Design and Analysis of Leaf Spring in ANSYS Using Different Materials

Umesh G. Chaudhari [1], Pratham V. Sonawane [2], Akash B. Dawbhat[3], Harsh S. Avhad[4], Harsh B. Waghchaure [5], Pratik A. Gosavi[6], Yashraj S. Thorat[7], Rushikesh V. Salunke[8]

[1] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[2] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[3] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[4] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[5] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[6] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[7] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

[8] *Department of Mechanical Engineering, Sanjivani College of Engineering, Kopargaon*

---***---

Abstract - This study presents the design and structural analysis of an automotive leaf spring using Spring Steel and Phosphor Bronze to optimize performance and determine the most suitable material. A leaf spring is a long, narrow, and flexible beam composed of layers of plates (called leaves) stacked together in a curved shape. It is commonly used in vehicle suspension systems and is designed to absorb shocks, support the vehicle weight, and provide stability during motion. In this study, a 3D model of a leaf spring was created using SolidWorks, and Structural analysis was performed on the leaf spring using ANSYS Workbench by varying the materials of the spring to Spring Steel and Phosphor Bronze. Structural analysis was performed to validate the strength of the springs for different materials. A comparison was made between the two materials to determine the best material for Leaf Spring. This study began with an analysis of vehicle suspension requirements, encompassing performance and durability requirements, as well as various technical parameters. The key components of the leaf spring modeled in SolidWorks and Structural analysis were conducted in ANSYS using Finite Element Analysis (FEA) to assess the total deformation, stress distribution, and performance under loading conditions.

Key Words: ANSYS, Deformation, Finite Element Analysis, SolidWorks, Stress distribution, Structural analysis.

1. INTRODUCTION

The components in a suspension system have to provide comfort, durability, and safety in today's automotive world. This study will detail a design and analyze a leaf spring using Spring Steel and Phosphor Bronze for the most efficient material. A 3D model was developed on SolidWorks and then subjected to structural analysis on ANSYS Workbench using FEA for stress distribution, total deformation, and load performance. This comparison of both materials helped to assess their strength and flexibility, suitable for suspension applications in vehicles. The aim of the study is to facilitate an improved ride quality and life through the selection of the optimum material for an efficient and reliable leaf spring design.

2. WORKING OF LEAF SPRING

A leaf spring works in such a way that it absorbs and distributes the shocks and vibrations generated in the process of movement of a vehicle over uneven roads. It is mounted between the axle and the chassis of the vehicle. As the wheel passes through a bump, the spring bends and flexes to let the axle move upward while controlling the motion to prevent a jarring impact. As the spring straightens, it smoothly dissipates the stored energy, maintaining vehicle stability and ride comfort. This flexible action helps support the load of the vehicle, dampens vibrations, and maintains a smooth and safe drive.

3. LITERATURE REVIEW

Research on automobile suspension systems has mainly focused on the enhancement of ride comfort, weight reduction, and durability with advanced material applications and structural optimization. Studies have confirmed that composite or alternative metal alloy springs can replace conventional steel leaf springs, with significant improvements in performance and efficiency.

The findings from the studies reviewed above are unanimous in terms of the use of lightweight and high-strength materials along with advanced analysis methods to enhance vehicle stability, comfort, and energy efficiency. These insights form the backbone of the current study regarding the design and analysis of a leaf spring using Spring Steel and Phosphor Bronze, with the aim of determining the most efficient and durable material through simulation-based comparison.

Table- 1: Summary of Literature Review

Author(s)	Title	Key Findings
Pankaj Saini, Ashish Goel, Dushyant Kumar	Design and Analysis of Composite Leaf Spring for Light Vehicles	Composite materials like E-glass/epoxy reduce weight by up to 90% while maintaining strength and stiffness.
M. Venkateshan, D. Helmen Devraj	Design and Analysis of Leaf Spring in Light Vehicles	Used FEA to evaluate stress and deformation; validated composite leaf spring as lightweight alternative to steel.
Rajendran I., Vijayarangan S.	Design and Analysis of a Composite Leaf Spring	Suggested replacing steel with composite to improve fatigue life and reduce vehicle weight.
Kumar Krishan, Aggarwal M. L.	FEA Comparison of Mono Steel and GRP Leaf Spring	Compared steel and glass-reinforced plastic; GRP showed improved flexibility and lower stress concentration.
V. K. Aher, P. M. Sonawane	Static and Fatigue Analysis of Multi Leaf Spring for LCV	FEA validated that optimized design reduces stress and increases fatigue life for lightweight vehicles.

