IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 4, Issue 2, April - May, 2016
ISSN: 2320 – 8791 (Impact Factor: 2.317)
www.ijreat.org

Design and Finite Element Analysis of Leaf Spring

Pawan Jaiswal[1], Debayan Das[2], Ankit Basnet[3], Saurav Rajgadia[4], Anush karki[5], Anupam Raj Jha[6], Rakesh Jaiswal[7],

Rabindra Nath Barman[8]

[1,2,3,4,5,6,7]B.Tech students, Department of Mechanical Engineering, NIT Durgapur, Durgapur-713209, India

[8]Assistant Professor, Department of Mechanical Engineering, NIT Durgapur, Durgapur-713209, India

Abstract

In present era it's difficult to find Automobile parts builder with enough money for building and testing real prototypes of the Automobile parts model, instead they use virtual prototypes. Leaf Spring has got less attention towards the stress, strain behaviour. The present study of this paper is to analyze the stress, strain and deformation of the current state of virtual prototyping of a Leaf spring which directly improve the quality of product through design aspect. The stress and strain analysis is done to compare with Finite Element Method in order to get the optimum design of the Leaf Spring element for modeling and analysis Solidworks and ANSYS Workbench has been used.

Keywords: *Solidworks 12.0, ANSYS Workbench, Leaf Spring, Finite Element Method (FEM).*

1. Introduction

Leaf sprigs are crucial suspension elements used on both light and heavy duty vehicle. It is necessary to minimize the vertical vibrations impacts and bumps due to road irregularities and to create a comfortable ride to the passenger. Leaf springs are widely used for automobile and rail road suspensions. The leaf spring should absorb the vertical vibrations and impacts due to road irregularities by means of variations in the spring deflection so, that the potential energy is stored in spring as strain energy and then released slowly by increasing the energy storage capabilities of a leaf spring and ensures a more compliant suspension system.[1-2]

Three dimensional finite element analysis of the leaf spring consists of a computer model or design that is stressed and analyzed for specific results. A company that is able to verify a proposed design will be able to perform to the clients specifications prior to manufacturing or construction.[3-6]

The leaf spring is analyzed for stress and strain acted, deflection using 3D finite element analysis. The general purpose finite element analysis software ANSYS is used for present study. The variations of bending stress, strain and displacement values are predicted. As the Indian market is becoming one of the leading companies globally in compare to multinational companies, a cut throat competition has a rise between Indian companies and multinational companies. To struggle in the market globally, it has become necessary for the Indian industries to improve and innovate their product. Awachat industries Limited, Wardha is one of the leading tractor trolley manufacturers in the region manufacturing 3000 trolley and above. Now they are manufacturing them with much economical and technical consideration. One of the important areas where one can improve the product quality by increasing the material quality in the design aspect. One can design the product in such a way that its performance increases as compared to the same product of other companies.[7]

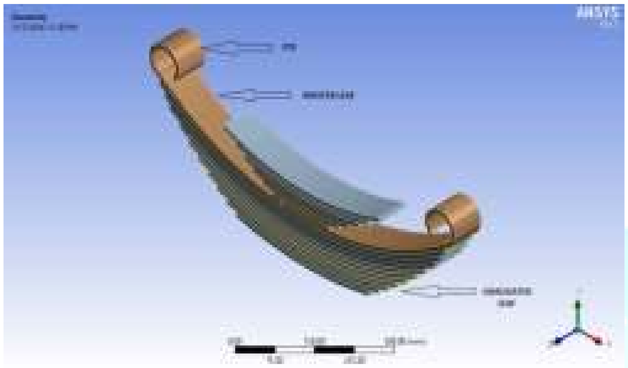

Fig 1: 3-D Model of Leaf Spring

1.1 Material and its properties

Aluminium alloys are alloys in which aluminium (Al) is the predominant metal. Aluminium alloys are widely used in engineering structures and components where light weight or corrosion resistance is required. Leaf spring made from aluminium is light and can absorb high impact at expense of durability and strength at expense of affordability.[8]

