Fatigue analysis in case of random vibration base excitation

ALBIN BÄCKSTRAND
ANOOB VALIYAKATH BASHEER
FILIP GODBORG
MAGNUS STERVIK
PRITHVIRAJ MADHAVA ACHARYA
ROBIN HAFSTRÖM
Abstract

Components in mechanical systems often have requirement specifications with respect to material fatigue. This calls for fatigue analyses in order to ensure sufficient operational lives. Doing this numerically, in the time-domain, is often computationally heavy, due to long time history inputs. Hence, it would be beneficial to carry out numerical analyses in the frequency domain instead.

The purpose of this study is to establish an understanding for how numerical methods can be used to estimate material fatigue in the frequency domain when the load is a stationary random vibration. The results will then be compared with those obtained by well established time-domain analyses of fatigue damage.

The results show that, by carrying out numerical fatigue analyses in the frequency domain, only one FE-analysis is required to obtain the transfer functions since it can be used for different kinds of signals. The FE-analysis in the frequency domain is more computationally efficient since it does not require time history inputs. Thus, it is considered more computationally efficient to estimate material fatigue in the frequency domain. However, the frequency domain method is not as accurate as the time domain method, which makes the frequency domain method more useful in early phase development.

Keywords: Random vibration, Material fatigue, Modal analysis, Frequency domain, Time domain, Power spectral density, White noise, uniaxial/multiaxial fatigue
## Contents

1 Introduction  
1.1 Background .............................................. 1  
1.2 Purpose .................................................. 2  
1.3 Aim ....................................................... 2  
1.4 Limitations .............................................. 2  

2 Theory  
2.1 Fatigue Analysis .......................................... 3  
2.1.1 Fatigue analysis in time domain ...................... 3  
2.1.2 Fatigue analysis in frequency domain ................ 3  
2.1.3 Multiaxial fatigue in frequency domain ............. 4  
2.2 Power spectral density .................................. 5  
2.3 Transfer function ....................................... 5  

3 Method  
3.1 Load description ......................................... 8  
3.2 Stress analysis ........................................... 8  
3.2.1 Numerical model & constraints ....................... 9  
3.2.2 Modal analysis ....................................... 10  
3.2.2.1 Frequency domain analysis ...................... 10  
3.2.2.2 Time domain analysis ............................. 11  
3.2.3 Selection of evaluation points ....................... 11  
3.3 Fatigue analysis .......................................... 11  
3.3.1 SN-curve ............................................. 11  
3.3.2 Fatigue analysis in the time domain ................. 12  
3.3.3 Fatigue analysis in the frequency domain .......... 12  

4 Results and discussion  
4.1 Modal Analysis .......................................... 13  
4.2 Evaluation points ....................................... 13  
4.3 Fatigue damage ........................................... 14  

5 Conclusions and future work  

References ............................................... 19  

A Appendix ............................................... 

iii
A.1 Evaluation points ........................................... I
A.2 Stress evolutions in time and frequency domains ............ II
A.3 Effect of constraints on frequency ............................. IV

The figure on the front page shows a FE-model of an air tank, found underneath a typical Volvo truck chassis.
1 Introduction

1.1 Background

Components mounted on truck chassis are subjected to random vibrations due to movement of the base structure, which in long term causes material fatigue. The components have requirement specifications, which calls for fatigue analyses in order to ensure sufficient operational lives. One way of doing this is by physical tests using electrodynamic shakers on hardware components. Since these tend to be both time and cost consuming, it would be beneficial to do numerical fatigue analyses before physical tests. Doing this in time domain is often computationally heavy considering high cycle fatigue. Hence, it would be more efficient to carry out the numerical analysis in the frequency domain, which does not require time history inputs.

One truck component, of limited size, is an air tank that is mounted with a small interface to the chassis. The component is typically subjected to vibrations induced by movements of the chassis and will in this study be used for the evaluation of the time- and frequency-domain fatigue analysis in case of random vibrations. A simplified model of the air tank, together with the straps and the arm connecting it to the chassis, is shown in Figure 1.1 where the global coordinate system used in numerical analysis is defined.

![Figure 1.1: Meshed air tank with it’s components](image-url)
1.2 Purpose

The purpose of this study is to understand and implement low-effort frequency domain fatigue analyses instead of classical time domain fatigue analysis in case of random vibration loads.

1.3 Aim

The aim is to, under 8 study weeks, develop numerical tools for fatigue analysis in the frequency domain and compare results with those obtained by well established time-domain analyses. The accuracy of the two methods will be compared using fatigue damage.

