacturing techniques. Special care has to be taken in setting up the optimization process since the choice of the constraints, i.e. the performance targets, of the mesh size and quality and of the optimization parameters directly affect the outcome of the process. The optimal configuration for the connecting rod is illustrated. Finite element displacement analysis and von-Mises stress distribution for the connecting rod are shown. From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to optimize, this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is also very less. So, the material from low stressed region is can be removed without affecting its strength and is within the yield strength. The results clearly indicate that the new design much lighter and has more strength than initial design of connecting rod.

ACKNOWLEDGEMENT

The authors are grateful to the Taif University for providing all the required facilities to carry out the present research. The present research was supported by The Council of Scientific Research (in the project No. 1-438-5949), Taif University, KSA.

REFERENCES

[1] J.P. Long, S.J. Hollister, and S.A. Goldstein, "Paradigm for the development and evaluation of novel implant topologies for bone fixation: In vivo evaluation," J. of Biomechanics 2012 45(15): 2651–2657.

[2] X. Wang, S. Xu, S. Zhou, W. Xu, M. Leary, and P. Choong, "Topological design and additive manufacturing of porous metals for bone scaffolds and orthopaedic implants: A review," Biomaterials 2016 83: 127–141.

[3] H.A. Almeida, and P.J. Bártolo, "Topological optimization of scaffolds for tissue engineering" Procedia Engineering 2013 59: 298–306.

[4] M. P. Bendsoe, and O. Sigmund "Topology Optimization, Theory, Methods and Applications" New York, Springer-Verlag (2004)

[5] M. P. Bendsoe, and O. Sigmund "Material interpolation schemes in topology optimization" Archives of Applied Mechanics 1999 69: 635–654.

[6] M. P. Bendsoe, and N. Kikuchi "Generating optimal topologies in structural design using a homogenization method" Computer Methods in Applied Mechanics and Engineering 1988 71: 197–224.

[7] Y. L. Mei, and X. M. Wang "A level set method for structural topology optimization and its applications" Advances in Engineering Software 2004 35: 415–441.

[8] Z. Luo, and L.Y. Tong "A level set method for shape and topology optimization of large-displacement compliant mechanisms" International Journal for Numerical Methods in Engineering 2008 76: 862–892.

[9] K. J. Lu, and S. Kota "Compliant mechanism synthesis for shape-change applications preliminary results" Proceedings of the 2002 SPIE Modeling Signal Processing and Control Conference, San Diego, 2002, p.161–172

[10] K.J. Lu, and S. Kota "Design of compliant mechanisms for morphing structural shapes" Journal of Intelligent Materials Systems and Structures 2003 14: 379–391.

[11] Y.M. Xie, and G.P. Steven "A simple evolutionary procedure for structural optimization" Computers and Structures 1993 49: 885–896.

[12] Y.M. Xie, and G.P. Steven "Evolutionary Structural Optimization, London, Springer (1997)

[13] G. Rozvany "A critical review of established methods of structural topology optimization" Structural and Multidisciplinary Optimization 2008 37: 217–237.

[14] U. Larsen, O. Sigmund, and S. Bouwstra "Design and fabrication of compliant micro-mechanisms and structures with negative poisson's ratio" Journal of Microelectromechanical Systems 1997 6: 99–106.

[15] S. Shenoy and A. Fatemi "Connecting rod optimization for weight and cost reduction" SAE International (2005)

[16] M. HedayatiZadeh, A. Jafari, and M. Omid, "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS)" Journal of Agricultural Technology 2008 4: 21–27.

[17] G. Gangwani, and R.M. Metkar "Computer aided modelling and finite element analysis of connecting rod fouling on camshaft of di engine of mahindra and mahindra tractor" Second International Conference on Emerging Trends in Engineering and Technology, ICETET- 09.

[18] Y. Kun, O. Guangyao, and Y. Lina "Computer aided numerical analysis upon the connecting rod bearing cave corroding fault" WASE International Conference on Information Engineering (2009)

[19] X. Hou, C. Tian, D. Fang, F. Pengand, and F. Yan, "Sensitivity analysis and optimization for connecting rod of lj276m electronic gasoline engine" IEEE 2009, pp 978.

[20] M. Rasekh, M. Asadi, A. Jafari, and K. Kheiralipour "Obtaining maximum stresses in different parts of tractor (Mf-285) connecting rods using finite element method" Australian Journal of Basic and Applied Sciences 2009 3: 1438–1449.

[21] P. Charkha and S. Jaju "Analysis & Optimization of Connecting Rod" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09.

[22] V. Pathade, B. Patle, and A. Ingale, "Stress Analysis of I.C. Engine Connecting Rod by FEM" Int. J. of Engineering and Innovative Technology 2012 1(3): 12–15.

[23] P. Sharma, and B. Rajendra "fatigue analysis and optimisation of connecting rod using finite element analysis" Int. J. of advance research in Science and Engineering 2012 1(I): 3367–3371.

[24] K. Sudershan, K. Tirupathi, and S. Hussan "Modeling and analysis of two Wheeler connecting rod" Int. J. of Modern Engineering Research 2012 2(5): 3367–3371.

[25] H.B. Ramani, N. Kumar, and P.M. Kasundra, "Analysis of connecting rod under different loading condition using ANSYS software" International Journal of Engineering Research and Technology 2012 1(9): 1–6.

[26] R. Bansal "Dynamic simulation of connecting rod made of aluminium alloy using finite element analysis approach" IOSR Journal of Mechanical and Civil Engineering 2013 5(2): 1–5.

[27] L. Vegi, and V. Vegi, "Design and analysis of connecting rod using forged steel" International Journal of Scientific & Engineering Research 2013 4(6): 2081–2090.

[28] B. Arun, and M. Faheem "Analysis and optimization of connecting rod using AlFASiC composites" International Journal of Innovative Research in Science, Engineering and Technology 2013 2(6): 2480–2487.

[29] Gupta, and M. Nawajish "Design and analysis of two wheeler connecting rod using different materials" International Journal of Advance Research In Science And Engineering 2014 3(1): 390–395.

[30] M. Priya "Designing and modeling of connecting rod for four stroke single cylinder engine with different cross sections" International Journal of Core Engineering & Management, NCETME, College of Engineering and Technology (2017)

[31] A. Chauhan, H. Saxena, and N. Mehta "A Review on FEA of connecting rod of I.C engine " International Journal of Innovative" Research in Science, Engineering and Technology, 2017 6(3): 3712–3716

[32] M. Cavazzuti, A. Baldini, E. Bertocchi, D. Costi, E. Torricelli, and P. Moruzzi. High performance automotive chassis design: a topology optimization based approach. Struct Multidisc Optim 2011 44: 45–56.

[33] P. Venkataraman "Applied Optimization with Matlab Programming" Wiley, New York (2002)

[34] ANSYS Inc.: ANSYS Theory Reference Release 18.0 (2017)

[35] M. Erdogan, and G. Ibrahim "The Finite Element Method and Applications in Engineering Using ANSYS, New York, Springer (2006)

[36] O. Sigmund, and J. Petersson "Numerical instabilities in topology optimization: a survey on procedures dealing with checkerboards, meshdependencies and local minima" Structural Optimization 1998 16: 68–75.

[37] K. Zuo, L. Chen, Y. Zhang, and J. Yang "Study of key algorithms in topology optimization" Int J Adv Manuf Technol. 2007 32: 787–796.

[38] L.N. Wankhade, and V. Zolekar. Finite element analysis and optimization of i.c. engine piston using RADIOSS and OptiStruct. Altair Technology Conference, India 2013 p. 1–8.