Aluminium include low density and therefore it includes low weight, high strength, superior malleability, easy machining, excellent corrosion resistance and good thermal and electrical conductivity which are the most important properties of aluminum. It is also easy to recycle. Aluminium alloys commonly have tensile strengths of between 70 -700 MPa. Unlike most steel grades, aluminium does not become brittle at low temperatures. Instead, its strength increases. At high temperatures, its strength decreases. At temperatures continuously above 100°C, strength is affected to the extent that the weakening must be taken into consideration.[9]

2. Objective

The objective of problem is to design Leaf spring made of Aluminium alloy using Solidworks 12.0, and to carry out the Finite Element Analysis (FEA) on the prepared model in ANSYS 14.5, and determine the values of Von Misses equivalent stress, strain and deflection values.

3. Analytical calculation

The leaf spring behaves like a simply supported beam and the simply supported beam is subjected to both bending stress and transverse shear stress. Flexural rigidity is an important parameter in the leaf spring design and test out to increase from two ends to the center.

Stiffness Existing Leaf Spring: Material Aluminium alloy, bending strength (fb) = 1200Mpa. The stiffness of the upper and lower springs is calculated by using the relation given below as:-

K= (f/v) = ((8enbt^3)/(3*l*^3))

The stiffness k_1= 11537 N/mm, k_2=3455 N/mm.

As the springs are connected in series and there are two Leaf springs used as isolators in parallel.

3.1 Dynamic Loading

There are different model available for calculating the load factor. The following spring mass having one degree of freedom is selected because since the leaf spring is attached to the trolley is single axle the Centre of gravity of the trolley lies nears the wheels. The mass (my_0'') represents the chassis of the trailer and the spring constant K_y represents the shock-isolator. The spring is considered as linear near the wheels.

y_0''/g =0.64 $V_s F_n$

Where, *V*s = shock velocity changes in m/sec

F_n=natural frequency of isolator experienced by chassis is expressed as dimensionless multiple of the acceleration due to gravity. The value of F_n is calculated by:-

F_n =1/2*3.1415$[K_y * 1000/\mathrm{M}]^{1/2}$

M=Maximum Load Acting on the System=6000kg

F_n =2.54 cps

To find out *V*s vertical downward velocity.

we have, s= $ut+\frac{1}{2}gt^2$

Where, u= initial downward velocity

S= $\left(\frac{1}{2}gt\right)^2$ also Vs = u +gt

We get, Vs=$(2gs)^{1/2}$

Considering a sudden drop of depth to 0.15 the various values for different values of the natural frequencies are calculated. Table 1 shows the equivalent load capacity.

Table1: Equivalent load capacity

Static load (kg)	F_n	$y_0^{"}$	***Equivalent static capacity (kg)***
1000	6.20	6.61	6610
2000	4.38	4.69	9380
3000	3.58	3.82	11460
4000	3.10	3.30	13200
5000	2.77	2.95	14750
6000	2.53	2.70	16200

Equivalent static load for static load capacity at various values of multiple factor $y_0^{"}$ for different natural frequency. Thus considering the maximum load capacity of 6000 kg which is the capacity of the system and a sudden depth of 0.15m the dynamic load factor is 2.70.thus it means that the trolley leaf spring should have static strength sufficient to support 2.70 times their normal static.

3.2 Bending stresses and Deflections of leaf spring

Bending stress developed:

Considering the maximum load capacity of 6000 kg which is the capacity of the system and multiplying it with the dynamic load factor of 2.70 for the equivalent static capacity. We get the equivalent static capacity as:

Static load (kg) = 6000 kg

Load factor = 2.70

Equivalent static capacity = static load × load factor

Therefore, equivalent static capacity = 6000×2.70

Therefore F = (16200×9.8)/2 = 79461 N

Now bending moment σ_b= 3×F×L/(2×N×B× T^2)

