Design and development of optimized sprocket for Track hoe

Mr. Laxmikant P. Sutar¹, Prof. Prashant G. Karajagi², Prof. Rahul Kulkarni³

¹PG Student, Siddhant College of Engineering, Pune, India
²Assistant Professor, Dr. D.Y. Patil School of Engineering, Pune, India
³Head of Department, Siddhant College of Engineering, Pune, India

Abstract — in this thesis the weight of Sprocket for Track hoe is optimized and validated experimentally for various torque condition. The current sprocket was having the wearing in the tooth region. The various field complaints are studied and then decided to redesign the current sprocket to sustain the durability without compromising the performance. Firstly for the weight optimization the analytical data should be developed as datum for re design. The loading conditions for the sprocket are studied which are useful for FEA. The sprocket is validated by using test special purpose rig with strain gauges and different loading conditions. The weight of the sprocket is optimized and the stress values derived from FEA, Analytical and experimental method are compared. The induced stresses are less than the yield stresses hence design is safe.

Keywords — Weight optimization, Finite Element Analysis, NX.

I. INTRODUCTION

TRACK HOE is the earth moving equipment used for material handling and civil work. The under carriage consist of sprocket which is a toothed wheel that engages with a chain or track to transmit rotary motion. NX (Siemens product) is used for CAD modeling and Finite element analysis. For optimization only stresses developed in the sprocket is considered and not the chain and sprocket assembly. The target value for the weight optimization is limited to 25% and the stressed (Von misses) allowed is not more than 75% of the ultimate strength of sprocket material. Finite element analysis results are verified by using analytical method and by the experimental method using strain gauges.
III. MESHING

Meshing is carried out in NX Nastran with CTERA(10) element type and size 14.4mm.

IV. MATERIAL INFORMATION

1. Material: Structural steel A36
2. Material type: Isotropic
3. Young’s Modulus: 200 GPA
4. Poisson’s ratio: 0.26
5. Shear Modulus: 79.3 GPA
6. Yield strength: 250 N/mm2
7. Ultimate tensile strength: 450 N/mm2
8. FOS: 2
9. Upper limit = 30.0mm, Lower limit = 10.0mm

V. BOUNDARY AND LOAD CONDITIONS

Bolt pretension assigned is 63000N to all mounting holes. Internal diameter of sprocket is held fixed [1]. Instead of modeling and meshing the chain, the load acting on teethes assigned is 333000/9 = 37000N

VI. GEOMETRY OPTIMIZATION BY NX NASTRAN

Geometry optimization is a process that helps to find the best solution for a design objective that we specify, such as reducing weight. To set up the optimization, we specify design constraints, such as minimizing stress or displacement, and design variables, such as feature dimensions. The software performs a series of iterations, adjusting the design variables within the design constraints, until it converges on a solution.

Based on our design objective, software modifies our master model geometry or idealized part geometry by adjusting any of the following design variables:

- Section properties of 1D element
- Shell properties of 2D elements
- Feature dimensions
- Sketch dimensions
- NX expressions

The software uses the results from an analysis, typically displacement or stress, as input to evaluating the design constraints. If a solution has multiple sub cases, the optimization uses only the results from the first sub case. During every iteration, the software compares the value of each constraint attribute against its defined limit. If a constraint value falls outside its limit, the model is considered to be in an invalid state. The optimization returns the model to a valid state, and in the next iteration, tries different values for the design variables.

After we solve the optimization solution, we can analyze the results in Post Processing.

Following are the input parameters defined for Geometry optimization:

1. Define Objective: Weight 400.5 N
2. Define Constrain: Upper limit = 534.329243 N
3. Define Design variables: Upper limit = 30.0mm, Lower limit = 10.0mm (Flange thickness)
4. Control Parameter: Max. number of iteration=20

Figure 6.2 Optimization summary

It is observed from the generated optimization report is that the converged result obtained at the seventh iteration. The resulted flange thickness is 11.75mm.

VII. MODIFIED CAD MODEL

By using the results from optimization summary, the pockets in the flange area are added in the sprocket with 11.75mm thickness. The weight for new sprocket model is 40.86 Kg. The model is meshed again and applied the same boundary conditions to get the new results.

