Vol. XI & Issue No. 4 April - 2018
INDUSTRIAL ENGINEERING JOURNAL

DESIGN OPTIMIZATION OF HYDRAULIC LIFT CONNECTING ROD

Ganesh S. Shetiye
Bharat Kungwani

ABSTRACT

The presented work focusses on the design optimization of hydraulic lift connecting rod by the implementation of Finite Element Analysis. The objective is achieved by optimizing the profile of the stem region of the component to reduce weight and hence the overall cost. The optimization was done as per the safety considerations and Von Mises Criterion was used for validation of design. The work resulted in considerable weight reduction of the connecting rod without affecting other critical parameters.

1. INTRODUCTION

One of the major equipment required in the agricultural industry is a tractor. The success of the tractor is largely based on the variety of tools that it can be coupled with such as a plow, tiller and rake. These tools are mounted and controlled by the use of a hydraulic lift installed at the rear end of the tractor [1]. There is a high magnitude of forces involved in this apparatus owing to the considerable loads acting. The lift, being the link between the tractor and the tool, is a critical part and therefore it is needed that its components be designed keeping long-life, cost effectiveness and weight considerations in mind. The force transmission from the piston of the lift to the actuating mechanism is through the connecting rod. Ever since the conception of the lift, the shape of the connecting rod has not been drastically changed. It can be described as having a cylindrical stem with spherical or hemispherical ends depending on the manufacturer. One end provides the bearing surface for the piston whereas other side has a hole and is connected to the actuating mechanism using a pin joint. The important properties required in a connecting rod are rigidity, hardness for the bearing end and minimum weight.

The primary manufacturing method employed in the production of connecting rods is forging. Forging is preferred over casting due to the realignment of the internal grain structures that takes place which results in increased strength characteristics [2].

This paper focuses on the re-design and optimization of connecting rod by changing some dimensional parameters subject to various constraints such as the load applied, allowable stresses, critical dimensional constraints, manufacturing considerations such as tooling and die life and the safety requirements. The objective was the re-designing of the stem region to save weight. Shape optimization approach was used and Finite Element Method (FEM) was applied for validation of the design. SolidWorks 2016 was used for the modelling of the component and ANSYS Student 17.0 was used for the Finite Element Analysis (FEA) of the same.

2. RELATED WORK

Significant work has been done in the field of design of Internal Combustion (I.C.) engine connecting rods whereas the design of hydraulic lift connecting rod has been relatively untapped. The function and operational requirements of both these components is similar and therefore works in the design of IC engines connecting rod can be used as references for this work.

For the optimization of the design of the connecting rod, Seraget al [3] developed an approximate mathematical model taking weight and cost of the connecting rod as objective functions along with the constraints. A Geometric Programming technique was used to achieve the optimization. Constraints were imposed on the bearing pressure at the crank and piston pin ends and the compressive stress developed without addressing the fatigue factor. The cost function expressed with the geometric parameters in some geometric form.

A three-dimensional finite element analysis of a high-speed diesel engine connecting rod was performed by Webster et al [4]. The maximum compressive load was measured experimentally, and the inertial load of the piston assembly mass was taken as the maximum tensile load. These loads were taken into consideration for the analysis and the load distributions on the crank end and the piston end were also determined experimentally. The modelling of the connecting rod cap was done separately whereas the bolt pretension was modelled using beam elements and multi point constraint equations.

Design methodology in use at the Piaggio was reported by Hippoliti [5], incorporating an optimization session. The details of the optimization or the load under which optimization was performed, were however, not discussed. The authors applied two finite element procedures using 2D plane stress and 3D approach. The results were found to have shown good agreement with the experimental results and the optimization procedure was developed based on the 2D approach.

For theoptimization of the wrist pin end, Sarihan and Song [6] used a fatigue load cycle consisting of maximum tensile load equal to maximum inertia load and maximum compressive gas load as per the maximum torque. The maximum loads in the whole operating range of the engine were used. Modified Goodman equation withmean octahedral shear stress and alternating octahedral shear stress was used to design for fatigue. For optimization, an approximate design surface was generated and optimization of this design surface was performed. To obtain precise values, objective andconstraint functions were updated. The process was repeated till convergemponents of the stress. Their work resulted in reduction of the weight of the connecting rod by nearly 27%.

