THE PENNSYLVANIA STATE UNIVERSITY
SCHREYER HONORS COLLEGE

DEPARTMENT OF ENGINEERING SCIENCE AND MECHANICS

DESIGN OF HIGH CYCLE BIAXIAL FATIGUE SAMPLE NEAR RESONANCE USING FINITE ELEMENT MODELING

MATTHEW A. SORNA

SPRING 2017

A thesis submitted in partial fulfillment of the requirements for a baccalaureate degree in Engineering Science with honors in Engineering Science

Reviewed and approved* by the following:

Manton Guers
Research Associate, Applied Research Laboratory
Thesis Supervisor

Clifford Lissenden
Professor of Engineering Science and Mechanics
Honors Advisor

Judith A. Todd
Department Head
P.B. Breneman Chair and Professor of Engineering Science and Mechanics

*Signatures are on file in the Schreyer Honors College and Engineering Science and Mechanics Office.
Abstract

Fatigue, the gradual accumulation of damage in a material due to microstructural damage and subsequent crack growth, is prevalent among almost all components who sustain prolonged periods of cyclic loading. The number of loading cycles until failure is known as the fatigue life of a material, and is a function of the applied cyclic stress history. For certain materials and operating conditions, it is common to assume infinite fatigue life for stress amplitude levels below a certain value, known as the endurance limit.

In this work, rectangular fatigue specimens were designed and simulated using finite element analysis software. Both free-free and fixed-free boundary conditions were studied to achieve a near resonance response assuming a commercially available shaker as the source of harmonic excitation. Eigenfrequency studies were used to determine resonant frequencies and time domain studies were used to simulate loading cycle histories. In the time domain analysis, special consideration was required for defining boundary conditions for valid finite element solutions. The results showed that nearly constant stress amplitude cycles were obtainable at the target stress of 140 MPa. By manipulating the free parameters of the shaker, a wide range of relatively low stress amplitudes can be generated to fully explore the high cycle ($> 10^8$ cycles) fatigue regime. Future work can be done to validate the simulated results, as well as design the appropriate mounting fixtures used to deliver the two boundary conditions used in the simulation. The proposed technique would allow measurement of high cycle fatigue properties, which can inform design choices and reduced costs associated with system failure and costs incurred through system downtime.
Table of Contents

List of Figures v

Acknowledgements vii

1 Introduction 1

1.1 Project Statement ...................................................... 2

2 Theory 3

2.1 Fatigue ................................................................. 3

2.1.1 Introduction ......................................................... 3

2.1.2 Crack Propagation .................................................... 3

2.1.3 Fatigue Life ........................................................... 7

2.1.4 Damage Accumulation ............................................... 8

2.1.5 High Cycle Fatigue .................................................. 9

2.2 Vibration ............................................................. 9

2.2.1 Natural Frequency ..................................................... 10

2.2.2 Forced Vibration and Resonance .................................. 11

2.2.3 Multiple Degrees of Freedom and Mode Shapes ................. 14

2.3 Finite Element Method ................................................ 17

2.3.1 Discretization ......................................................... 18

2.3.2 Building the Global Stiffness Matrix ............................ 18
2.3.3 Displacement, Strain, and Stress Relationship .............................................. 20

3 Methodology ................................................................. 23

3.1 Resources ............................................................... 23

3.1.1 TA250-S062-PB Shaker ....................................................... 23

3.1.2 COMSOL Multiphysics and ICS-ACI Services ................................. 23

3.1.3 MATLAB ............................................................... 24

3.2 Design ................................................................. 24

3.2.1 Defining the Geometry ..................................................... 24

3.2.2 Material Selection ......................................................... 26

3.2.3 Building the Model ......................................................... 26

3.2.4 Applying Boundary Conditions ........................................... 28

3.2.5 Adjusting the Time Step .................................................... 30

4 Results ................................................................. 34

4.1 Time Domain Analysis ..................................................... 34

4.1.1 Initial Results and Model Revision ........................................ 34

4.2 Stress Cycle Generation .................................................... 36

4.2.1 Free-Free Boundary Conditions ........................................ 36

4.2.2 Fixed-Free Boundary Conditions ....................................... 37

5 Conclusion ............................................................... 41

5.1 Specimen Design .......................................................... 41

5.1.1 Impact ............................................................... 41
5.2 Future Work ......................................................... 42

5.2.1 Experimental Test Design ................................. 42

5.2.2 Multiaxial Fatigue ............................................. 43

Bibliography ............................................................ 45
List of Figures

2.1 Crack length growth as a function of the number of cycles .......................... 6
2.2 Three distinct regions of crack growth propagation ................................. 6
2.3 S-N curve of a brittle aluminum sample, Al 6351-T6 ................................. 7
2.4 Periodic stresses simplified into "block" stress histories ............................ 8
2.5 Diagram of low and high cycle fatigue regimes ....................................... 10
2.6 Example of a mass-spring system .......................................................... 12
2.7 Example of a mass-spring-damper system with sinusoidal forcing function ... 12
2.8 Amplitude response as a function of frequency ratio ................................. 14
2.9 Example of a multiple degree of freedom system .................................... 15
2.10 Tapered beam with an applied point load ............................................. 22
2.11 Discretized model of a tapered beam with an applied point load ............... 22
3.1 S-N curves for Al 6061/T6, at various stress ratios .................................. 27
3.2 Geometry of finite element model, free-free boundary conditions ............... 28
3.3 Discretized component model, with user defined mesh parameters .............. 28
3.4 Modified component geometry for fixed-free boundary conditions .............. 30
3.5 Conceptual diagram for fixed-free boundary conditions utilizing shaker (not to scale) 31
3.6 Normalized displacement plots of the first four natural frequencies of a fixed-free beam .................................................................................................................. 31
3.7 Normalized displacement plots of the first four natural frequencies of a free-free beam ................................................................. 32
3.8 Conceptual diagram for free-free boundary conditions utilizing shaker (not to scale) 32

4.1 Snapshot of transient response stress amplitude plot of free-free beam .............. 35
4.2 Initial stress history of fixed-free fatigue specimen, unstable response ............... 35
4.3 Stress cycle plot of Al 6061-T6 bar, free-free boundary conditions ................... 38
4.4 Stress cycle plot of Al 6061-T6 bar, increasing to constant transition region, free-
free boundary conditions ........................................................................ 39
4.5 Stress cycle plot of Al 6061-T6 bar, fixed-free boundary conditions ................. 40

5.1 Potential asymmetrical fatigue specimen used to test multiaxial fatigue ............. 43
5.2 First five modes of L-shaped multiaxial fatigue specimen ............................... 44
Acknowledgements

Over the course of my three years at Penn State, I owe an incredible amount of thanks to those who have put in tremendous effort to help me succeed, most notably my parents and siblings, my friends both in the Department of Engineering Science and Mechanics and in the Penn State Blue Band, and my professors all throughout the College of Engineering. Specifically I would like to thank both my honors adviser, Dr. Cliff Lissenden, and my thesis adviser, Dr. Matt Guers, who have spent countless Thursday afternoons helping me navigate my project with their invaluable insight. Without their help, I would most likely still be figuring out how to log into my SSH client. Also special thanks to Dr. Wixom of the Applied Research Laboratory whose key advice helped advance along the project out of its most prolonged rut.
1. Introduction

Mechanical components found throughout vehicle support structures, highway infrastructure, and industrial machinery can often endure tens of millions of cyclic stresses throughout their lifetimes. These stresses are generated through the typical operation of a system, and need to be accounted for. If these cyclic stresses are high enough, flaws and defects throughout the body of a material serve as the location of crack initiation. As these cyclic stresses continue to be applied to a material, these minor cracks begin to grow until the remaining intact material is unable to support the applied stresses, and fractures. This process is known as fatigue.

