ISSN 2395-1621

Finite Element Analysis and Optimization of Diesel Engine Connecting Rod

#1Ajinkya.A.Bhilare, #2Akash.R.Suryavanshi

1ajinkya.bhilare5@gmail.com
2akash.suryavanshi@zealeducation.com

1 Department of Mechanical Engineering, Dnyanganga College of Engineering & Research, Pune-41,Maharashtra, India
2Department of Mechanical Engineering, Dnyanganga College of Engineering & Research, Pune-41, Maharashtra, India

ABSTRACT

The automobile engine connecting rod is a high volume production component. It undergoes high cyclic loads of order which range 10^8 to 10^9 cycles from high compressive loads due to combustion, to high tensile loads due to inertia. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders. Connecting rods for automotive applications were typically manufactured by various manufacturing process like by forging from either wrought steel or powdered metal. They could also be cast, but castings could have blown-holes which are critical from durability and fatigue points of view. Forgings produce blow-hole-free and better rods gives them an advantage over cast rods. But between the forging processes, powder forged or drop forged, each process has its own pros and cons. So in order to reduce the material cost and thus production cost it is better to optimize the weight or volume. This project entitled weight optimization process through Ansys software. In this with the help of generate code through Ansys API, we run the test to achieve the optimized weight condition of diesel engine connecting rod. It achieve saving in material, weight and cost for manufacturing diesel engine connecting rod.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

***Keywords*— Ansys, Connecting Rod, Cost, Optimize, Weight.**

I. INTRODUCTION

Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. The connecting rod is subjected to a complex state of loading. Therefore, durability of this component is of critical importance. Due to these factors, many research work is always going on the connecting rod and it is always area of research for different domains such as production technology, materials, performance simulation, fatigue and many more. Connecting rod of automotive applications should be lighter and lighter, should consume less fuel and at the same time they should provide comfort and safety to passengers, that unfortunately leads to increase in weight of the vehicle. As we want the connecting rod with less weight, we done optimization work. In that, we modelled the connecting rod of diesel engine with ANSYS software by parametric modelling and thus we performed finite element analysis of connecting rod under static compressive load and static tensile load by giving standard boundary

conditions. We get the structural behaviour of connecting rod under the static load conditions. After the static finite element analysis of connecting rod we moved towards optimize the connecting rod process. On the basis of study of static finite element analysis of connecting rod we concluded that there is much scope of weight reduction in existing connecting rod. As weight is the main concern factor for any design issues. We try for optimized it and there is structural improvement. Every designer and manufacture always want the lighter and strong connecting rod. Connecting rod with less weight is always best option for further engine assembly. Therefore after static finite element analysis, optimization of connecting rod for weight is carried out under specified limits. We get the connecting rod with optimized weight. That is our requirement.

II. LITERATURE REVIEW

Various studies were carried out for finding optimized weight of connecting rod. Cad Data modelling and analysis was one of the commonly used methodology. Based on analysis results obtained geometry parameters were changed

Prof. Vivek C. Pathade [1] proposed the major stress induced in the connecting rod was a combination of axial and bending stresses in operation. The axial stresses were produced due to cylinder gas pressure (compressive only) and the inertia force arising in account of reciprocating action (both tensile as well as compressive), whereas bending stresses were caused due to the centrifugal effects. The result of which was, the maximum stresses were developed at the fillet section of the big and the small end. Hence, the paper deals with the stress analysis of connecting rod by Finite Element Method using Pro/E Wildfire 4.0 and FEA WORKBENCH 11.0 software. The comparison and verification of the results obtained in FEA was done experimentally by the method of Photo elasticity (Optical Method). The method of Photo elasticity includes the casting of Photo elastic sheet using Resin AY103 and Hardner HY951, preparation of the model from Photo elastic sheet calibration of the sheet to determine material fringe value.

Material replacement was one of the most commonly used process for reducing the weight parameters of the geometry. Kuldeep B et.al [2] proposed the material of connecting rod was replaced by aluminium based composite material reinforced with silicon carbide and fly ash. And they also performed the modelling and analysis of connecting rod. FEA analysis was carried out by considering two materials. The parameters like von misses stress, von misses strain and displacement were obtained from FEA software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 43.48% of weight, with 75% reduction in displacement.