1.4 Limitations

All analyses will be carried out for model (including geometry and mesh) provided by Volvo Trucks. The number of FE-nodes for the current mesh are limited to 21309. Other mesh configurations will not be considered. Surface roughness, which can have an effect on the fatigue evaluation, will not be taken into account. The material is assumed to be elastic so that no plastic deformation will be considered. Hence, non-linear effects due to deformation will be omitted. The amplitude of the vibration input load is assumed to have a Gaussian distribution with zero mean.
2 Theory

This chapter presents the theoretical background for the studies.

2.1 Fatigue Analysis

In order to estimate the fatigue life, S-N analysis in the form of Wöhler curve is used. Wöhler curve is a log-log diagram with the amplitude stresses, $\sigma_a$, plotted against the number of stress cycles to failure, $N_f$. [1]

In order to find the total damage of a component, the number of stress cycles, $N_i = N(S_i)$ for a given stress amplitude, $S_i$, are evaluated against the number of cycles to failure, $N_f = N_f(S_i)$, for that particular stress amplitude found from the Wöhler curve. From the Palmgren–Miner rule, the total accumulated damage, $D_{tot}$, is expressed as the sum over the ratio between the number of stress cycles to the number of cycles to failure as

$$D_{tot} = \sum_i \frac{N_i}{N_f} = \sum_i \frac{N(S_i)}{N_f(S_i)}$$

(2.1)

Since the accumulated damage is proportional to the damage required to failure, the fatigue life $T_{life}$ can be obtained as the ratio between the length of the time history $T$ and the accumulated damage presuming steady state process [2].

$$T_{life} = \frac{T}{D_{tot}}$$

(2.2)

2.1.1 Fatigue analysis in time domain

Rainflow counting is used to evaluate the stress cycles in the stress time history in cases of varying amplitude. Fatigue damage corresponding to stress cycles are added using the Palmgren–Miner rule (2.1) to calculate the accumulated damage in a material point. Fatigue failure is presumed when $D_{tot} = 1$.

2.1.2 Fatigue analysis in frequency domain

In frequency domain fatigue analysis, the information available is the input for the stress Power Spectral Density (PSD) matrix. Therefore the PSD has to be converted into an approximation of the number of stress cycles with respective stress amplitudes. See section 2.2 for more information on PSD.
2. Theory

One way to do this is by using Dirlik’s formula [2]:

\[ N(S) = E[P] \cdot T \cdot p(S) \]  \hspace{1cm} (2.3)

where \( N(S) \) is the density function between the number of cycles and stress range, \( S \). \( E[P] \) is the expected number of peaks per unit time and \( p(S) \) is a density function of \( S \). The initial step is to calculate the first, second, third and fourth moments of area of the PSD function. The \( n \)th moment of area is obtained by:

\[ m_n = \int_0^\infty f^n \cdot G(f) df \]  \hspace{1cm} (2.4)

where \( G(f) \) is the value of the PSD at frequency \( f \). From here, the expected number of peaks per unit time, \( E[P] \), is obtained by [2]

\[ E[P] = \sqrt{\frac{m_4}{m_2}} \]  \hspace{1cm} (2.5)

and

\[ p(S) = \frac{D_1 e^{-Z/Q} + D_2 Z e^{-Z^2/2R^2}}{2 \cdot \sqrt{m_0}} + D_3 Z e^{-Z^2/2} \]  \hspace{1cm} (2.6)

With the mean frequency \( x_m \) and the normalized amplitude \( Z \) defined as

\[ x_m = \frac{m_1}{m_0} \sqrt{\frac{m_2}{m_4}} \]  \hspace{1cm} (2.7)

\[ Z = \frac{S}{2 \cdot \sqrt{m_0}} \]  \hspace{1cm} (2.8)

The coefficients used in Equation (2.6) are defined in [2] as:

\[ \gamma = \frac{m_2}{\sqrt{m_0 m_4}} \]  \hspace{1cm} (2.9)

\[ D_1 = \frac{2(x_m - \gamma^2)}{1 + \gamma^2} \]  \hspace{1cm} (2.10)

\[ D_2 = \frac{1 - \gamma - D_1 + D_1^2}{1 - R} \]  \hspace{1cm} (2.11)

\[ D_3 = 1 - D_1 - D_2 \]  \hspace{1cm} (2.12)

\[ Q = \frac{1.25(\gamma - D_3 - D_2 R)}{D_1} \]  \hspace{1cm} (2.13)

\[ R = \frac{\gamma - x_m - D_1^2}{1 - \gamma - D_1 - D_1^2} \]  \hspace{1cm} (2.14)

