International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 09 | Sept -2020 ISSN: 2582-3930

Design and Analysis of Automobile Leaf Spring using Ansys

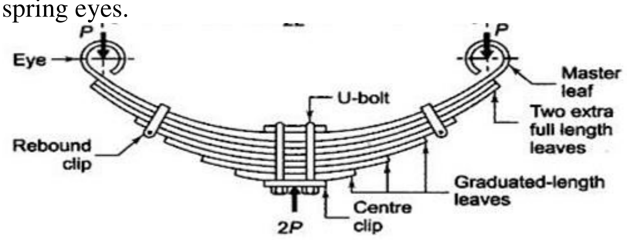
[1]A.G.Chikte, [2]A.V.Patil
[1]PG Student, Mechanical-Design, [2]HOD Assistant Professor, Mechanical Department
[1]SSGB-COET, Bhusawal, Maharashtra, India
[2]SSGB-COET, Bhusawal, Maharashtra, India

Abstract: A multi-leaf spring is one of the most important components of automobile suspension system. Leaves are basically a series of flat plates, usually of semi-elliptical shape. Generally, a multi-leaf spring used in automobile suspension, consists of two types of leaves i.e. graduated-length leaves and full-length leaves. The present work is an attempt to estimate the magnitude of bending stresses in the above mentioned leaves for a semi-elliptic multi-leaf spring made of steel. A lot of research work has been carried out in the context of leaf spring considering its material and a significant progress has been observed in the field of weight reduction, improvement of load carrying capacity when we replace the material of the spring by any advanced material like composites as E-glass/epoxy, carbon/epoxy etc. Finally referring to the results obtained in these research studies, the present work proposes a new idea regarding the construction of multi-leaf spring based on practical applications. Dimensions of the multi-leaf spring are taken from practical understanding and calculate dimension manually from standard chart of automobile spring. The multi-leaf spring was modelled in CATIA V5R18 and the same were analysed under similar conditions using ANSYS (Workbench 12.0) software considering structural-steel as the spring material.

I. INTRODUCTION

A spring is an elastic machine element which undergoes deflection for the application of any load and intends to regain its original shape depending upon the magnitude of the applied load. The major applications of spring may include its use as a shock and vibration absorber and storing potential energy by its deflection during the application of load. A multi-leaf spring or laminated spring is a very important component in automobile suspension system. It is one of the oldest suspension components too and still today it is extensively used in all the heavy and light duty commercial vehicles, railway wagons and usually in the rear suspension of passenger vehicles. It differs from the conventional helical spring in a way that it can be guided along a definite path and it deflects under the application of load while acting as a structural member. This concept is employed during the analysis of bending stresses in different leaves by consideration of cantilever beam. The present work makes an attempt to validate the above concept by performing static structural analysis using ANSYS software for the evaluation of maximum bending stress and subsequently bending stresses in different leaves, which in all, construct the entire spring. Finally, the reader may get an exposure regarding the bending stress variation for multi-leaf spring, which obviously differs from that of any cantilever beam section.

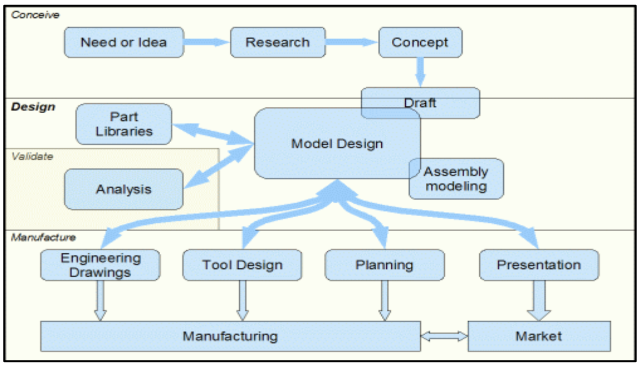
In its construction the leaf spring consists of a series of flat plates or leaves, usually of semi-elliptic shape, which are held together with the help of U-bolts and centre clip. Generally two types of leaves may be observed in a multi-leaf spring i.e. some graduated -length leaves and a few extra full-length leaves. The length of the leaves gradually decreases from top to bottom as shown in fig. . The longest leaf in the top is known as master leaf which is bent at both the ends to form spring eyes.