Zheng Bin Liu Yongqi et.al [3] analysed stress distribution, safety factor and fatigue life cycle of connecting rod by using 3D finite element method. The results show that the exposed destructive position was the transition location of small end and connecting rod shank at maximum compression condition. Maximum stress was 303MPa. Safety factor was 1.24. At maximum stretch condition, the exposed destructive position was I-shaped cross-section at big end. Maximum stress was 118MPa. Safety factor was 3.19. And structure of connecting rod was improved. Safety factor and fatigue life cycle of connecting rod increases. After structural improvement, maximum stress decreases and both safety factor and fatigue life cycle increases.

Mr. Pranav G Charkha et.al [4] performed load analysis on connecting rod. Their study were deals with two subjects, first, static load stress analysis of the connecting rod, and second, optimization for weight. They performed finite element analysis on single cylinder four stroke petrol engine. Structural systems of connecting rod could be easily analysed using Finite Element techniques. Optimization was performed to reduce weight. They found that weight can further more reduced by changing the material of the forged steel connecting rod to crack able forged steel (C-70). The optimized geometry was 20% lighter than the current connecting rod. Current connecting rods could be replaced by fracture split able steel forged connecting rods. These connecting rods were lighter in weight than existing connecting rod, with similar or better fatigue behaviour.

Om Parkash et.al [5] found the existing design performs by modelling and evaluates critical regions in the connecting rod under fatigue loading. The main objective of their work was to re-optimize the existing design of connecting rod of universal tractor (U650) by changing some of the design variables. Optimization of connecting rod was done under same boundary and loading conditions for variation in the few stress and fatigue parameters i.e. stresses, weight, life, damage and safety factor. The allowable numbers of cycles under fully reversed fatigue loading were increased and assumed up to a maximum limit. Stress concentration coefficient was varied to obtain the maximum cycles condition. The critical regions under both static and fatigue analysis were identified and improved. The connecting rod was then modelled and optimized for the reduction in weight.

Zhou Qinghui et.al [6] they obtain the vibration characteristics and vibration frequency distributions, structural characteristics of the connecting rod mechanism using modal analysis. Then they prepare a physical model of connecting rod mechanism using CAD software. Then finite element analysis and simulation of the model is taken by Hyperworks and MSC. Nastran software's. Then its flexible multi-body dynamic model was established by ADAMS/View and the fatigue stress of connecting rod under the max combustion pressure and Inertia force condition was calculated using the durability Module. The stress was mainly produced on the joint of connecting rod shell and the bottom end or the top end. The simulation result showed that the stability of the mechanism was well. The simulation analysis was really an economical and efficient method to study.

III.OBJECTIVES

The main objective of this study was to optimize an alloy steel connecting rod for its weight. Commercial software such as ANSYS was used to obtain the variation of quantities such as dimensional properties and loads.

The following objectives were carried out with ANSYS software:

- Modelling of diesel engine connecting rod of four stroke engine. Complete model of the connecting rod done using ANSYS software. As the model is symmetric about two planes, it is further reduced to one forth. Then this quarter model is used for the analysis.

- Static finite element analyses of connecting rod under static compressive load and static tensile load. There is compressive force acting on the piston and so on connecting rod due to gas pressure, at some phase of the cycle and the tensile load acting due to inertia of the moving components. As the load is varying from compressive load to tensile load, Maximum compressive load and maximum tensile load conditions are two cases taken for the analysis. The details of boundary conditions.

IV. SCOPE

Optimization is done to reduce the weight and the cost of the connecting rod, which is subject to constraints. Generally modelling in the ANSYS is performed by using two approaches

GRAPHICAL USER INTERFACE (GUI)

COMMAND LINE INTERFACE (CLI)

From the above two approaches command line interface is more accurate for the modelling of connecting rod in ANSYS. We have performed command line interface to model the connecting rod. Various dimensional parameters as shown in the table are fed into the program, thus this makes the parametric approach under command line interface, and thus programming is called PARAMETRIC MODELING. One of the main scope of parametric program modelling is that once the basic program for the modelling is generated we can model the connecting rod of various sizes by just changing the values of the parameters in program. Thus, modelling has been achieved using geometric programming technique (i.e. parametric modelling).

