Design and development of optimized sprocket for Track hoe

Mr. Laxmikant P.Sutar, Prof. Prashant.G. Karajagi

***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 stressed 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.

Mr. Laxmikant P.Sutar
Mechanical Engineering Department,
Siddhant College of Engineering,
Sudumbare, Pune,
Tal-Maval, Dist-Pune,
Maharashtra, India-412109.
(e-mail: laxmikant_sutar@yahoo.com)

Prof. Prashant.G. Karajagi
Mechanical Engineering Department,
Siddhant College of Engineering,
Sudumbare, Pune,
Tal-Maval, Dist-Pune,
Maharashtra, India-412109.

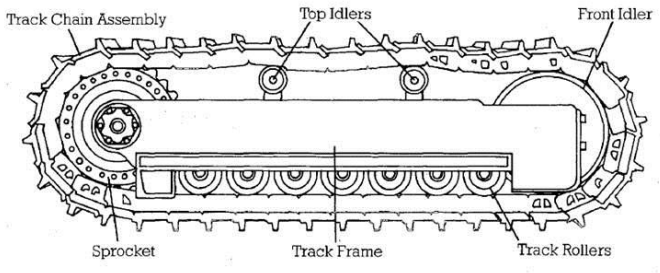


Fig.1.1 Generic Track hoe Undercarriage assembly.

II. GEOMETRY OF CURRENT SPROCKET

Current sprocket contains the solid flange section. The weight of the sprocket is 54.86Kg and the maximum elemental stresses developed is 161.19 N/mm^2 . The stresses developed are well below than the 75% of ultimate strength of structural steel i.e. 400-500 N/mm^2 .

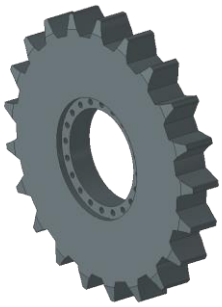


Fig.2.1 Current sprocket CAD geometry.

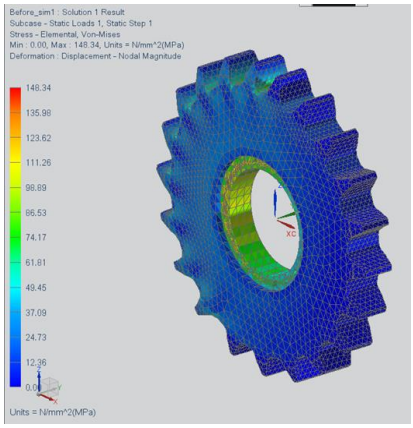


Fig2.2 Current sprocket CAD geometry.

III. MESHING

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

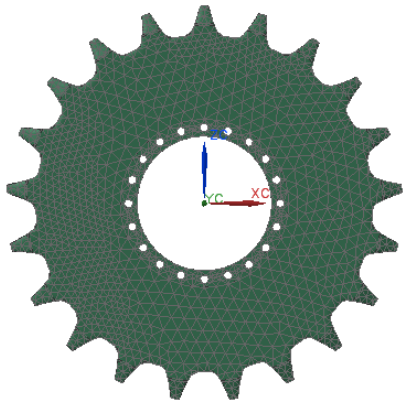


Fig.3 Meshed model.

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/mm^2
7. Ultimate tensile strength: 450 N/mm^2
8. FOS:2
9. Upper limit = 30.0mm, Lower limit = 10.0mm

V. BOUNDARY AND LOAD CONDITIONS

Bolt pretension assigned is 63000N to all to 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

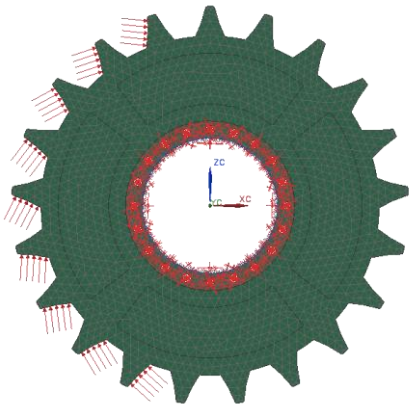


Fig.5 Boundary Conditions

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 elements
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 each 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