II. COMPUTER AIDED ENGINEERING (CAE)

A. CAD (Computer Aided Designing)

Computer-aided design (CAD) is the use of computer systems to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. Each stage requires specific knowledge and skills and often requires the use of specific software.



A. Need or Idea

Usually, the design process starts with a defined need. The need can be defined by market research, by the requirements of a larger body of work (for example airplane part). Sometimes, but more rarely than you may think, the design process is begun with a new idea or invention. At any rate, a needs analysis should precede any decision to undertake a project. This includes defining the need in a highly detailed way, in writing. This is similar to the requirements specification process in software engineering.

B. Research

Professionals tend to research available solutions before beginning their work. There is no need to "reinvent the wheel". You should study existing solutions and concepts, evaluating their weaknesses and strengths. Your research should also cover available parts that you can use as a part of your design. It is obvious, that Internet and search engines like Google are very helpful for this task. There are also many libraries of standardized parts which you can import into your project.

C. Concept

Based on your research, start with a high level concept. You should specify the main principles and major parts. For example, you can consider Diesel or Sterling engines for stationary electric generators.

D. Draft

You can choose to create a draft by pen and paper. Some prefer to use simple vector graphics programs, others even simple CAD (for example Smart Sketch), yet others prefer to start directly in their main CAD system.

E. Model Design

2D and 3D modelling in CAD. The designer creates a model with details, and this is the key part of the design process, and often the most time consuming. This will be described in greater detail in further lessons. asaceva

F. Part Libraries

Standard parts, or parts created by other team members, can be used in your model (you don't have to reinvent the wheel). Files representing a part can be downloaded from the Internet or local networks. They are also distributed on CD ROMs or together with CAD as an extension (library). By putting these predefined parts into your project, you ensure that they are correct and save a lot of time and effort. When working on a large project, this becomes a requirement to ensure the parts operate together, swap out equivalent parts, and coordinate distributed teams' work. This was, a standard part can be inserted into the project by one team member.

G. Assembly modelling

Parts are assembled into a machine or mechanism. Parts are put together using mating conditions such as alignment of the axis of two holes. More about how to do this in further lessons. Cad is used in industries.

H. Engineering Drawings

From your 3D models, you generate a set of engineering drawings for manufacturing. These drawings are then distributed to the departments and individuals responsible for producing that work. Also, these drawings must be tolerance for proper manufacturing.

I. About CATIA V5

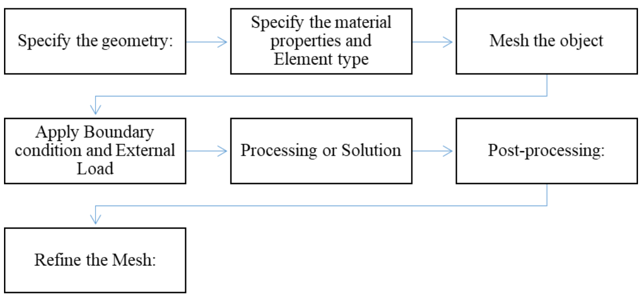
CATIA (an acronym of computer aided three-dimensional interactive application) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), PLM and 3D, developed by the French company Dassault Systems. CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CADAM software to develop Dassault's Mirage fighter jet. It was later adopted by the aerospace, automotive, shipbuilding, and other industries.