V. PROJECT WORK

A. Selection of FEA Software

This is initial stage of our project. Depending on our requirement, we chose the suitable FEA software package. We take the information of different software packages like- Hypermesh, Nastran, Ansa, Ansys. As we decide the use of command line interface then Ansys is the most suitable software package for our project. ANSYS is the tool developed by ANSYS, Inc. for the design and analysis purpose i.e. by using this software modelling as well as the analysis can be performed. The solver used in ANSYS is much stronger than the other software's. Analysis in this software is more user friendly than any other software. While part modelling is not so much user friendly as for creating any model as its dimensions are given in vector coordinates. Analysis of this software used a technique called "Finite Element Analysis". This method gives results which are very closer to actual one. Lot of elements are there in ready to use format associated with ANSYS, so it is good for different Analyses from different fields.

B. Creating the Model for Analysis

1) *Studying the Commands for Preparation of Model:* We are doing the analysis for diesel engine connecting road. Preparation of model is the main part of project. As we are creating the test model first by command line interface so first our task is to study the command line interface. For that various commands are need to studied first. Ansys contains hundreds of commands for generating geometry, applying loads and constraints, setting up different analysis types and post-processing.

TABLE I

SOME VARIOUS ANSYS COMMANDS

Sr. No	**Ansys Command and Syntax**		
	Command	**Description**	**Syntax**
1	k	key point definition	reference item (partial)
2	l	Straight line creation	abstract heading (also in Bold)
3	larc	Circular arc line (from key points)	level-2 heading, level-3 heading, author affiliation
4	spline	Spline line through key points	spline,kp1,kp2, ... kp6
5	a	Area definition from key points	a,kp1,kp2, ... kp18
6	al	Area definition from lines	a,l1,l2, ... l10
7	v	Volume definition from key points.	v,kp1,kp2, ... kp8
8	va	Volume definition from areas	va,a1,a2, ... a10
9	rectng	Create volume from area extrusion	rectng,x1,x2,y1,y2
10	cylind	Cylindrical volume creation	cylind,rad1,rad2,z1,z2,theta1,theta2

TABLE II

SOME VARIOIUS DIMENSIONAL PARAMETERS

Font Size	**Ansys Command and Syntax**		
	Regular	**Bold**	**Italic**
1	lng	Connecting rod length	**65**
2	bid	Crank end inner dia.	17
3	bodb	Crank end bottom outer dia	24
4	bodt	Crank end top outer dia	23
5	sid	Pin end inner dia	13
6	sodb	Pin end bottom outer	19.5

		dia.	
7	sodt	Pin end top side outer diameter	19
8	ethk	End thickness	10
9	shthk	Shank Thickness	6
10	shsbd	Shank slot big dia	8
11	shssd	Pin end slot thickness	2

Then we use above commands and defied variable to construct solid model step by step like below some command line, it is the combination of syntax and value from table-

```
C*** Crank end
k,1,bid/2
k,2,bid/2,ethk/2
k,3,bodt/2,ethk/2
k,4,bodb/2
a,1,2,3,4
k,5,,1
k,6,,-1
vrot,1,,,,,,5,6,-60
vrot,6,,,,,,5,6,-120
```

Test connecting rod model from which we are taking dimension for writing our program.

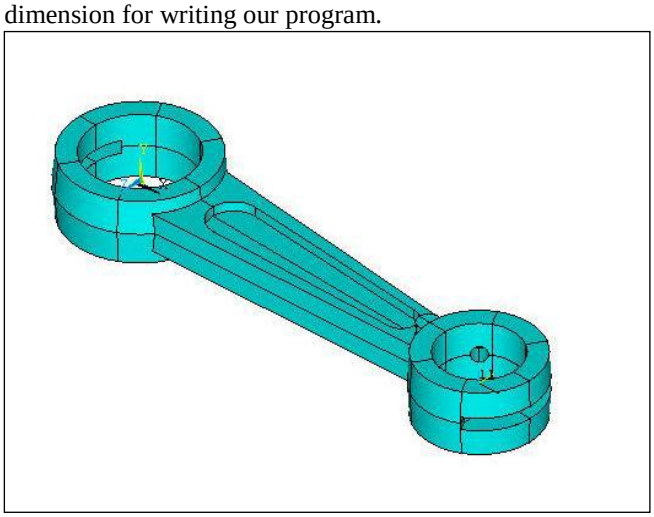

Fig 1 Connecting Rod from where, we are taking reference Dimension to generate the code

Optimization process has will be also performed with ANSYS software which follows the parametric program in which various instructions are given:

- Various dimensional parameters with maximum and minimum limits.
- Maximum allowable stress.