4. METHODOLOGY

The design and analysis of the leaf spring in ANSYS consists of logical and systematic steps. First, the key performance requirements of an automotive suspension system, such as the load-bearing capacity, flexibility, and vibration absorption, were identified. Based on these requirements, two materials, spring steel and phosphor bronze were chosen for comparison, based on their mechanical strength and elasticity. A leaf spring 3D model was developed in SolidWorks, whose dimensions and geometrical details were accurate and similar to those of practical vehicle suspension components.

The designed model was then imported into ANSYS Workbench for structural analysis. The FEA was conducted to assess the total deformation, equivalent stress, and strain under static loading conditions. Loads and appropriate boundary conditions were applied to simulate the real vehicle operating conditions. The material properties for each model were then assigned to compare performance of the Spring Steel and Phosphor Bronze. The results obtained from the FEA helped in comparing the stress distribution and deformation characteristics of both materials.

This systematic method ensured that the design was mechanically efficient and durable for real-world applications, thereby determining the material that provided the best balance between strength, flexibility, and weight.

5. MODELING

SolidWorks is powerful and widely used CAD software. developed by Dassault Systèmes is known for its friendly interface and powerful functionality, the main uses of SolidWorks are in mechanical engineering, product design, and manufacturing in order to create detailed 3D models, assemblies, and simulations. It enables designers and engineers to manifest ideas into tangible form through the use of parametric design; thus, allowing them to Create complex parts and assemblies that are fully editable and adaptable. With integrated 3D modeling, analysis, and documentation tools, SolidWorks provides an all-inclusive Solution for creating everything from individual parts to large-scale mechanical systems. Besides, the software includes features for rendering, animation, and stress analysis, This helps designers assess and visualize their designs under realistic conditions. The software's extensive library of parts and its ability to import data from most CAD formats. This versatile tool facilitates collaboration and productivity in product development.