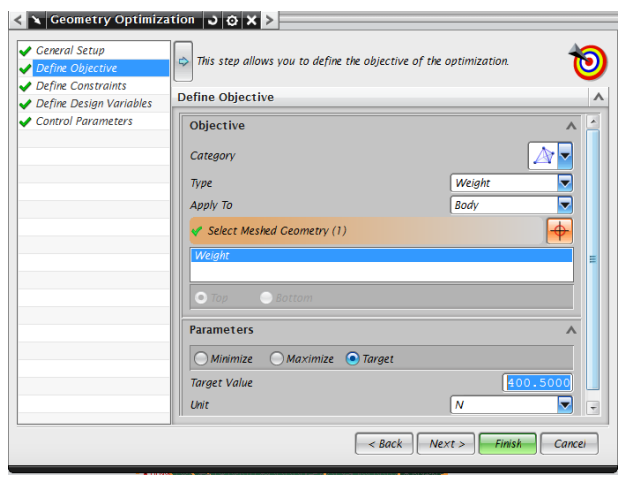
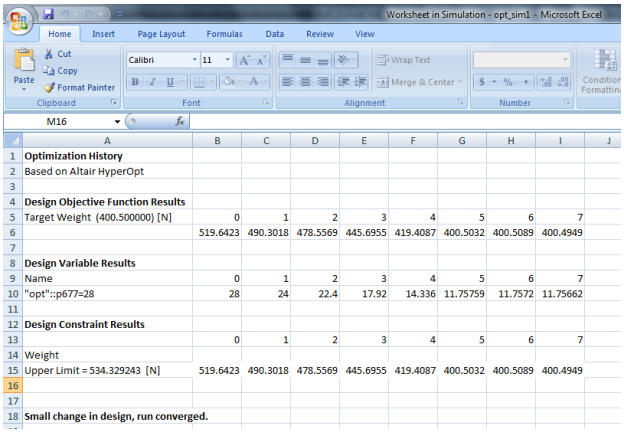


Fig. 6.1 Meshed model.

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



	A	B	C	D	E	F	G	H	I	J
1	**Optimization History**									
2	Based on Altair HyperOpt									
3										
4	**Design Objective Function Results**									
5	Target Weight (400.500000) [N]	0	1	2	3	4	5	6	7	
6		519.6423	490.3018	478.5569	445.6955	419.4087	400.5032	400.5089	400.4949	
7										
8	**Design Variable Results**									
9	Name	0	1	2	3	4	5	6	7	
10	"opt"::p677=28	28	24	22.4	17.92	14.336	11.75759	11.7572	11.75662	
11										
12	**Design Constraint Results**									
13		0	1	2	3	4	5	6	7	
14	Weight									
15	Upper Limit = 534.329243 [N]	519.6423	490.3018	478.5569	445.6955	419.4087	400.5032	400.5089	400.4949	
16										
17										
18	**Small change in design, run converged.**									

Fig.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.

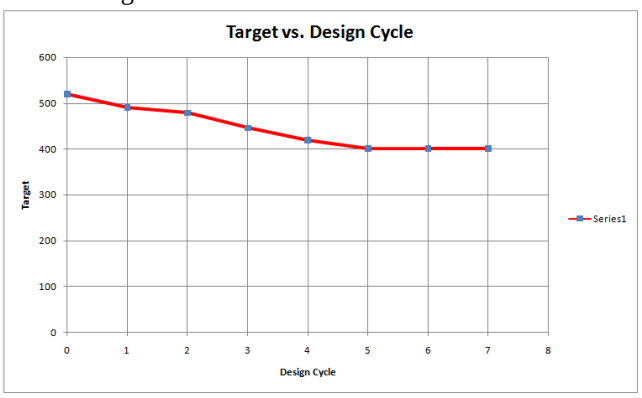


Fig.6.3 Target Vs Design Cycle

Figure 6.3 shows the convergence of targeted weight against the design iterations.

