Investigation of Stresses and Deflection in Multi Stage Leaf Spring of Heavy Duty Vehicle by FEM and Its Experimental Verification

Ruchik Tank  
VE Commercial Vehicles Ltd  
102, Industrial Area No. 1,  
Pithampur -454775, INDIA  
rdtank@vecv.in

Srinivas Kurna  
VE Commercial Vehicles Ltd  
102, Industrial Area No. 1,  
Pithampur -454775, INDIA  
skurna@vecv.in

Abstract
In commercial vehicle, Leaf Spring design is an important milestone during product design and development. Leaf springs are the most popular designs having multiple leaves in contact with each other and show hysteresis behavior when loaded and unloaded. Commonly used methods for evaluation of leaf spring strength like endurance trials on field and Rig testing are time consuming and costly. On the other hand, virtual testing methods for strength and stiffness evaluation give useful information early in the design cycle and save considerable time and cost. They give flexibility to evaluate multiple design options and accommodate any design change early in development cycle.

A study has been done in Volvo-Eicher Commercial Vehicles to correlate Rig result with Finite Element Analysis (FEA) simulation result of Multi-stage Suspension Leaf Spring, entirely through Finite Element Analysis route. Correlations have been achieved in both Stiffness and Stress at strain gauge locations between the Rig test and Computer Aided Engineering (CAE) results.

Correlation has helped in reducing product design time and cost of running the rig for failure prediction. Thus, this process has indirectly reduced the cost by using soft validation.

Introduction
The automobile chassis is mounted on the axles, not directly but with the help of some form of springs. This is done to isolate the vehicle body from the road shocks which may be in the form of bounce, pitch, roll or sway. These tendencies give rise to an uncomfortable ride and also cause additional stress in the automobile frame and body. The main parts which perform the function of isolating the automobile from the road shocks are collectively called the suspension system. Leaf spring is one such device which is used in suspension system to safeguard the vehicle and the occupants. These systems are also responsible for ride comfort.

A Leaf spring is as an elastic body, whose function is to distort when loaded and to recover its original shape when the load is removed. Leaf spring absorbs the vehicle vibrations, shocks and bump loads (induced due to road irregularities) by means of spring deflections. This deflection generates potential energy in the leaf spring and which is then relieved slowly. Ability to store and absorb more amount of strain energy ensures a comfortable suspension system. Semi-elliptic leaf springs are almost universally used for suspension in light and heavy commercial vehicles. There are also other types of leaf springs like Parabolic, Bogie Suspensions, Bell crank Suspensions etc.

Leaf spring consists of a number of leaves. The leaves are varying in length. The leaves are usually given an initial curvature or camber so that they will tend to straighten when loaded. The leaf spring design is based upon the theory of a beam of uniform strength.

The longest leaf has eyes on its ends. This leaf is called main or master leaf, the remaining leaves are called graduated leaves. All the leaves are bound together by means of steel straps. The spring is mounted on the axle of the vehicle. The entire vehicle load rests on the leaf spring. The front end of the spring is connected to the frame with a simple pin joint, while the rear end of the spring is connected with a shackle. Shackle is the flexible link which connects between the leaf spring rear eye and the vehicle frame.

Simulate to Innovate
When the vehicle comes across a projection on the road surface, the wheel moves up, leading to deflection of the spring. This changes the length between the spring eyes. If both the ends are fixed, the spring won’t be able to accommodate this change of length. Hence, a shackle is provided at one end for a flexible connection. The front eye of the leaf spring is constrained in all translational directions keeping rotation free, whereas the rear eye is not constrained in longitudinal direction. This rear eye is connected to the shackle. During loading the spring deflects and moves in the direction perpendicular to the load applied.

The design of Semi-elliptical leaf-springs along with helper springs being a multi-stage operation has different stiffness corresponding to the two stages of the leaf spring. During the first stage of leaf spring only the top five of the nine leaves are essentially carrying the load and deforming. The second stage comes into force when the gap between the fifth and the sixth leaves closes after the deformation in top five leaves becomes greater than the initial gap. In this stage all the nine leaves share the load and contribute to the stiffness leading to an increase in the overall spring stiffness.