An approach akin to that applied in design and analysis of IC engine connecting rods [7-9] has been employed in this work for optimization of the hydraulic lift connecting rod.

3. OPTIMIZATION STATEMENT

The objective of this work was to minimize the weight of component acting under the specified loading conditions, thereby minimizing its effect onthe overall cost of the component. The optimization task subject to the constraints may be stated as:

Objective: To minimize cost and weight

Subject to:

- Load applied
- Maximum equivalent stress < Permissible equivalent stress
- Dimensional constraints (critical)
- Manufacturing constraints
- Safety considerations

4. METHODOLOGY

In order to achieve the optimization task, the re-designing of the stem region of the connecting rod was done while pertaining to the aforementioned criteria. Originally, the stem was of a circular profile. In the new design, the stem was made in the form an I section as it is highly effective in both shear and bending. The flanges resist the bending moments acting on the I shaped beam whereas the web resists the shear forces acting on it.The I section was adopted for its enhanced rigidity in reduced weight. The original design and proposed design are depicted in figures 1 and 2.

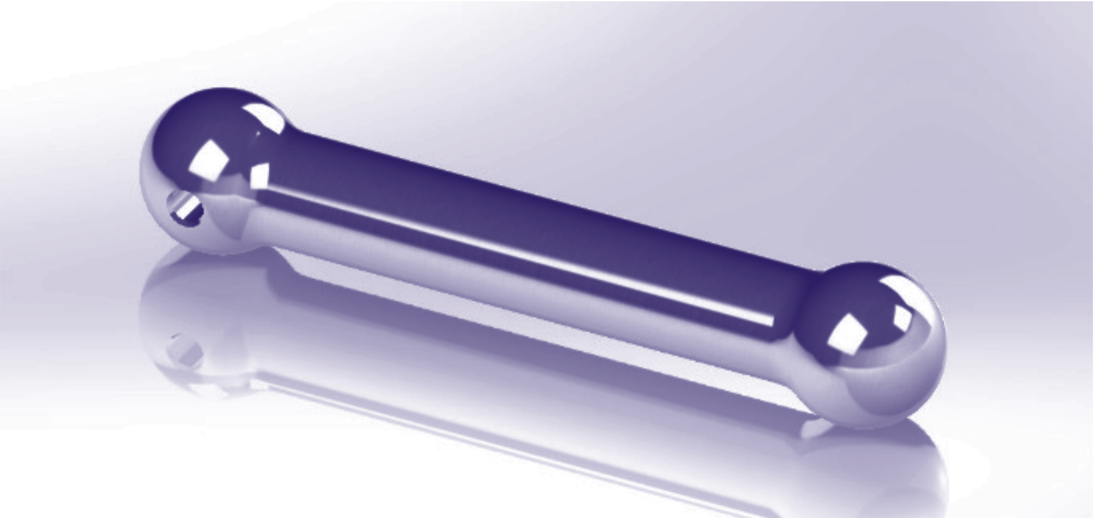

Fig. 1 – Original design

Fig. 2 – Proposed design. (a) Side View

(b) Isometric View

For validation of the design, FEA was carried out and the original and proposed designs were tested under the given loading conditions. Von Mises Yield Criterion was chosen as the judging parameter as the material being used is a ductile material and Von Mises Theory is better applicable for ductile materials than Maximum Shear Stress or Tresca Criterion. Both models were tested for Von Mises Stresses along with total deformation occurring in the component. The permissible stress as per the Von Mises criterion was found using equation (1). The analysis and the results are reported in the following sections.

$$\sigma_o = [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{1/2} \quad (1)$$

Where,

σ_o = Tensile Yield Strength (MPa)

σ_1 = Axial stress in X direction (MPa)

σ_2 = Axial stress in Y direction (MPa)

σ_3 = Axial stress in Y direction (MPa)

5. FINITE ELEMENT ANALYSIS

Finite element analysis of the connecting rod was done using ANSYS Student 17.0 software package. The model was created in SolidWorks 2016 and imported into ANSYS and the appropriate loading conditions were applied. The material used for the analysis was EN9 steel, an alloy widely used for forging applications. Properties of the material used in the analysis are listed in table 1. The meshed modelsare shown in figures 3 and 4. The results of the analysis of original and proposed model for the Von Mises Stresses and total deformation are represented in the following figures.