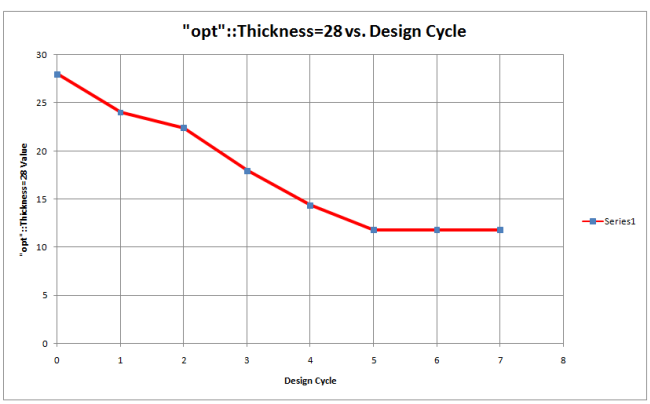


Fig.6.4 Target Vs Design Cycle

Figure 6.3 shows the convergence of thickness against the design iterations.

VII. MODIFIED CAD MODEL

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

Fig.7 Modified CAD model

VIII. FEA RESULTS

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

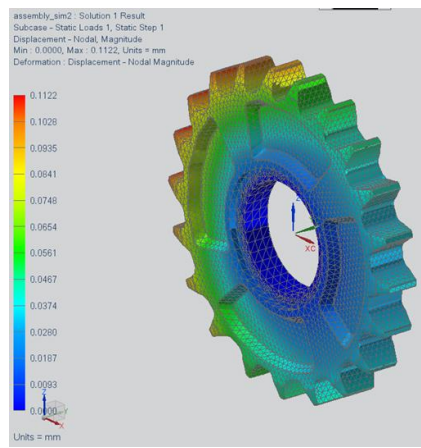


Fig.8.1 Displacement-Nodal

The resulted elemental Von-Mises stress is 161.19 N/mm^2.

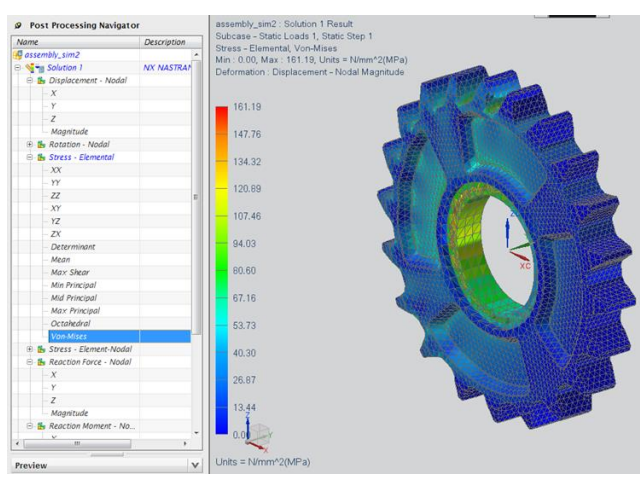


Fig.8.2 Stress-Elemental (Von-Mises)

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.

$$\sigma = \frac{6 \times Ft \times h}{b \times h^2} \quad (1)$$

1Max. gear box Torque at min. speed = 4100000 N-m

1. Radius of wheel= 200 mm
2. Force (Ft) = Torque/ Radius

   = 4100000 / 200

   Ft = 20500 N

3. h (height of tooth) = 40 mm
4. b (Width of tooth) = 35 mm
5. t (Thickness of tooth) = 30mm

Refer to "(1)"

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

σ = 156.19 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.