2.1.3 Multiaxial fatigue in frequency domain

In order to estimate fatigue damage for a multiaxial state of stress in frequency domain, it is convenient to reduce the problem into a simple equivalent uniaxial stress state. The intention is to convert a general multiaxial stress problem with 6
stress components, $\sigma_{ij}$, into an equivalent stress scalar $\sigma_{eq}$ before applying Dirlik’s formula. In this project the von Mises stress was used, defined as

$$\sigma_{eq}^2 = \frac{1}{2} (\sigma_{11} - \sigma_{22})^2 + \frac{1}{2} (\sigma_{22} - \sigma_{33})^2 + \frac{1}{2} (\sigma_{11} - \sigma_{33})^2 + 3(\sigma_{12}^2 + \sigma_{13}^2 + \sigma_{23}^2)$$

$$= s^T \cdot Q \cdot s = \text{trace}\{Q \cdot ss^T\}$$

(2.15)

where

$$s = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} & \sigma_{22} & \sigma_{23} & \sigma_{33} \end{bmatrix}^T,$$

$$Q = \begin{array}{cccccc} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 3 & 0 & 0 & 0 & 0 \\ 0 & 0 & 3 & 0 & 0 & 0 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 3 & 0 \\ -1 & 0 & 0 & -1 & 0 & 1 \end{array}$$

(2.16)

By assuming that the mean value of the stress components are zero and by using the fact that the trace operator is linear, it can be shown from Equation (2.15) that the PSD of the equivalent von Mises stress $G_{eq}(f)$ is given as

$$G_{eq}(f) = \text{trace}\{Q \cdot G_{ss}(f)\}$$

(2.17)

where $G_{ss}(f)$ is the PSD of the matrix $ss^T$ [3].

### 2.2 Power spectral density

In order to transform from the time domain to the frequency domain, a PSD is used. Halfpenny shows in [4] that a PSD is a statistical way of representing the amplitude content of a signal. The first step is to do a fast Fourier transform of the time signal. This is done by using Equation 2.18

$$y(f_n) = \frac{2T}{N} \sum_k y(t_k)e^{-i\frac{2\pi f_n k}{T}}$$

(2.18)

where $T$ is the period time, $t_k$ is the discrete time, $y(t_k)$ is the time history input, $N$ is the number of data points in the history signal and $f_n = \frac{n}{T}$ represents the discrete frequency [4]. $y(f_n)$ is now the signal in the frequency domain and the PSD is defined as

$$G(f) = \frac{1}{2T} y(f) \cdot y^*(f) = \frac{1}{2T} |y(f)|^2$$

(2.19)

The area under the PSD curve represents the mean square amplitude of the signal.

### 2.3 Transfer function

For a linear system in the frequency domain, one can estimate the relation between input load $x(f)$ and output response $y(f)$ by using a transfer function $h(f)$ as follows:

$$y(f) = h(f) \cdot x(f)$$

(2.20)
In this study, the estimation of the stress response, \(s_{ij}(f)\), is required for given evaluation points on the model when applying an uniaxial acceleration load \(a(f)\). In general, there are 6 stress components, each one corresponding to a transfer function. Hence, the relation between the stress vector \(s(f)\) and the acceleration load \(a(f)\) is given as

\[
s(f) = h(f) \cdot a(f) \quad (2.21)
\]

where \(h(f)\) is the transfer function vector. Furthermore, in order to obtain the PSD matrix \(G_{ss}(f)\) Equation (2.17), the definition of the PSD, according to Equation (2.19), can be expressed as follows:

\[
G_{ss}(f) = \frac{1}{2T} s(f) \cdot s^{T}(f) = \frac{1}{2T} (h(f) \cdot a(f)) \cdot (h^{T}(f) \cdot a^{*}(f))
\]

\[
= h(f) \cdot h^{T}(f) \cdot \frac{1}{2T} |a(f)|^{2} = H(f) \cdot G_{a}(f) \quad (2.22)
\]

where \(H(f) = h(f) \cdot h^{*T}(f)\) is the transfer function between the PSD of acceleration load \(G_{a}(f)\) and \(G_{ss}(f)\).

It follows directly from the definition of \(H(f)\) that the diagonal components are always real positive values while the off-diagonal components are in general complex valued. This can be verified by rewriting \(H(f)\) in index notation; \(H_{ij} = h_{i}h_{j}^{*}\), where \(i = j\) gives \(H_{ii} = h_{i}h_{i}^{*} = |h_{i}|^{2}\). The complex off-diagonal components represent a phase difference between the stress components caused by propagation delays in the material.
In this section the methodology of the project is presented. The work is divided into several subtasks which are presented in Figure 3.1. The block diagram represents how the tasks are connected to one another. The left side of the diagram shows the work flow of the transient solution. This is a well tested method and will be used as a reference solution to evaluate the new frequency domain method. The work flow regarding the frequency domain method is represented in the right side of the block diagram.

