ISSN 2229-5518

Design of Connecting Rod For Weight Reduction Using C70S6 Material

Deepak G. Gotiwale, Shailesh D. Ambekar

Abstract— This paper presents a study of redesigning of connecting rod for its weight reduction using c70s6 material. During its operation connecting rod undergoes various types of loads. Fatigue as well as static stresses are mainly responsible for failure of a connecting rod. Initially fatigue testing was carried out for studying failures during its life cycles. But after recognizing these failures during fatigue testing, fatigue life further enhanced by incorporating few changes and then analyzed with the help of FEA for highlighting critical points on connecting rod. These critical points are divide with respect to five different zones at connecting rod. Considering these five different zones and critical points, connecting rod was subjected to static FEA for tensile and compressive loading for both small and big end. In this process, first small end is restrained and simultaneously compressive and tensile load is applied at crank end. Similarly, crank end is restrained and simultaneously compressive and tensile load is applied at pin end. Stresses near these points are studied and the regions where stresses are less, are considered for material reduction areas from connecting rod. Then the connecting rod design is also supported by analytical calculations for its thickness. All these findings are incorporated in new digitized connecting rod. Therefore the aim of this paper is to study these causes or areas of failure with the help of FEA and redesigning the connecting rod by focusing on the scope of weight and cost reduction. Thus the component was redesigned with reduction in its weight.

Index Terms— Connecting rod Design, Connecting rod, Fatigue testing, Finite Element Analysis, Analytical approach, Weight reduction.

1 Introduction

The connecting rod or conrod is bridge between the piston to the crank or crankshaft in reciprocating engines. The connecting rod with piston and crank converts linear motion into rotary motion and also rotary motion into linear motion with a simple mechanism.
The aim of this study is to redesign a connecting rod for its light weight. The weight reduction carried out here, is not only helping in reducing mass from connecting rod, but also helping in for enhancing manufacturing feasibility and cost reduction parameters. In addition to this, while performing weight reduction under fatigue life, software used for this work shows restriction and this is the constraint. Therefore, in weight reduction, the structure modification is carried out by fatigue testing results, FEA results and the design is supported with numerical values.
Serag et al. [1] developed approximate mathematical formulae to define connecting rod weight and cost as objective functions as well as constraints. The optimization was achieved using a geometric programming technique. Pai[2] presented an approach to optimize the shape of a connecting rod subjected to a load cycle which consisted of the inertia load deducted from gas load as one extreme and peak inertia load exerted by the piston assembly mass as the other extreme.

Yoo et al. [3] performed shape optimization of an engine connecting rod using variational equations of elasticity, material derivative idea of continuum mechanics, and an adjoint variable technique to calculate shape design sensitivities of stress. The results were then used in an iterative optimization algorithm to numerically solve for an optimal design solution.

1.2 Necessity

In the last 50 years, cars have learned to think, adjust, and even protect. High performance is more demanding. Not only for putting a little smile on the face, the majority of people want a machine that will get them from spot A to spot B as easy as possible. Mainly this smile comes by a quick strike of the accelerator. Keeping it in mind, manufacturer has to design lighter, faster, and more efficient engines for this job.
In this project, one component of an engine in particular, the connecting rod, will be analyzed. Being one of the most integral parts in an engine's design, the connecting rod must be able to withstand tremendous loads and transmit a great deal of power. It is no surprise that a failure in a connecting rod can be one of the most costly and damaging failures in an engine.

2 Material And Methods

Fig. 1 shows a 3D solid model of a connecting rod made with the help of Pro-E and later on imported in ANSYS for FEA. [7]

• Deepak G. Gotiwale is student of masters degree program in Mechanical design in Government College of Engineering, Aurangabad(M.S), India , PH-919423455287, E-mail: omdeepak2001@gmail.com
• Shailesh D. Ambekar is Faculty in Department of Mechanical Engineering in Government college of Engineering, Aurangabad(M.S.),India E-mail: sd.ambekar@gmail.com

Fig. 1. 3D Solid model of Connecting rod.

2.1. Forces Calculation

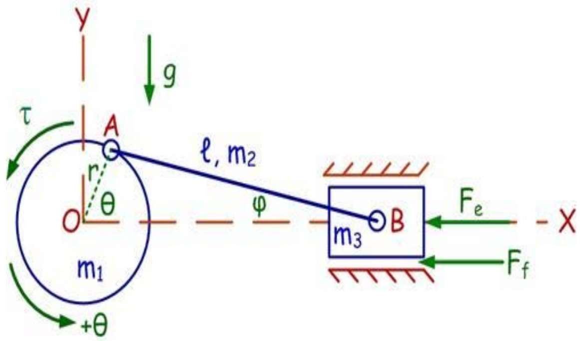


Fig. 2. Forces acting on Connecting Rod

After having a solid model of connecting rod, it can be use for further process and for that, forces or loading need to be calculated.