Therefore, it is necessary to understand the nature of the relationship between applied stresses and the number of loading cycles to failure. Knowing the fatigue life of a specific component allows for preemptive maintenance to be done, replacing damaged components before catastrophic failure. This understanding allows for the safety of those operating or utilizing these components to be guaranteed, while also allowing for economic responsibility. Replacing a worn-out component before failure prevents any unnecessary damage to be incurred by the surrounding system, resulting in costly repairs. Additionally, this preemptive replacement reduces the amount of implicit costs incurred by systems from being out of order, affecting overall system efficiency and productivity.

Analyzing the relationship between stress amplitudes and fatigue lives has long led to the assumption that below a certain stress amplitude, certain materials experiences an infinite fatigue life, known as an endurance limit. This assumption was partially influenced by the fact that fatigue testing, since its inception in the late 19th century, required sophisticated machinery or an incredibly long time to reach appreciably high cycle counts. However, with innovations in material testing, this claim has been called into question [1]. For example, secondary fatigue mechanisms have been observed above $10^8$ [2]. In order to explore this area of high cycle fatigue, sophisticated test rigs are typically required to deliver such high intensity, high frequency forces necessary to generate appropriate stress levels in a material. In an effort to better understand this phenomenon
of a secondary fatigue mechanism, existing test machinery can be creatively re-purposed to gen-
erate these cyclic stresses in a material at a high enough frequency to reach these necessary high
cycle counts.

1.1 Project Statement

The goal of this project was to use finite element analysis software to design a biaxial fatigue
specimen, that upon cyclic excitation, generates stress amplitudes at a desired stress level. The
cyclic excitation of the component geometry was based on operating parameters of a commercially-
available shaker, thus re-purposing existing test equipment to aid in the characterization of a still
relatively unknown region of material response.
2. Theory

2.1 Fatigue

2.1.1 Introduction

Fatigue is the continual accumulation of microstructural damage, that leads to eventual failure of components subjected to stresses less than the tensile strength of the material [3]. Fatigue is first initiated at some type of flaw or defect in a material. This could be an external surface flaw, such as some sort of accidentally created mark on the surface of the component during the handling or operation of a component. Additionally, the flaw could be an inclusion in the bulk material that had arisen due to the introduction of impurities in the fabrication process. In either case, the presence of this stress concentrator creates a localized area of high stress within a material. Other kinds of stress concentrators include notches, holes, filets, corners and even stamped lettering on the surface of metallic components. In metals, unit cells of atoms begin to move along favored slip planes as the applied tensile stresses is resolved into shear stresses, leading to net microstructural damage in these localized area of high stress [4].

2.1.2 Crack Propagation

When an object with an initial flaw is subjected to cyclic loading perpendicular to the plane of the crack, the crack will begin to propagate in Mode I. In Mode I of fracture mechanics, the crack can be thought of as being ripped apart through the applied tensile stresses. These cyclic stresses can be fully reversed, meaning that the amplitude of the tensile stress applied is exactly equal to the amplitude of the compressive stresses. The stresses applied could also consist of a mean value at which the stresses oscillate, meaning that the component may never experience compressive stress, and only vary from a maximum to a minimum tensile stress centered at some average tensile stress.
Through the use of the Goodman relation given by Equation 2.1, any cyclic loading pattern can be transformed into that of a fully reversed loading [5].

\[
\sigma_{a0} = \sigma_a [1 - \frac{\sigma_m}{\sigma_u}]
\]  

(2.1)

Where \(\sigma_a\) represents the uncentered stress amplitude, \(\sigma_{a0}\) is the newly transformed and fully reversed stress amplitude, \(\sigma_u\) is the ultimate tensile strength of the material, and \(\sigma_m\) is the mean stress of the uncentered stress pattern. Getting stress histories into this fully reversed form allows for both fatigue lives and damage accumulation approximations to be calculated.

The propagation of a crack continues from its initial flaw size, \(a_0\) until it reaches a critical flaw size, which is typically denoted in most fracture mechanics diagrams as \(a_c\). Once this critical flaw size is reached, the material can no longer support the applied stresses and violently fractures. The number of cycles a component endures before fracturing is known as its fatigue life. However, it is of major importance to understand the behavior of the crack propagation prior to fracture.

Figure 4 illustrates the change in crack length (a) vs. the number of cycles (N). It can be seen that this crack growth is exponential in nature. As the crack grows, the stress concentration factor also begins to increase, with the same load being applied to a component with constantly changing geometry. While the size of the crack is increasing over time, so too does the rate at which the crack is growing itself. This can perhaps be better visualized by a plot of the crack growth rate, \(da/dN\), versus the stress intensity factor range, \(\Delta K\).

In Figure 2.2, three distinct regions of crack growth can be seen. In Region I, the crack growth begins incredibly slow until it slowly transitions into Region II, which is a period of a stable increase in the rate of crack growth. Most importantly, however, is the behavior of Region III where the crack growth rate begins to rapidly increase until material fracture [6].

This relationship between the stress intensity factor range and crack growth is known as the Paris-Erdogan law, or more commonly, Paris’ law, which is one of the most popular fatigue crack
growth models in fracture mechanics. Paris’ law, Equation 2.2, allows for the number of cycles until failure to be calculated given only the initial flaw size, critical flaw size, applied stress amplitude and a series of material constants. When assuming a relatively small crack, separation of variables and subsequent integration can be used to find the number of cycles until failure.

Integrating Equation 2.2 to get Equation 2.3, \( m \) and \( C \) are material constants, while \( \Delta \sigma \) is the applied stress amplitude, \( Y \) is a stress multiplication factor dependent on crack geometry, and \( a_i \) and \( a_c \) are the initial and critical crack sizes, respectively.

\[
\frac{da}{dN} = C \Delta K^m \tag{2.2}
\]

\[
\int dN = N_f = \frac{2(a_c^{2-m} - a_i^{2-m})}{(2 - m)C(\Delta \sigma Y \sqrt{\pi})^m} \tag{2.3}
\]
Figure 2.1: Crack length growth as a function of the number of cycles [6]

Figure 2.2: Three distinct regions of crack growth propagation [6]
2.1.3 Fatigue Life

The S-N curve, or Wöhler curve, is one of the most widely used visualizations for the relationship between the applied completely reversed stress amplitude, $S$, and number of cycles until failure, $N$ for a mechanical component. An example of the S-N curve for a brittle aluminum with an ultimate tensile strength of 320 MPa is given by Figure 2.3 [7]. These curves provide crucial information regarding the number of cycles a material can withstand before failure. Despite the fact that this aluminum alloy has an ultimate tensile strength of 320 MPa, failure will occur after only $10^4$ cycles when subjected to a cyclic load of 150 MPa (less than half the ultimate tensile strength).

It is important to realize that in this particular example, the assumption is made that only a single cyclic stress amplitude is applied to the component until failure. If the component was to experience a series of varying stress amplitudes, as shown in Figure 2.4, the methodology for calculating its fatigue life is much different.