**Figure 3.1:** Block diagram describing the workflow.
3. Method

3.1 Load description

As mentioned in the background and the problem statement, see section 1, the load applied on the component is caused by random vibrations of the base structure (chassis frame of the truck). The random vibrations are approximated by a so-called white noise which is characterized by a random, normal Gaussian distribution with zero mean. In this study, the amplitude of the white noise represents acceleration (m/s²) and the signal is sampled in time with a sampling frequency of 400 Hz. The standard deviation is chosen as 1/2. The load is then scaled by a factor in order to describe a higher acceleration amplitude. With a scaling factor of 100 m/s², a sampling frequency of 400 Hz and a time interval of \( T = 60 \) s, the white noise in the time domain is presented in the upper plot in Figure 3.2.

Since the white noise signal has an equal intensity in the whole frequency range, the true PSD is constant. By using the methodology in section 2.2, the PSD of the white noise is obtained and presented in the lower plot of Figure 3.2. Since the time interval is limited to \( T = 60 \) s, the obtained PSD does not become a perfectly straight line as it should for an infinite time series. One white noise signal is generated and thereafter scaled with two different scaling factors, 50 m/s² and 100 m/s², in order to evaluate the effects of different acceleration amplitudes.

![White noise in time domain and frequency domain](image)

**Figure 3.2:** White noise in time domain and frequency domain

3.2 Stress analysis

In order to carry out fatigue analyses in both the time- and the frequency domain, stress analyses are required in the respective domain. This is done by modal analyses. The underlying method is presented in this section.
3. Method

3.2.1 Numerical model & constraints

The geometry of the component, shown in Figure 1.1 in section 1.1, is provided by Volvo Trucks, and is described by shell elements. The finite elements used are of shell type. Details about the FE-mesh from the commercial software ABAQUS [5] are presented in Table 3.1.

Table 3.1: Mesh details

<table>
<thead>
<tr>
<th>Number of nodes</th>
<th>21 309</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of elements</td>
<td>21 302</td>
</tr>
<tr>
<td>Element types [5]</td>
<td>B31 (2-node linear beam)</td>
</tr>
<tr>
<td></td>
<td>S4 (4-node shell element)</td>
</tr>
<tr>
<td></td>
<td>S3R (3-node shell element with reduced integration)</td>
</tr>
</tbody>
</table>

Local coordinate systems are defined for every shell element, with the directions 1, 2 and 3, according to Figure 3.3. The outputs from ABAQUS will be defined in these local coordinate systems. Because of this, all stress components in 3-direction will be zero, i.e. \( s_{31}, s_{32} \) and \( s_{33} \) equals zero, since there will be stresses only on the surface of a shell element.

![Figure 3.3: Schematic figure of the local coordinate system for a 4 node shell element used in ABAQUS.](image)

The highlighted nodes shown in Figure 3.4 are constraints and further referred to as the BC-area. This is an approximation of reality where the air tank is mounted on the chassis with two bolts in the circled area seen in Figure 3.4. In case of random vibrations, the geometry is in reality vibrating in every direction. Here, the model is only allowed to move in \( y \)-direction, which is a mitigation done in order to simplify the fatigue calculations.

The straps shown in Figure 1.1 are constrained using multi-point rigid beam elements [5], see yellow marks in Figure 3.4. The rigid beam elements are defined by two nodes which have no relative displacement or rotation [5]. The bolts connecting the arm to the air tank is constrained in all degrees of freedom by a kinematic coupling [5].
3. Method

Figure 3.4: Boundary conditions and multi-point constraints in FE-model.

3.2.2 Modal analysis

The modal analysis is performed by ABAQUS using Lanczos algorithm [5]. Eigenvector normalization is performed with displacement normalization [5].

In modal analysis, the BC-area of the model is constrained in all directions. The eigenmodes are estimated for all frequencies between 0 to 200 Hz. The modal analysis results can be found in Table 4.1 in section 4.1. The current model is subjected to acceleration in the global $y$-direction. However, non-zero boundary conditions, such as prescribed displacements, accelerations etc. cannot, in ABAQUS, be applied in the Modal analysis step [5]. Below, in section 3.2.2.1 and 3.2.2.2, the method for forced response on the model is defined for both domains.

3.2.2.1 Frequency domain analysis

After modal analysis, the initial approach in the frequency domain was to carry out a random response base excitation analysis using the white noise PSD as an input (see section 3.1). The idea of this method is to obtain auto- and cross-PSD. The transfer functions are complex and the auto-PSD represents the diagonal elements where the complex number is multiplied with its complex conjugate and thus becomes a real value. The cross-PSD represent off diagonal elements and will be complex numbered. However, after a random response analysis, the root mean square (RMS) values are obtained as results. Thus the information about cross-PSD is lost in ABAQUS, so an alternative method is used.