III. Finite Element Analysis (FEA)

FEA is a numerical method. It is very commonly used in finding the solution of many problems in engineering. The problem includes deigning of the shaft, truss bridge, buildings heating and ventilation, fluid flow, electric and magnetic field and so on. The main advantage of using finite element analysis is that many designs can be tried out for their validity, safety and integrity using the computer, even before the first prototype is built. Finite element analysis uses the idea of dividing the large body in to small parts called elements, connected at predefine points called as nodes. Element behavior is approximated in terms of the nodal variables called degrees of freedom. Elements are assembled with due consideration of loading and boundary condition. This results in a finite number of equations. A solution of these equations represents the approximate behavior of the problem. The design and analysis have done with the 3D modeling software and FEA technique standard FEM tool. The analysis is carried out by using the ANSYS software. This gives the comparison between analytic and numerical value. Part is drawn in CAD software. The CAD software which is involved in this is CATIA and this part is a call to ANSYS in (.igs) format.

IV. Procedure for FE Analysis

There are a number of steps in the solution procedure using finite element method. All finite element packages require going through these step.



A. Specify the geometry: In this import the geometry from CAD software to FEA software.
B. Specify the material properties and Element type:
In this step, the selection of element type is done and the material properties are given. The Young's modulus and Poisson's ratio are the input for material properties.
C. Mesh the object: Here the object is broken in to small elements. This involves defining the type of element into which structure will be broken as well as specifying how the structure will be divided in to the element. This subdivision in to elements can either be input by the user or with same finite element programs can be chosen automatically.
D. Apply Boundary condition and External Load: This is followed by specifying the boundary condition and the external loads are specified.
E. Processing or Solution: The modified algebraic equations are solved to find the nodal values of the primary variable.
F. Post-processing: It involves improving the result of processing in to the model. These results are graphically displaced to enable user case of high deflection and stress.
G. Refine the Mesh: For the case of a judge of the accuracy of the result, there is need to increase or decrease no of elements of an object.

V. MATHEMATICAL FORMULATION

For the purpose of analysis, the leaves are divided in two groups as master leaf along with graduated length leaves forming one group and the extra full-length leaves forming the other group.

Let,

n_f = number of extra full-length leaves,

n_g = number of graduated-length leaves including the master leaf,

$n = n_f + n_g$ =Total number of leaves present in the multi-leaf spring,

b = width of each leaf (mm),

t = thickness of each leaf (mm),

L = half the length of the semi-elliptic spring or the length of the cantilever (mm),

P = force applied at the end of the spring (N),

Pf = portion of P taken by the extra full-length leaves (N),

P_g. = portion of P taken by the graduated-length leaves (N),

So,

$$P = P_f + P_g.$$

Now, from practical considerations for an automobile leaf spring, that is of semi-elliptical shape usually, for a length of 2L and a load of 2P acting at the center, the entire beam can be considered as a double cantilever. If the leaves are cut into two equal halves in longitudinal plane and then combined accordingly, to form almost a triangular plate then,

The maximum bending stress is given by

$$(Sb)\ max = 6P.L/n.b.t2$$

The bending stress in the graduated-length leaves is given by

$$(Sb)\ g= 12P.L/\ (3.\ nf +2.ng).b.t2$$

The bending stress in the extra full-length leaves is given by

$$(sb)f = 18P.L/(3.\ nf +2.ng).b.t2$$

It is to be noted that, the maximum bending stress occurs at the supports for such a plate. The above relations hold good for the leaves (or plates), having uniform cross-section. Also, it is seen that the bending stresses in extra full-length leaves are 50% more than that of the graduated length leaves

$$\text{i.e. } (sb)f = 1.5(sb)g$$

Calculation of length of different leaves:

For the calculation of length of different leaves, following relations were used and subsequently, the results were implemented while modeling the multi-leaf spring in CATIA V5R18 software.

- Length of the smallest leaf = 1*Effective length/ (n-1) + Ineffective length
- Length of the next leaf = 2*Effective length/ (n-1) + Ineffective length
- Similarly, Length of the (n-1)th leaf = (n-1)*Effective length/ (n-1) + Ineffective length
- Length of master leaf = 2.L+ 2.P.(d+t)
- Ineffective length = Distance between the centres of U-bolts = l
- Effective length = 2.L – 2.l/3 and

- d = inside diameter of eye.