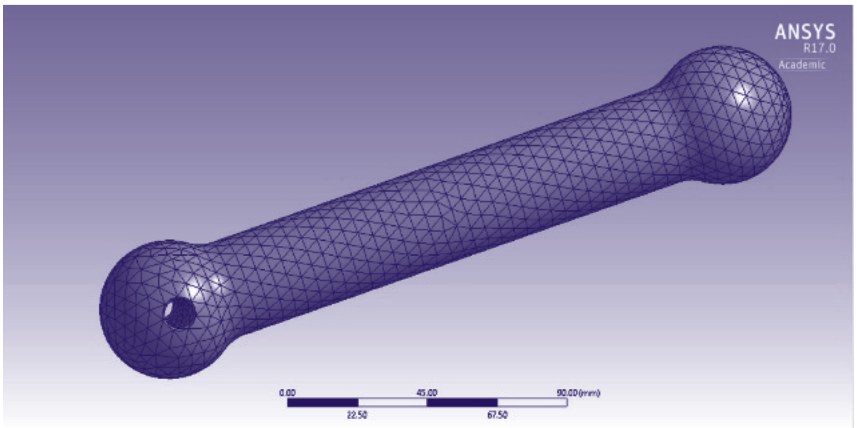

Fig. 3 – Meshing of original model

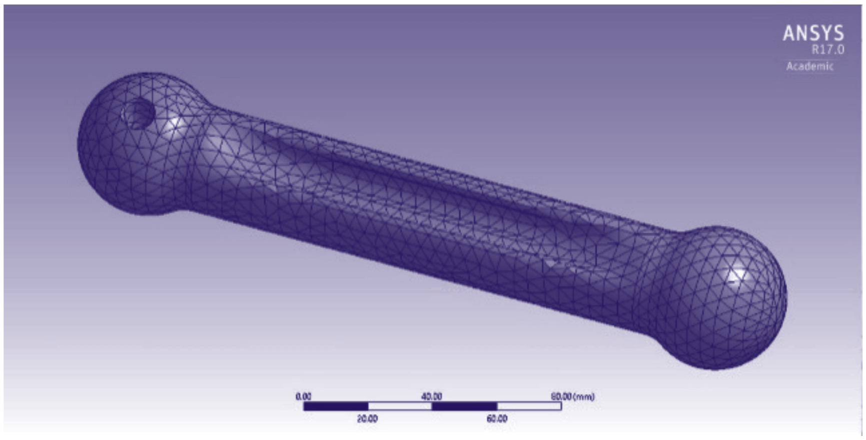

Fig. 4 - Meshing of proposed model

Property	Value
Density	7800 kg/m^3
Tensile Yield Strength	450 MPa
Compressive Yield Strength	450 MPa
Young's Modulus	2.06 E+05MPa
Poisson's Ratio	0.3
Tensile Ultimate Strength	930 MPa