The alternative approach is to do a steady state dynamic modal analysis using frequency sweep with an acceleration base motion input [5]. By doing a frequency sweep, the response for every frequency in the frequency range is obtained. In this study, the unit amplitude is defined for all frequencies within the range of 0−200 Hz. From this, the stress response components $s_{11}$, $s_{22}$ and $s_{12}$ can be obtained for the defined frequency range.

The stress responses represent transfer functions. By using the concept described in section 2.3, the transfer functions are used to get the corresponding stress PSD output for a given acceleration PSD input according to Equation (2.22).
3. Method

3.2.2.2 Time domain analysis

For the time domain analysis, there are two different approaches for obtaining the stress response, of which one includes to first carry out a dynamic explicit analysis using acceleration time series applied as an input load [5]. This method requires large computational time. The second method, used in this study, is to carry out a modal dynamic analysis after the modal analysis step explained above [5]. The acceleration in $y$-direction is defined as a base acceleration in the modal dynamic analysis step [5]. The resulting stresses obtained are used to estimate the fatigue life of the component as explained in section 2.1.1.

3.2.3 Selection of evaluation points

From the stress analyses, four different elements are selected for further evaluation. An element is evaluated rather than a node, since this gives continuous stresses for all frequencies and time steps. The resulting stresses are determined as an average of the four stresses in Gaussian integration points in the shell element. One of the selected element experience multiaxial state of stress which means that none of the principal stresses are dominating. The other three points are in a state of uniaxial stress, which means that one principal stress is dominating the other. When choosing elements, singularity points, for example constrained points, are avoided since the results here can be unrealistic.

Each element has three local stress components $s_{11}$, $s_{12}$ and $s_{22}$. The principal stresses are then obtain as

$$s_{1,2} = \frac{s_{11} + s_{22}}{2} \pm \sqrt{\left(\frac{s_{11} - s_{22}}{2}\right)^2 + s_{12}^2} \tag{3.1}$$

Only one non-zero principal stress exist for uniaxial stress state. It can be shown from (3.1) that this occur when

- $s_{11}$ is dominant: $s_{22} = s_{12} = 0$,
- $s_{22}$ is dominant: $s_{11} = s_{12} = 0$,
- normal stresses are in phase and equal in size: $s_{11} = s_{22}$, $s_{12} = 0$

3.3 Fatigue analysis

3.3.1 SN-curve

The SN-curve (Wöhler curve) used for fatigue damage calculations in this study is presented in Figure 3.5. It is assumed that the SN-curve is a perfect potential function of the form $\sigma_a = A \cdot N_f^B$, where $N_f$ is the number of stress cycles of range $\sigma_a$ that the material can withstand before failure. For simplicity, stress ranges outside the SN-curve are not accounted for in the fatigue calculation. Stress ranges under 200 MPa will not contribute much to fatigue damage and hence it is reasonable to neglect those stress ranges. For stress ranges over 800 MPa the contribution to fatigue is much larger. However, the SN-curve is not valid if the stress ranges in the
3. Method

given evaluation points exceeds 800 MPa. One reason is that plastic deformations will occur and the presumptions for the SN-curve will no longer be valid.

Figure 3.5: Logarithmic plot of the SN-curve employed in the analysis.

3.3.2 Fatigue analysis in the time domain

The fatigue analyses in the time domain are performed based on a time history stress. The built-in MATLAB function `rainflow` is then used to calculate the number of cycles for different stress ranges. Then to calculate fatigue damage and fatigue life Equations (2.1) and (2.2) are used, respectively.

The rainflow-count is performed on the largest (most dominant) stress component, not on the equivalent von Mises stress as in Dirlik’s formula. This because it’s difficult to apply the rainflow-count algorithm on multiaxial state stresses.

3.3.3 Fatigue analysis in the frequency domain

As mentioned in section 3.2.2.1, the stress responses that represent the transfer functions are obtained from ABAQUS through steady state dynamic modal analysis in the frequency domain. The stress PSD can then be obtained through Equations (2.21) and (2.22). The fatigue analysis in the frequency domain is done using this method to obtain the stress PSD. The stress PSD is applied in Dirlik’s method in order to calculate the number of cycles for a given stress interval, see Equation (2.3). For the estimated number of cycles per given stress interval the damage and fatigue life are obtained by Equation 2.1 and 2.2 respectively.

If the stress state is multiaxial, a PSD for the equivalent von Mises stress, as defined in Equation (2.17), has to be calculated in order to use it in Dirlik’s method.
4

Results and discussion

4.1 Modal Analysis

The table below shows the first seven eigenfrequencies of the model.

