FEA-based durability assessment: A case study using a parabolic leaf spring

1S. ABDULLAH, 1F.N. AHMAD REFGAH, 1A. JALAR, 2L.B. CHUA, 1A.K. ARIFFIN AND 1A. ZAHARIM
1Universiti Kebangsaan Malaysia
43600 UKM Bangi, Selangor
MALAYSIA
2AM Spring Sdn. Bhd. Klang, Selangor, MALAYSIA

Abstract: - This paper presents the about fatigue life prediction based on finite element analysis and variable amplitude loading (VAL). Parabolic spring is one of the vital component in a vehicle suspension system, and it is commonly used in trucks. It needs to have an excellent fatigue life, and recently, manufacturers rely on constant loading fatigue data. This study objective is to simulating the variable amplitude loading for the fatigue life analysis. Service loading of parabolic spring has been collected using data acquisition system. Finite element analysis (FEA) was performed on the spring model so stress and damage distribution can be observed. Experimental works was done in order to validate the FEA result. The main finding can be obtained when VAL as the input, and it was then combines with the FE model. Fatigue life simulation was performed and analyzed. From the results, the fatigue damage using VAL was predicted and the result was correlated with FEA.

Key-Words: - Fatigue life; variable amplitude; parabolic spring; simulation; FEA

1 Introduction

Large vehicle need a good suspension system that can deliver good ride and handling. At the same that component need to be lightweight and had an excellent fatigue life. Fatigue is one of the major issues in automotive component. It must withstand numerous numbers of cycles before it can fail, or never fail at all during the service period.

In reality, most engineering components and structures are subjected to variable amplitude loading conditions at which stress–strain cycles fluctuate with time [1]. At this condition, components are tend to fail under various source of loading. Fatigue failure of mechanical components is a process of cyclic stress/strain evolutions and redistributions in the critical stressed volume [2].

From the viewpoint of engineering applications, the purpose of fatigue research consists of the prediction of fatigue life of structures, increasing fatigue life and simplifying fatigue tests [3]. All these can be obtained by applying fatigue tests of full-scale structures under a random load spectrum that give us more challenging for the real test.

In order to simplify the test, it then comes to the application of related computational simulation. Using this technique, the fatigue life of components can be firstly predicted before proceed to the durability test. One of the simulation scope related to this issue is finite element analysis (FEA) approach.

Parabolic spring is widely used in automotive and one of the components of suspension system. It consists of one or more leaves. The spring was thicker at the center and has varying thickness along its length [4,5]. At least the main leaf of which is design in such manner that the cross sectional moment of inertia of the leaf varies in the longitudinal direction of the sprang, so that the bending stress in the leaf will be essentially equal over the major portion of the length of the spring [6]. Parabolic leaf springs required relatively little space but on the other hand their production is costly [4]. This spring serves to absorb and store energy and then release it. During the operation, the stress in the spring must not exceed a certain maximum in order to avoid settling or premature failure [7].

Parabolic springs are subjected to cyclic compression and tension load when the truck was drove on the road. In industry, manufacturer only manage to test the fatigue life of these spring using constant amplitude loading. This is because VAL fatigue test is time consuming and adding more cost.

This paper presents the fatigue life prediction based on finite element analysis and variable
amplitude loading. Data collection of service loading was carried out and this data was used as an input in the simulation. Critical areas were obtained from the FEA. Finally, the life of this parabolic spring can be predicted, showing the total life within the acceptable limit.

2 Literature Review

2.1 Fatigue life prediction

Fatigue life prediction is based on knowledge of both the number of cycles the part will experience at any given stress level during that life cycle and another influential environmental and use factors [8]. For strain-based fatigue life prediction, this rule is normally applied with strain-life fatigue damage models, such as the Coffin–Manson relationship [9,10], i.e.

\[ \varepsilon_a = \frac{\sigma'f}{E} \left(2N_f\right)^b + \varepsilon'f \left(2N_f\right)^c \]  

(1)

where \( E \) is the material modulus of elasticity, \( \varepsilon_a \) is a true strain amplitude, \( 2N_f \) is the number of reversals to failure, \( \sigma'f \) is a fatigue strength coefficient, \( b \) is a fatigue strength exponent, \( \varepsilon'f \) is a fatigue ductility coefficient and \( c \) is a fatigue ductility exponent. Recently, only fatigue test data or/and fatigue curve under constant amplitude loading have been given in many handbooks of the fatigue and mechanical properties of metals [11].