Multi stage leaf springs suspensions are used in automotive industry to gain maximum load rating while retaining good ride characteristics when the vehicle is in unladen condition. Currently the Leaf spring design was mainly based on simplified theoretical equations and trial-and-error methods. The simplified equation models used were limited to the three-link mechanism assumption and linear beam theory.

The aim of this study is to establish correlation between FEA simulation and Rig test data for Multi-stage semi-elliptical suspension leaf spring. This method would then be used to predict durability and other suspension characteristics like stiffness from the leaf geometry at design stage itself. The rates predicted can then be used in multi-body dynamic models or full vehicle Noise, Vibration and Harshness (NVH) models. This methodology would reduce product development time & cost significantly.

**Literature review**

Leaf springs designing have involved engineering calculations, finite element simulations with extensive fatigue tests in reliability labs. A study on effects of production parameters, geometrical tolerances and material variations was done through CAE simulations by Ahmet Kanbolat et al [4]. Non linear finite element analysis can be used to capture physical behavior of leaf springs on road loads as well as rig tests. Felipe Nogueira et al [5] did FEA based study on wind up deformation of parabolic leaf spring. This paper grants additional work on leaf spring simulation through non linear FEA to verify methodology & correlate between the physical test and CAE established.

**Methodology**

The main objective of this paper is to correlate CAE results with Rig test. Correlation between Test and Virtual will help in reducing product design time and cost of Rig Test. This FEA based correlation methodology will help in variable rate suspension design whose stiffness characteristics are more complicated than single rate suspension. Correlating this problem in FEA with Rig Test is challenging task as static and dynamic friction will play key role in simulation.

We have developed this methodology primarily for Semi-elliptical Leaf Springs & then used same approach for different types of Leaf Springs.

Every design method with CAE in the process must be based on validated virtual models. We have validated our finite element analysis models through experimental studies. These studies were carried out through controlled manufacturing processes, measured manufacturing variables and comparison of the test results and virtual model using real variables. Primary output obtained from test or finite element analysis of leaf springs were spring rate (i.e. force, displacement) and stresses.

**Correlation Parameters**

a. Spring Stiffness
The most important parameter for spring is its stiffness. Some of the factors which will affect the stiffness of spring are:

- **Geometrical stiffness:** depends on the number of leaves and their dimensions.
- **Contact stiffness:** during loading leaves slide over each other and load transmits through the contact, this makes problem highly non-linear for simulation. Contact stiffness depends on the pre load of the U-bolts and friction between leaves. Static and Dynamic friction have major role on the contact stiffness and will vary as per increasing displacement.

**b. Stress**

Stress is the second parameter which is taken for correlation to predict the durability of the leaf spring. Durability of the leaf spring is of high importance as failure of a leaf spring on a running vehicle can lead to severe road safety issue.

**Reference Study**

There are three methods to evaluate the strength of leaf spring. They are:

- a. Theoretical Calculation
- b. Experimental setup
- c. Virtual Simulation

**Theoretical Calculation**

The Theoretical stress calculations for Cantilever Beam shown in Figure 1 are done using equations 1, 2 and 3 (for variable definitions refer Definitions/Abbreviations table on page no. 9).

\[
\sigma = \frac{6FL}{dnb^2} \quad (1)
\]

\[
S = \phi \frac{4FL^3}{Enb^2} \quad (2)
\]

\[
\phi \approx \frac{3}{2 + \frac{n^2}{m}} \quad (3)
\]

The theoretical stress as calculated by the above equation near the mounting of the U bolt comes out to be nearly 1100 MPa.

**Experimental Setup**

Test rig setup (Figure 2) should be modelled similar to the vehicle assembly condition. In order to model this setup, one half of the Axle-spring assembly is taken. A leaf spring type suspension consists of leaf spring panels, a center bolt, a U-bolt, and a bushing. Multiple leaf spring panels are fixed by the center bolt and the U-bolt. The frame can be considered rigid as no frame deformation between the two mounting locations of the leaf springs is observed on the vehicle while in action. Thus, the leaf spring suspension brackets are rigidly connected to the base of the test rig for the simulation.
The axle is replaced by a hydraulic actuator, which gradually applies displacement in the vertical direction. The pin joints at eye location and shackle location are same as those on vehicle. When the load is applied the spring will move only in longitudinal direction with help of revolute joint and shackle.

