Stress analysis and optimization of connecting rod using finite element analysis

Yogesh Kumar Bharti¹, Vikrant Singh¹, Afsar Hussain¹, Dipanshu Singh¹, Shyam Bihari Lal², Satish Kumar Dwivedi²

¹B. Tech. Students, Department of Mechanical Engineering, Buddha Institute of Technology
²Asst. Professor, Mechanical Engineering, Buddha Institute of Technology, GIDA, Gorakhpur - 273209 (U.P.) India

ABSTRACT

Design is process by which human desired fulfill. Design of connecting rod is very complex because the connecting rod works in very complicated condition. To obtain a best & suitable design of connecting rod to sustain various stress & forces, finite element analysis suggest the minimum design specification with the help of ANSYS. For this purpose Pro/Engineer wildfire 4.0 develop a solid modeling and ANSYS v13.0 for finite element analysis. Optimization is process which provides best suited option and result. After observation and modeling the analysis for optimization was performed. It relates with weight comparison & reducing of size of connecting rod by analysis various factors which affect the performance like stress.

Key words- Pro-E, ANSYS, Stress analysis, Connecting rod, FEA.

I. INTRODUCTION:

The intermediate component between crank and piston is known as connecting rod. The objective of C.R. is to transmit push & pull from the piston pin to the crank pin and then converts reciprocating motion of the piston into the rotary motion of crank. The components are big shank, a small end and a big end. The cross section of shank may be rectangular, circular, tubular, I-Section, + -section or ellipsoidal-Section. It sustains force generated by mass & fuel combustion. The resulting bending stresses appear due to eccentricities, crank shaft, case wall deformation & rotational mass.

FEA approach deals with structural analysis along with various parameters which affects its working & define best solution to overcome the barriers associated with it. The structural analysis allows stresses & strains to be calculated in FEA, by using the structural model. The structural analysis performed to create high & low stresses region from the input of the material, loads, boundary condition. FEA approach was adopted in structural analysis to overcome the barriers associated with the geometry & boundary condition. It is used to improve optimize design.
The main objective of this work is to determine shear stresses and optimization in the existing connecting rod, which are in different cross-section as plus (+) section, I-section and ellipsoidal section. The failures of existing design suggest the minimum design changes in the existing connecting rod.

II. MATERIAL STUDY:

The C-70 materials have been widely present in culture. Alloysing elements in the material enables hardening of forged connecting rod when they undergo controlled cooling after forging. The properties of material are initial input for optimization task thus it play a very important role in optimization task. Connecting rod was design & modeled by using Pro/E 4.0 v. It was then imported to ANSYS for analysis.

Fig. 1: Equivalent (Von-Mises) stress
Fig. 2: Shape Optimization
Fig. 3: Equivalent (Von-Mises) Stress

Fig. 4: Shape Optimization

Fig. 5: Equivalent (Von-Mises) Stress

Fig. 6: Shape Optimization
Finite element analysis of connecting rod has been carried out for investigation of critical stresses, elastic strain and total deformation.

III. OPTIMIZATION:

Objective of the optimization task was to minimize the mass of the connecting rod of various cross-section such as I-section, plus(+)section and ellipsoidal-section. The weight of optimized connecting rod is certainly lower than the weight of original connecting rod. The factors have been addressed during the optimization- load factor, stresses under the loads.

Objective: Minimize Mass and Cost

Subject to:

a) Compressive load = peak compressive gas load.

b) Maximum stress < Allowable stress

c) Side constraints (Component dimensions)

d) Manufacturing constraints

e) Buckling load > Factor of safety x the maximum gas load

f) Optimized Model

Shape Results (Ellipsoidal Section):

<table>
<thead>
<tr>
<th>Name</th>
<th>Scope</th>
<th>Target Reduction</th>
<th>Predicted Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>“Geometry”</td>
<td>20%</td>
<td>10.56%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Name</th>
<th>Original</th>
<th>Optimized</th>
<th>Marginal</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>1.9274 kg</td>
<td>1.7192 kg</td>
<td>1.344e-02 kg</td>
</tr>
</tbody>
</table>

Shape Results (I-Section):

<table>
<thead>
<tr>
<th>Name</th>
<th>Scope</th>
<th>Target Reduction</th>
<th>Predicted Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>“Geometry”</td>
<td>20%</td>
<td>11.23%</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Name</th>
<th>Original</th>
<th>Optimized</th>
<th>Marginal</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>1.7385 kg</td>
<td>1.5433 kg</td>
<td>1.22e-02 kg</td>
</tr>
</tbody>
</table>
**Shape Results (+ Section)**

<table>
<thead>
<tr>
<th>Name</th>
<th>Scope</th>
<th>Target Reduction</th>
<th>Predicted Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>“Geometry”</td>
<td>20%</td>
<td>12.65%</td>
</tr>
</tbody>
</table>

**Table: 6**

<table>
<thead>
<tr>
<th>Name</th>
<th>Original</th>
<th>Optimized</th>
<th>Marginal</th>
</tr>
</thead>
<tbody>
<tr>
<td>“Shape Finder”</td>
<td>2.1083 kg</td>
<td>1.8416 kg</td>
<td>1.034e-02 kg</td>
</tr>
</tbody>
</table>

**RESULT AND DISCUSSION:** As compared to table no. 1 to 6. It will be the most significant finding was find from the table no 1 to 6 according to mass wise analysis shown I table. When target reduction kept that ellipsoidal has removed less amount of mass(10.56%) and more mass will be removed shown in + section in table no 5. The main theme at + section more superior than the other to the masses among them but the target is shifted on weight among all the section than will find that + section superior to other section. At a glance after the optimization of connecting rod it will be predicated equivalent stress is more suitable in I section. Therefore I section will be consider for connecting rod design under the sustain various stresses and forces.

**CONCLUSION:**

The result find out by FEA tool in this present work the value of Equivalent Stress, Shear Elastic Strain and Total Deformation. The weight optimization of connecting rod, consider following properties. This work investigates the weight and cost reduction opportunities that steel C-70 connecting rods offer. The maximum stresses developed near the big end connecting rod, it the region which is more susceptible for failure. The following conclusions can be drawn from this work.

1. The parameter taken in account to reduce 20% weight of steel C-70 connecting rod.
2. The stress is found maximum at the crank end so the material is increased in stressed portion to minimize the stress.
3. The weight of the steel C-70 connecting rod (I-section) is reduced by 0.1952 kg, which is about 11.23% of original weight of connecting rod.
4. The weight of the steel C-70 connecting rod (+-section) is reduced by 0.2667 kg, which is about 12.65% of original weight of connecting rod.

5. The weight of the steel C-70 connecting rod (ellipsoidal-section) is reduced by 0.2082 kg, which is about 10.56% of original weight of connecting rod.

REFERENCES:


[27] Dr. NWM Bishop and Dr. F. Sherratt, 2000, “Finite Element Based Fatigue Calculations” NAFEMS, RLD Ltd., Hutton Roof, Eglinton Rod, Tilford Farnham, Surrey, GU102DH.