Where,

F= Maximum static force = 79461 N

L= Length of the leaf spring = 435mm

N= number of leafs= 17

B= width of leaf = 70mm

T=thickness of the leafs= 10mm

σ_b = 3×79461×435/ (2×17×70×10^2)

= 435.70 N/mm^2

Developed stress i.e. bending stress is 435.70 N/mm^2

Developed Deflections of Leaf Spring:

δ (developed) = 3×F×L^3/(8×E×N×B× T^3)

Where, F= Maximum static force = 79461 N

L= Length of the leaf spring = 435mm

N= Number of Leafs= 17

B= Width of Leaf = 70mm

T=Thickness of the Leafs= 10mm

E=Modulus of Elasticity for (Al alloy) multiplate Leaf= 7.1*10^4N/mm^2

δ = 3×79461×435^3/ (8 × 2.04×105 × 17 × 70 × 10^3)

So, developed deflection = 29.03 mm

Also, permitted stress σ_b (permitted) = 1200 N/mm^2

σ_b (Permitted) = 1100×S_z

S_z=0.8+2.5/t

= 1.05

σ_b = 1200×1.05

=1260 N/mm^2

For this permitted value of bending stress the permitted deflection can be calculated as:

1260= 3×F×435/(2×17×70×100)

F = 229793.1N δ

We get the permitted deflection as:

δ (permitted)=3×229793.1×435^3/(8×7.1*10^4×17×70×10^3)

= 83.95 mm

Deflection using the number of graduated Leafs and number of full length Leafs:

Where, F= Maximum static force = 79461N

L= Length of the leaf spring = 435mm

N= number of leafs= 17

B= width of leaf = 70mm

T=thickness of the leafs= 10mm

N_f=number of full length leaves=1
N_g=number of graduated leaves=16
δ= 3×F×L^3/8×E×N_g×B×T3 [1/ (1+1.5(N_f/N_g))]
δ = 28.20mm
Developed bending stress σ_b (developed)
σ_b=3×F×L/(2×N×B× T^2)[(1.5/(1+1.5*(N_f/N_g))]
=597.53 N/mm^2

The values are approximately similar to that of calculated before without considering the number of graduated leaves and full length leaves independently.
As from above δ (developed) ‹ δ (permitted)
Hence, the design is safe.
Also σ_b (developed) ‹ σ_b (permitted)
Hence, the design is safe.
Also the factor of safety can be calculated as:
F.O.S = σ_b (permitted)/ σ_b (developed)
= 1260 N/mm^2 /435.70 N/mm^2
= 2.8

3.3 Specifications of Leaf spring

Table 2: specification of leaf spring

Description	***Upper Spring (mm)***	***Lower Spring (mm)***
Number of leaves (n)	6	11
Width of leaves (b)	70	70
Thickness of leaves (t)	10	10
Effective length	410	750
Young modulus (e)	7.1*10^4 N/mm^2	7.1*10^4 N/mm^2

4. Methodology

For the analysis of Leaf spring we have designed our model in Solidworks 12.0 and then save it as IGES format for exporting the part into ANSYS 14.5 Workbench environment.

4.1 Meshing:

The figure 2. Shown is the meshed model of rigid flange coupling in the ANSYS analysis for the static structural process. To analyze the FEM, default type of mesh is used for the Leaf spring in the ANSYS environment. The number of elements used in this meshing is 285984 and the number of nodes is 1406340. In this process default type of meshing is done to analyze the process. Using the working condition of leaf spring a dynamic loading comes into picture consequently. The determination of the stress, strain and deformation over leaf spring is essential. So, the model is meshed and then analyzed to get the detail and authentic result of the stress, strain and deformation at dynamic condition.[10]