Figure 2: Test Rig Setup showing Leaf Spring and Actuator.

Four strain gauges are used to measure the strain on the leaf. They are located on the top leaf at different locations shown in Figure 3. Test is performed by applying load gradually on leaf spring on rig. Output data is measured (Strain and reaction force) on top leaf spring.

Figure 3: Strain gauge locations
Material

The rudimentary necessity of selected grade of steel for springs is that it must have sufficient hardness to ensure a full martensitic structure throughout the section. In general terms higher alloy content is mandatory to ensure adequate hardenability when thick leaf sections are used. Refer Table 1 for material properties details of leaf spring.

Table 1: Specifications of the Leaf Spring

<table>
<thead>
<tr>
<th>Material</th>
<th>SUP 09</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus of the spring (MPa)</td>
<td>210000</td>
</tr>
<tr>
<td>Ultimate Tensile Strength (MPa)</td>
<td>1240</td>
</tr>
<tr>
<td>Total Length of the spring (Eye to Eye) (mm)</td>
<td>1500</td>
</tr>
<tr>
<td>Distance between U bolts (mm)</td>
<td>112</td>
</tr>
<tr>
<td>Free Camber (At no load condition) (mm)</td>
<td>55</td>
</tr>
<tr>
<td>No. of full length leaves</td>
<td>02</td>
</tr>
<tr>
<td>No. of First Stage Leaves</td>
<td>05</td>
</tr>
<tr>
<td>Total No. of Leaves</td>
<td>09</td>
</tr>
<tr>
<td>Thickness of each leaf (mm)</td>
<td>13</td>
</tr>
<tr>
<td>Width of each leaf spring (mm)</td>
<td>90</td>
</tr>
<tr>
<td>Maximum Deflection (mm)</td>
<td>190</td>
</tr>
</tbody>
</table>

Experimental Load-Deflection Curve

A static load is applied by a universal testing machine and the corresponding deflection and stress values are observed. A graph shown in Figure 4 is plotted between load and deflection.

Virtual Simulation

Computer Aided Engineering tools are widely used in the automotive industries. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulation rather than physical prototype testing.

Virtual simulation of leaf spring assembly test set up was performed in three inherent steps -

1. Finite element modeling (Preprocessing in CAE software)
2. Identification and incorporation of analysis parameters (setting solver FE inputs to get realistic spring behavior under the load)
3. Results interpretation and revisit above steps, if required.

In present study Radioss is used as solver and Altair Hyper works as pre and post-processor.

**Finite Element Modeling**

The geometry of leaf spring components are modeled in Pro-E software in freely assembled condition. Geometry with no load condition was taken to ensure that initial zero stress condition geometry in analysis are consistent with rig testing.

**Meshing**

FE mesh generation over the 3-d Computer Aided Design (CAD) model was done using Altair Hyper mesh as pre-processor. Each leaf is modeled with HEXA and PENTA elements having global element size 8mm shown in Figure 5. It was ensured that each leaf has three layers of element across its thickness. Other components of assembly, shackle, U-bolt and clamp brackets are also modeled using solid elements. Appropriate material properties with their stress-strain curves are assigned to the elements.

![Figure 5: Meshed model of Leaf Spring for CAE Analysis.](image)

**Setting Contact Region**

Contact conditions are formed where bodies meet or are in touch with each other. Contacts are used to transfer structural loads and heat flows across the contact boundaries and connect the various bodies. Depending on the type of contact, the analysis can be linear or nonlinear. Type-7 interface is used in this particular model to define the contact between all the parts.

**Joints**

A joint is an idealized kinematics linkage that controls the relative movement between two bodies. Joint types are characterized by their rotational and translational degrees of freedom as being fixed or free.

In this assembly revolute joints are used at front and Rear suspension pin locations shown in Figure 6. The suspension brackets are modeled as rigid cylinders.

The general spring property (Type-8) is used to model this joint connecting between bottom hole of shackle & rear eye of the spring. This joint is fixed in all degrees of freedom except rotation about x-axis, which is perpendicular to the plane of spring movement.