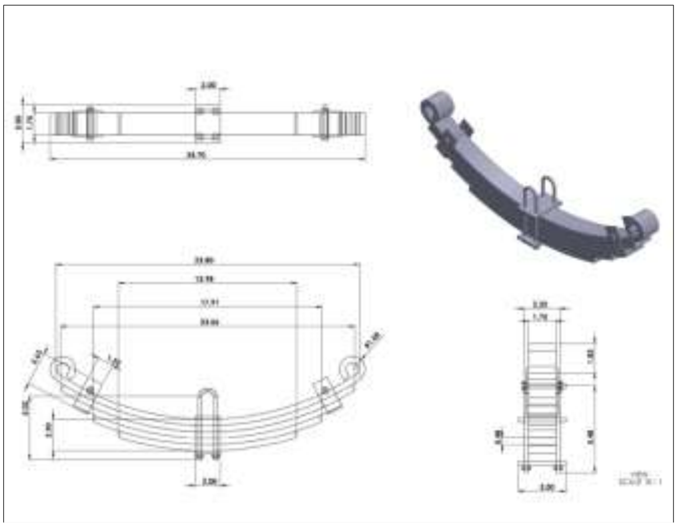

Fig- 1: Drafting of Leaf Spring

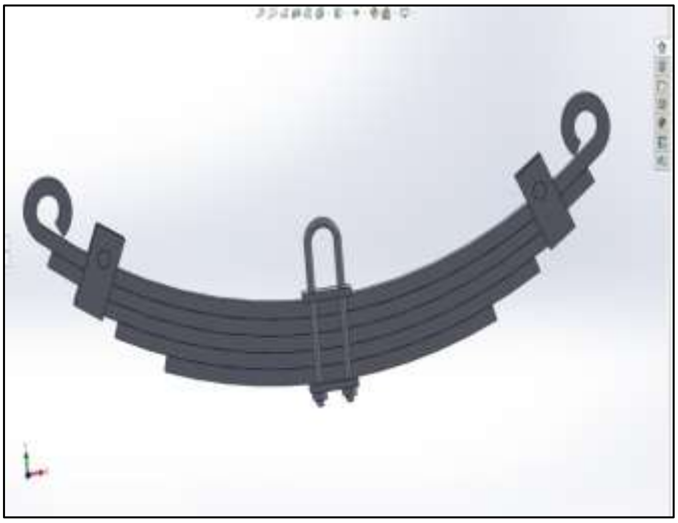

Fig- 2: Assembly of Leaf Spring

6. FINITE ELEMENT ANALYSIS

Finite element analysis (FEA) is a powerful computer-based tool wherein engineers study how a structure or component will act when subjected to real-world conditions, such as load, pressure, heat, or vibration. Rather than testing an actual model, FEA enables simulation on a computer to predict performance, safety, and durability. In this technique, a complicated structure is divided into many smaller and simpler parts called elements, which are connected at specific points referred to as nodes. Each of these elements follows a set of mathematical equations that identify how they will deform or react if forces are applied. Furthermore, the software combines all these equations to determine how the entire structure will respond. This process helps calculate the stress, strain, and deformation in different areas of the design.

Finite element analysis (FEA) has become very important because it identifies weak points and potential failures before actual manufacturing, thereby saving time and cost. They are widely used in various industries, such as automotive, aerospace, civil, and mechanical engineering, to enhance design reliability and performance. Advanced tools, such as ANSYS, enable FEA to be easily performed with greater efficiency and accuracy. Overall, FEA helps engineers make products safer, stronger, and more efficient without requiring repeated physical testing.

ANSYS is a leading engineering simulation software suite that is used for analyzing and Complex engineering problems in various fields. ANSYS Inc. created, This software utilizes FEA (Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD). and other simulation techniques to give deep insights into the behaviour of designs under real-world conditions. ANSYS allows the engineer or designer to simulate the effects of physical forces such as heat, stress, fluid flow, electromagnetics, and vibration on materials and structures. By simulating design performance before physical prototypes are built, ANSYS This helps to reduce the time, cost, and risk associated with the development process. Additionally, the Software provides scalability for high-performance computing, enabling engineers to perform highly detailed simulations and complex multi-physics analyses.

