© DEC 2025 | IRE Journals | Volume 9 Issue 6 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I6-1712485

Analysis and Optimization of sprocket using FEA

PRAVIN HUJARE[1], SHUBHANKAR KALE[2], ANUSHKA KADLAG[3], SANIKA KALE[4], ATHARVA KAKAD[5], SAHIL KALE[6]
[1, 2, 3, 4, 5, 6]*Mechanical Engineering, Vishwakarma Institute of Technology*

***Abstract:* This study uses FEA to perform an in-depth mechanical analysis and subsequent geometrical optimization of a typical sprocket with the aim of effectively and significantly improving its performance and durability under service conditions. This was done to consider in detail the critical mechanical parameters, including stress concentrations, total deformation, and the resultant factor of safety when the component is subjected to a series of representative service loads. Further to the initial analysis, the geometry of the sprocket was strategically refined to minimize mass without any deleterious reduction in structural integrity. The resultant optimized design exhibited a marked improvement in both its load-bearing capacity and operational efficiency, thus conclusively establishing the effectiveness of an FEA-driven approach toward mechanical component design and optimization.**

***Index Terms:* Computer-Aided Design (CAD), Finite Element Analysis (FEA), Mechanical Optimization, Stress Analysis**

I. INTRODUCTION

Sprockets are very important mechanical components in the transmission of power and find applications in virtually every area of bicycles, industrial machinery, and conveyor systems. The design of a good sprocket is indispensable for smooth operation, long service life, and high energy efficiency. However, conventional designs face numerous problems, such as rapid wear, high-stress concentrations, and material over-utilization, which may culminate in operational failures or increased production expenses.

The integration of FEA-a powerful modern computational tool-has transformed the ways of evaluation and refinement of mechanical components. This research project focuses on the FEA-based analysis and optimization of a sprocket. The methodology simulates **actual loading conditions** to precisely locate the areas of maximum stress and significant deformation. These diagnostic results inform the subsequent geometrical refinement process, aimed at improving inherent strength while achieving a reduction in material waste.

The resultant optimized design thus offers superior stress distribution, an improved factor of safety, and allows the manufacturing of products at a more economical cost. This work, therefore, highlights how important it is to adopt FEA in achieving advanced component designs.

IDEA

The necessary background knowledge and contextual data were meticulously gathered through the following methods:

1. Systematic Literature Review: A comprehensive review was conducted on published academic journals, peer-reviewed conference proceedings, and specialized industrial reports. This activity was crucial for achieving a deep understanding of:
    - Existing sprocket designs and established performance benchmarks.
    - Common failure mechanisms (e.g., pitting, fatigue, wear) observed in operational sprockets.
    - Prior optimization methodologies and their reported effectiveness.
2. Targeted Online Research: Various high-quality academic databases, technical repositories, and engineering articles were systematically searched. This focused effort was aimed at accumulating specific information regarding the successful application of Finite Element Analysis (FEA) in the design and refinement of sprockets and gears.
3. Industry Engagement and Professional Development: Participation in relevant workshops, seminars, and webinars focused on mechanical design, power transmission dynamics, and advanced FEA techniques provided essential insights into

© DEC 2025 | IRE Journals | Volume 9 Issue 6 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I6-1712485

current industry best practices and emerging technological trends.

4. Theoretical and Terminological Foundation: A dedicated effort was made to thoroughly study and internalize critical scientific terminology and fundamental engineering theories. Key concepts examined included:

II. The difference between alternating stress (σ_a) and mean stress (σ_m).

III. o Fatigue life prediction criteria, namely Soderberg and Modified Goodman criteria, in order to ensure their accurate and appropriate application during the subsequent FEA-based fatigue analysis.WRITE DOWN YOUR STUDIES AND FINDINGS

Equations

Alternating Stress (σa):

$\sigma a = (\sigma max - \sigma min) / 2$

Mean Stress (σm):

$\sigma m = (\sigma max + \sigma min) / 2$

Soderberg Criterion (more conservative):

$(\sigma a / \sigma'e) + (\sigma m / \sigma y) \leq 1 / FOS$

Modified Goodman Criterion:

$(\sigma a / \sigma'e) + (\sigma m / \sigma u) \leq 1 / FOS$

σa = alternating stress

σm = mean stress

$\sigma'e$ = corrected endurance limit

σu = ultimate tensile strength

σy = yield strength

FOS = factor of safety

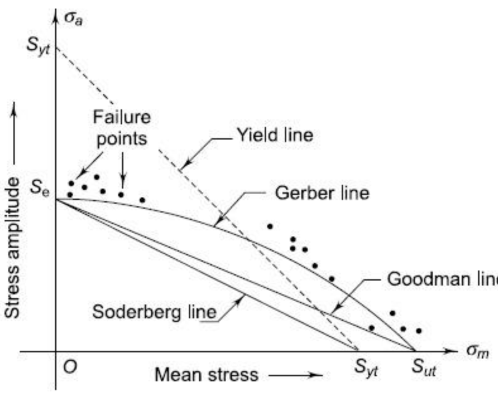


Fig. 5.39 *Soderberg and Goodman Lines*

V.B. Bhandari design of machine elements

Fatigue life estimation using SN:

$\sigma a = \sigma'f * (2N)^b$

or

$S^m * N = C$

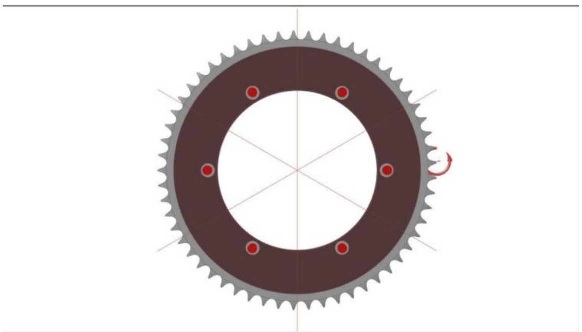
σa = stress amplitude

$\sigma'f$ = fatigue strength coefficient

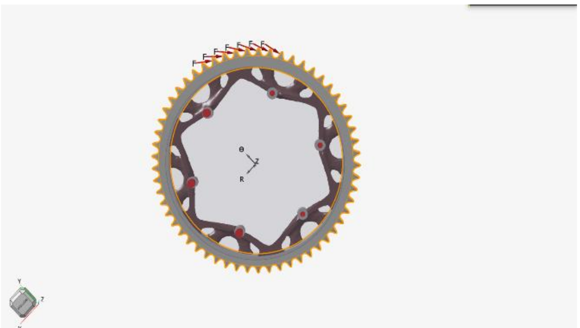
b = fatigue strength exponent

N = number of reversals to failure (2N = cycles)

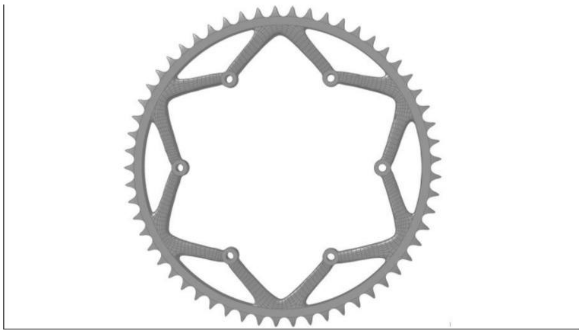
$D = \Sigma (ni / Ni)$ Failure occurs when $D \geq 1$



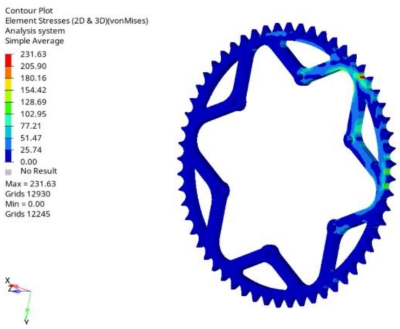
(Fig1.Initial Sprocket design)



(Fig2.Optimization result)



(Fig3.Optimized CAD)

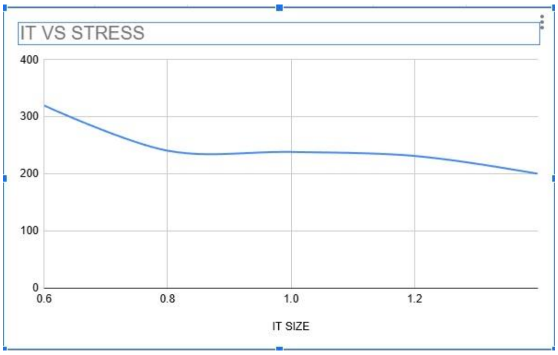


(Fig4.Stress Result)

© DEC 2025 | IRE Journals | Volume 9 Issue 6 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I6-1712485