*Figure 2.3: S-N curve of a brittle aluminum sample, Al 6351-T6 [7]*
2.1.4 Damage Accumulation

In instances where a series of blocks of loading is applied, the Palmgren-Miner equation can easily calculate the composite fatigue life of a component, given by Equation 2.4 [8]. The Palmgren-Miner equation, or Miner’s rule, is an empirical rule that involves the summation of the fractions of partial damages from each block of loading sustained by a component. The numerator of each summed term, $N_k$, is the number of cycles sustained by a component at a given stress amplitude, while the denominator, $N_{fk}$, is the number of cycles a component would be capable of sustaining at that single amplitude until failure. All of the partial damage terms are summed and failure is typically said to occur if their sum is greater than or equal to 1.

$$\frac{N_1}{N_{f1}} + \frac{N_2}{N_{f2}} + \ldots + \frac{N_k}{N_{fk}} = 1$$  \hspace{1cm} (2.4)

While the use of Miner’s rule is incredibly widespread, there exists a number of important limitations and shortcomings that must be discussed. First and foremost, Miner’s rule neglects to take into account the order in which these blocks of loading are applied. Evidence shows a difference in fatigue life between a component subjected to incredibly high stresses followed by those of considerably lesser magnitudes and the converse. In some cases, incredibly high stresses developed in a component can lead to the formation of residual compressive stresses that effectively
slow crack growth rates. In an effort to combat this shortcoming of the Miner rule, a correction factor, (typically between 0.7 and 2) is used to better account for the order in which stresses occur. Secondly, Miner’s rule is only a widely-adopted approximation of the linear nature of fatigue crack growth. It is quite possible that a fatigue life calculated by the Miner rule could be an extreme over-approximation or under-approximation of the true performance of component, as there is inherent uncertainty in the fatigue characteristics of materials. Therefore designers must expect variability in material performance when using Miner’s rule to make an approximation of fatigue life [11].

2.1.5 High Cycle Fatigue

As shown in Figure 2.5, a threshold of $10^5$ cycles often serves as a distinction between tests done in the low cycle fatigue (LCF) regime and the high cycle fatigue (HCF) regime. A tertiary region known as ultra-high cycle fatigue (UHCF) is sometimes also defined, although this distinction is inconsistent among researchers. Cycle counts about $10^7$ will be defined simply as high cycle fatigue for this work. As cycle counts increase, a secondary region of linearly decreasing fatigue life may be observed. In an article by Mughrabi, this new understanding of fatigue behavior of metals at increasingly higher cycle counts is discussed. Mughrabi also provides insight into what is believed to be the sources of discrepancy between the low and high cycle fatigue regimes [2].

Advanced test rigs can apply cycles at much fast rates than before, shortening the amount of time necessary to run fatigue tests out in to the region past $10^8$ cycles that was once impractical to achieve. Nowadays, fatigue testing can be run well into counts of more than $10^{10}$ cycles, to reveal secondary mechanisms.

2.2 Vibration

Vibration is a mechanical phenomenon describing the motion of a particle or body which oscillates about a point of equilibrium. As an example, the driving force behind the motion of strings on a
The guitar is due to vibration. Similarly, vibrations can travel through solid mechanical structures such as building frames, bridges, car axles, and airplane wings. It is of great importance to understand the manner in which these solid structures behave when subjected to vibrations in the form of periodic loading [12].

2.2.1 Natural Frequency

One of the simplest ways to understand vibration within a solid is to construct a mass-spring system. This mechanical system is a macroscopic representation of the interactions between atomic particles in a solid structure. With only two parameters to control, spring stiffness, $k$, and particle mass, $m$, relationships between the two can be readily discovered and utilized to discover the vibratory response of a system. In addition to these two parameters, the number of degrees of freedom of a given system is important. The number of degrees of freedom a system has can be defined as the number of independent coordinates needed to fully describe the motion of its particles. Consider for example, a simple mass-spring system consisting of a single mass, $m$, connected to fixed walls on either side by two springs of stiffness, $k$ in Figure 2.6. It can be
assumed that the mass in this example is constrained to move only in the x-direction.

In this example, the system has one degree of freedom. The mass, \( m \), can be fully defined in terms of its horizontal displacement, \( x_1 \). Using Newton’s Second Law, the equation of motion for this single particle is found:

\[
m\ddot{x}_1 + 2kx_1 = 0
\]  
(2.5)

From Equation 2.5, the natural frequency of the system is obtained. In a mass-spring system under completely free vibration, the natural frequency is the frequency which a mass oscillates about a system’s equilibrium point, assuming an initial displacement from equilibrium or an initial velocity of the mass.

In this example, the natural frequency of its system, \( \omega_n \), is defined as shown in Equation 2.6.

\[
\omega_n = \sqrt{\frac{2k}{m}}
\]  
(2.6)

This concept of natural frequency is not only to be applied to physically vibrating systems, but all systems that can be described by a second-order differential equation. Natural frequencies are also found in electrical circuits, like in LC and RLC circuits where natural frequency is proportional to the inverse square of the product of a circuit’s inductance and capacitance [13]. More generally, the natural frequency of any second-order differential equation is defined as the square root of the coefficient in front of the zeroth derivative term over the coefficient in front of the second derivative term.

### 2.2.2 Forced Vibration and Resonance

A phenomenon known as forced vibration occurs when the system as described by Figure 2.6, is excited by a sinusoidal force of the form, \( F \cos(\omega t) \) to the mass, \( m \). The addition of a mechanical
Figure 2.6: Example of a mass-spring system

damper, or dashpot to the system causes vibratory energy to be dissipated through frictional losses within the damper. In solid mechanics, most materials have intrinsic damping effects present [14]. In the case of most metals, however, these effects are relatively small. With the inclusion of a damper into the mass-spring-damper system shown in Figure 2.7, parallels between this simple example and the true vibratory response of a solid component can be drawn.

Through a series of trigonometric identities and manipulations, it is possible to derive the response of the mass, $m$, as a function of time, $x_1(t)$, as defined by Equation 2.7.

$$x_1(t) = X e^{\zeta \omega_n t} \cos(\sqrt{1 - \zeta^2} \omega_n t - \phi)$$  \hspace{1cm} (2.7)

In Equation 2.7, $\phi$ is simply a function of the initial conditions of the system, representing

Figure 2.7: Example of a mass-spring-damper system with sinusoidal forcing function
the phase shift of the signal. The value of $\zeta$, or the damping ratio, can be found by dividing the damping constant, $c$, by the quantity $2\sqrt{km}$.

Perhaps the most interesting quantity of Equation 2.7, is $X$, the amplitude of vibration. The amplitude of displacement for the mass, $m$ in Figure 2.7 can be described by Equation 2.8. The amplitude of the mass-spring-damper system is a function of the magnitude of the applied sinusoidal force, $F$, and equivalent system stiffness, $k$, as well as the ratio of the sinusoidal forcing frequency to the natural frequency of the system, $r$.

$$X = \frac{F}{k} \frac{1}{\sqrt{(1 - r^2)^2 + (2\zeta r)^2}}$$  \hspace{1cm} (2.8)

To characterize the response of a system to an applied sinusoidal force, the amplitude response, $X$, of the system can be normalized by dividing Equation 2.8 through by $F/k$, which is known as the static deflection of the system. Static deflection is the distance a given particle would move if a stationary force of magnitude, $F$, was applied to a system of stiffness, $k$. By comparing this value to the maximum displacement amplitude under a sinusoidal forcing function, a frequency response of the system can be obtained, as shown in Figure 2.8.