Step 1: Import the Solid works 3D model into ANSYS

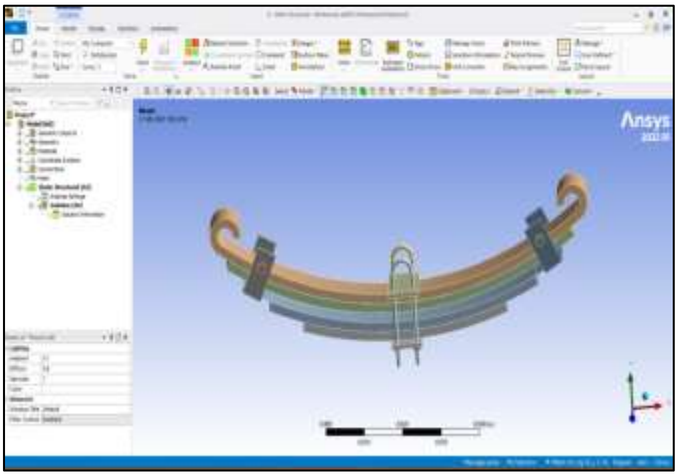

Fig- 3: Imported Model in ANSYS

Step 2: Meshing of the design

Meshing Size- 1.e^3

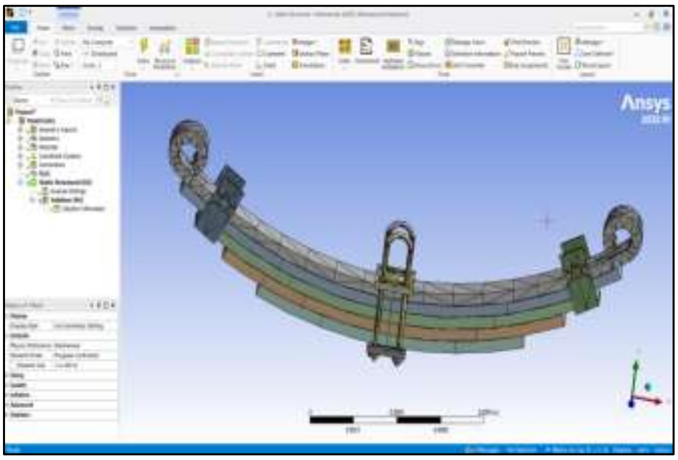

Fig- 4: Generating Mesh on Model

Step 3: Fixed Supports

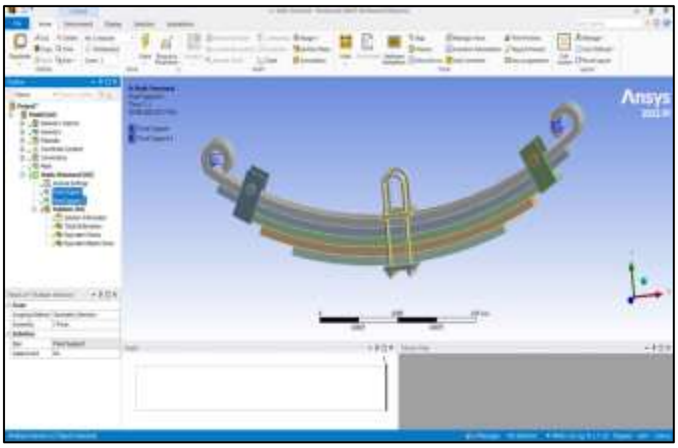

Fig- 5: Adding Fix Supports

Step 4: Applying Load

Load= 98066 N = 10,000 KG

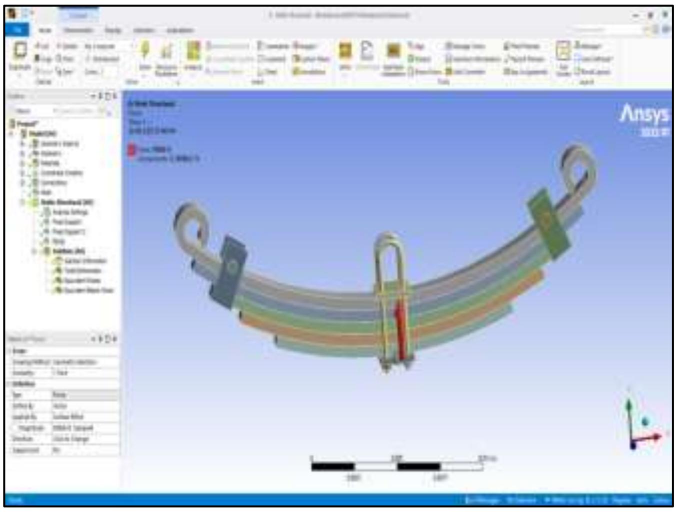

Fig- 6: Adding load

7. ANALYSIS RESULTS

Table- 2: Material Properties of Spring Steel

Properties	Value
Poisson ratio	0.28
Density (Kg/m^3)	7800
Young's Modulus (GPa)	200

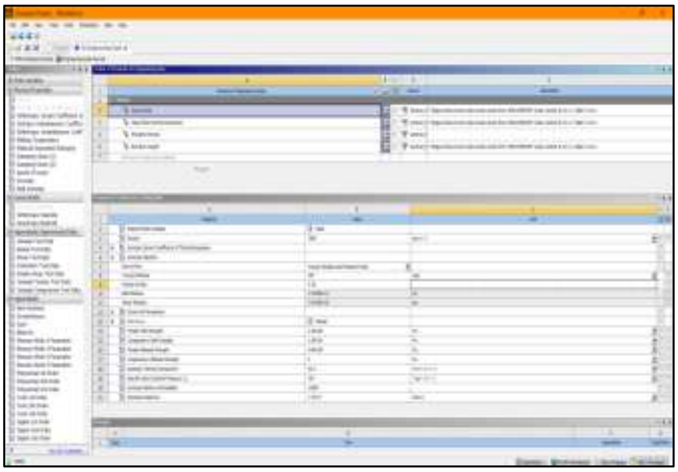

Fig- 7: Adding Material Properties for Spring Steel

Table- 3: Material Properties of Phosphor Bronze

Properties	Value
Poisson ratio	0.34
Density (Kg/m^3)	816
Young's Modulus (GPa)	103