Fig 2: Mesh model of Leaf Spring

4.2 Finite element method (FEM)

The Finite element method (FEM) is a numerical technique for finding the approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler part, called finite elements and solve the problem by minimizing an associated error function. Finite element model is constructed that simulates boundary condition of the vehicle and test rig. Boundary conditions of finite elements are simplified suitably and implemented to the model. Leaf spring is modeled with rectangular type of default elements. At the beginning, the leaf spring was tried to be simulated within the entire vehicle Finite Element Model. However, not only the time of calculation will considerably increase but also the accuracy of the results will decrease because of the possible errors in the modeling of high nonlinearities. Therefore, a simulation method was developed based on the Finite Element Analysis. Moreover, the stress, strain and deformation were taken into account carefully, especially for multi-layer leaf springs. In this study, the mesh and model were generated with the software ANSYS and Solidworks

5. Analysis of Leaf spring

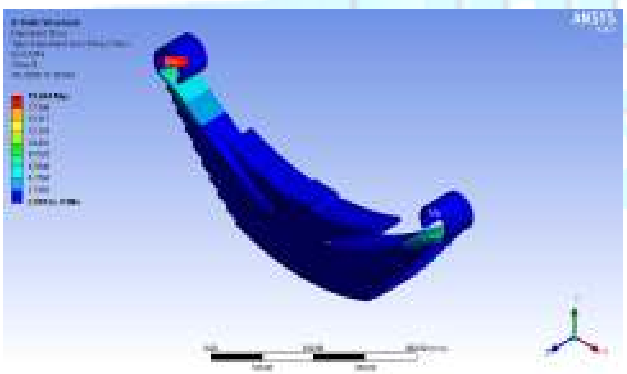

Fig 3: Equivalent stress of Leaf Spring

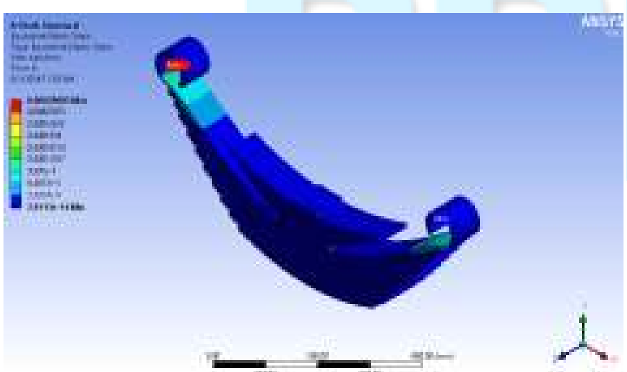

Fig 4: Equivalent elastic strain of Leaf Spring

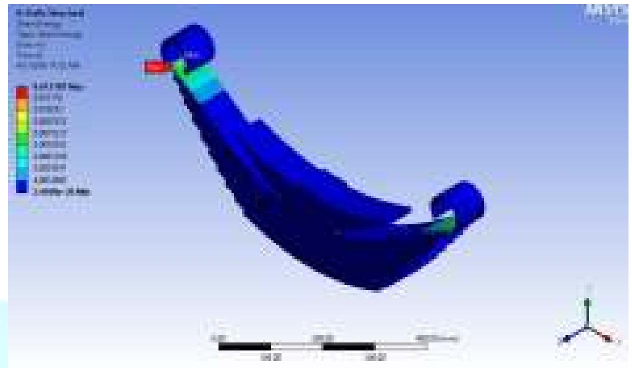

Fig 5: Strain energy of Leaf Spring

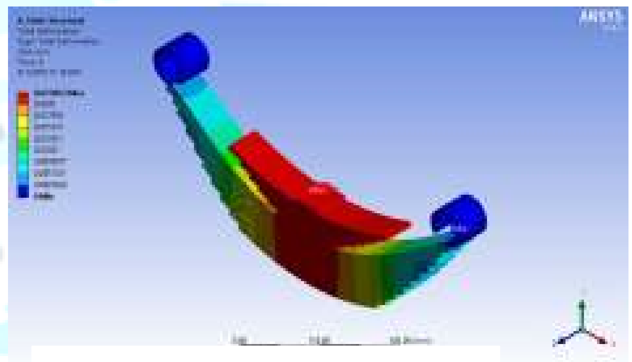

Fig 6: Total Deformation of Leaf Spring

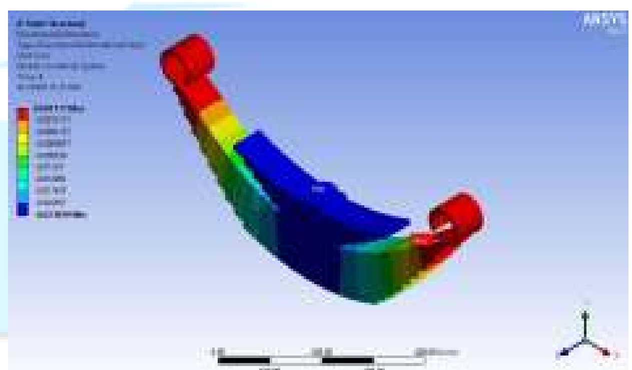

Fig 7: Directional deformation of Leaf Spring

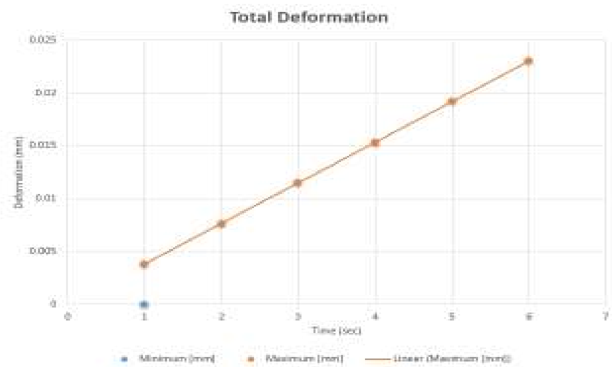

Fig 8: Total deformation graph of Leaf Spring

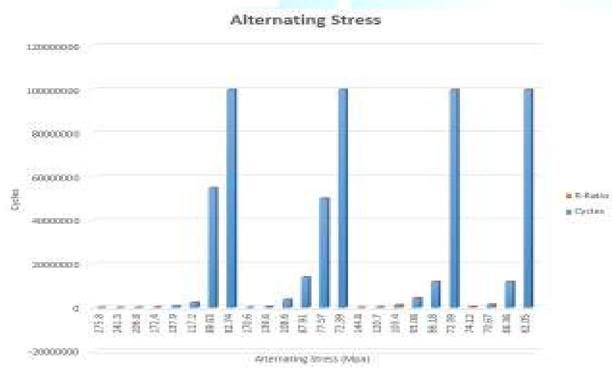

Fig 9: Alternating stress graph of Leaf Spring