As the frequency ratio, $r$ approaches 1, or as the forcing frequency approaches the natural frequency of the system, the amplitude ratio begins to grow. The rate at which and to what extent the amplitude grows by is determined by the damping coefficient, $\zeta$. In an undamped system, where $\zeta = 0$, the amplitude ratio rapidly approaches infinity as $r$ approaches 1. When damping is introduced into the system, the amplitude response follows a similar pattern, although the maximum amplitude ratio now reaches only a finite value. This condition is known as resonance [15].

Quite often, designers must perform a frequency response analysis of a mechanical system to discover any resonance conditions. Designers can adjust accordingly to ensure that normal operating levels do not coincide with such conditions, as to keep the system under safe operation. For example, rotating turbines can experience subtle vibrations at normal operating conditions that are
Figure 2.8: Amplitude response as a function of frequency ratio

well within system specifications. However, if the same turbines were to be driven at or near a frequency that coincides with their natural frequency, the amplitude of these once negligible vibrations would become incredibly violent and unpredictable, causing system instability that would most likely lead to damage. This damage could be caused by vibrating components distorting in such a way that they exceed predetermined safety tolerances and result in catastrophic failure. Another possible cause of system failure could be the introduction of higher amplitude stresses into a component that causes non-negligible damage accumulation, unlike normal operating conditions, as described in Section 2.1 [16].

2.2.3 Multiple Degrees of Freedom and Mode Shapes

The number of degrees of freedom in a multiple degree of freedom system corresponds to the number of independent variables needed to fully define the particles in a system. This can be thought of as having several masses connected by a network of springs and dampers, where there is no relationship in position between masses. The other way in which a system can be thought of having more than one degree of freedom is by how many orthogonal directions a particle can
move in. Looking at Figures 2.6 and 2.7, the particles in these systems were constrained to move in only the x-direction.

A system of multiple degrees of freedom can be modeled by a set of numerous, linearly independent second-order differential equations. For example, a two degree of freedom system containing two independent particles of masses $m_1$ and $m_2$, and three springs of stiffness $k_1$, $k_2$, and $k_3$ is modeled by Figure 2.9. Since systems such as these can be described by a set of second-order differential equations, it is often beneficial to write these equations in matrix form.

The most general form of these systems of equations is given by Equation 2.9, where $M$ is the mass matrix of the system, $C$ is the damping matrix and $K$, the stiffness matrix.

The zeroth, first and second derivatives of position are given by the column vectors, $x$, $\dot{x}$ and $\ddot{x}$ respectively. Lastly, the applied forces to each particle in the system are given by $F$, the force column vector. A fully expanded form of these matrices is given by Equation 2.10, assuming a system of i, linearly independent second-order differential equations.

$$M\ddot{x} + C\dot{x} + Kx = F \tag{2.9}$$

![Figure 2.9: Example of a multiple degree of freedom system](image)
\[
\begin{bmatrix}
m_{11} & m_{12} & \cdots \\
m_{21} & m_{22} & \cdots \\
\vdots & \vdots & \ddots \\
m_{i1} & m_{i2} & \cdots 
\end{bmatrix}
\begin{bmatrix}
\ddot{x}_1 \\
\ddot{x}_2 \\
\vdots \\
\ddot{x}_i
\end{bmatrix}
+ \begin{bmatrix}
c_{11} & c_{12} & \cdots \\
c_{21} & c_{22} & \cdots \\
\vdots & \vdots & \ddots \\
c_{i1} & c_{i2} & \cdots 
\end{bmatrix}
\begin{bmatrix}
\dot{x}_1 \\
\dot{x}_2 \\
\vdots \\
\dot{x}_i
\end{bmatrix}
+ \begin{bmatrix}
k_{11} & k_{12} & \cdots \\
k_{21} & k_{22} & \cdots \\
\vdots & \vdots & \ddots \\
k_{i1} & k_{i2} & \cdots 
\end{bmatrix}
\begin{bmatrix}
x_1 \\
x_2 \\
\vdots \\
x_i
\end{bmatrix}
= \begin{bmatrix}
F_1 \\
F_2 \\
\vdots \\
F_i
\end{bmatrix} \tag{2.10}
\]

For the two degree of freedom system shown in Figure 2.9, the governing system of equations is described by Equation 2.11. It should be noted that for simplicity, this system contains only springs, and has no externally applied sinusoidal forces.

\[
\begin{bmatrix}
m_1 & 0 & \cdots \\
0 & m_2 & \cdots \\
\vdots & \vdots & \ddots \\
m_{i1} & \cdots & m_{ij}
\end{bmatrix}
\begin{bmatrix}
\ddot{x}_1 \\
\ddot{x}_2 \\
\vdots \\
\ddot{x}_i
\end{bmatrix}
+ \begin{bmatrix}
k_{11} + k_2 & -k_2 \\
-k_2 & k_2 + k_3
\end{bmatrix}
\begin{bmatrix}
x_1 \\
x_2
\end{bmatrix}
= \begin{bmatrix}
0 \\
0
\end{bmatrix} \tag{2.11}
\]

Once a matrix is constructed from a set of differential equations used to describe the motion of a system, analysis can be done to determine natural frequencies and their corresponding mode shapes.

Through a series of matrix manipulations, the natural frequencies of a system can be readily obtained. By taking the determinant of the quantity \( K - \omega^2 M \), and setting it equal to zero, a set of natural frequencies can be calculated, equal to the number of degrees of freedom of the system. With these natural frequencies, the modal vectors or mode shapes of the system can be obtained. These mode shapes are column vectors that describe the relative displacements of individual particles oscillating at a natural frequency about equilibrium. Through the law of superposition, the composite response of the entire system can be described, analyzed. Further derivations allow for the specific initial conditions necessary to excite a given mode shape to be calculated. Looking at Figure 2.9, it is easiest to visualize the mode shapes.

With this specific system having two degrees of freedom, there are two natural frequencies expected to be found, corresponding to two different mode shapes. The first mode shape can be thought of as the two masses \( m_1 \) and \( m_2 \) moving in tandem, oscillating in the x-direction with
frequency $\omega_{n1}$. The second, more complex mode shape can be visualized as $m_1$ and $m_2$ oscillating in opposite direction with the same frequency, $\omega_{n2}$. There are several sets of initial conditions able to excite these mode shapes. One of the simplest sets of initial conditions could be a displacement of one unit of length to the right to both masses for mode one, and a displacement of one unit of length to the left and right for masses $m_1$ and $m_2$ respectively for mode two [17].

2.3 Finite Element Method

Consider a vertical, tapered beam of length $L$, fixed at the top, with a concentrated load of magnitude $P$ applied at the bottom in the positive $y$-direction as shown in Figure 2.10. Because of this loading, the beam will deform in the $y$-direction.

However, unlike a beam of uniform cross section, this beam of relatively complex geometry does not have a simple analytic relationship to relate displacement as a function of distance along the bar. Such a system is defined as continuous, and has an infinite number of degrees of freedom. From discussions in Section 2.2, a system of infinite degrees of freedom would require an infinite number of differential equations to describe its motion, thus being unsolvable. Instead, the geometry can be simplified into a finite number of segments of simple geometry, known as elements. These elements are connected by points known as nodes. If these elements are of a geometry with an incredibly well-documented response to tensile loading, like a series of smaller rectangular bars, the response of the true continuous geometry can be approximated. This process of breaking up a complex geometry of infinite degrees of freedom into a system of smaller elements with a finite number of degrees of freedom is known as discretization.