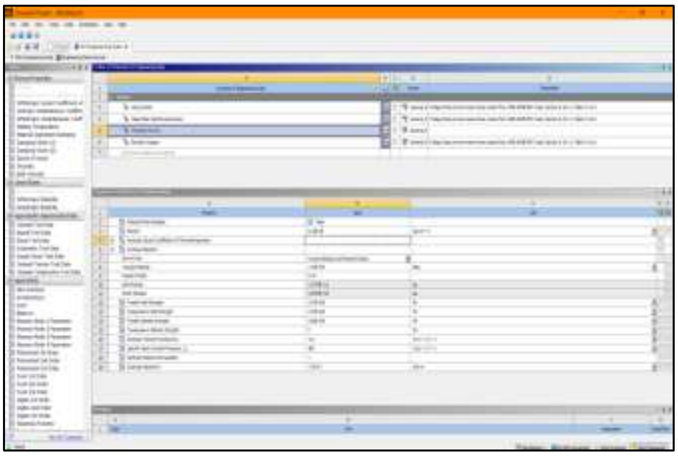

Fig- 8: Adding Material Properties for Phosphor Bronze

A. Total Deformation

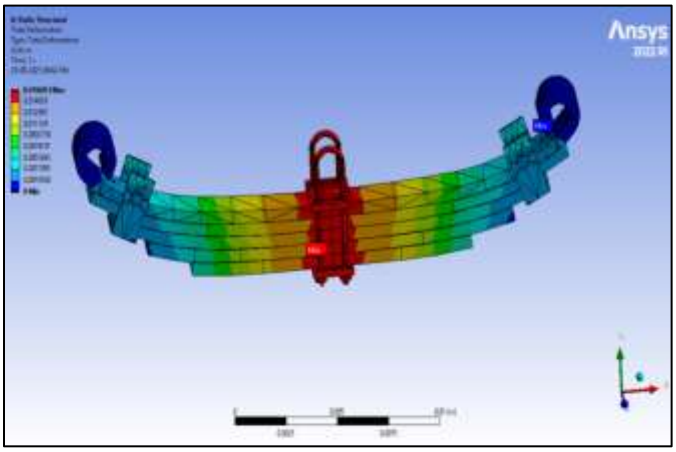

Fig- 9: Total Deformation in Spring Steel

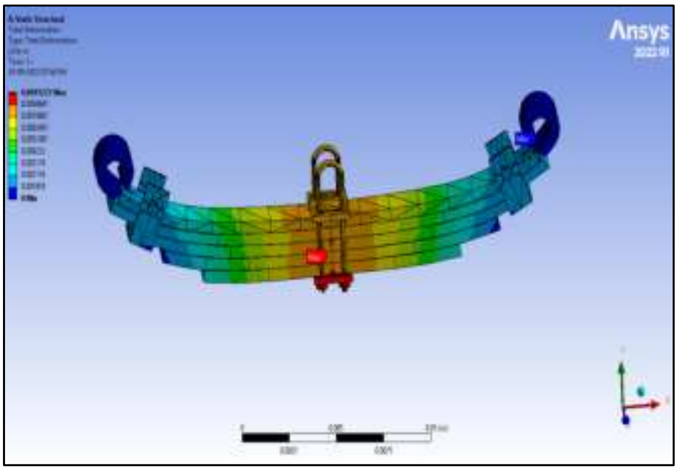

Fig- 10: Total Deformation in Phosphor Bronze

Table- 4: Summary of Total Deformation

Material	Total Deformation (mm)
Spring Steel	**0.0166**
Phosphor Bronze	**0.00952**