The load acting on the connecting rod is divided into two types,

1. Maximum compressive load

This load is calculated on the basis of peak firing pressure.

2. Maximum tensile load

This load is calculated on the basis of inertia masses at both the ends.

Graph 1 shows tensile and compressive forces acting on a connecting rod with respect to crank angle. This graph can be drawn with the help of equations,

$$\emptyset = \arcsin[\frac{R}{L} \times \sin\theta] \tag{1}$$

$$r = R\cos\theta + L\cos\emptyset \tag{2}$$

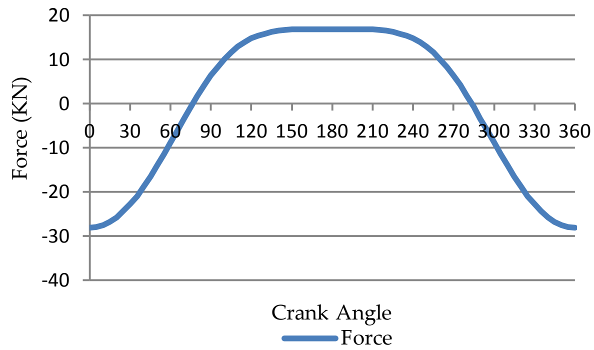
$$LVel = \frac{R}{L} \times \omega \times \frac{\cos\theta}{\cos\emptyset} \tag{3}$$

$$Vel = -R \times \sin\theta \times (\omega + LVel) \tag{4}$$

$$Lacc = \frac{(-R\times\omega^2\times\sin\theta)+(L\times LVel^2\times\sin\emptyset)}{L\times\cos\emptyset} \tag{5}$$

$$Acc = -R \times \omega \times \cos\theta \times (\omega + LVel) - R \times \sin\theta \times LAcc \tag{6}$$

$$Force = \frac{W\times Acc}{110\times\cos\emptyset} \tag{7}$$



Graph 1. Crank angle Vs Forces

2.2 Fatigue Testing

In a fatigue test, the infinite life condition is assumed to meet at 5 - 6 million cycles for steels. Therefore for 5 million cycles test, a targeted factor of safety is decided. Therefore for infinite life conditions FOS is the require factor of safety. The targeted failure probabilities of less than 0.01% are followed in automotive industries. For forged components a factor of safety of 1.7 is consider is sufficient considering a normal scatter band width is between 10 and 90%.Generally in Europe, during component sourcing from OEM's they required FOS of only is 1.6 to 1.7. Here mainly considering the possible scatter of components strength during volume production and also considering variation in engine load (in peak firing pressure variation, miss usages condition), the targeted factor of safety is 2.0. Approach used for testing of connecting rod is as below,

- Take first sample
- Apply a load level, which will be little higher than the estimated fatigue strength until it either fails or sustain at the targeted life.
- If sample fails before achieving its targeted life, then decreased load level by pre-selected decrement.
- Take second sample and tested at this new lower load level.
- If the first sample sustains this load, the load level is increased and second sample is tested at this new increased load level.

Thus specimen is tested in such a manner and sequence that, it tested above or below the load levels to see whether it sustain or fails at this load.

Thus from the above explanation it is very clear that, the main focus of this test is on the man value of load level for failure or fatigue strength for the targeted life. However, to evaluate the scatter observed during this testing, the actual testing is little deviated from the above mentioned procedure.

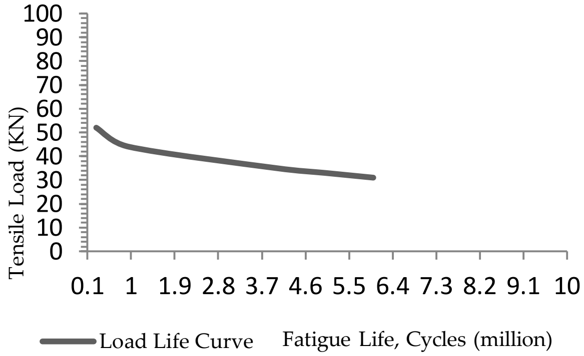
Though the loads acting on small end and big ends are different, here testing is performed on small end only considering a critical area of con rod.