2.2 Finite element method

Finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. With the advances in computer technology and CAD system, complex problem can be modeled with relative ease [12]. It is very useful, more in prototyping stage. It helps to reduce cost and save times. Previous work on leaf spring simulation had been done by Liu [13], but it only discuss about the mono leaf spring. Finite element analysis of leaf spring have been made using beam element to predict the jounce condition [14] and others using solid element [15, 16].

3 Methodologies

3.1 Finite element modeling

Parabolic spring was modeled using a commercial software and all the specification was accordingly followed the relevant drawing standard. The spring geometry is shown in Fig. 1, and it consist of two leafs. In this study, several assumptions have been made i.e. the chosen material was homogenous, no interleaf friction was defined. To reduce the complexity of simulation, shackle and bush was not model together, only represent by boundary condition and shot peening stress effect and nip stress also omitted.

For the FEA simulation, 20-nodes hexahedral element was used. Model of a parabolic spring was partition to get more accurate geometry and easier to mesh. Boundary condition was set according to real static load test which is the front eye was allowing only rotational at y axis and the rear eye was constrained in y and z translation and x and z rotation, allowing free x translation and y rotation [15, 16]. Contact from main-to helper leaf also been defined. Helper leaf was constrained at 2nd degree of freedom to represent the clip that holds that to spring together. Finally, vertical load was applied at the center flat of the spring.

3.2 Experimental works

In order to validate the analysis, the experimental work of static loading was performed. Parabolic spring was measured in order to locate the strain gauges for which these gauges were used to measure the static strain. After that, the area was polished in order to remove coating element and strain gauge was then attached to the spring using cyanoacrylate adhesive material. The parabolic spring was then placed at the static load test rig. Lastly, strain gauge was attached to data logger and static load was applied vertically. Fig. 2 was taken from [16] shows the similar test that has been performed for the static loading.
3.3 Random signal measurement

SoMat eDAQ data acquisition system was used to collect strain data and Fig. 3 shows the experimental set up. Strain gauge had been attached to the parabolic spring of a truck. Locations of the strain gauges were shown in Fig. 4. In this study, both left and right side spring was attached with the strain gauge. Then truck was driven on a public road in Malaysia with average speed of 60-70 km per hour.

The collected data was sampled at 200 Hz ad containing 17000 discrete data points, thus, it gave 86 seconds of total record length. The range of amplitude of the collected data (as shown in Fig. 5) is at -910 µE minimum and 760 µE maximum.

The collected strain loading was then used as an input for FEA-based fatigue simulation. In this simulation, the Morrow approach had been used for correcting the damage calculation with considers the mean stress effect. In a case of the loading being predominantly compressive, particularly for wholly compressive cycles, the Morrow model provides more realistic life estimates [18].

The material that had been used for the parabolic spring was SAE5160H low carbon alloy steel, which normally used for the spring fabrication. The material properties are shown in Table 1.

Table 1. The material properties of SAE 5160H steel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ultimate tensile strength, $S_u$</td>
<td>1537</td>
</tr>
<tr>
<td>(Mpa)</td>
<td></td>
</tr>
<tr>
<td>Modulus of elasticity, $E$ (GPa)</td>
<td>207</td>
</tr>
<tr>
<td>Fatigue strength coefficient, $\sigma_f$ (MPa)</td>
<td>2063</td>
</tr>
<tr>
<td>Fatigue strength exponent, $b$</td>
<td>-0.08</td>
</tr>
<tr>
<td>Fatigue ductility coefficient, $\epsilon_f$</td>
<td>9.56</td>
</tr>
<tr>
<td>Fatigue ductility exponent, $c$</td>
<td>0.26</td>
</tr>
</tbody>
</table>
4 Results and discussions

4.1 Finite element analysis using static loading

From the FEA simulation, the critical locations of the parabolic have been predicted. Fig. 6 shows the stress distribution in a simulated parabolic spring. In this figure, stress value which was found at maximum loading applied was 1018 MPa. Also, from this figure, high stresses were distributed around the spring seats towards the spring eye.

![Stress distribution on simulation](image)

The stress distribution patent is resulting from the thickness of the spring. Center flat is the thickest part of all the spring, but that area deflected the most. From Fig. 6, the spring was almost straight after maximum load been applied. In Fig. 7, the deflection of the spring was gradually increased. For the simulation purpose, the deflection curve was linear, because the linear static analysis that had been performed. This is to reduce complexity of the simulation.