B. Equivalent Stress

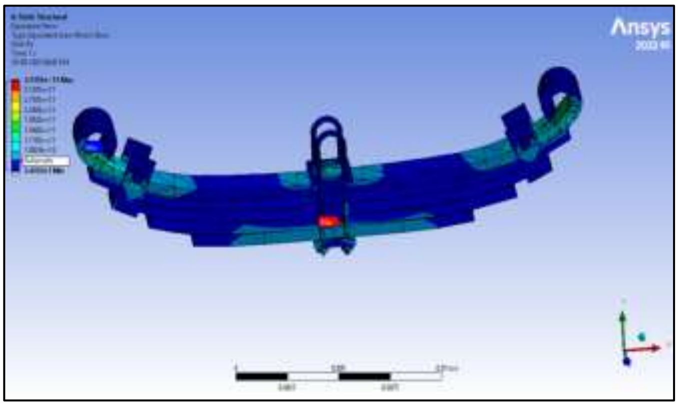

Fig- 11: Equivalent Stress in Spring Steel

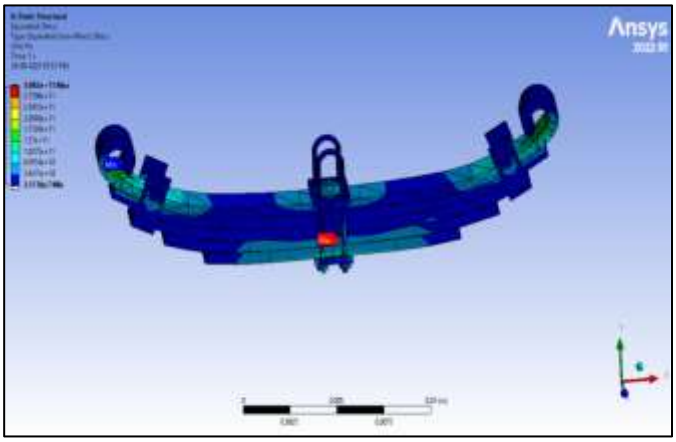

Fig- 12: Equivalent Stress in Phosphor Bronze

Table- 5: Summary of Equivalent Stress

Material	Equivalent Stress (N/mm^2)
Spring Steel	**3.510*10^{11}**
Phosphor Bronze	**3.082*10^{11}**

8. CONCLUSIONS

In this project, a leaf spring was designed and analyzed using SolidWorks and ANSYS to compare the performance of two materials: Spring Steel and Phosphor Bronze. The most important parameters considered during this structural analysis were total deformation and equivalent stress.

The results showed that the total deformation in Spring Steel was 0.0166 mm, whereas for Phosphor Bronze, it was 0.00952 mm. Therefore, Phosphor Bronze deforms slightly less, which makes it relatively flexible under load. Nonetheless, it also resulted in equivalent stress for Spring Steel of 3.510×10^{11} N/mm², which is higher compared to that of Phosphor Bronze of 3.082×10^{11} N/mm², demonstrating that Phosphor Bronze can have lower stress without failure.

Considering both the deformation and stress results, Phosphor Bronze is stronger and stiffer; hence, it would be better for applications that require a higher load-bearing capacity and durability. Based on the analysis conducted in this study, Phosphor Bronze is considered a better material for leaf spring design.

REFERENCES

1. Saini, P., Goel, A., Kumar, D.: Design and Analysis of Composite Leaf Spring for Light Vehicles. Int. J. Eng. Res. Appl. (2013).

2. Venkateshan, M., Devraj, D.H.: Design and Analysis of Leaf Spring in Light Vehicles. Int. J. Modern Eng. Res. (2012).

3. Rajendran, I., Vijayarangan, S.: Design and Analysis of a Composite Leaf Spring. J. Sound Vib. (2002).

4. Krishan, K., Aggarwal, M.L.: FEA Comparison of Mono Steel and GRP Leaf Spring. Int. J. Eng. Res. Technol. (2012).

5. Aher, V.K., Sonawane, P.M.: Static and Fatigue Analysis of Multi Leaf Spring for LCV. Int. J. Mech. Eng. Technol. (2012).