ISSN 2229-5518

TABLE 1
LOADS ON CONNECTING ROD

Loads	Forces (N)
Tensile	17000
Compressive	28000

Connecting rods were tested for small end as per the sequence mentioned above and the results are as shown in graph 2.
As stated above, after qualifying dimensional and metallurgical inspection a connecting rod was taken for testing.



Graph 2. Applied load Vs fatigue life.

Before and after testing it is necessary to check all components for crack and surface defects by dye penetrate method. Three design iterations were done during the fatigue evaluation of the connecting rod. As per the graph shown above for small end failure, considering FOS as 2, the life ranging from 0.5 million to 5 million cycles shows excessive scatter in life of conrod and inconsistency of small end strength. This failure is may be due to variation in metallurgical / physical properties, tolerances and manufacturing quality.
Now using Ansys software the FE analysis of connecting rod was carried out to determine the stress concentration areas. The boundary conditions are as shown in fig. 3. FEA results for small end are shown in figure and, indicate stress concentration is more at oil hole chamfer. During testing failures also shows crack initiations in the same region. To overcome this problem it was decided to change oil hole chamfer angle to 90° from 120^0 to avoid sharp merging with oil hole inner diameter. Fig. 3 shows proposed modification profile of oil hole.

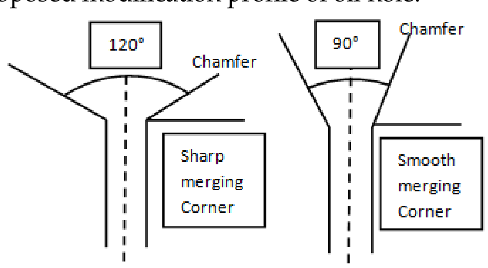


Fig. 3. Oil hole modification

2.3. Boundary Conditions for FEA Analysis

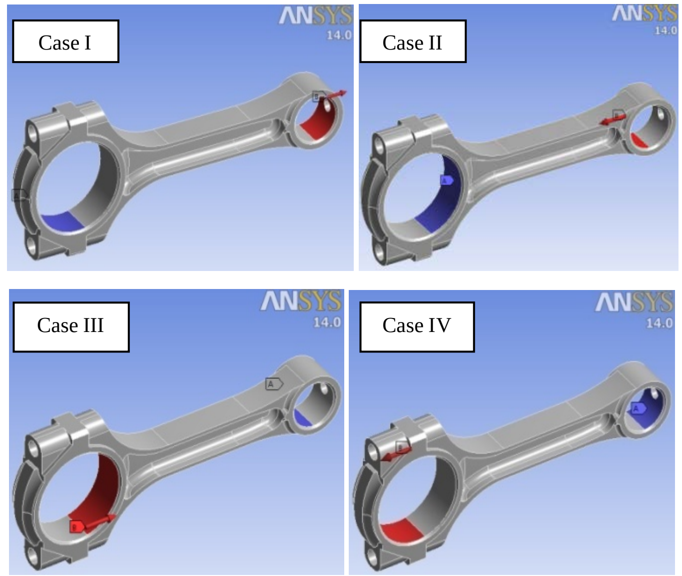


Fig. 4. Boundary conditions.

Fig. 4 shows 4 cases of boundary conditions for connecting rod in which, in case I, crank end is fixed and a tensile load is applied at pin end. In case II crank end is fixed and a compressive load is applied at pin end. In case III pin end is fixed and a compressive load is applied at crank end. In case IV pin end is fixed and a tensile load is applied at crank end.

2.4. Geometry And Meshing of Connecting rod

Meshing is the process used to "fill" the solid model with nodes and elements, i.e., to create the

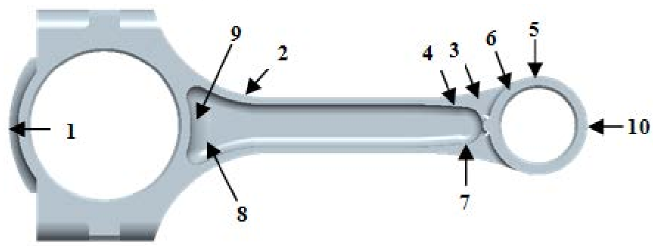


Fig. 5. Point locations for meshing

FEA model. Remember, you need nodes and elements for the finite element solution, not just the solid model. The solid model does not participate in the finite element solution.
In this connecting rod, before finalization of element size for meshing, a meshing convergence is performed by tetrahedral element with various element lengths.