For the validation, the strain-based FEA stresses were then compared to the experimental finding (refer to Fig. 2 for the test set-up), as tabulated in Table 2. The deviation value was calculated based on the actual values, and the negative value means experimental result was found to be higher than FEA but most of the FEA stress value was higher than experimental result. This may due to residual stress by shot peening, and this factor was not included in the simulation.

![Load vs deflection plot](image)

The deviation situation was occurred because of, the interleaf friction and nip stress inside the parabolic spring was not included in the FEA simulation. Also, the analysis was performed using linear static loading assumption. These assumptions were taken in order to simplify the analysis and also reduce the simulation time. In order to obtain the better result, non-linear analysis is preferable [13-16]. In non-linear analysis, a large deformation of the geometry will be taken account during the simulation.

<table>
<thead>
<tr>
<th>Stress (MPa)</th>
<th>At load 20544.93 N</th>
<th>At load 40373.98 N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation</td>
<td>Experimental</td>
<td>Deviation (%)</td>
</tr>
<tr>
<td>17.68</td>
<td>27.67</td>
<td>-56.51</td>
</tr>
<tr>
<td>54.67</td>
<td>56.45</td>
<td>-3.25</td>
</tr>
<tr>
<td>56.64</td>
<td>36.23</td>
<td>36.04</td>
</tr>
<tr>
<td>56.17</td>
<td>56.13</td>
<td>0.075</td>
</tr>
<tr>
<td>54.87</td>
<td>39.88</td>
<td>27.32</td>
</tr>
<tr>
<td>15.02</td>
<td>6.78</td>
<td>54.85</td>
</tr>
<tr>
<td>25.20</td>
<td>35.14</td>
<td>-39.45</td>
</tr>
<tr>
<td>77.60</td>
<td>76.11</td>
<td>1.92</td>
</tr>
<tr>
<td>79.98</td>
<td>55.92</td>
<td>30.08</td>
</tr>
<tr>
<td>80.00</td>
<td>76.53</td>
<td>4.34</td>
</tr>
<tr>
<td>77.62</td>
<td>59.26</td>
<td>23.65</td>
</tr>
<tr>
<td>25.29</td>
<td>13.75</td>
<td>45.64</td>
</tr>
</tbody>
</table>
4.2 Fatigue damage analysis

Fig. 8 shows the damage histogram plotted using the analyzed variable amplitude loading analysis. The fatigue damage distribution was then plotted based on particular fatigue cycles. From this histogram, cycles within the range below 5000 µE may not contribute to the damage of this spring. Damage occur at cycles from range above 5000 µE to 12000 µE, but not very significant. The highest damaged, which is 7.033 x 10^4 occur at highest cycles range. The highest cycles came when vehicle went through high bump or deep hole, which mean higher deflection occur to the spring. This shows that in regular operation of vehicle and on normal road surface, will not cause a lot of damage to the spring. In the worst case of scenario, there is a condition called metal-to-metal contact. It happens when the spring seat was completely pushes the jounce to the curb frame.

In Fig. 9 (a), damage area can be found when spring was applied with random loading and this image was correlated well with the FEA stress distribution area. From spring seat towards the spring eye was where the high damage occur, but the eye itself didn’t show significant or no damage at all. The eye only experiencing rotational movement, where the magnitude deflection was relatively small compared to the critical area. The center of the leaf was connected to axle and clamp with U-bolt and it makes the fatigue damage effect became lower.

Fig. 9 (b) shows the life prediction of the parabolic spring. The lowest life cycles was predicted at the highest damage area. Generally, in constant loading fatigue test, parabolic spring need to surpass 300000 cycles, and as predicted in simulation, the spring can sustain up to 1x10^7 cycles with the maximum load, 48396 N was applied.

5 Conclusion

Fatigue life prediction was performed based on finite element method and fatigue life simulation method. Using the variable amplitude loading, it predicted the actual case that happen when the vehicle run on the road. Both analysis results were well correlated. This simulation gives the prediction about the damage that occurs. This study hopes to give more information for the manufacturer to improve the fatigue life of the parabolic spring. It can reduce cost and times to do the research and development. Recently, manufacturer only rely on fatigue test with constant amplitude loading, and this study will help to understand more the behavior of the parabolic spring.

References:
[11] JB. Conway, LH. Sjodahl, Analysis and


Fig. 9. Simulated FEA-based fatigue prediction: (a) Damage contour, (b) Life to failure contour