If the beam in Figure 2.10 is discretized into a series of four rectangular bars connected by five corresponding nodes, Figure 2.11, the displacement along the bar can be estimated using the Finite Element Method using a series of approximations.
2.3.1 DISCRETIZATION

One approximation of the finite element method in regards to solid mechanics, is that the geometry of the component in question is simplified or reconfigured in order to make useful simplifications in calculation. In the case of Figure 2.11, the four elements used to represent the tapered beam in Figure 2.10 allows for the behavior of these localized rectangular beams to be exploited to estimate overall component behavior.

In order to solve for the displacement of the individual nodes of a finite element model, all forces on the model need to be equivalently applied to its nodes. In the case of Figure 2.11, only the point load P is being considered, applied directly at node 5. However, if the body force from the acceleration due to gravity was to be included, the overall forces on the model would have to be equivalently distributed to all of the five nodes to have the same overall effect on nodal displacement.

The displacement of a series of nodes can be described by assembling a so-called global stiffness matrix, \( K \), used to solve the matrix equation given by Equation 2.12. \( Q \) is a column vector of nodal displacements, while \( F \) is a column vector of the forces applied to their corresponding nodes.

\[
KQ = F
\]

(2.12)

In order to build this global stiffness matrix, \( K \), the stiffness matrix of each individual element must be determined first.

2.3.2 BUILDING THE GLOBAL STIFFNESS MATRIX

Beginning with Element 1 of Figure 2.11, its corresponding nodes are 1 and 2. The corresponding nodes of Element 2 are 2 and 3, and so on. Assuming that Element 1 has a rectangular cross-section
of area, $A_1$, length, $L_1$ and a Modulus of Elasticity, $E$, its elemental stiffness matrix, $k_1$, can be shown as in Equation 2.13.

$$k_1 = \frac{EA_1}{L_1} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$  \hspace{1cm} (2.13)$$

The row and column positions in the elemental stiffness matrix correspond to the position of these individual elements into the larger global stiffness matrix. Following the same procedure for the remaining four elements, a global stiffness matrix for this finite element model can be constructed. It is under the assumption that each element has a different, rectangular cross-sectional area, and length, while having a uniform Modulus of Elasticity. Applying Equation 2.12 to the system defined in Figure 2.11, the following matrix equation is given in Equation 2.14 [18].

$$\begin{bmatrix} \frac{EA_1}{L_1} & -\frac{EA_2}{L_1} & 0 & 0 & 0 \\
-\frac{EA_2}{L_1} & \frac{EA_1}{L_1} + \frac{EA_2}{L_2} & -\frac{EA_2}{L_2} & 0 & 0 \\
0 & -\frac{EA_2}{L_2} & \frac{EA_2}{L_2} + \frac{EA_3}{L_3} & -\frac{EA_3}{L_3} & 0 \\
0 & 0 & -\frac{EA_3}{L_3} & \frac{EA_3}{L_3} + \frac{EA_4}{L_4} & -\frac{EA_4}{L_4} \\
0 & 0 & 0 & -\frac{EA_4}{L_4} & \frac{EA_4}{L_4} \end{bmatrix} \begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \\ Q_5 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ P \end{bmatrix}$$  \hspace{1cm} (2.14)$$

Drawing parallels to the matrix equation developed in Equation 2.11 for a multiple degree of freedom system, it is useful to highlight the symmetry of the stiffness matrices in both cases, $K$, along their diagonals. Also, given the nature of the example, the value of $Q_1$ is already prescribed to be zero.

Once such an equation exists, a number of different methods are available to solve for the vector, $Q$, giving the nodal displacements of the four elements. With these displacements defined, a number of other important quantities can be calculated such as stress and strain, through a series of simple relationships.
2.3.3 Displacement, Strain, and Stress Relationship

Strain is defined as percent change in length. Given the nodal displacements of the five nodes in Figure 2.11, it is possible to write an equation for displacement as a function of \( y, u(y) \). This equation is assumed to be piece-wise linear, given the nature of linear elastic behavior or solids. In this form, strain in the \( y \)-direction, \( \epsilon_y \), can be readily obtained through the one-dimensional displacement-strain relation, shown in Equation 2.15 [19].

\[
\epsilon_y = \frac{du}{dy} \tag{2.15}
\]

Differentiating this piece-wise linear displacement equation yields a constant strain value for each individual element. Using the stress-strain relationship established in Hooke’s Law, shown in Equation 2.16, a uniform tensile stress, \( \sigma_y \), can be obtained for each of the four elements.

\[
\sigma_y = E \epsilon_y \tag{2.16}
\]

Realistically, the stress along the bar is not segmented into four discrete values, highlighting one of the drawbacks of approximation when using the finite element method. Had the number of elements been increased from 4 to 400 rectangular prisms, the stresses in the bar would begin to resemble that of their true, continuous values. As the number of elements in this model increases, the size of the global stiffness matrix, the displacement vector, and the load vector increase as well. A one-dimensional model consisting of 4 elements and 5 nodes required a 5-by-5 global stiffness matrix, consisting of 25 individual entries. Increasing the number of elements in the model to 400 requires the creation of 401 nodes, resulting in a 401-by-401 global stiffness matrix. This larger matrix consists of 160,801 entries, which increases the amount of time necessary to solve for the displacement of the 401 nodes.

Understanding the balance between model resolution and required computational time is an
invaluable skill to possess when utilizing the finite element method, especially as problems begin to increase in complexity. The same process of discretization followed by the construction of a global stiffness matrix is used in both 2 and 3-dimensional finite element problems. Fortunately, the required assembly and resulting computation is most often handled by commercially-available finite element software.

However, it is still in the best interest of those utilizing finite element software to be extremely familiar with the process of discretization and its limitations. Blindly following the results of any computational tool without having an understanding of what your expected results should look like is sure to lead to incorrect conclusions. Throughout the design process outlined in Section 3, proper documentation provided by the finite element software utilized was constantly referenced, making sure that any changes made to the fatigue specimen had a supported and legitimate reason for doing so. Keeping to this standard guaranteed results which were to be trusted.
Figure 2.10: Tapered beam with an applied point load

Figure 2.11: Discretized model of a tapered beam with an applied point load
3. Methodology

3.1 Resources

3.1.1 TA250-S062-PB Shaker

In an effort to ground the theoretical nature of this project in terms of realistic constraints, a specific, commercially-available shaker was used as a reference when assigning maximally allowable values for parameters such as applied frequency and armature displacement. These parameters, along with component geometry dictate the range of possible stresses capable of being generated in a given component.

The machine in question, a TA250-S062-PB model shaker from the Unholtz-Dickie Corporation is currently maintained by the Applied Research Laboratory at Penn State. The shaker has a nominal maximum operating frequency around 3.5 kHz, although the range of applicable frequencies depends on the weight of the component and required mounting equipment. For example, a payload of 8.6 kg can only reach a maximum frequency of 2 kHz. As will be discussed later, any constructed component should be well below this weight threshold, making the value of 2 kHz as its maximum operating frequency a reasonable one. The shaker also has a maximum armature displacement of 38 mm, and a corresponding maximum acceleration of 60 g. The acceleration of the shaker is also a function of the payload weight, decreasing as weight increases [20].

3.1.2 COMSOL Multiphysics and ICS-ACI Services

All finite element modeling was done using the commercially available finite element analysis software, COMSOL Multiphysics. Specifically, the Structural Mechanics module was utilized, taking advantage of the frequency and transient responses of mechanical components [21]. This
software is hosted on Hammer, an interactive, Linux-based GUI environment maintained by the Penn State Institute for CyberScience Advanced CyberInfrastructure, or ICS-ACI.