*Simulate to Innovate*
When load is applied on leaf spring, adjacent leaves slide over each other. This phenomenon creates continuous and forced edge in addition to surface contact between the leaves with dynamic friction. Simulation of such events requires robust contact algorithms.

The spring has six DOF computed in a skew system frame: $R_x$, $R_y$, $R_z$, $\dot{R}_x$, $\dot{R}_y$ and $\dot{R}_z$.

**Boundary Conditions**

In rig test, Leaf spring suspension bracket were considered rigidly connected to the test bed with rotational bushes at leaf end & shackle. Therefore in simulation, the ends are fixed in all degrees of freedom except rotation about X-Axis.

**Application of Test Loads**

After the bolt preload, test actuator displacement is applied to the centre of the leaf spring and ramped up to sufficient duration as per Figure 7. The rate of application of load is kept similar to that of test rig.

**Identification and incorporation of analysis parameters**

After detail FE modeling of leaf spring assembly, it was desirable to set certain analysis parameters to correctly represent analysis conditions. These parameters have significant impact on structural behavior of leaf spring under loading. On the other hand they are very effective in taking virtual simulation further level up towards actual. A summary of them is presented below.
The U-bolt assembly process introduces preloading stress and makes it difficult for the user to simulate leaf spring type suspensions.

U-Bolt is modeled using HEXA and PENTA elements with CBEAM elements at a section as shown in Figure 9.

Bolt pretension load is applied as shown in Figure 8 at the start of analysis using Type 32 Spring and dynamic relaxation Control Card. The U-bolt is put under preload condition during dynamic relaxation stage. The bolts remain in preload condition for complete duration of the analysis. For preload value calculations refer Appendix 1.

Contact Parameter: Friction

Iform = 2, Stiffness (incremental) formulation i.e. It will take care of dynamic friction w.r.t. loading is used in this analysis.
Contact Parameter: Penetration

Deep Penetrations are not tolerated in Interface Type-7. It Leads to:
- high penalty forces,
- small time step,
- infinite loop message

Therefore, it is important for the user to remove all the penetration manually from the model for best results.

However, there are some parameters which can be used to avoid initial penetration.

Option-1:
- \( \text{Inact} = 3 \), change node coordinates to avoid initial penetration

This is not recommended as it will forcefully move the penetrated node which may cause instability in the model & unrealistic results.

Option-2:
- Initially, the gap is variable with time but initial gap is adjusted (the node is slightly depenetrated) - up to max 5%
- \( \text{gap} = 0.5 \times (\text{Th} - \text{L}) \)

If the penetration is above 5%, run will be terminated by penetration error.
- This will avoid human error.
- (Recommended parameter)

Output Requested

Stress and strain output on the top leaf at four locations similar to test strain gauge and reaction force are measured. Kinetic energy, interface energy and total energy are requested in analysis output to verify the behavior of virtual model.

Advance Mass Scaling

Time plays a major role when it comes to our Corporate World. In order to save time without compromising accuracy, smart work is required. In this case it is achieved by AMS.

AMS is a Method to increase computing Speed & Maintain Accuracy. This is obtained by artificially increasing the time step, similarly to what is done in traditional controlled mass scaling.

Note: Advanced Mass Scaling is specific to Radioss. It is called advanced as it allows its application to the entire model without degrading computing performances and result quality.
Results and Correlation

Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems due to the fact that virtual analysis cannot predict all variables in complex assemblies, therefore the validation of CAE results is important.

Correlations have been achieved in both spring rate and Stress at measured locations between the Rig test and CAE results.

Correlation

a. Spring Stiffness

High correlation between stiffness of CAE and Rig test of achieved as shown in the Figure 10.

![Figure 10: Correlation of Stiffness between Rig Test (Red line) and CAE Results (Blue line)](image)

b. Stress

Very good correlation (minimum 95%) achieved between Rig test data and FEA results at SG1, SG2, SG3 and SG4 measured locations correspondingly as shown in Figure 11 & 12

![Figure 11: Stress contour in Computer Aided Engineering](image)
Figure 12: Correlation of Stresses between Rig Test (Red line) and CAE Results (Blue line) for 
(a) Strain Gauge 1   (b) Strain Gauge 2 
(c) Strain Gauge 3   (d) Strain Gauge 4
Conclusions

In the current work, leaf spring is modeled and Quasi-static analysis is carried out by using Radioss software and it is concluded that for the given specifications of the leaf spring FEM gives quite accurate prediction of critical area stresses from the viewpoint of Quasi-static loading. The results of Quasi-static analysis of the leaf spring using the commercial solver and analytical results shows better correlation at all measured locations at all the measured displacements.