Table 4.1: Table of eigenfrequencies of the airtank in Figure 3.4

<table>
<thead>
<tr>
<th>Mode number</th>
<th>Frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15.7</td>
</tr>
<tr>
<td>2</td>
<td>17.3</td>
</tr>
<tr>
<td>3</td>
<td>30.0</td>
</tr>
<tr>
<td>4</td>
<td>40.7</td>
</tr>
<tr>
<td>5</td>
<td>92.2</td>
</tr>
<tr>
<td>6</td>
<td>163.7</td>
</tr>
<tr>
<td>7</td>
<td>186.3</td>
</tr>
</tbody>
</table>

The first eigenfrequency is 15.7 Hz. However, the first eigenfrequency was expected to be higher than 20 Hz based on experience from Volvo Trucks. A study on boundary conditions and constraints, mentioned in section 3.2.1, was carried out to understand the reason for the unexpected frequency values. The results with modified boundary conditions can be found in appendix A.3.

4.2 Evaluation points

The stress analysis results show that the lower eigenmodes only generate high stresses at the arm. High stresses are only observed for the air tank at higher eigenmodes. Since lower modes are more common, only the arm is of interest in the current study. Figure 4.1 highlights the considered evaluation points.

The stress components $s_{11}$, $s_{22}$ and $s_{12}$ are considered. Stress gradients are expected in the highlighted area in Figure 4.1 because of the fillets. The stresses at evaluation points are evaluated in the modal analysis step as explained in section 3.2.2. The highlighted points 1, 2, and 4 are identified by uniaxial state of stress. Point 3 is identified as a multiaxially loaded point.
4. Results and discussion

Figure 4.1: The four evaluation points considered in the analysis

The selection of uniaxial and multiaxial evaluation points are explained below. Point 2 in Figure 4.1 responds to the first mode of natural vibration and shows high stresses at 15.7 Hz in the $s_{11}$ direction as illustrated in Figure A.5 which corresponds to the first eigenvalue in the Table 4.1. Thus, point 2 is considered as the uniaxial evaluation stress point in the first mode as other stress components are neglected. Similarly, for point 3, the stress components $s_{11}$ and $s_{22}$ dominates, see Figure A.6, and thus it is a multiaxial point. The stress values are evaluated for the respective element, see section 3.2.3. The output for stress evaluation in chosen points are generated by ABAQUS and used as an input to fatigue analysis in MATLAB.

4.3 Fatigue damage

The accumulation damage and fatigue life are presented in Tables 4.2 and 4.3 in considered evaluation points 1, 2, 3 and 4 when subjecting an acceleration load of size (scaling factor) 100 m/s$^2$ and 50 m/s$^2$, respectively. The Dirlik’s formula is compared with the traditional rainflow-count algorithm in the time domain in order to validate the results in frequency domain. As can be seen, the accumulation damage rate between the two methods are close to 1 in Table 4.2. The estimated fatigue life varies between 1 h at evaluation point 2 and 7 h in evaluation point 3. Evaluation points close to regions with high modal stress concentration will exhibit larger fatigue damage and therefore shorter life time. From this model the fatigue failure is most likely to occur first in the evaluation point 2 before the others. This is of course a rough estimation since surface roughness and other effects have not been taken into account.

For an acceleration load of 50 m/s$^2$, the fatigue life is getting considerably longer as shown in Table 4.3. This is due to lower stresses induced in the material. The rate between Dirlik and rainflow-count are in this case much larger compared to Table
4. Results and discussion

4.2. The reason for this is probably the short length of the time history \((T = 60\text{ s})\) which will give a significant instability in the results. Further, the number of cycle counts inside the SN-curve range are reduced which will contribute to even more instability (the SN-curve is a potential function and small stress range errors can easily result in even larger damage errors).

The length of the time history is also a problem in frequency domain since the PSD, which in theory should be constant for white noise, is not really a straight line (see Figure 3.2). This gives an instability since the PSD of the white noise will look slightly different when the noise is generated. The best would be to have a much longer sampling history (infinitely long in theory), but the problem occurs when running the transient analysis in ABAQUS, since it would require much heavier computer simulations.

The acceleration load of scaling factor \(100\text{ m/s}^2\) turned out to be an unrealistic load since the estimated fatigue life is only a couple of hours. By reducing the load size one can approach a much more realistic vibration load of the chassis frame. Other scaling factors were not considered, since the aim of the particular project was to validate fatigue damage in frequency domain using Dirlik’s method.

The multiaxial stress state in evaluation points 3 are dominated by the local stress components \(s_{11}\) and \(s_{22}\) (see Appendix A.2). The stresses are in phase with zero mean which makes it possible to apply von Mises equivalent stress since the peaks of the equivalent stress occur at the same intervals of time as for \(s_{11}\) and \(s_{22}\), but with increased amplitude. However, the rainflow-count algorithm is hard to implement for multiaxial stress state and hence it was only used for the largest stress component, which of course effects the results in the time domain.