3.1.3 MATLAB

All post-processing of stress time history results were done using MATLAB. Importing .csv files exported by COMSOL Multiphysics allowed for the plotting of stress histories as well as the identification of key values of generated stress cycles such as maxima, averages and percent variation.

3.2 Design

3.2.1 Defining the Geometry

When designing the fatigue test specimen, it was important to keep in mind the fixed constraints put in place by the nature of the project. Several of these constraints came with the selection of the shaker to be used in possible experimental testing, such as the TA250-S062-PB model shaker table from Unholtz-Dickie Corp as described in Section 3.1.1. Keeping in mind the likely laboratory setting of such testing, and the payload weight limits put in place by the shaker, a component with the dimensions on the order of 10-20 cm in any one direction was deemed reasonable. The selected shaker was also only nominally rated to reach a maximum frequency of roughly 2 kHz. This frequency maximum is also a function of the max amplitude of acceleration of the shaker, so any increase in the applied acceleration of the shaker would most likely require a reduction in the maximum operating frequency.

Looking at standard fatigue life test setups commonly encountered, as in Section 2.1.5, it was a likely assumption that the geometry of the test specimen would end up as a bar, with either a circular or rectangular cross-section. With the maximum length of the bar constrained to be no more than 20 cm as per the initial geometric constraints, the width and height of the component
could then be selected. Knowing that the natural frequencies of the test specimen would be of consider-
considerable importance, preliminary work was done using equations that relate the natural frequency
of a simply supported beam to its geometry, as well as its material properties, as seen in Equation
3.1 [22].

\[ f_n = \frac{K_n}{2\pi} \sqrt{\frac{EIg}{wl^4}} \]  

(3.1)

The natural frequency of a beam, loaded only in the sense of its self-weight can be calculated
using Equation 3.1, where \( E \) is the material’s Modulus of Elasticity, or Young’s Modulus, \( I \) is the
area moment of inertia, \( g \), the acceleration due to gravity, \( w \), the uniform load on the beam, which
in this case is only the force on the bar due to gravity, \( l \), the length of the beam, and \( K_n \), a constant,
where \( n \) relates to the mode of vibration. Different sets of \( K_n \) are calculated depending on the
boundary conditions of the beam. In the case of a simply supported beam, \( K_1 \) is found to be 9.87,
while in a beam where both ends are fixed, \( K_1 \) is slightly higher at 22.4 [23]. At least in the first
mode of vibration for this bar, these constants only vary within and order of magnitude. Several
sets of possible boundary conditions for this component will be explored later in Section 3.2.4.

With such a wide variety of criteria, including material selection, component dimensions, vibra-
tory mode and boundary conditions to choose from, it is important to note that many combinations
of parameters would lead to satisfactory results when attempting to generate high cycle counts for
a given component. In an effort to converge on a solution quickly, several parameters were initially
set. The relationships between these fixed parameters and the remaining free parameters could
then be recognized and adjusted accordingly if need be.

One of the first assumptions made was the shape of the cross-section of the beam. Keeping
in mind the method by which the shaker delivers forces to the beam, it makes sense to have a
cross-section whose geometry favors bending in one direction over the other. This was the initial
justification behind selecting a rectangular cross-section over a circular one. Comparing the area
moment of inertia of a rectangle about the length-wise and width-wise axes, a clear distinction can
be seen, unlike in a solid or hollow cylindrical beam, where the circular area moment of inertia is uniform across any given axis. In this way, the vibration can almost be guided to excite mode shapes in one axis over the other by using a rectangular cross-section, thus simplifying analysis. With cross-section selected, and a length of 20 cm prescribed, a basic geometry can now be built in the environment of finite-element software.

3.2.2 Material Selection

Before using the finite element model to calculate the natural frequencies and mode shapes, material properties must be assigned in the model. These parameters include Young’s Modulus, density and Poisson’s ratio. In order to see a fatigue life in the range of $10^8$ cycles, the material class in consideration was limited to metals. While ceramics also undergo failure due to fatigue via crack growth, their fatigue lives are in general much shorter, and are unable to withstand the same stress magnitude as metals. For proof-of-concept testing, the raw material costs, and fabrication considerations (machinability and cost) must also be made. It is for this reason that Al 6061-T6 was selected as the material to apply to the component domain of the finite element model. Al 6061-T6 has an ultimate tensile strength of 310 MPa at room temperature conditions of 25°C (yet at $10^7$ cycles is only expected to have a fatigue strength of 20 ksi, or roughly 140 MPa, assuming a fully-reversed loading condition, or stress ratio of -1 [24]). An S-N curve for Al 6061-T6 is shown in Figure 3.1. Al 6061-T6 also has a density of 2700 kg/m3, and a Poissons ratio of 0.33. With this new information, further design criteria can be modified accordingly [25].

3.2.3 Building the Model

When first building the model in the COMSOL environment, a nominal width of 2 cm and a height of 1 cm were selected, making the component a 2 x 1 x 20 cm rectangular prism. As stated previously, these dimensions were selected arbitrarily, allowing for a possible solution to be fully defined before geometries and applied forces had to be manipulated to generate appropriate
Figure 3.1: S-N curves for Al 6061/T6, at various stress ratios [24]

stresses. The model geometry was divided into four smaller rectangular prisms as shown in Figure 3.2 to ensure the creation of nodes at specific points of interest.

Also, stress values can be readily probed along the two lines running along the x and y center axes of the component. It is along these lines that the highest stress magnitude is expected to occur; either at their intersection point or x-axis extrema, depending on the boundary conditions of the bar.

When defining the mesh size of a geometry, it is important to balance the accuracy of the model with the time it takes to generate results. A finer mesh will calculate a more accurate answer, until the solution is converged, albeit at the cost of added computational time. Keeping this in mind, several of the parameters of element size were modified accordingly. A discretized model of the component is shown in Figure 3.3. While also providing clarity of discretization around points and edges of interest, the selected mesh element size settings allowed for multiple elements to be constructed through the thickness of the bar. Initial testing provided unreliable results when only
single tetrahedral elements spanned the entire thickness of the bar. In the final discretized model of the beam, elements were designated to be no larger than 0.5 cm, and no smaller than 0.05 cm, allowing for clarity in the model through its thickness.

### 3.2.4 Applying Boundary Conditions

Depending on the method in which this component geometry is fixtured to the shaker, different vibratory responses are expected. Two of the various sets of boundary conditions possible for such a geometry include a fixed-free and free-free case.

The fixed-free boundary condition requires one end of the beam geometry to be fixed, with the other end free to move. Applying these boundary conditions to a bar in a frequency domain
analysis in finite element modeling software produces mode shapes and corresponding natural frequencies consistent with those found using Equation 3.1.

In the time domain analysis of a fixed-free beam, this boundary condition was handled differently. Simply applying a prescribed displacement to the 'fixed' face of the beam produced inaccuracies in stress solutions in local regions of the bar. From simple bending response of a beam, it is known that the region of highest bending stress will occur closest to the fixed end. To improve the compatibility between the fixed boundary and high stress region, the thickness of the specimen was increased at the fixed end. The modified geometry is shown in Figure 3.4. Instead of applying a prescribed displacement to the edge of the bar, this boundary condition can now be applied to the faces of the connecting support. The overall length of the beam section did not change, ensuring the same natural frequencies obtained in the frequency domain. A diagram of a conceptual scenario for the fixed-free case is shown in Figure 3.5, with the test component directly mounted in a right angle to the shaker armature.¹

Normalized displacement plots of the first four natural frequencies of a fixed-free beam are shown in Figure 3.6. The 'in-plane' modes will not be excited by 'out-of-plane' excitation. For the transient analysis of this bar with fixed-free boundary conditions, the first natural frequency of 205 Hz was examined for its simpler response behavior.