The method of optimization used here is First Order Method. Program for optimization has been prepared to follow loops or iterations. ANSYS software carry out each loop and finds out the weight of connecting rod and accordingly it changes the dimensional parameters within the specified limits. ANSYS continues subsequent loop until the optimized weight is obtained and it takes care that dimensional parameters do not go beyond specified limits and maximum allowable stress.

C. Static Finite Element Analysis of Connection Rod under Compressive Loading

Static structural analysis is one in which the load/field condition does not vary with time and the assumption here is that the load or field conditions are gradually applied. This stage is performed after modelling of connecting rod. Various loads are always acting on connecting rods. These are helpful to find the structural behaviour of connecting rod. We consider two cases here static compressive load and static tensile load. Now one thing is important that as the model is symmetric about two planes, it is further reduced to one forth. Then this quarter model is used for the analysis. This is the basic structure used for next stages of analysis. We take the reference dimension from this model.

1) *Constraints:* Since connecting rod is under static compressive loading, thus the crank end of the connecting rod has been restrained for 120 degrees. The FE model here is the quarter model so the all nodes of the area of 60 degrees from the axis of symmetry are selected and all three degrees of freedom (translations in x, y and z directions) are fixed.

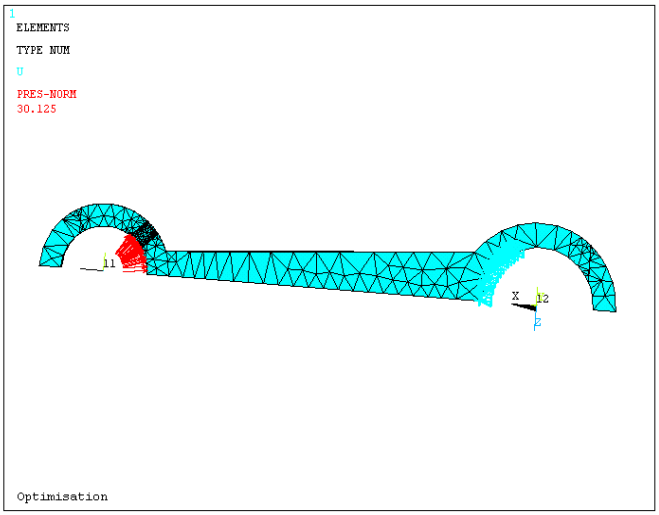

Fig 2 Boundary Conditions for Compressive Loading.

2) *Loading:* The crank end and piston end are assumed to have cosine distribution loading over the contact surface area of angle 60 degree, for quarter model, under static compressive loading (for full model, contact surface area will be for angle 120 degree). But there are very small amount of variation in results is observed when the constant pressure is applied instead of cosine loading. So the constant pressure is applied using surface load in ANSYS The pin end of the connecting rod has been applied standard gas force for 60 degree (as it is quarter model)