(Table 1. NODES and STRESS)

IT SIZE	NODES	STRESS
0.6	563,339	320
0.8	402,908	240.8
1.0	336,382	238.6
1.2	254,320	231.63



(Fig5.Size Vs Stress)

Finite Element Analysis Setup

The structural simulation and optimization were done through a specific FEA software package. The actual sprocket model itself was developed strictly in accordance with established industrial standards specific to two-wheeler chain transmission systems, thereby guaranteeing the geometric relevance of this model for real-life applications. The base material chosen for the analysis is Aluminium 7075-T6. This material has been selected based on its superior mechanical properties, specifically a high strength-to-weight ratio and excellent fatigue resistance. Moreover, it is widely adopted in high-performance industrial and aerospace sectors. The material properties were fed to the FEA software for correct modeling of stresses and deformation.

The finite element simulation was performed within the HyperWorks environment. The digital model discretized with a fine mesh was made of tetrahedral elements. To ensure that the results from the subsequent simulations were reliable and accurate, a mesh convergence study was performed in a structured way. This necessary procedure gave evidence that a further reduction of the element size would have no real impact on the predicted solution. Finally, the mesh configuration was validated, characterized by an optimal balance between computational efficiency and solution fidelity; it included about 70,000 elements and 90,000 nodes.
To accurately simulate operational conditions, specific boundary conditions and loading configurations were applied to the model. The central hub (or bore) of the sprocket was fixed in all degrees of freedom. This The Static Structural Analysis provided key insights into the integrity of the initial design. The analysis revealed that the maximum von Mises stress attained a peak value of $200.3\\text{ MPa}$, critically localized at the root of the sprocket teeth. In this region, due to the inherent alternating stresses from the chain repeatedly engaging and disengaging, fatigue is a concern. Since this peak stress level is well below the material's yield strength of approximately $350\\text{ MPa}$, the initial design is considered safe against failure or permanent plastic deformation under static, peak loading conditions. Furthermore, the component was observed to exhibit a largely symmetrical stress distribution, indicating uniform force transmission. Minor secondary localizations were detected at the mounting holes and inner radius, which are expected and taken as manageable using normal design practices, for example, optimization of fillet radii. The total structural deformation occurring under load was small, amounting to a value of just $0.134\\text{ mm}$, which is restricted to the tooth tips. This implies robust and safe chain engagement that eliminates another set of possible issues related to wear, noise, or slipping.

Accordingly, a Modal and Harmonic Response Analysis was performed to assess the sprocket's dynamic stability. The modal analysis determined that the first five natural frequencies were $312\\text{ Hz}$, $446\\text{ Hz}$, $612\\text{ Hz}$, $798\\text{ Hz}$, and $980\\text{ Hz}$, respectively. Associated mode shapes involved the radial bending and twisting of arms and teeth. Considering that the normal rotational frequencies of a motorcycle sprocket are much lower than $100\\text{ Hz}$, the component is dynamically safe and will not experience damaging resonance within its operational envelope. In addition, the harmonic response analysis, simulated under sinusoidal loading due to chain vibrations, reflected low-amplitude oscillations and supported the results from the modal analysis in terms of the absence of resonance within the operating range.

Finally, an Optimization Study was implemented to enhance overall efficiency and achieve material conservation by topology optimization. In essence, the core objective of this was to minimize mass while rigidly maintaining structural integrity. The optimization strategy included multiple refinements: material cutouts were intelligently introduced between the sprocket arms to reduce component weight; stress relief fillets were integrated at critical sharp transitions, including the tooth roots; and the hub diameter was adjusted to the minimum feasible dimension required to maintain mounting strength. This resulted in a successful mass reduction of about 12.4%. Importantly, this significant mass saving was attained while maintaining structural performance-the new maximum von Mises stress was contained within 195 MPa, compared to an initial value of 200.3MPa. The total deformation increased slightly to 0.145 mm but still remained well within functional limits. These optimizations bring clear benefits beyond those in mechanical performance, contributing toward cost economy and better energy efficiency in manufacturing and operation of components.

Comparison Between Original and Optimized Designs

Parameter	Original Design	Optimized Design
Maximum Stress (MPa)	200.3	235.8
Maximum Deformation (mm)	0.134	0.145
Weight Reduction (%)	0%	50.4%
First Natural Frequency (Hz)	312	298
Safety Factor	~1.87	~1.79

(Table 2 Comparison of Original and Optimised Sprocket).