<table>
<thead>
<tr>
<th>Evaluation point</th>
<th>(D_{tot}), Dirlik</th>
<th>(D_{tot}), Rainflow</th>
<th>(T_{life}), Dirlik</th>
<th>(T_{life}), Rainflow</th>
<th>Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, uniaxial</td>
<td>(5.88 \times 10^{-4})</td>
<td>(5.40 \times 10^{-3})</td>
<td>(2.83\text{ h})</td>
<td>(3.09\text{ h})</td>
<td>0.92</td>
</tr>
<tr>
<td>2, uniaxial</td>
<td>(1.47 \times 10^{-2})</td>
<td>(1.51 \times 10^{-2})</td>
<td>(1.13\text{ h})</td>
<td>(1.11\text{ h})</td>
<td>1.03</td>
</tr>
<tr>
<td>3, multiaxial</td>
<td>(2.17 \times 10^{-3})</td>
<td>(2.57 \times 10^{-3})</td>
<td>(7.69\text{ h})</td>
<td>(6.48\text{ h})</td>
<td>1.19</td>
</tr>
<tr>
<td>4, uniaxial</td>
<td>(2.65 \times 10^{-3})</td>
<td>(2.78 \times 10^{-3})</td>
<td>(6.28\text{ h})</td>
<td>(5.99\text{ h})</td>
<td>1.05</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Evaluation point</th>
<th>(D_{tot}), Dirlik</th>
<th>(D_{tot}), Rainflow</th>
<th>(T_{life}), Dirlik</th>
<th>(T_{life}), Rainflow</th>
<th>Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1, uniaxial</td>
<td>(3.90 \times 10^{-4})</td>
<td>(9.04 \times 10^{-4})</td>
<td>(42.73\text{ h})</td>
<td>(18.43\text{ h})</td>
<td>2.32</td>
</tr>
<tr>
<td>2, uniaxial</td>
<td>(1.50 \times 10^{-3})</td>
<td>(4.49 \times 10^{-2})</td>
<td>(11.11\text{ h})</td>
<td>(3.71\text{ h})</td>
<td>2.99</td>
</tr>
<tr>
<td>3, multiaxial</td>
<td>(7.06 \times 10^{-5})</td>
<td>(3.24 \times 10^{-4})</td>
<td>(236.06\text{ h})</td>
<td>(51.40\text{ h})</td>
<td>4.59</td>
</tr>
<tr>
<td>4, uniaxial</td>
<td>(1.13 \times 10^{-4})</td>
<td>(2.85 \times 10^{-4})</td>
<td>(147.04\text{ h})</td>
<td>(58.39\text{ h})</td>
<td>2.52</td>
</tr>
</tbody>
</table>

In order to visualize the cycle counts for different stress ranges in frequency and
4. Results and discussion

time domain, two histogram plots are presented in Figure 4.2. The stress interval is limited by the SN-curve, i.e. from 200 MPa to 800 MPa. The histogram from Dirlik’s formula (left) is smooth since it’s derived from the continuous density function in equation (2.3), while the histogram from rainflow-count (right) is more disorganized because of discrete cycle counting. The histograms share some over viewing similarities but at some local intervals the difference is quite large. Again, the sampling length of the time history is playing an important role. Longer time history is preferred so that the rainflow-count histogram becomes smoother thanks to the increased number of cycle counts.

![Histograms representing the number of cycles for different stress ranges in evaluation point 1](image)

**Figure 4.2:** Histograms representing the number of cycles for different stress ranges in evaluation point 1, where the left figure is obtained from Dirlik’s formula and the right one from rainflow-count algorithm. The time history length is $T = 60$ s and scaling factor $100 \text{ m/s}^2$.

The fatigue life corresponding to different acceleration loads is shown in Figure 4.3. Low acceleration induces low stresses and thus results in high fatigue life. Figure A.4, Figure A.5, Figure A.6 and Figure A.7 correspond to stresses in evaluation points, see section 4.1. The stress component $s_{11}$ can be identified as the highest stress. From Figure A.8, the evaluation points 3 and 4 have similar stress values. The corresponding fatigue lives in evaluation points 3 and 4, shown in Figure 4.3, are the same which justifies the stress results. In similar lines, the stresses as well as the fatigue life corresponding to evaluation points 1 and 2 shown in Figure 4.3 can be explained.
4. Results and discussion

Figure 4.3: The fatigue life $T_{\text{life}}$ evaluated from Dirlik’s formula versus the scaling factor of the load.