When analyzing the free-free boundary conditions of the bar, both the frequency domain and time domain models needed to be considered. In the frequency domain, there are no constraints applied to the bar (all boundaries are free), with the component isolated in free space. The first bending mode of the beam corresponds to a first natural frequency of 1281 Hz. This frequency is well below the roughly 2 kHz maximum operating frequency of the TA250-S062-PB shaker. With the natural frequency falling inside of this accepted range, no further modifications to the geometry of the component need to be done. Instead, the amplitude of force applied to the component can be varied to generate appropriate stress levels capable of displaying high-cycle fatigue behavior.

¹Note that fixturing would need to be studied in greater detail and that other factors may require a different fixturing approach.
Normalized displacement plots of the first four natural frequencies of a free-free beam are shown in Figure 3.7. In the time domain analysis of the free-free case, the TA250-S062-PB shaker can be utilized in such a way that two symmetric pin-like support attachments can be made along either side of the top edges of the bar, parallel to the x-axis, as shown in Figure 3.8. Through the fine-tuned control of the displacement and frequency of the shaker’s movement, ‘applied forces’ can be delivered to both sides of the bar simultaneously. Pin-like supports allow for the slope of deflection at both edges of the bar to be non-zero, providing an approximately free-free excitation setup.\footnote{Note that fixturing would need to be studied in greater detail and that other factors may require a different fixturing approach.}

3.2.5 Adjusting the Time Step

As described in Section 2.3.3, the relationship between mesh quality and computation time is directly proportional. However, in a transient finite element analysis, the time step settings of a given software package is of equal or greater importance when dictating computation time. Knowing that the selected shaker will be operating between 200 - 1300 Hz yields an expected period for gener-
Figure 3.5: Conceptual diagram for fixed-free boundary conditions utilizing shaker (not to scale)

Figure 3.6: Normalized displacement plots of the first four natural frequencies of a fixed-free beam: a) first bending mode (out of plane), b) first bending mode (in plane), c) second bending mode (out of plane), d) second bending mode (in plane)
Figure 3.7: Normalized displacement plots of the first four natural frequencies of a free-free beam: 
a) first bending mode (out of plane), b) first bending mode (in plane), c) second bending mode (out of plane), 
d) first torsional mode

Figure 3.8: Conceptual diagram for free-free boundary conditions utilizing shaker (not to scale)
ated stress cycles to be on the order of milliseconds. Keeping this in mind, a small enough step size is required to ensure proper calculation of stress amplitudes. If the step size for these stress cycles is too coarse, inaccurate solutions are produced. Once this time step is properly adjusted, stress histories such as those found in Section 4 can be generated.

Through the described design process, the number of freely adjustable design parameters has been limited to two, the amplitude and frequency of the applied vibration. Section 4.1 discusses the results for selection of these final two parameters, and explores the transient response of the finite element model in both the free-free and fixed-free cases, determining whether or not such a geometry is capable of generating the desired stress amplitude.
4. Results

4.1 Time Domain Analysis

Before adjustment of the available free parameters, an arbitrary amplitude and frequency of the prescribed displacement were selected. It was then possible to verify the section of the bar experiencing the most significant tensile stresses. A frequency of 1000 Hz was selected, with a shaker displacement of 1 mm. A snapshot of a color-legend plot of stress amplitudes is given in Figure 4.1, with red regions designating stresses of more than 500,000 Pa. The point of highest stress concentrations, and thus the region of likely fatigue failure, was shown to be at the top or bottom face of the beam. With the location of the highest tensile stresses confirmed, a boundary probe could then be assigned to record the x-component stresses at that point.

4.1.1 Initial Results and Model Revision

Before the free parameters of the shaker could be adjusted to produce constant amplitude stress cycles at the target stress level, an initial shaker displacement and operating frequencies were selected. In this case, the component was excited with a sinusoidal prescribed displacement on both top edges in the yz-plane as defined in Figure 4.1 to simulate the free-free boundary conditions discussed in Section 4.2.1.

Expected stress history of this component was near constant stress cycles measured at the center of the fatigue specimen. However, as shown in Figure 4.2, unstable, yet repetitious wave packets are seen. Locally, these wave envelopes grow to a maximum, and then rather unexpectedly decrease back down to zero before repeating. Initially it was thought that this could have been an issue with the time step of the finite element software solver settings. After continued alterations to the time step settings without any changed behavior of the stress history, alternative explanations
Figure 4.1: Snapshot of transient response stress amplitude plot of free-free beam

Figure 4.2: Initial stress history of fixed-free fatigue specimen, unstable response
were explored.

Utilizing the advice of Dr. Wixom of the Applied Research Laboratory, the way in which the prescribed displacement was applied to the model in the transient analysis was altered. As per his advice, the sinusoidal prescribed displacement on both ends of the bar was linearly ramped up in magnitude from zero to its eventually required constant value. This approach also mimics the way in which the shaker slowly ramps up the displacement of its armature upon startup. Linearly ramping up the prescribed displacement to its maximum value over the course of 0.05 ms produced the nearly constant stress amplitudes that were to be expected. A detailed discussion of these results is presented in Section 4.2.

4.2 Stress Cycle Generation

4.2.1 Free-Free Boundary Conditions

Having corrected the boundary condition issues, initial stress cycles were then generated for the free-free bar. Given that the first natural frequency of the free-free bar is roughly 1281 Hz, a prescribed displacement oscillating at a frequency of 1300 Hz was selected. The amplification factor is very large so close to resonance. A decision was made to drive slightly off of resonance to improve stability. The amplification achieved by driving near resonance allows for large stress levels to be obtained in the beam. In order to obtain a stress amplitude of 140 MPa, the shaker displacement has to be finely tuned.

Furthermore, driving at a frequency slightly higher than resonance allows for an improvement in the total amount of time necessary to reach the expected range of $10^8$ cycles. Driving at resonance (1291 Hz), it would take nearly 22 hours to reach a cycle count of $10^8$. Increasing the driving frequency to 1300 Hz cuts down that time by about a half an hour. While this decrease may not seem significant at a count of $10^8$ cycles, the added benefit of slightly increasing the driving frequency beyond resonance is valuable, as very high cycle fatigue testing is done into the gigacycle,
cycles, regime and beyond.

A shaker displacement of 0.9 mm at a frequency of 1300 Hz, results in the stress cycle plot shown in Figure 4.3. A more focused view of the region of transition between linearly increasing and constant stress amplitudes is shown in Figure 4.4. Following the period of 50 ms in which the displacement amplitude is linearly increased to its maximum value, the stress amplitudes reach and then maintain an average value of 137.51 MPa. Although there is a slight variation in amplitude over time, the maximum difference between any two peaks over the course of 500 ms is only 1.24 MPa. This variation amounts to only a 0.9% change in stress amplitude. Some of this variation may be due to discretization, and the numerical rounding inherent to the finite element method.