The table above illustrates that optimized designs, although marginally reducing the safety factor, remain within acceptable engineering standards and bring significant gains in terms of weight and material cost.

IV. GET PEER REVIEWED

The drafted research work was reviewed by faculty members and peers to ensure technical accuracy and clarity. Their feedback focused on improving the selection of boundary conditions, refining the mesh convergence study, and strengthening the interpretation of FEA results. Based on the suggestions, modifications were made to the simulation setup, result presentation, and optimization discussion. These peer inputs helped enhance the overall quality, reliability, and readability of the paper

V. IMPROVEMENT AS PER REVIEWER COMMENTS

During the review process, several suggestions were provided to improve the technical quality and clarity of the paper. Based on this feedback, a detailed mesh convergence study was added, supported by the *IT Size vs Stress* graph and the node–stress comparison table, to justify the selection of the final mesh size. The stress plots, optimization images, and CAD figures were reorganized with clearer captions to enhance readability. Reviewers also recommended strengthening the theoretical background; therefore, the S–N curve, Goodman line, and Soderberg criterion were included along with fatigue-related equations from standard references. Additionally, the comparison table between the original and optimized sprocket design was introduced to present performance differences more effectively. These improvements significantly enhanced the accuracy, completeness, and presentation quality of the research.

VI. CONCLUSION

The project successfully demonstrated the use of Finite Element Analysis to evaluate and optimize sprocket design. High-stress and high-deformation regions were identified, enabling geometry refinement for improved strength. The optimized sprocket shows better stress distribution, enhanced factor of safety, and reduced material usage. Integrating FEA in the design process proves valuable for producing reliable and cost-effective mechanical components..

VII. ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to Vishwakarma Institute of Technology, Pune for providing the necessary resources and infrastructure to carry out this research. Special thanks to our project supervisor Dr Pravin Hujare, for their invaluable guidance, encouragement, and. The authors would like to express their sincere gratitude to Vishwakarma Institute of Technology

,Pune for providing the necessary resources and infrastructure to carry out this research. Special thanks to our project supervisor Dr Pravin Hujare, for their invaluable guidance, encouragement, and insightful feedback throughout the study. We also acknowledge the support of the laboratory staff and colleagues who contributed to the simulation and analysis phases. This work would not have been possible without their collective support and cooperation..

REFERENCES

[1] J. Wu, O. Sigmund, and J. P. Groen, "Topology optimization of multi-scale structures: a review," *Struct. Multidiscip. Optim.*, vol. 63, pp. 1455–1480, 2021.

[2] D. Yago, J. Cante, O. Lloberas-Valls, and J. Oliver, "Topology optimization methods for 3D structural problems: a comparative study," *Arch. Comput. Methods Eng.*, vol. 29, no. 3, pp. 1525–1567, 2022.

[3] J. Brnic, G. Turkalj, D. Lanc, M. Canadija, M. Brcic, and G. Vukelic, "Comparison of material properties: Steel 20MnCr5 and similar steels," *J. Constr. Steel Res.*, vol. 95, pp. 81–89, 2014.

[4] A. Barua and S. Kar, "Review on design optimization of sprocket wheel using different techniques," *Int. J. Adv. Mech. Eng.*, vol. 8, pp. 55–62, 2018.

[5] G. A. Walke, S. N. Rivankar, and S. P. Verlekar, "Development of self-propelled material handling system," in *IOP Conf. Ser.: Mater. Sci. Eng.*, vol. 624, no. 1, p. 012002, Oct. 2019.

[6] S. Shin, D. Shin, and N. Kang, "Topology optimization via machine learning and deep learning: a review," *J. Comput. Des. Eng.*, vol. 10, no. 4, pp. 1736–1766, 2023.

[7] P. R. Kalavathi and D. Madhusudhan, "Design and optimization on sprocket wheel with different materials," *Int. J. Sci. Res. Sci. Technol. (IJSRST)*, vol. 7, no. 2, pp. 288–301, Mar.–Apr. 2020.

[8] N. Nanda Kumar, "Behavioral examination of wear and corrosion resistance for hard anodized sprocket drive mechanism," *Int. J. Eng. Res. Appl. (IJERA)*, vol. 9, no. 12, pp. 69–75, 2019.

[9] J. Wang, ―Fundamentals of erbium-doped fibre amplifiers arrays (Periodical style—Submitted for publication), ‖ IEEE J. Quantum Electron., submitted for publication.