VI. Design Specification

Here Weight and initial measurements of four wheeler "TATA ACE" Light commercial vehicle is taken.

Weight of vehicle= 1200kg

Maximum load carrying capacity= 1000 kg

Total weight= 1200+ 1000 = 2200 kg

Taking factor of safety (FS) = 2

Acceleration due to gravity (g) = 9.81m/s2

Therefore;

Total Weight = 2200×9.81 = 21582N

Since the vehicle is 4-wheeler, a single leaf spring corresponding to one of the wheels takes up one 4th of the total weight.

21582/4 = 5395.5 N

But 2F = 5395.5 N,

F = 2697.75 N

Span length,

2L = 1072 mm,

L= 536mm.

Now the Maximum Bending stress of a leaf spring is given by the formula

$$\sigma = \frac{6 \times f \times l}{n \times b \times t^2}$$

$$\sigma = \frac{6 \times 2697.75 \times 536}{3 \times 60 \times 8^2}$$

σ = 753.12N/mm2

The Total Deflection of the leaf spring is given by

$$\delta = \frac{6 \times f \times l^3}{E \times n \times b \times t^3}$$

$$\delta = \frac{6 \times 2697.75 \times 536^3}{2.1 \times 10^5 \times 3 \times 60 \times 8^3}$$

$\delta = 128.79mm$

Measured data of the above stated light weight four wheeler vehicle.

Straight length of the parabolic leaf spring (L) =1072mm

The ratio of camber length of parabolic leaf spring

$$C/L = 0.089$$

$$C = 0.089 \times 1072$$

$$C = 95.4 \text{ mm}$$

Since the leaf spring is fixed with the axle at its centre, only half of it is considered for analysis purpose. Since analyzing half of the leaf spring is enough, half of the applied force would have been taken, but here we took as it is to account over loadings of the vehicle and flexures of the leaf spring.

Hence,

$L/2= 536mm$

F=2697.75N t=? b=?

Calculation for "t" and "b" dimensions which are capable of withstanding the loading behaviour of the conventional and composite parabolic leaf spring is the result of this design. From equation of strength of material we have, now the Maximum Bending stress of a leaf spring is given by the formula,

Bending Stress,

$$\sigma = \frac{6 \times f \times l}{n \times b \times t^2}$$

The Total Deflection of the leaf spring is given by

$$\delta = \frac{6 \times f \times l^3}{E \times n \times b \times t^3}$$

Solving these two equations the thickness and width of the parabolic leaf spring can be, formulated, respectively, as follows;

$$\sigma = \frac{6 \times 2697.75 \times 536}{3 \times b \times 8^2}$$

$$753.12 = \frac{6 \times 2697.75 \times 536}{3 \times b \times 8^2}$$

b=60mm

The Total Deflection of the leaf spring is given by

$$128.79 = \frac{6 \times 2697.75 \times 536^3}{2.1 \times 10^5 \times 3 \times 60 \times t^3}$$

t=8mm

The objective of the present course of study includes, determination of maximum bending stress for three different loading conditions and to establish the relationship for the magnitude of bending stresses existing in the graduated-length leaves and extra full-length leaves (i.e. (σb)f = 1.5(σb)g), with the following dimensional specifications, Table 1 Specification of multi-leaf spring Parameters Value Notation Total length of the semi-elliptic spring (distance between two eyes) 1243 mm 2L width of each leaf 100 mm b thickness of each leaf 14 mm t number of extra full-length leaves 2 nf number of graduated-length leaves including the master leaf 8 no Total number of leaves present in the multi-leaf spring 10 n Load acting at the center of the spring 15000 N,10000 N,5000 N 2P The dimensions are taken from practical understanding and the standards available in the market. Structural steel was considered as the leaf material with an

Elastic modulus E = 2.1*105 MPa,

Tensile yield strength Syt = 250 MPa

And

Poisson's ratio=0.3

While performing the analysis in ANSYS (Workbench 16.0) software.