The stiffness of the leaf spring is studied by plotting load versus deflection curve for whole working load range. Under the given loading conditions, stiffness correlated well between CAE and Test.

There is very good correlation of CAE results and Rig test measurements for spring rate and stresses. This study will help to understand more the behavior of the spring and give information for the manufacturer to improve the strength of the spring using CAE tools. It can help to reduce cost and time required for research and development of new products as shown in Figure 14.

Thus with the help of this study we are able to successfully establish the correlation between FEA simulation and Rig test data for Multi-stage elliptical suspension leaf spring & then using the same approach for different types of leaf-springs.
Simulate to Innovate

Figure 14: Benefits of Virtual Prototype Process.

REFERENCES

ACKNOWLEDGEMENTS

The authors would like to express their gratitude to Mr. Rajinder S. Sachdeva, Executive Vice President, Technology Development, VE Commercial Vehicles Ltd, for granting permission to publish this work. Special thanks to Mr. V.Chandrasekhar and Mr. Uma Shanker Gupta for their guidance and encouragement to develop present work. Also, the contributions and support of all system engineers and respective testing engineers are highly appreciated.

Definitions/Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer Aided Engineering</td>
</tr>
<tr>
<td>( \sigma )</td>
<td>Bending stress of the spring material (Mpa)</td>
</tr>
<tr>
<td>( F )</td>
<td>loading of spring (N)</td>
</tr>
<tr>
<td>( L )</td>
<td>functional spring length (mm)</td>
</tr>
<tr>
<td>( n )</td>
<td>total number of spring leaf</td>
</tr>
<tr>
<td>( n_e )</td>
<td>number of extra full-length leaves</td>
</tr>
<tr>
<td>( b )</td>
<td>width of spring leaf (mm)</td>
</tr>
<tr>
<td>( t )</td>
<td>thickness of spring leaf (mm)</td>
</tr>
<tr>
<td>( \delta )</td>
<td>spring deflection (mm)</td>
</tr>
<tr>
<td>( \nu )</td>
<td>shape coefficient</td>
</tr>
<tr>
<td>( E )</td>
<td>modulus of elasticity in tension (Mpa)</td>
</tr>
<tr>
<td>( T )</td>
<td>Installation Torque (N mm)</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>Helix Angle</td>
</tr>
<tr>
<td>( \mu_e )</td>
<td>thread coefficient of friction</td>
</tr>
<tr>
<td>( \mu_c )</td>
<td>collar coefficient of friction</td>
</tr>
<tr>
<td>( D )</td>
<td>Diameter of Bolt (mm)</td>
</tr>
<tr>
<td>( p )</td>
<td>Pitch of bolt (mm)</td>
</tr>
</tbody>
</table>
Appendix 1

Bolt Preload Calculation

Bolt preload comes from the installation torque $T$, used to tighten the bolts. The torque is converted into bolt preload by the inclined plane of the bolt thread helix.


$$ P_t = \frac{T}{KDP} $$

(4)

Where torque coefficient $K$ is a function of thread helix angle ($\psi$), thread coefficient of friction ($\mu_t$), and collar coefficient of friction ($\mu_c$). (Shigley, Mechanical Engineering Design, 5 ed., McGraw-Hill, 1989, p. 346, Eq. 8-19)

$$ K = \left\{ \left[ \frac{1 + 0.5}{\mu_t} \right] + \left[ \frac{D - \pi p \frac{\mu_t}{\sin(\psi)} - \pi u_2}{\sin(\psi)} \right] + \left[ 0.525 \mu_c D \right] \right\} $$

(5)

Replacing the variables with corresponding values of our test model in the above equations

$T = 62000$ N mm
$D = 20$ mm
$p = 2.5$ mm
$\psi = 60^\circ$
$\mu_t = 0.15$
$\mu_c = 0.15$

From this, we get the initial preloading force value of 160kN, which will be used in virtual simulation.