Table 1:Material Properties of EN9

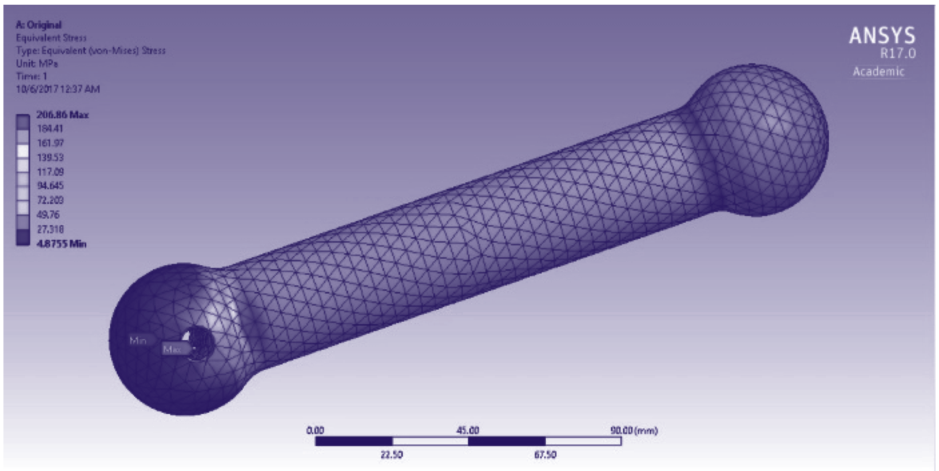

Fig. 5 – Equivalent (Von Mises) Stresses in original model

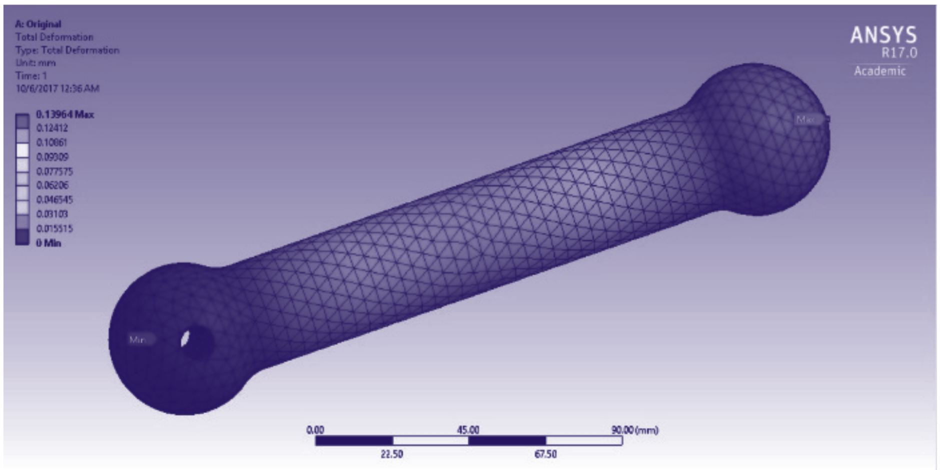

Fig. 6 – Total Deformation in original model

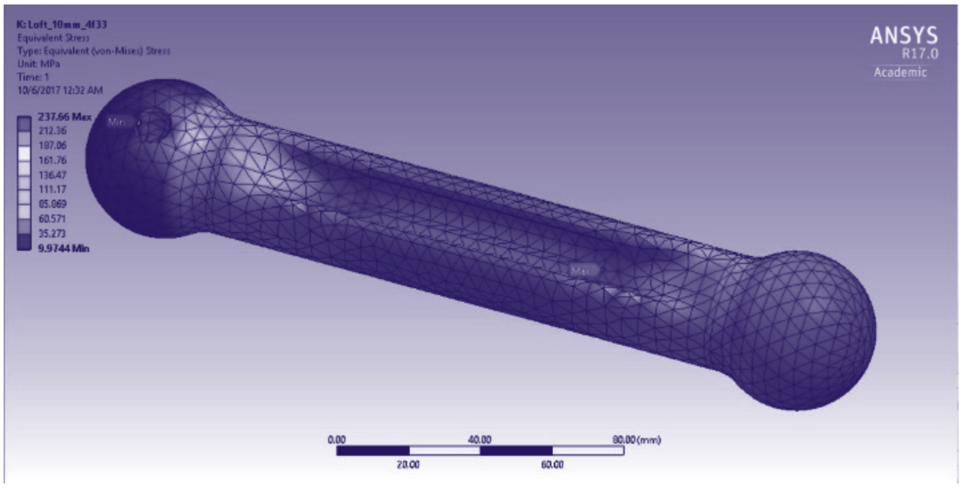

Fig. 7 – Equivalent (Von Mises) Stresses in proposed model

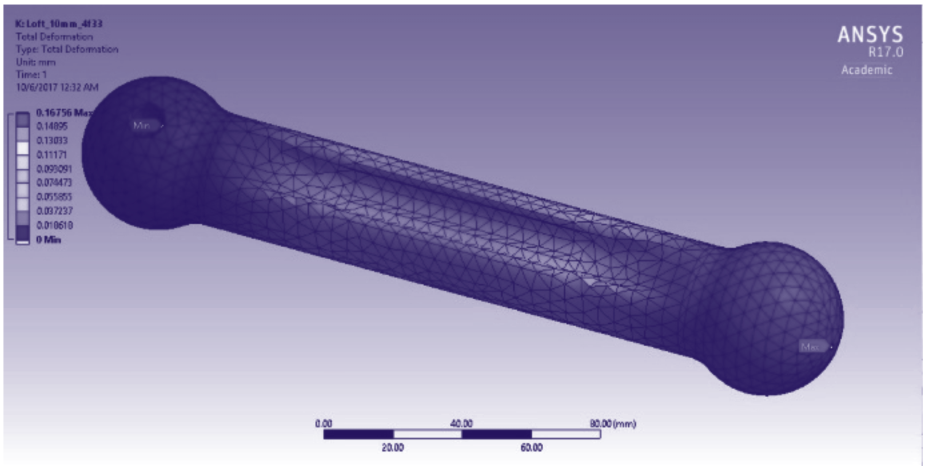

Fig. 8 – Total Deformation in proposed model

6. RESULTS

The maximum permissible Equivalent (Von Mises) stress was evaluated to be 259 MPa. The maximum equivalent stress and maximum total deformation in the original design were found to be 206.86 MPa and 0.14 mm respectively (fig. 5 and fig 6). The maximum equivalent stress in the proposed design was obtained to be 237.66 MPa and was well within permissible limits (fig. 7). The maximum total deformation in the proposed design is 0.17 mm (fig. 8). Weight reduction of 20% was observed in the proposed design.

7. CONCLUSION

The proposed design was safe under the given working conditions. It adhered to the dimensional constraints and safety considerations. The objective of the optimization task was achieved and there was significant weight reduction achieved resulting in lowering of overall cost of component. Critical regions were identified where the I section stem connects to the spherical ends. The manufacturing of the re-designed connecting rod can be accomplished by simply addinginserts to the existing die, thus there is no excessive cost of tooling involved. The design was so made as to maximize the die life

REFERENCES:

1. *Wilson, R. (1973). U.S. Patent No. 3,753,467. Washington, DC: U.S. Patent and Trademark Office.*