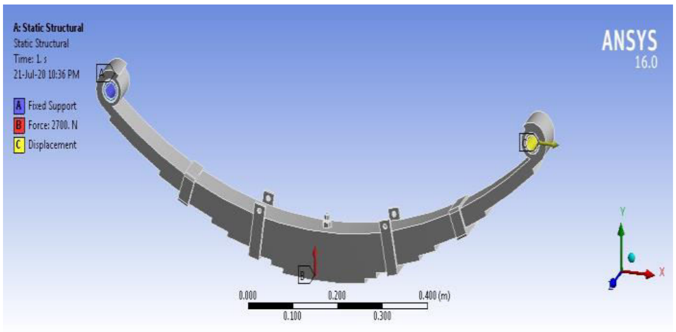
VII. PROPOSED DESIGN

A. STATIC ANALYSIS

The objective of this analysis is to investigate the stresses in the multi-leaf spring within the desirable limits to obtain a practical validation for the theoretical results. After geometric modeling in CATIA V5R18 software the spring is subjected to static analysis, performed in ANSYS (Workbench 16.0) software. The computer compatible mathematical description of the geometry of the object is called geometric modeling. CATIA is basically CAD (computer-aided design) software that allows the mathematical description of the object to be displayed and manipulated as the image on the monitor of the computer [7], whereas, ANSYS is a engineering simulation software that predicts with confidence about the performance of the

product under the real-world environments incorporating all the existing physical phenomena. The layout of static analysis involves meshing, boundary conditions and loading.

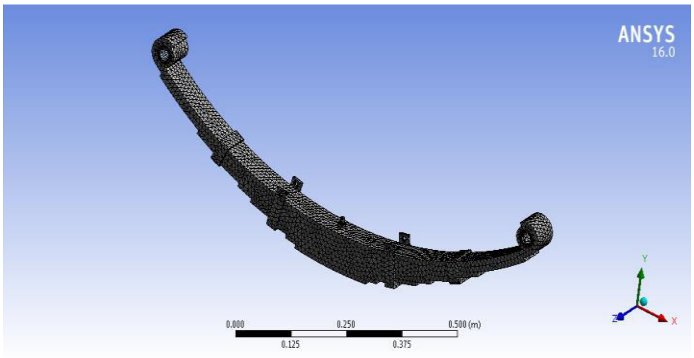
Boundary Condition:



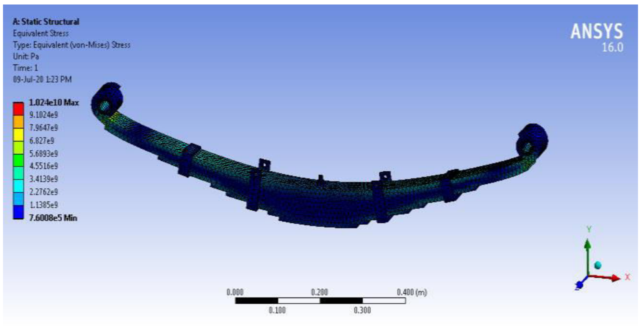
As shown in geometry is fixed at eye ad force applied at bottom face

Meshing

Meshing is basically the division of the entire model into small elements. In order to maintain the shape, it is convenient to select the free mesh, as the geometry of leaf spring contains sharp curves. Element size of 20 mm with medium smoothing was considered for mesh generation. Minimum edge length of the elements was 14 mm. Within the solution domain under the Adaptive Mesh Refinement segment, Max. Refinement Loops was taken as 3 and Refinement Depth as 2. For the convergence plot, the maximum allowable change was considered as 4%. All the 44 faces were selected for mesh generation and total number of nodes and elements were observed as 13423 and 7358 respectively. Figure 1 shows the meshed model of multi-leaf spring. Load 2P (newton) 15000 10000 5000 Maximum Bending stress (σb)max (MPa) 142.691 95.127 47.563