4.2.2 Fixed-Free Boundary Conditions

Analyzing the stress history generated from the fixed-free boundary conditions yields similar results to the free-free case. With a first bending mode, and first natural frequency of 205 Hz, a driving frequency just off of resonance, 210 Hz was selected. In order to generate stresses near the desired stress level, a corresponding shaker displacement of 0.31 mm was used. Looking at the plot of the fixed-free stress history shown in Figure 4.5, stress amplitude behavior is similar to what is seen in Figure 4.4. The prescribed displacement amplitude again linearly increases until 50 ms, at which point the stress amplitudes would be expected to be nearly constant. However, looking at the stress amplitudes in Figure 4.5, a much larger variation amplitude is apparent than was shown in the free-free case. Stress varies in the fixed-free case by approximately 15.42 MPa, which is roughly 11% of the target of 140 MPa.
Figure 4.3: Stress cycle plot of Al 6061-T6 bar, free-free boundary conditions
Figure 4.4: Stress cycle plot of Al 6061-T6 bar, increasing to constant transition region, free-free boundary conditions
Figure 4.5: Stress cycle plot of Al 6061-T6 bar, fixed-free boundary conditions
5. Conclusion

5.1 Specimen Design

In this work, rectangular fatigue specimens were designed and simulated using finite element analysis software. Both free-free and fixed-free boundary conditions were studied to achieve a near resonance response when utilizing a commercially available shaker as the source of harmonic excitation. Eigenfrequency studies were used to determine resonant frequencies and time domain studies were used to simulate loading cycle histories. In the time domain analysis, special consideration was required for defining boundary conditions for valid finite element solutions. The results showed that nearly constant stress cycles were obtainable. By manipulating the free parameters of the shaker, a wide range of stress amplitudes can be generated to fully explore the high cycle (> $10^8$ cycles) fatigue regime.

5.1.1 Impact

With a clearer understanding of fatigue characterization curves past $10^8$ cycles, more informed design decisions can be made in order to maximize the operating life of a metallic component subjected to high cycle counts. In doing so, systems can be properly designed and maintained to minimize failure and the indirect cost of operation outages. Understanding how a material responds at very high cycle counts allows for designers to push components to their maximum efficiency, ultimately saving money and the time of replacement.

Tangentially, this project shows the impact of components and systems operating near resonance. While in the case of this study the resonance response of a geometry was utilized to help generate appropriately high stress cycles, quite often operation at resonance is to be avoided to decrease the likelihood of damage and overall instability. The impact of resonance operation must
not be ignored, and should always be addressed when analyzing a system which undergoes cyclic stresses.

5.2 Future Work

5.2.1 Experimental Test Design

Due to the restricted time frame of the project, there were specific aspects of the project that were not addressed fully, and others that are recommended to others wishing to advance the work done in the field of high cycle fatigue.

The parameters defined in Section 4.2 yielded a maximum stress amplitudes of 140 MPa. Decreasing the prescribed shaker displacement and adjusting the operating frequency even further off of resonance conditions can yield lower stress cycle amplitudes. Carefully varying these variables in a parametric sweep could allow high cycle behavior to be analyzed. With the use of finite element analysis, preliminary geometries and simulations of various boundary conditions can be constructed to find the parameters necessary to drive fatigue testing of any material.

Once the finite element models of these components of various metals have been evaluated, the task of physical testing of the fatigue specimen would begin. Rudimentary discussions on the ways in which boundary conditions could be handled in a physical laboratory environment were included in this project. However, any future work would have to spend a considerable amount of effort designing the proper support structures. Additionally, some consideration would need to be made into the fabrication of such fatigue specimen. The rectangular shape of these fatigue specimen allow for easy fabrication, and the various boundary conditions can be implemented through the use of various support structures and fixtures.
5.2.2 **Multiaxial Fatigue**

In the preliminary outline of this thesis project, an asymmetrical component was to be modeled using finite element software, shown in Figure 5.1. Due to the asymmetrical nature of this proposed component, the stresses generated from the harmonic excitation of a shaker would be multiaxial in nature. Consider, for example, the first five mode shapes of an L-shaped, cantilever component as shown in Figure 5.2.

Similarly to how the fixed-free cantilever specimen had been attached to the shaker in Section 3.2.4, this L-shaped specimen could be connected to and then excited by this shaker near resonance conditions. Due to the asymmetrical nature of this component, excitation of multiaxial stress states throughout the bar is to be expected. With such a setup, the relationship between multiaxial states of stress and fatigue life could be explored. While work in this field of multiaxial fatigue is not uncommon, the test equipment required is oftentimes complex, consisting most often of a biaxial torsion-tension testing rig. In an effort to provide material testers with a method to rework existing and fairly common machinery to research multiaxial fatigue, this similar process can be utilized.

*Figure 5.1: Potential asymmetrical fatigue specimen used to test multiaxial fatigue*
Figure 5.2: First five modes of L-shaped multiaxial fatigue specimen:
a) first bending mode (out of plane), b) first bending mode (in plane), c) first torsional mode, d) mixed mode,
e) second bending mode

Additionally, it would then be possible to explore the joint interests between the fields of multiaxial fatigue, and high cycle fatigue. As discussed in Section 2.1.5, a possible reason behind a change in fatigue behavior at higher cycle counts may be a change in fracture mechanism. Introducing high cycle, multiaxial fatigue could further explore this question.
Bibliography


Education

Master of Science in Engineering in Mechanical Engineering and Applied Mechanics
The University of Pennsylvania Fall 2017 – Fall 2018 / Philadelphia, PA

Bachelor of Science (Honors) in Engineering Science / Minor in Engineering Mechanics
The Pennsylvania State University May 2017 / University Park, PA
- Engineering Science is a multidisciplinary Honors Program within the College of Engineering
- Specialization in Engineering Mechanics, completing undergraduate degree in three years
- For a full list of completed courses, programming examples, projects: [http://www.personal.psu.edu/mvs6004/](http://www.personal.psu.edu/mvs6004/)

Academic Experience

- **Spring 2017**: Strength Design, Mechanics of Fluids, Finite Elements, Electronic Properties and Applications of Materials

Honors and Awards

- **Schreyer Honors Scholar** - member of Penn State’s Schreyer Honors College
- **Mirna Irquidi-Macdonald and Digby D. Macdonald Student Award in ESM** – scholarship recognizing outstanding academic performance in Engineering Science and Mechanics
- **The President's Freshman Award** - presented annually to undergraduate degree candidates and provisional students who have earned a 4.00 (A) cumulative grade-point average

Research Experience

**Design of High Cycle Biaxial Fatigue Sample Near Resonance Using Finite Element Modeling**
Senior Honors Thesis and Research Project Spring 2016 – 2017
- FE modeling and analysis of biaxial vibrations on a cantilever specimen using Comsol Multiphysics, computational post processing of cyclic fatigue done using MATLAB
- Hoping to pursue future work in physical testing to corroborate computational results

Work Experience

**SerDes Characterization Intern** Summer 2016
Globallfoundries – second largest semiconductor foundry in the world East Fishkill, NY
- Programmed lab equipment with digital signal processing algorithms in an effort to characterize the improvement in the bit error rate of high speed serial data transfer
- Wrote code to interface with test equipment in order to automate lab-based data collection
- Integrated an embedded Linux processor onboard a Beaglebone Black into the next generation of in-house lab equipment, used in testing most recent 14nm ASIC designs from Globalfoundries

**Teaching Assistant** Fall 2015 – Spring 2016
Introduction to Engineering Design – Penn State School of Engineering Design Technology, and Professional Programs University Park, PA
- Teaches the basics of Engineering Design to first year Engineering students, using Solidworks, Sketchup, HTML coding as well as discussions regarding ethics in the Engineering field

Skills

- **Programming**: MATLAB, C, Python, LaTeX, Lua, HTML, Bash
- **Operating systems**: Windows, Linux/Debian
- **Foreign language**: Italian

Activities and Organizations

The Pennsylvania State University Marching Blue Band August 2015 – May 2017