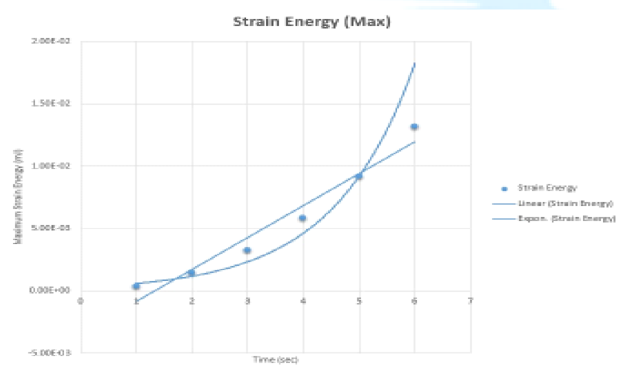

Fig 10: Maximum strain energy graph of Leaf Spring

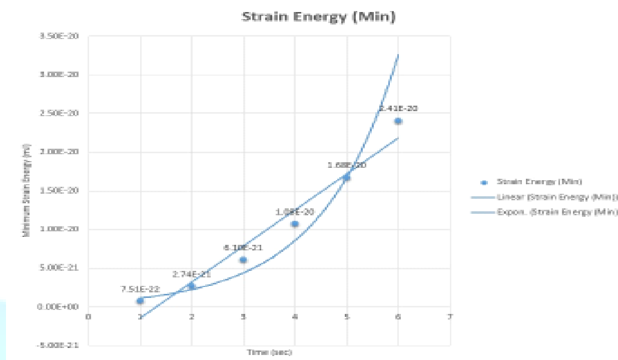

Fig 11: Minimum strain energy graph of Leaf Spring

5.1 Results

This paper shows the importance of analytical calculation and analysis of leaf spring. Finite element method (FEM) is done by using ANSYS 14.5 and it also shows the importance of Stress, strain analysis and deformation in which load become maximum from (1000- 6000N) when time reaches from (1-6)sec. The paper highlights the need of FEM analysis in industries from small to large scale which will improve accuracy. Using the mechanical software Engineers are able to reduce the stress and strain and thereby making it strengthened enough to handle the critical load.

6. Conclusion

This paper, not only deals with the design of Leaf Spring of tractor trolley but also the bending stress, strain effect and deflection applied to the Leaf spring. The Leaf spring chosen is of tractor trolley in which failure results in replacement of the whole leaf spring assembly. In design, the Leaf spring is taken to have maximum weight possible with ability to withstand high bending, deflection and static and dynamic forces. For Leaf spring design, default type of rectangular section has been considered with reference to dynamic load evaluation. The stresses were found to be high at end of leaf spring but they were relatively uniformly distributed. The developed stress and deflection values have been found to be 435.70 N/mm^2 and 29.03 mm

7. References

[1] V.B. Bhandari, Design of Machine Elements, ISBN: 0-07-0681791-1, Design of IC Engine Component, Page no. 437-441

[2]https://en.wikipedia.org/wiki/Leaf_spring