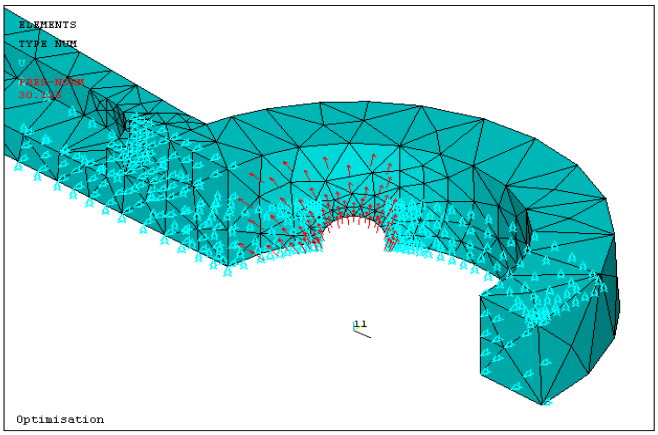

Fig 3 Pressure acting as a compressive load

Gas force at pin end F= 951 N

Area of compression A=31.568

Therefore, the Normal pressure on the contact surface is given by

Normal pressure P= F/A

=951/31.568

TABLE III

PARAMETERS OF CONNECTING ROD MATERIAL

Sr. No	Parameters of Connecting Rod Material		
	Parameter	**Unit**	**Scalar Value**
1	Modulus of elasticity	N/m2	2.1e5
2	Poisson's Ratio	Unit less	0.30
3	Mass Density	Kg/m3	8030
4	Ultimate strength	MPa (N/mm2)	537
5	Yield Strength	MPa (N/mm2)	966

Then we prepare the parametric program for compressive loading. Result of analysis is also obtained with this program as below-

3) *Result of Finite Element Analysis under Compressive Loading:* Figure 4 shows the principle stress distribution in connecting rod under compression. Under compressive load, the critical regions are the crank end transition and pin end transition, also the web at the crank end has a high stress region. Thus plot gives us the general idea of the stress variation along the length of connecting rod. The static loads for which these stresses are plotted is a compressive load of 951 N. Figure shows the maximum stress produced under axial compressive load comes out to be 232.077 MPa which is at the pin end transition and crank end transition. Mainly more stresses are coming in the shank part because it is the weaker part under compression. There is increase in the stresses in shank with reduction in the cross section of the shank. It goes to highest value at the joint with the end.

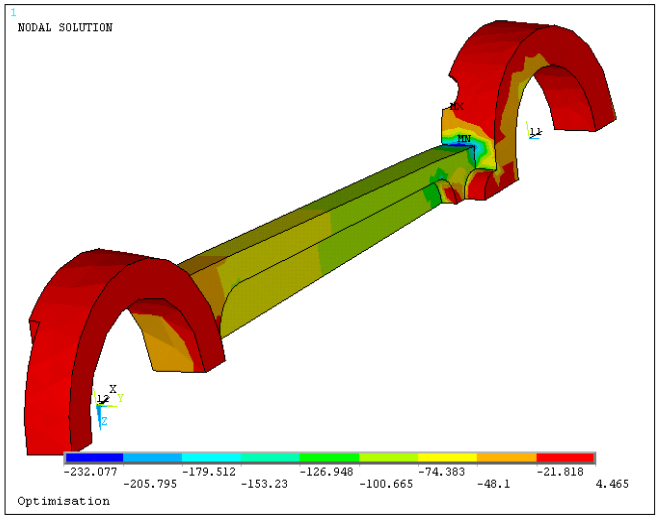

Fig 4 Principle Stress Distribution over the Connecting Rod under Compressive Loading

4) *Deflection of Connecting Rod*: Connecting rod is applied under compressive load thus maximum deflection will occur at pin end of the connecting rod and minimum deflection will occur at crank end of the connecting rod. Figure 5 shows the deflection at various sections of connecting rod. Maximum deflection produced is 0.23759mm at the pin end and minimum deflection produced is 0.00264mm.

Maximum deflection = 0.23759mm
Minimum deflection = 0.00264 mm

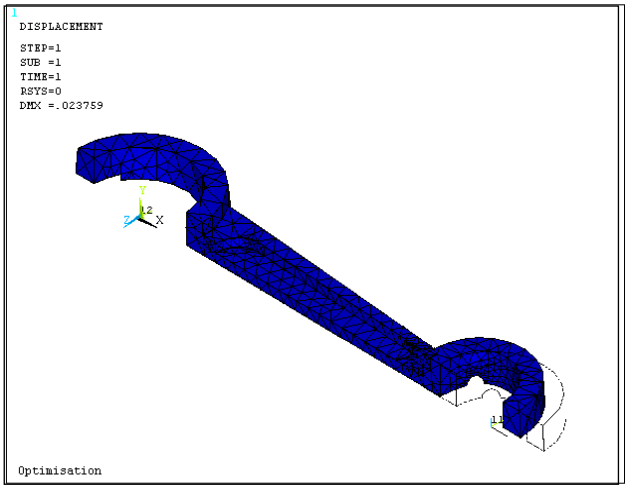

Fig 5 Deflection of the Connecting Rod

D. Static Finite Element Analysis of Connection Rod under Tensile Loading.

Figure shows quarter model in which tensile load is applied at the crank end and pin end is restrained.