- 2.5 mm (17773 elements)
- 2 mm (24699 elements)
- 1.5 mm (42177 elements)
- 1 mm (95989 elements)

With reference to fig. 5, ten points are located at connecting rod for checking Von Misses stresses for convergence. As per the above explanation, all results of these element lengths are combined to plot a graph. After studying this graph we come to know that, 1mm element length is best suited for meshing, as it helps to achieve connecting rod convergence with uniform element length. This resulted in a mesh with 95989 elements. It can be seen that convergence has been achieved with 1 mm global mesh size. Now comparing two models, it is observed that, percentage difference is only 2.5% between stress values. Hence, the mesh with 95989 elements was used for Finite element analysis of connecting rod.

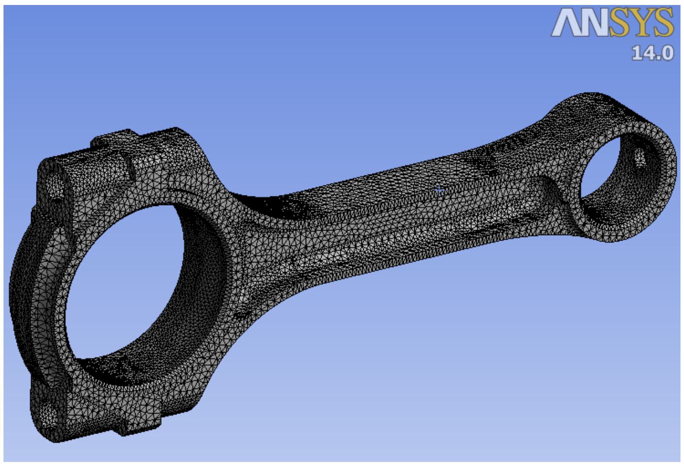


Fig. 6. Connecting rod meshing with tetrahedral 1mm element length.

As shown in fig. 6, meshing is performed with 1mm global element length with tetrahedral mesh type. It gives uniform distribution of all elements on connecting rod. [6]

2.5. Static Stress Analysis

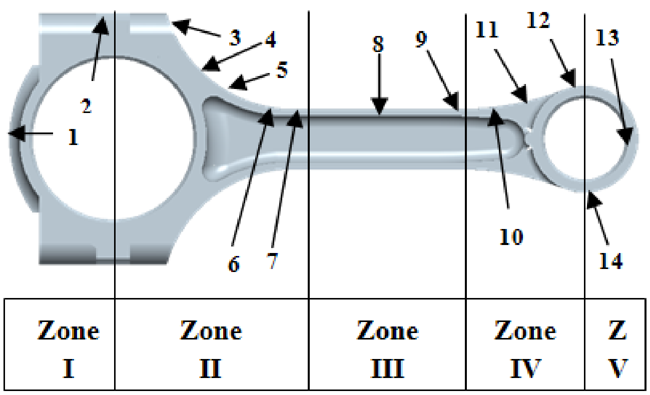


Fig. 7. Stress distribution zones.

In this chapter, to obtain loads acting on the connecting rod and to perform FEA, the load analysis is carried out. For the design and analysis of connecting rod most of the people have used static axial loading criteria only. However, some people have used inertia loads (axial load varying along the length) during the design process. In this paper the investigation of connecting rod is carried out with static axial loading criteria. Fatigue testing range can be simulating by the maximum and minimum static load. As a result of this, finite element analysis was carried out under axial static load with no dynamic / inertia loads. Therefore in this chapter, the results of the above mentioned analysis are shown and discussed with a view to use them for optimization of connecting rod.

3 Results

In this chapter results are obtained for static stress analysis in the form of Von-Misses stresses. While performing this static stress analysis in ANSYS, connecting rod was divided in to five different zones for understanding stresses at different locations of connecting rod.

3.1. Static Stress Results

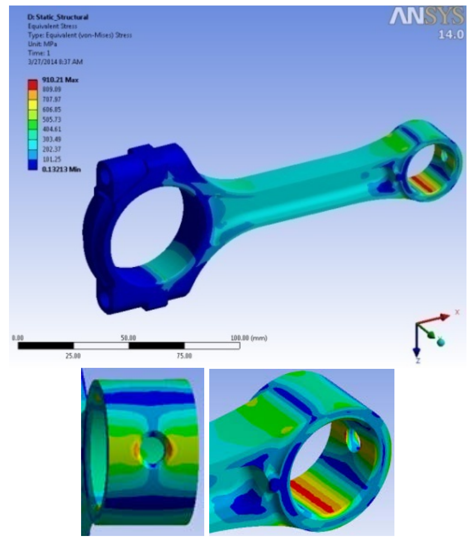


Fig. 8. Von Mises stresses with Crank end fixed and static tensile load at piston pin end.

