STATIC ANALYSIS & OPTIMIZATION OF CONNECTING ROD USING CAE TOOLS

Om Parkash
Department of Mechanical Engineering,
Amity University Haryana
om.mech8@gmail.com

ABSTRACT

The main objective of the work was to carry out the design optimization of connecting rod, which was obtained by changing the design variable in the existing connecting rod design. The model was developed in CATIA and saved in igs file and this file is further saved in parasolid x_t file using solid works for no data loss and then imported in ANSYS workbench. In this work finite element analysis of the single cylinder four stroke engine (specifically Hero Honda splendor) connecting rod was taken as a case study. The Von Mises stress, strain and total deformation determined for the same loading condition and compared with the existing results. The work consists of static structural analysis. The static analysis was carried out under axial and buckling load.

Key words - Connecting Rod, CATIA, Finite Element Analysis, Optimization, ANSYS workbench

---

INTRODUCTION

The function of the connecting rod is to transfer the reciprocating motion of the piston into rotary motion of the crankshaft. The maximum stress occurs in the connecting rod near the piston end due to thrust of the piston. The tensile and compressive stresses are produced due to the gas pressure. And bending stresses are produced due to centrifugal effect. So the connecting rods were designed generally of I-section to provide maximum rigidity with minimum weight. The maximum stress produced near the piston end could be decreased by increasing the material near the piston end. Pravardhan s. Shenoy and Ali Fatemi [1] presented the FE analysis procedure for optimization for connecting rod weight and cost reduction. A study was performed on a forged steel connecting rod with a consideration for improvement in weight and production cost. Weight reduction was achieved by an iterative procedure. The study results in an optimized connecting rod that was 10% lighter and 25% less expensive, as compared to the existing connecting rod. A. Mirehei et al. [2] carried out the fatigue analysis of connecting rod of universal tractor (U650). The objective of the research was to determine the lifespan of connecting rod due to cyclic loading. The results were carried out under fully reversed loading. The numbers of critical points were also located from where the crack propagation initiates. Allowable number of load cycles and using fully reverse Loading was gained 10^8. It was suggested that the results obtained could be useful to bring about modifications in the process of connecting rod manufacturing. Vasile George Cioata, Imre Kiss [3] presented a method used to verify the stress and deformation in the connecting rod using the finite element method with Ansys v.11. The study only analyses the connecting rod foot. The obtained results provided by this method were compared to the results obtained by classic calculation, in similar conditions of application, and after wards conclusions were drawn. Pranav G Charkha and Dr Santosh B Jaju[4] carried out the finite element analysis and optimization of connecting rod using ANSYS work bench 9.The study consists of two types of analysis. i.e. static analysis and fatigue failure analysis. The main objective of this study was to explore weight reduction opportunities for a production forged steel connecting rod. The results determine the ten gram reduction in weight and thereby reducing the inertia force of connecting rod. M.Omid S.S.Mohtasebi, S.A. Mireei and E. Mahmoodi[5] calculate the stresses and static displacements under the maximum compressive and tensile loading in the connecting rod of universal tractor U650.According to this study the critical point of connecting rod is the end of the shank and near piston pin hole. The main objective of the work to optimize the design of connecting rod. In this paper, only the static FEA of the connecting rod was performed. To give the optimum design parameter a number of random designs were generated by varying the values of the design variables within the specified limits till optimized design had been reached. The results were determined under the same weight and loading condition as for the existing connecting rod [4].

MATERIAL PROPERTIES

Material Data: Material for connecting rod is considered structural steel. This material is having following properties.
E = 200000 MPa Young's modulus
υ = 0.3 Poisson's ratio
ρ = 7850 kg/m Density

σ_y = 250 MPa Tensile Yield stress
σ_u = 460 MPa Tensile Ultimate stress
The behaviour of the material is assumed to be isotropic.