1) *Constraints:* Pin end of the connecting rod has been restrained for 90 degree for quarter model (but for full model 180 degree) .The nodes in the area are selected and all degrees of freedom at all nodes are fixed (constrained).

2) *Loading:* The crank end of the connecting rod has been applied standard tensile force for 90 degree (as it is quarter model)

Tensile force at crank end F= 707.85 N

This force is acting on the area of 180 degrees (on 90 degrees here in quarter model).All nodes in the corresponding area are selected and the radial force is applied on the nodes. As the load is varying sinusoidal, the maximum load is multiplied by the value of sine of the angle at which it is acting. The total nodes in the area are divided in to 9 parts, each of 10 degrees. So the net force is decreasing with the factor 'sin θ', from 1 to 0.

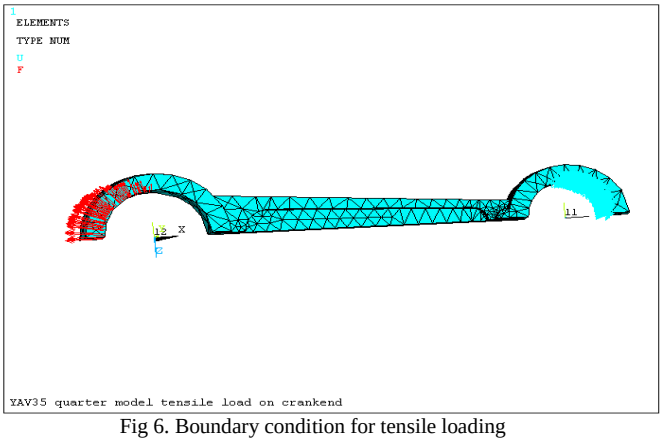

Fig 6. Boundary condition for tensile loading

Then we prepare the parametric program for tensile loading. result of analysis is also obtained with this program as below-

3) *Result of Finite Element Analysis under Tensile Loading:*

In tensile load condition, the critical regions are the crank end transition and pin end transition, also the web at the crank end has a high stress region. Thus plot gives us the general idea of the stress variation along the length of connecting rod. The static loads for which these stresses are plotted is a tensile load of 707.85N.Figure shows the maximum stress produced under axial compressive load comes out to be 90.303 MPa which is at the region of slot section.

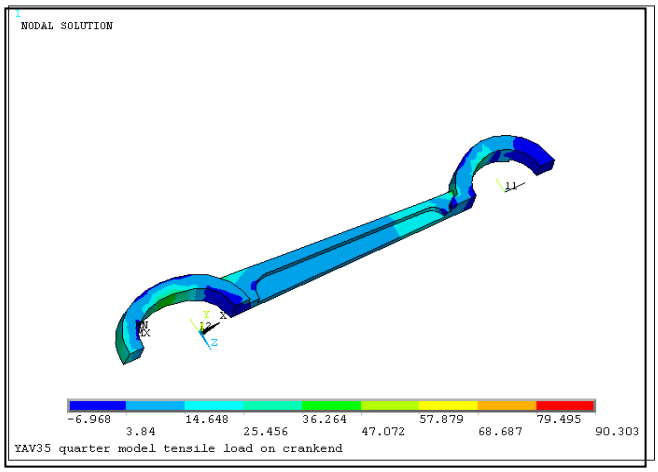

Fig 7 Principle Stress Distribution over the Connecting Rod under Tensile Loading

The tensile stresses are here due to tensile load coming because of the inertia force (inertia of the moving parts). The inertia forces are generally very less compared to load due gas pressure; so naturally the stresses are not much important compared to compressive stresses in the previous stresses.

4) *Deflection of Connecting Rod:* Deflection of the connecting rod in the axial direction is also shown figure. It is increasing with distance from the fixed end, so minimum at small end and maximum at big end. This deflection reading is taken under tensile load applied.