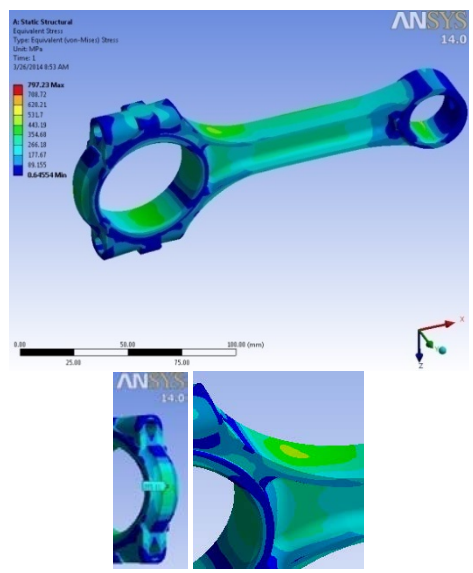


Fig. 9. Von Mises stresses with Crank end fixed and static tensile load at piston pin end.

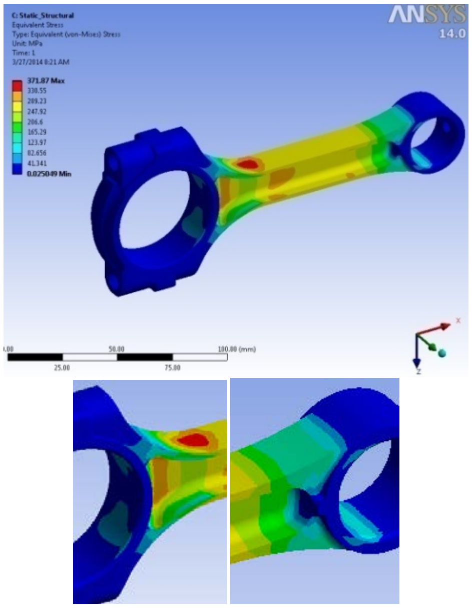


Fig: 10. Von Mises stresses with Crank end fixed and static compressive load at piston pin end.

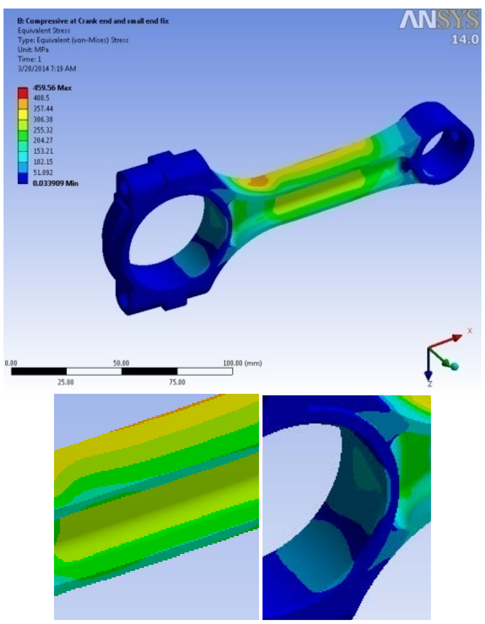


Fig: 11. Von Mises stresses with piston pin end fixed and static compressive load at the crank end.

Fig. 8 to 11 shows results in terms of Von Misses stresses of the connecting rod under static loading. Fig. 8 shows the von Misses stresses in which tensile load applied at the piston pin end, and crank end is fixed. Fig. 9 shows results in terms Von Misses stresses, in which tensile load applied at the crank end, and piston pin end is fixed. Fig. 10 shows results in terms of Von Misses stresses, in which compressive load applied at the piston pin end, and crank end is fixed. Fig. 11 shows results in terms of Von Misses stresses, in which compressive load applied at the crank end, and piston pin end is fixed. In all cases applied load is 28 KN. Now we will discuss the differences between above mentioned cases. Considering this, as shown in Fig. 7, connecting rod is divided into five zones and comparison is made on the basis of obtained nodes. As per the different zones stress distribution is shown in Fig. 8 to 11 and these results are compared. After comparison with fig. 8 (case I) and 9 (case II) major differences are observed between the results of zone I, and zone II, IV, and V, having nodes 3, 4, 5. In zone II and zone III, node numbers 6 and 7 and node number 8 respectively have very close stress values. In case I, the crank end is fixed while in case II the pin end is fixed. These above mentioned fixed ends not pin joints. Therefore, we can say that, to predict the structural behavior perfectly, results received from zone I, II and zones IV, V from case-I and case-II respectively are not suitable. In case III and case IV, the differences between the stresses at nodes 6, 7, 8, and 9 are small for compressive loading.