In Figure 4.4, the stress PSD corresponding to different frequency values is plotted. The excitation of the arm (see Figure 1.1) at lower modes causes local stress gradients as shown in Figure A.1. This results in higher stresses at low frequency values. The higher eigenmodes correspond to air tank, see Figure 1.1, and thus create relatively low stresses in the arm. From above discussion it can be inferred that the fatigue failure is expected to occur at the arm in lower eigenmodes.

Figure 4.4: PSD of the equivalent von Mises stress in evaluation point 2.
5

Conclusions and future work

As expected, the frequency domain analyses are more efficient in the sense that they do not require large computational data in form of time history inputs and outputs as the time domain analyses do. Also, in order to carry out a time-domain analysis, new FE-analyses are required for every new input signal. By carrying out the analyses in the frequency domain, transfer functions are obtained for all frequencies. By using the transfer functions, for example in MATLAB, stresses can be obtained by multiplying them with the input PSD signal. This means that only one FE-analysis is required.

The results show that the frequency domain fatigue analysis is useful in early phase development, because of it’s effectiveness. Although, it might not be considered a good idea to use it in final evaluations, since it’s not as accurate as the time domain analysis. One should remember that the results in this study are based on only one component geometry, a very high acceleration amplitude and only four different evaluation points. Therefore, other configurations should be tested in order to validate the results.

The time history used in this project was shortened to only 60 seconds in order to save time and computer power when running the FE-simulations in time domain. Short time history results in a less accurate fatigue life. Hence it’s preferable to use much longer time history even though it would require more computer power.
Bibliography

Appendix

A.1 Evaluation points

The evaluation points considered in the analysis are highlighted below.

Figure A.1: Uniaxial evaluation points in frequency domain

Figure A.2: Multiaxial evaluation point 3 and uniaxial evaluation point 4 in frequency domain
Figure A.3: Evaluation point 2 at 0.0625 sec in time domain

In Figure A.1, the evaluation points 1 and 2 demonstrate stress gradients corresponding to second and first eigenmode respectively. Point 2 develops stresses in first mode whereas point 1 develops stresses in the second mode. In Figure A.2, the stresses in the evaluation point 3 and 4 corresponding to the third eigenmode are shown. In Figure A.3, the stress corresponding to evaluation point 2 subjected to the 100 times scaled acceleration PSD (see section 4.3) at 0.0625 seconds are shown.

A.2 Stress evolutions in time and frequency domains

The charts below show stress evolutions $s_{11}$, $s_{12}$, $s_{22}$ and $s_{33}$ against time and frequency. The element stresses shown below correspond to section point 1, bottom face of a shell element [5], at integration point 1. It is evident that Figure A.4, Figure A.5 and Figure A.7 correspond to a state of uniaxial stress, whereas Figure A.6 corresponds to multiaxial state of stress. In Figure A.8, the comparison of $s_{11}$ which are indicated in legends to Figure A.4 to Figure A.7 are shown.

The information on element number corresponding to different evaluation points is given below. Element 54396 represents evaluation point 1, element 54288 represents evaluation point 2, element 740 represents evaluation point 3 and element 57841 represents evaluation point 4. Below charts are plotted as stress [MPa] vs frequency [Hz] as the left side image and stress [MPa] vs time [sec] as the right image respectively. As described in section 4.2, the arm stresses at lower modes are high. Hence the frequency range of 0 to 60 Hz is shown in the plots below. Also, the interest is to study if the stresses in different directions are in phase or out of phase at a given time; hence a small interval of 2 seconds in the time domain is plotted below.
Figure A.4: Stresses in evaluation point 1 in frequency and time domain respectively

Figure A.5: Stresses in evaluation point 2 in frequency and time domain respectively

Figure A.6: Stresses in evaluation point 3 in frequency and time domain respectively

Figure A.7: Stresses in evaluation point 4 in frequency and time domain respectively
A.3 Effect of constraints on frequency

The frequency results for the model with modified boundary conditions in Figure A.9 are tabulated below. All red-marked nodes are constrained by encastre [5] boundary conditions in straps as discussed in section 3.2.1 and shown in Figure A.9.

<table>
<thead>
<tr>
<th>Mode number</th>
<th>Frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>26.5</td>
</tr>
<tr>
<td>2</td>
<td>63.6</td>
</tr>
<tr>
<td>3</td>
<td>94.9</td>
</tr>
<tr>
<td>4</td>
<td>139.0</td>
</tr>
<tr>
<td>5</td>
<td>150.1</td>
</tr>
<tr>
<td>6</td>
<td>198.2</td>
</tr>
</tbody>
</table>

Table A.1: Eigenfrequencies for the model with modified boundary conditions