Equivalent Stresses:



Load (Newton)	Von-mises Stress N/mm^2	Bending Stress N/mm^2
1000	139.628	145.507
2000	282.615	291.015
3000	426.152	436.522
4000	568.83	582.0302
5000	712.642	727.540
6000	852.345	873.045
7000	993.515	1018.550
8000	1136.106	1164.060
9000	1277.672	1309.570
10000	1420.076	1455.076
11000	1458.076	1600.583
12000	1602.463	1746.091
13000	1746.091	1891.598
14000	1891.599	2037.106
15000	2037.106	2182.613

It is seen that at load 12000 N, it crosses the yield stress (yield stress is 1680 N/mm^2). By considering the factor of safety 1.5 to 2, it is obvious that the allowable design stress is 840 to 1120 N/mm^2. So the corresponding loads are 6000 to 8000 N. Therefore it is concluded that the maximum safe pay load for the given specification of the leaf spring is 7700N.

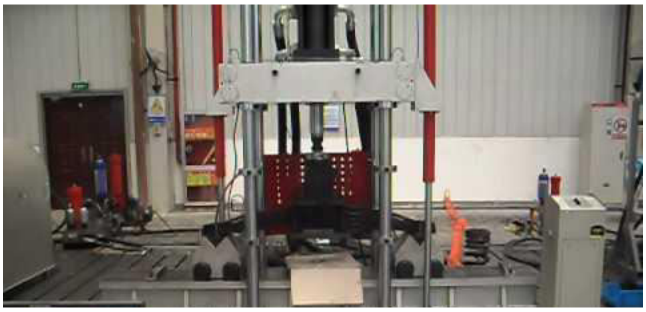
	Von-misses stress N/mm^2	
Load N	Practical	ANSYS
7500	1091.310	1072.133
7600	1105.857	1084.372
7700	1120.408	1097.681

VIII. TESTING OF LEAF SPRING

Static Test In the experimental analysis, the comparative testing of different steel leaf springs will help for drawing the conclusions. The steel and steel leaf springs are tested by using leaf spring test rig. The experimental set up is shown in Fig 2. The leaf springs are tested by recommended standard procedure. The deflection or bending tests of selected spring

Nodes	2087
Element	10870

for comparative study is taken on the universal testing machine. The spring is loaded from zero to the prescribed maximum deflection and back to zero. The load is applied at the centre of spring. In the testing, firstly move the plunger up to desired height so that we can fix the fixture and leaf spring for test. Fix the position of fixture. On the fixture place the specimen. Set the universal testing machine. Apply the loads in steps.



Experiments setup

To conducted the load testing of leaf spring use of UTM (Universal Testing Machine) The experiment use following object to carried out successful analysis.

1. UTM
2. Height Gauge
3. Strain Gauge
4. Loading Movable support

Following figure show the two condition of test one before load applied and after load applied.

IX. REFERENCE

[1] Rohit Ghosh (2016) . STATIC ANALYSIS OF MULTI-LEAF SPRING USING ANSYS WORKBENCH 16.0 International Journal Of Mechanical Engineering And Technology (IJMET) Volume 7, Issue 5, September–October 2016, Pp.241–249,

[2] 1Suraj B. Pawar (2015). Design & Analysis Of Multi Steel Leaf Spring , International Engineering Research Journal (IERJ) Special Issue 2 Page 4468-4474

[3] G HARINATH GOWD (2012) STATIC ANALYSIS OF LEAF SPRING , International Journal Of Engineering Science And Technology (IJEST)

[4] C.K. Clarke (2005). Evaluation Of A Leaf Spring Failure , Journal Of Failure Analysis And Prevention

[5] S. Surya Teja (2016)DESIGN AND STATIC ANALYSIS OF COMPOSITE LEAF SPRING FOR HEAVY VEHICLE ,International Research Journal Of Engineering And Technology (IRJET)