Fig. 8 and 9 shows stress distribution for tensile loading and after analyzing these results of tensile loading, we come to know that, oil hole is the critical region and for this purpose, a enlarge image of that critical region is shown. Similarly, after analyzing stress distribution in fig. 9 we come to know that, web of the connecting rod is the critical region and for this purpose a enlarge image is shown. As shown in fig. 10 and 11 crank end and pin end regions are the critical regions.

In this way, with help of FEA results, few critical regions are targeted for minimizing its stress intensity.

3.2 Design Calculations

Connecting rod Specifications:

Diameter of Piston(d)	=	68.3 mm
Length of Conrod(2L)	=	144 mm
Stroke Length(L)	=	72 mm
Speed(n)	=	3000 R.P.M
Maximum Explosion	=	37.20 Bar

TABLE 2
Material Properties

Parameters	Value
Tensile Strength MPa	900-1050 min
Fatigue Strength MPa	345
Poisson's Ratio	0.3
Yield Strength	550 min.

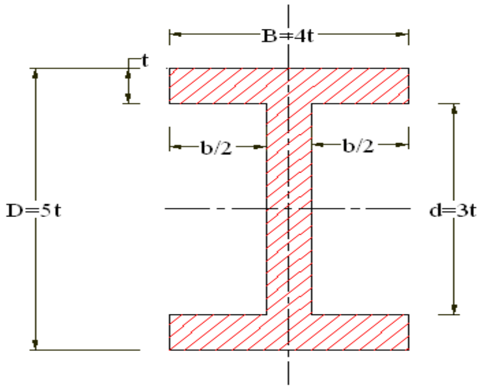


Fig. 12. Conrod I Section

$I_{XX} = 4I_{YY}$

$\frac{I_{XX}}{I_{YY}} = \frac{34.91t^4}{10.91t^4}$

AREA OF C/A (A)

$$\boldsymbol{A = (5t \times 4t) - (3t \times 3t) = 11t^2} \quad (8)$$

$$\boldsymbol{K = \sqrt{\frac{I}{A}} = 1.78t} \quad (9)$$

CRANK RADIUS (R)

$$r = \frac{Stroke\ of\ pison}{2} = \frac{L}{2} = \frac{72}{2} = 36\ mm$$

$$n' = \frac{Length\ of\ Conrod}{Crank\ radius} = 4$$

ANGULAR SPEED (W)

$$\omega = \frac{2\pi N}{60} = \frac{2\pi \times 3000}{60} = 314\ rad/sec$$

INERTIA FORCES OF RECIPROCATING PARTS (F)

$$\boldsymbol{F = \frac{1000WrV2}{gr} \cos\theta \pm \frac{\cos 2\theta}{n'}} \quad (10)$$

$$Wr = mg = 19.62N$$

CRANK VELOCITY (V)

$$V = r \times \omega = 36 \times 10^{-3} \times 314 = 11.30\ m/s$$

Now, Inertia forces, from Eq. (10)

$$F = \frac{1000 \times 19.62 \times (11.30)^2}{9.81 \times 36} \cos\theta \pm \frac{\cos 2\theta}{4}$$

$$F = 7094.11\ N$$

TOTAL FORCES ON CONROD

$$\boldsymbol{Fc = Fp - Fj = Fp - F} \quad (11)$$

$Fp = Piston\ force$
$Fp = P \times A$
$Fp = 13622.41\ N$

Therefore,
$Fc = 13622.41 - 7094.11 = 6528.3\ N$

Now,

$$\boldsymbol{Fc = \frac{fcA}{1 + K\left(\frac{l}{k}\right)^2}} \quad (12)$$

Where,
$A = Area\ of\ c/s\ = 11\ t2$
$L = Length\ of\ connecting\ rod = 144\ mm$
$K = Radius\ of\ gyration\ about\ X - X\ axis = 1.78\ t$
$fc = Allowable\ unit\ stress\ for\ desig\ N/mm2$