3D CAD MODEL OF CONNECTING ROD

The model of connecting rod was generated in CATIA V5 R19.
The steps in modelling are

- Creating the complete sketch on xy plane using two circle, line and fillets with the help of sketcher option.
- Fill the material in desired sketch with the help of pad command.
- Creation of hole on piston end and crank end with the help of pocket command.
- Creation of second sketch in shank portion of the connecting rod.
- Pocket the second sketch on both sides of the shank upto desired depth to make the I-section.
- Select an arbitrary plane at an angle in piston end and make circular entities on this plane.
- Creation of hole at the piston end with the help of pocket command.

The model of connecting rod is shown in fig.1

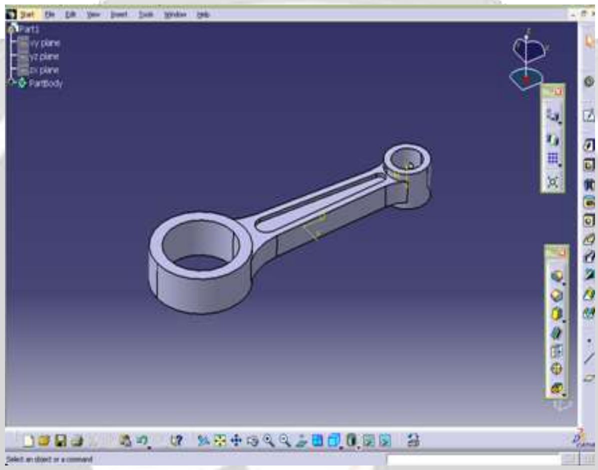

Fig. -1 :3D CAD model of connecting rod

Solid works is only used to save the file in parasolid x_t format.

MESHED MODEL

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called Nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as Mesh. This mesh along with material properties is used to mathematically represent the stiffness and mass distribution of structure. The 3-dimensional model of connecting rod is meshed in the ANSYS-V12. Meshed Model connecting rod is having 7820 nodes and 4266 elements.

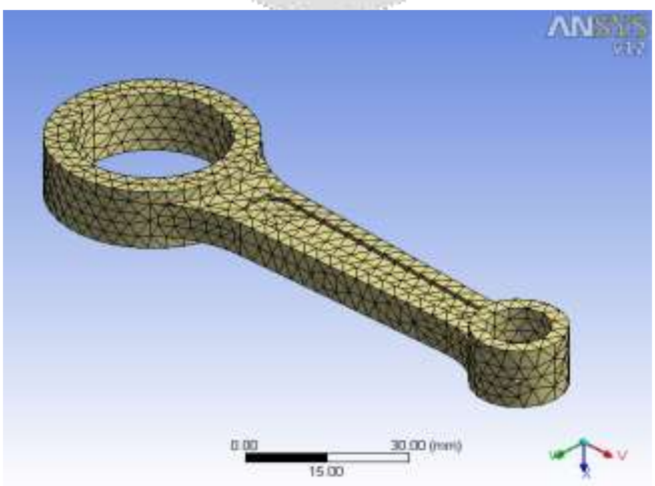

Fig-2: Meshed model of connecting rod

RESULTS AND DISCUSSIONS

The load applied at the piston end and cylindrical support was given at the crank end. The analysis carried out under axial and buckling loads. Here the tensile or compressive load was equal to 4319N and buckling load is equal to 21598N [4]. The distortion-energy theory is also called the Von-Mises theory, which is the most suitable theory to be used in ductile materials and after that comparisons were made for optimization purpose. The static results are shown in figures given below.