VIII. FEA RESULTS

The resulted max displacement is 0.1122mm which is comparatively very small. The design is hence safe for the displacement criteria.

The resulted elemental Von-Mises stress is 161.19 N/mm².
IX. ANALYTICAL CALCULATIONS

The analytical calculations are carried out based on the Lewi’s gear equation [4] to validate the FEA results obtained from NX Nastran.

\[
\sigma = \frac{6 \times Ft \times h}{b \times h^2}
\]  

(1)

1. Max. gear box Torque at min. speed = 410000 N-m
2. Radius of wheel= 200 mm
3. Force (Ft) = Torque/ Radius = 4100000 / 200 
   Ft = 20500 N
4. h (height of tooth ) = 40 mm
5. b (Width of tooth) = 35 mm
6. t (Thickness of tooth) = 30mm
7. Refer to "(1)"

\[
\sigma = \frac{6 \times 2500 \times 40}{35 \times 30^2}
\]

\[
\sigma = 156.19 \text{ N/mm}^2
\]

X. EXPERIMENTAL SETUP

The test rig is used for the experimental validation. Test rig consist of hydraulic cylinder, mounting block for cylinder, strain gauges, FFT analyzer, bearing mounted rotating structure, fixed bolted structure for sprocket etc.

XI. EXPERIMENTAL RESULTS

The experimentation is carried out for three different load cases and the induced stresses are obtained at the different strain gauge positions.

<table>
<thead>
<tr>
<th>TABLE I Experimental Results</th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Case</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ram Pressure (N)</td>
<td>37000</td>
<td>35489.8</td>
<td>52102.0</td>
</tr>
<tr>
<td>Gauge-1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Disp.(mm)</td>
<td>0.049</td>
<td>0.047</td>
<td>0.069</td>
</tr>
<tr>
<td>Stress (N/mm2)</td>
<td>40.967</td>
<td>39.29</td>
<td>57.688</td>
</tr>
<tr>
<td>Gauge-2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Disp.(mm)</td>
<td>0.048</td>
<td>0.045</td>
<td>0.068</td>
</tr>
<tr>
<td>Stress (N/mm2)</td>
<td>26.896</td>
<td>25.2</td>
<td>38.102</td>
</tr>
<tr>
<td>Gauge-3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Disp.(mm)</td>
<td>0.06</td>
<td>0.07</td>
<td>0.08</td>
</tr>
<tr>
<td>Stress (N/mm2)</td>
<td>40.908</td>
<td>39.08</td>
<td>44.669</td>
</tr>
</tbody>
</table>
XII. CONCLUSION

1. It is observed that the weight of the old sprocket was 54.86 Kg while new sprocket is 40.86 Kg.
2. The overall optimization in weight is 25.5%.
3. The results obtained from FEA, Analytical calculations and experimental are compared.

<table>
<thead>
<tr>
<th>Sr. No</th>
<th>FEA Stress (N/mm²)</th>
<th>Analytical stress (N/mm²)</th>
<th>Experimental Stress (N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>161.19</td>
<td>156.19</td>
<td>168.416</td>
</tr>
</tbody>
</table>

It is observed that the results obtained from the analytical calculations, FEA and experimental for the stress value are close enough.

4. As the induced stress values are less than the yield strength of material hence design is safe.
5. Cost saving details- 25 % saving per year

<table>
<thead>
<tr>
<th>Sr . No</th>
<th>Cost (Rs)</th>
<th>QTY (Piece/Year)</th>
<th>Cost</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12000</td>
<td>2000</td>
<td>24000000</td>
<td>Before</td>
</tr>
<tr>
<td>2</td>
<td>9000</td>
<td>2000</td>
<td>18000000</td>
<td>After</td>
</tr>
</tbody>
</table>

XIII. ACKNOWLEDGEMENT

The authors would like to thank the Siddhant College of Engineering and for the timely and important help in the field of Design & Finite Element Analysis. The authors also wish to thank the reviewers for their very detailed review of the manuscript and for their help in clarifying some aspects of the reported results.

XIV. REFERENCES