$$fc = \frac{Yield\ point\ stress}{FOS} = \frac{500}{2} = 275\ MPa$$

Therefore,

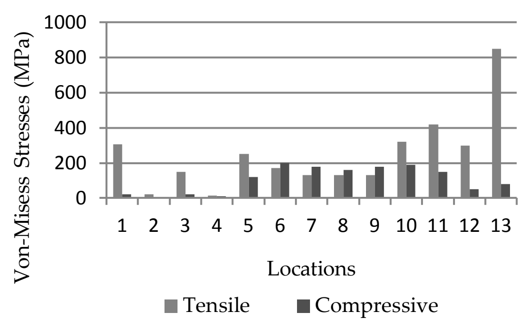
From Eq.(12)

$$3025t^4 - 6528.3t^2 - 68364.3 = 0$$

t = 2 mm

4. DISCUSSION

Fig. 3 shows the Von Misses stress distribution at different point locations of connecting rod. This plot is a type of summary of above calculated stresses, which tells about critical regions/ stresses.



Graph 3. Von Misses stress distribution at different point locations of conrod

Now to understand this graph or stress distribution, we will divide this connecting rod in 3 parts:

- The crank end Zone:
  As shown in graph 3 and fig. 6, very low or negligible stresses are generated near the bolt holes area. The highest von Mises stress in the region is about 177 Mpa.
- The small end zone:
  As shown in graph 3 and fig. 6, stresses near small end are 443 Mpa. Details of these results are made with table 4.3. Very high stress are generated at oil hole region and

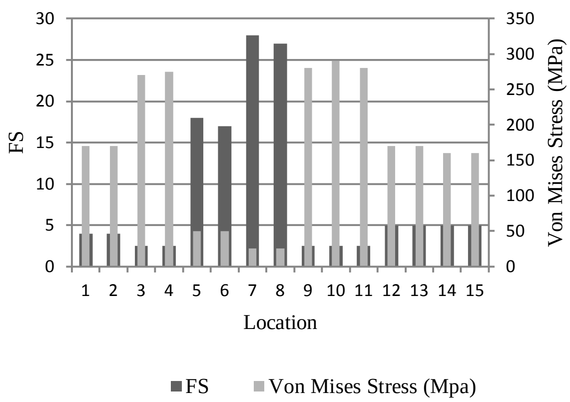
as shown in Graph 3 and fig. 7, it is 707 Mpa, which exceeds its yield strength.

- Web zone:
  As shown in graph 3 and figure 6, this region faces with 404 Mpa stresses.

Graph 4 shows the factor of safety (FS) on vertical axes, the ratio of yield strength to maximum von Mises stress under service operating condition for fifteen locations with reference to figure 6, over the entire operating load range of the connecting rod. Though the factor of safety for conrod design is unknown, factor of safety used for conrod can be known.

As shown in Graph 4, there is large margin for material removal from locations 1, 2, 12, 13, 14 and 15. But for locations 3, 4, 9, 10 and 11, the martial removal scope may or may not exist due to factor safety used.

Therefore by choosing different locations from connecting rod will definitely gives a scope for Weight reduction. Instead of



Graph 4. FS and the maximum Von Mises stress in the whole operating range.

considering stresses at just few locations, the stresses at all the nodes are considered. As per the results, analysis shown in graph 3 and graph 4, several iterations are performed in FEA for

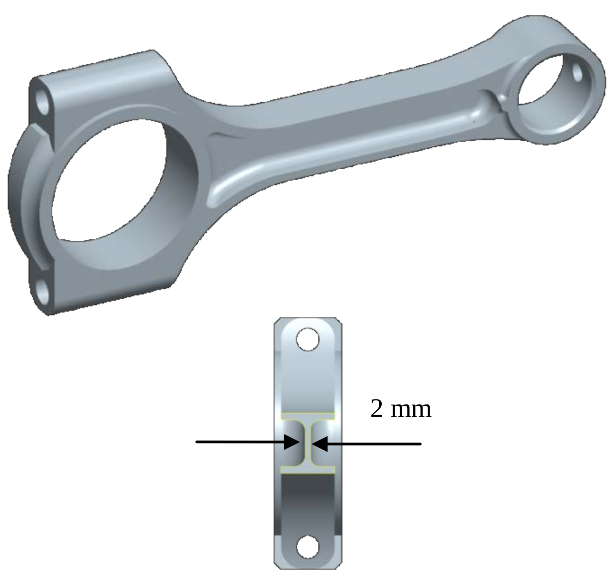


Fig. 13. Modified Connecting rod

obtaining weight reduction in the connecting rod geometry. Fig.13 shows a modification in connecting rod geometry having a mass reduced to 0.380 Kg which is less than that of original connecting rod. In original connecting rod, rib thickness was 3 mm which is reduced to 2 mm and also the crank area is also modified.