2. *Black, J. T., &Kohser, R. A. (2017). DeGarmo's materials and processes in manufacturing. John Wiley & Sons.*
3. *Serag, S., Sevien, L., Sheha, G., & El-Beshtawi, I. (1989). Optimal design of the connecting-rod. Modelling, Simulation and Control, B, 24(3), 49-63.*
4. *Webster, W. D., Coffell, R., & Alfaro, D. (1983). A three dimensional finite element analysis of a high speed diesel engine connecting rod (No. 831322). SAE Technical Paper.*
5. *Hippoliti, R. (1993, December). FEM method for design and optimization of connecting rods for small two-stroke engines. In Small Engine Technology Conference (pp. 217-231).*
6. *Sarihan, V., & Song, J. O. (1990). Optimization of the wrist pin end of an automobile engine connecting rod with an interference fit. Journal of Mechanical Design, 1121407.*
7. *Pathade, V. C., Patle, B., &Ingale, A. N. (2012). Stress Analysis of IC Engine connecting rod by FEM. IJ EIT,*

1(3).

8. *Noor, M. M., Shuhaizal, M. N., Kadirgama, K., Mohamed, J. J., Rejab, M. R. M., & Rose, A. N. M. (2008). Analysis Of Connecting Rod Based On Finite Eliment Approach. In Proceedings Of Mucet2008 Malaysian Technical Universities Conference On Engineering And Technology March (pp. 8-10).*
9. *Chikalthankar, S. B., Nandedkar, V. M., &Baratam, S. P. (2012). FEA Analysis of Geometric Parameters of connecting rod big end. International Journal of Engineering Science & Technology, 4(4)..*

AUTHORS

Prof Ganesh S. Shetiye, Assistant Professor, Industrial Engineering Department, Ramdeobaba College of Engineering and Management, Nagpur, India

Email : shetiyegs@rknec.edu

Bharat Kungwani, Mechanical Engineering Department, Ramdeobaba College of Engineering and Management, Nagpur, India