Static analysis (for load =4319N)

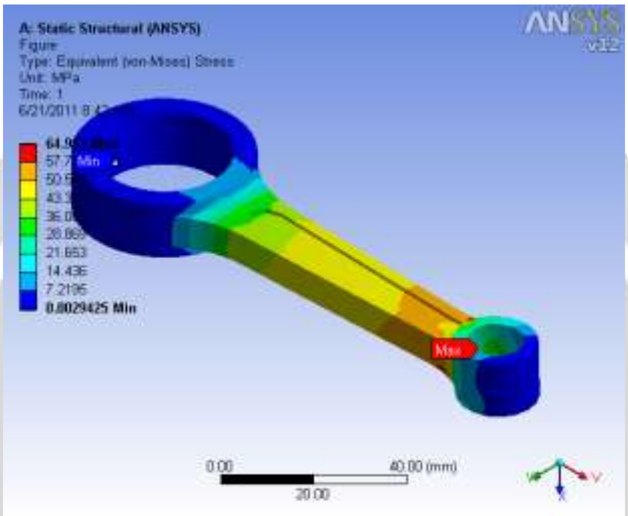

Fig-3: Equivalent Von Mises Stress (for load=4319N)

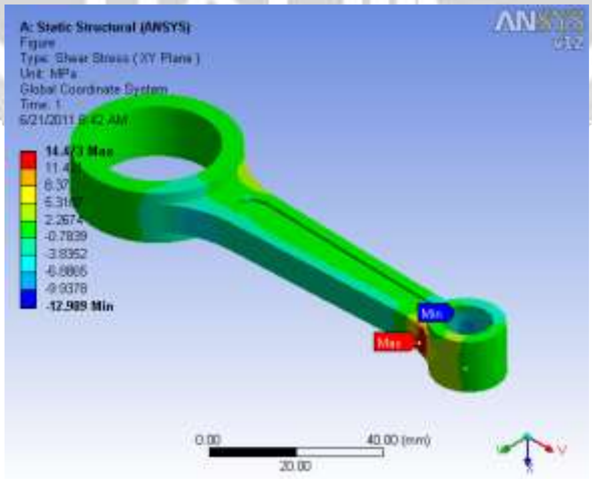

Fig - 4: Shear Stress (for load =4319N)

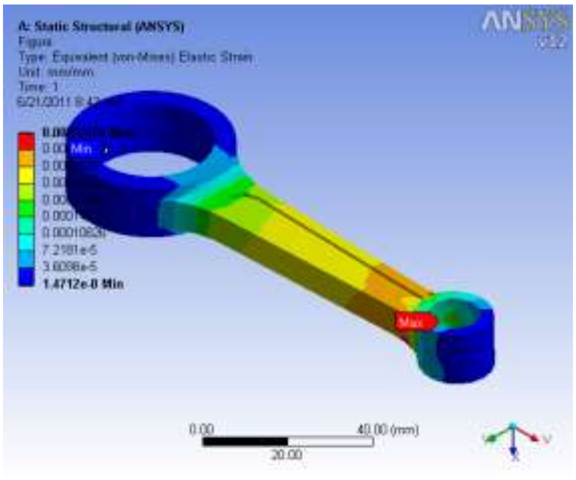

Fig-5: Equivalent Elastic strain (for load =4319N)

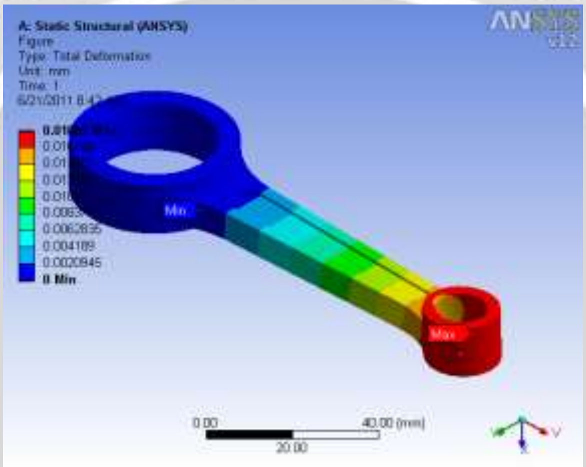

Fig-6: Total deformation (for load =4319N)

Static analysis (for load =21598N)