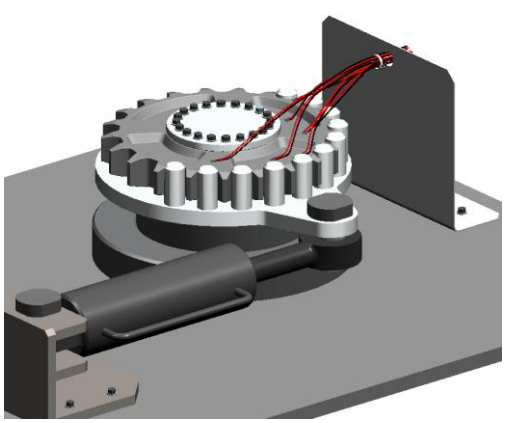


Fig.10.1 Test rig

Strain gauges will be applied at the positions in the area where the induced stresses will be maximum [3].

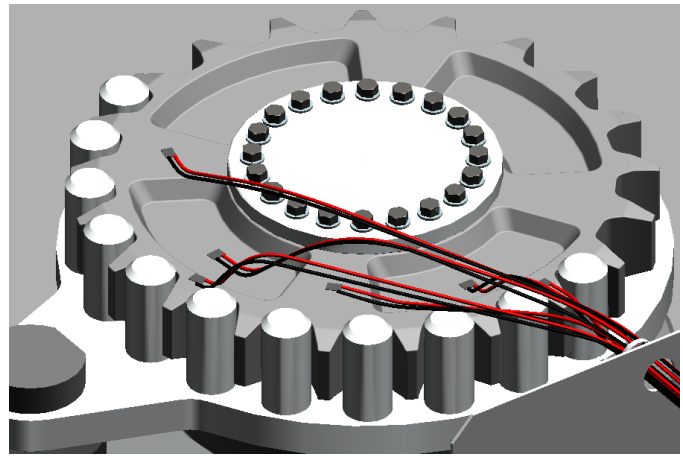


Fig.10.2 Strain gauge positions

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 I Experimental Results

Load Case		1	2	3
Ram Pressure (N)		37000	35489.8	52102.0
Gauge-1	Disp.(mm)	0.049	0.047	0.069
	Stress (N/mm2)	40.967	39.29	57.688
Gauge-2	Disp.(mm)	0.048	0.045	0.068
	Stress (N/mm2)	26.896	25.2	38.102
Gauge-3	Disp.(mm)	0.06	0.07	0.08
	Stress (N/mm2)	40.908	39.08	44.669
Gauge-4	Disp.(mm)	0.03	0.04	0.05
	Stress (N/mm2)	101.05	134.7	168.416
Gauge-5	Disp.(mm)	0.053	0.051	0.073
	Stress (N/mm2)	84.75	81.5	116.731
Gauge-6	Disp.(mm)	0.053	0.052	0.073
	Stress (N/mm2)	76.163	74.7	104.903

XII. CONCLUSION

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

TABLE II Results Comparison

Sr. No	FEA Stress (N/mm^2)	Analytical stress (N/mm^2)	Experimental Stress (N/mm^2)
1	161.19	156.19	168.416

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

3. As the induced stress values are less than the yield strength of material hence design is safe.

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

[1] M. Chalupa, R. Vlach "Simulation of vehicle track dynamic loading" 18thInternational Conference, ENGINEERING MECHANICS 2012 pp. 507–512 Svratka, Czech Republic, May 14 – 17, 2012

[2] Milan CHALUPA, Josef VEVERKA " Influence of Design Parameters on Vehicle Track Dynamic Loading" Final report of Research program POV MO 03171100014. VA Brno 2001.

[3] Mr. Swapnil Ghodake, Prof. Prashant Deshpande" Optimization of Excavator Sprocket and it's Validation by Test Rig Concept" ISBN: 978-1-63248-028-6 doi: 10.15224/ 978-1-63248-028-6-03-80

e33[4] "http://www.engineersedge.com/gears/lewis-factir.html"