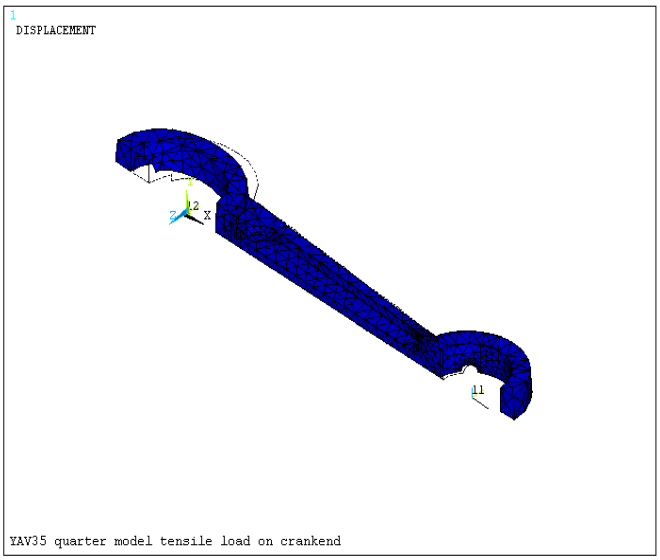

Fig 8 Deflection of the Connecting Rod under Tensile Load

E. Weight Optimization Process-

This work is going on. Now under preparing the final source Code. The weight of the new connecting rod is definitely lower than original connecting rod. Optimization has been performed by preparing parametric program under command line interface technique with ANSYS software. Program for optimization has been prepared to follow loops or iterations and finally it will give an optimized weight connecting rod.

VI. CONCLUSION.

From all above work we found the potential for weight reduction in the existing connecting rod. Program for optimization has been prepared to follow loops or iterations. ANSYS software carry out each loop and finds out the weight of connecting rod and accordingly it changes the dimensional parameters within the specified limits. ANSYS continues subsequent loop until the optimized weight is obtained and it takes care that dimensional parameters do not go beyond specified limits and maximum allowable stress.

ACKNOWLEDGEMENT

Thanks to our P G Coordinator of our DCOER College for giving the opportunity to present my work.

REFERENCES

[1] Prof. Vivek C. Pathade ,Dr. Dilip S. Ingole (2013) 'Stress Analysisof I.C. Engine Connecting Rod by FEM and Photoelasticity' IOSR Journal ofMechanical and Civil Engineering, PP 117-125.

[2] Kuldeep B, Arun L.R, Mohammed Faheem (2013) 'Analysis andOptimization of Using ALFASic Composites'. International Journal of Research in Science, Engineering and Technology.

[3] Zheng Bin Liu Yongqi, Ji Lixia(2010) 'Finite Element Analysis and Structural Improvement of Diesel Engine Connecting Rod'. Second International Conference on Computer Modeling and Simulation.

[4] Mr. Pranav G Charkha, Dr Santosh B Jaju (2009) 'Analysis & Optimization of Connecting Rod' Second International Conference on Emerging Trends in Engineering and Technology.

[5] Om Prakash et al.(2013) 'Optimizing the Design of Connecting Rod under Static and Fatigue Loading. International Journal of Research in Management, Science and Technology.

[6] Zhou Q, Wang Y, J. Wei (2010), 'The Finite Element Analysis of Connecting Rod of Diesel Engine', International Conference on Measuring Technology and Mechatronics Automation, IEEE.PP 870-873.

[7] P S Shenoy and A Fatemi (2005) 'Connecting rod optimization for weight and cost reduction', SAE Publications, Paper No 2005-01-987.

[8] DR.B.K.Roy (2012) 'Design Analysis and Optimization of Various Parameters of Connecting Rod using CAE Software'. International Journal of New Innovations in Engineering and Technology.

[9] Deng Banglin et al (2011), 'Fatigue and Bearing Analysis of An Engine Connecting Rod Based on Multi-body Dynamics' Third International Conference on Measuring Technology and Mechatronics Automation.

[10] E Musango Munyao et al (2014) 'Simulation of Thermal-Mechanical Strength for Marine Engine Piston Using FEA' Int. Journal of Engineering Research and Applications. PP319-323