With this modified connecting rod, FEA is performed and the stress distribution is as shown in fig. 14.

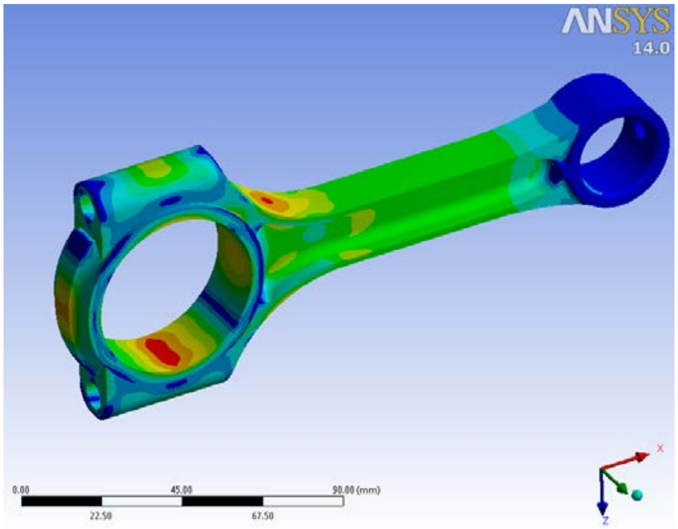


Fig. 14. FEA of Modified Connecting rod.

5. Conclusion

This overview research report studies the possibilities of weight reduction in forged steel connecting rod. For weight reduction process, fatigue strength and static strength were considered as a structural factor. First, the connecting rod was 3D modeled. Fatigue testing was performed, and corrections were made in the problematic area. After that load analysis was performed using ansys software.

From the results of the study, following conclusion can be made.

1. During fatigue testing, failure was occurred at the oil hole are and same thing was observed during FEA analysis of the connecting rod. But after changing angle at chamfer, failure was restricted as per the observation made in modified connecting rod geometry.
2. In connecting rod design, fatigue is most important factor.
3. The connecting rod is designed at its maximum engine speed and maximum gas pressure.
4. As per the results received from FEA, there is large margin of material removal from big end area, small end area and area connecting to small end the of connecting rod.
5. As per the results received from analytical calculations, there may be a scope of reduction in its “I-section” thickness.[4,5]
6. The new connecting rod geometry is lighter than original connecting rod.[3]
7. High carbon micro-alloyed steel i.e. C70S6 material has higher mechanical properties and better machinability and lower ductility.

ACKNOWLEDGMENT

The authors are very much grateful to mechanical Engineering department, Government College of Engineering Aurangabad, for giving some of the valuable suggestions for the design and analysis of the project with a proper guidance for completion of the project.

REFERENCES

1. Serag, S., Sevien, L., Sheha, G., and El-Beshtawi, I., 1989, *"Optimal design of the connecting-rod"*, *Modelling, Simulation and Control, B, AMSE Press*, Vol. 24, No. 3, pp. 49-63.
2. Pai, C. L., 1996, *"The shape optimization of a connecting rod with fatigue life constraint," Int. J. of Materials and Product Technology*, Vol. 11, No. 5-6, pp. 357-370.
3. Afzal, A. and Fatemi, A., 2004, "A Comparative Study of Forged Steel and PM Connecting Rods," *SAE Technical Paper Series*, Paper No. 2004-01-1529,
4. Leela Krishna Vegi and Venu Gopal Vegi. 2003," *Design and analysis of Connecting rod using forged steel", ISSN, Volume 4, Issue 6.*
5. RS Khurmi, JK Gupta, 2005, *"machine Design"*, S.Chand Publications, New Delhi, 4th edition.
6. ANSYS, "*Modeling and Meshing Guide*", Analysis Guide, Version 14.0. ANSYS, Inc. Canonsburg, pp. 28-265.
7. Pro-Engineer, *"Part Modelling Guide"* Pro-Engineer Guide, Version 4.0 Pro-E.
8. ASM Hand Book, 2005, "*Fatigue and Fracture Mechanics*", *Volume 19, ASM International, Material Park. Publications*", OH.(USA), pp. 15-989.
9. ASM Hand Book, 2005, "*Material selection and Design*", *Volume 20, ASM International, Material Park. Publications*", OH.(USA), pp. 385-521.