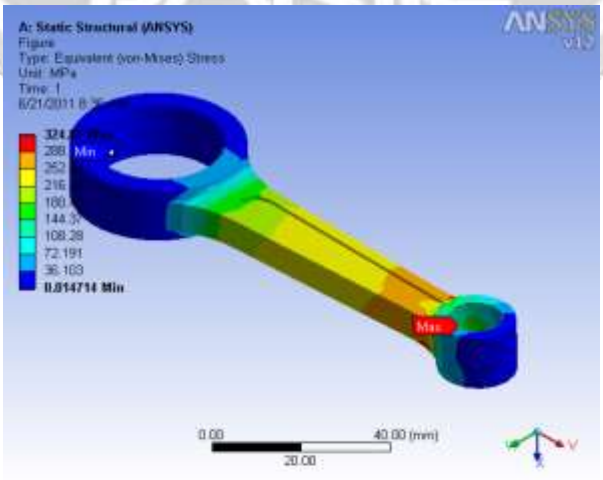

Fig-7: Equivalent Von Mises Stress (for load =21598N)

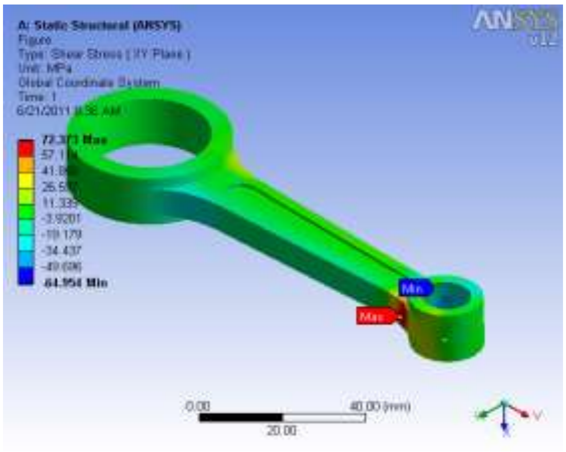

Fig-8: Shear Stress (for load =21598N)

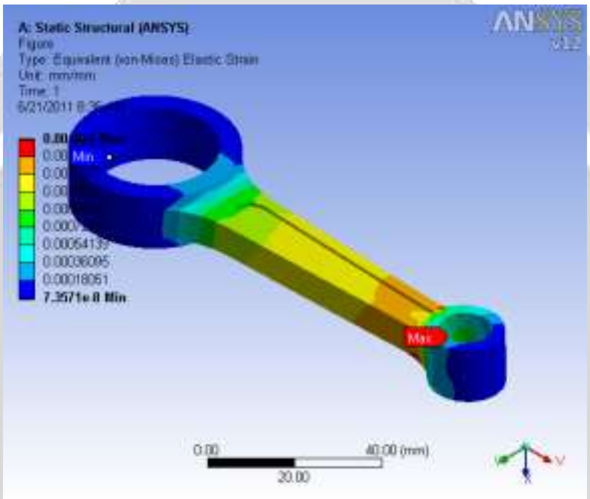

Fig-9: Equivalent Elastic strain (for load =21598N)

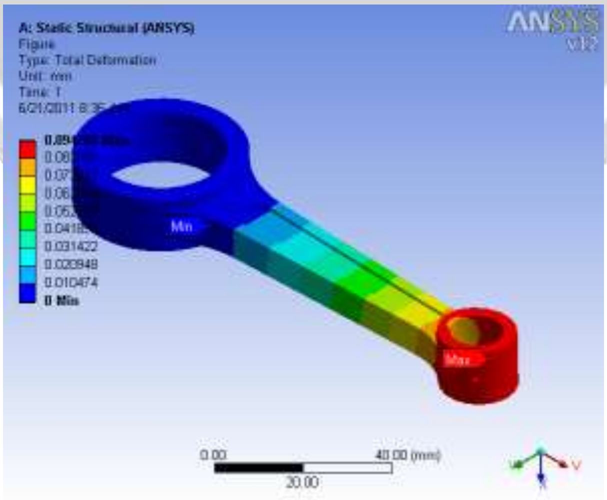

Fig -10: Total deformation (for load =21598N)