[3] Rajendran, I., Vijayarangan, S. "Optimal Design of a Composite Leaf Spring using Genetic Algorithms Int. Jr. of Computer and Structures" 79 2001: pp. 1121 – 1129.
[4] Rajendran, I., Vijayarangan, S. "Design and Analysis of a Composite Leaf Spring Journal of Institute of Engineers India" 82, 2002: pp. 180 – 187.
[5] Daugherty, R. L. Composite Leaf Springs in Heavy Truck Applications. K.Kawata, T.Akasaka (Eds). Composite
[6] Prof. N.P.Dhoshi, Prof .N.K.Ingole, Prof .U.D.Gulhane "Analysis and Modification of Leaf Spring of Tractor Trailer Using Analytical and Finite Element Method" (IJMER), Vol.1, Issue.2, pp-719-722, ISSN: 2249-6645
[7]https://www.google.co.in/webhp?sourceid=chromeinstant&ion=1&espv=2&ie=UTF8#q=Awachat+industries+Limited
[8] https://en.wikipedia.org/wiki/Aluminium_alloy
[9]http://www.aluminiumdesign.net/whyaluminium/properties-of-aluminium/
[10] Saurav Rajgadia, Debayan Das, Pawan Jaiswal, Ankit Basnet, Anupam Raj Jha, Rakesh Jaiswal, Anush Karki, Rabindra Nath Barman "Design and Stress-Analysis of a Rigid Flange Coupling using FEM" (IJIRSET), vol. 4, Issue 10, October 2015, IISN :2319-8753
[11] y.s. cong, m.z. omar, l.b. chua, s. abdullah "Fatigue life prediction of parabolic leaf spring under various road conditions"(ELSEVIER),www.elsevier.com/locate/engfailanal
[12] Mahmut Duruúa*, Levent KÕrkayakb, Aykut Ceyhana, Kagan Kozana "Fatigue Life Prediction of Z Type Leaf Spring and New Approach to Verification Method" Procedia Engineering 101 (2015) 143 – 150, www.sciencedirect.com
[13] Debayan Das, Saurav Rajgadia, Anush Karki, Ankit Basnet, Pawan Jaiswal, Rakesh Jaiswal, Anupam Raj Jha, Rabindra Nath Barman "Design And Finite Element Analysis Of Connecting Rod Using Solidworks And ANSYS Workbench" (IJREAT), Volume 3, Issue 4, Aug-Sept, 2015, ISSN: 2320 – 8791