Optimized shape

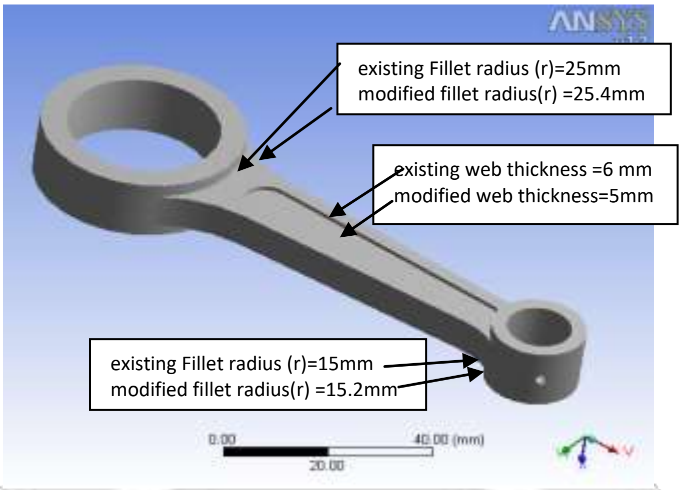

Fig-11: Optimized shape of connecting rod

Table 1 - Comparison Table (For Static Analysis)
For load =4319N

Parameter	Existing results	Improved FEA results	variation
Equivalent Von Mises stress	76.22 MPa	64.952MPa	14.78%
Shear stress	16.44 MPa	14.473MPa	11.96%
Euivalent Elastic strain	3.8e-4 mm/mm	3.2e-4 mm/mm	15.78%
Total deformation	_	0.01885mm	

Table 2
For load =21598N

Parameter	Existing results	Improved FEA results	variation
Equivalent Von Mises stress	381.17 MPa	324.81MPa	14.78%
Shear stress	82.21 MPa	72.37MPa	11.96%
Equivalent elastic strain	1.91e-3 mm/mm	1.6e-3 mm/mm	15.78%
Total deformation	_	0.09425mm	_

Table 3
Weight reduction in %

Sr. No.	Existing weight (kg)	Optimized weight (kg)	Reduction in weight
1	0.13kg	0.11kg	0.02kg or 20gm

CONCLUSION AND FUTURE SCOPE

1. It was found that the design parameter of connecting rod with modification gives sufficient improvement in the existing results.
2. The reduction in weight is obtained as 20gm which is not so more but signify the reduction in cost and inertia forces.
3. The other new design can also be obtained by changing the other design variables of connecting rod i.e.diameters of both ends,size of pin hole etc.
4. A connecting rod made up of composite material which is light in weight can also be analyzed under the same loading conditions for the reduction of inertia force.

REFERENCES

[1] Pravardhan S. Shenoy and Ali Fatemi "Connecting Rod Optimization for Weight and Cost Reduction", 2005, SAE International.
[2] A. Mirehei, M. Hedayati Zadeh, A. Jafari, and M. Omid "Fatigue analysis of connecting rod of universal tractor through finite element method (ANSYS)" Journal of Agricultural Technology,2008, Vol.4, pp 21-27.
[3] Vasile George Ciotia, Imre Kiss "Computer aided design of the connecting rod".
[4] Mr. Pranav G Charkha and Dr Santosh B Jaju "Analysis & Optimization of Connecting Rod" Second International Conference on Emerging Trends in Engineering and Technology, ICETET-09.
[5] M.Omid S.S.Mohtasebi, S.A. Mireei and E. Mahmoodi "fatigue analysis of connecting rod of U650 tractor in the finite element code ANSYS"journal of applied science 8(23):4338-4345,2008ISSN 1812-5654.
[6] Afzal, A., 2004, "Fatigue Behavior and Life prediction of Forged Steel and PM Connecting Rods," Master's Thesis, University of Toledo
[7] P. C. Sharma and Dr. D. K. Aggarwal "Machine Design", S.K. kataria and sons.
[8] ANSYS Workbench V12 Reference manual.
[9] CATIA V5R19 Reference manual.
[10] Solid works Reference manual.

IJARIIE