Finite element analysis of laser weld induced thermal strain

Fred W. Warning
Lehigh University

Follow this and additional works at: http://preserve.lehigh.edu/etd

Recommended Citation

This Thesis is brought to you for free and open access by Lehigh Preserve. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Lehigh Preserve. For more information, please contact preserve@lehigh.edu.
AUTHOR: Warning, Fred W., Jr.

TITLE: Finite Element Analysis of Laser Weld Induced Thermal Strain

DATE: October 9, 1994
Finite Element Analysis

of

Laser Weld Induced

Thermal Strain

by

Fred W. Warning Jr.

A Thesis

Presented to the Graduate and Research Committee

of Lehigh University

in Candidacy for the Degree of

Master of Science

in

Mechanical Engineering

Lehigh University

1994
Approval

This thesis is accepted and approved in partial fulfillment of the requirements for the Master of Science.

Sept. 12, 1994
Date

Dr. Arkady S. Voloshin
Advising Professor

Dr. Robert P. Wei
Chairperson of Department
Acknowledgments

I would like to thank Dr. Arkady Voloshin for his technical support. I would also like to thank the staff of AT&T for going the extra mile in order to make this project possible. This includes:

- Vince Kowalski: for the welded sample,
- Joe Stack, Tim Butrie, Stan Kaufman: for technical advice,
- Lisa Dower: for keeping ANSYS running and providing me with the computer resources I needed,
- Patricio McGowan, and Randy Rittle: for the help I got in the Computer User Center,
- Yiu-Man Wong, Sho Nakahara, Leo Voyden, and Ed Lane, for the weld photographs,
- Al Benzoni: for the use of his scanner.

I would also like to thank the staff at Mallet Software for support with the use of ANSYS; Dr. Robert Mallet, Dr. Abed Khaskia, and Chip Potter.

I would like to express my deepest appreciation to my Wife, Tammy, for her endless support and encouragement, and my Daughter, Lauren, whose arrival revealed the importance of completing this project in a timely manner.
# Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>1</td>
</tr>
<tr>
<td>CHAPTER 1 : INTRODUCTION</td>
<td>3</td>
</tr>
<tr>
<td>CHAPTER 2 : MODEL IMPLEMENTATION</td>
<td>7</td>
</tr>
<tr>
<td>Chapter 2.1 General Procedure</td>
<td>7</td>
</tr>
<tr>
<td>Chapter 2.2 Model Setup Conditions</td>
<td>12</td>
</tr>
<tr>
<td>Chapter 2.2.1 Model Geometry</td>
<td>12</td>
</tr>
<tr>
<td>Chapter 2.2.2 Thermal Boundary and Loading Conditions</td>
<td>14</td>
</tr>
<tr>
<td>Chapter 2.2.3 Mechanical Boundary and Loading Conditions</td>
<td>16</td>
</tr>
<tr>
<td>Chapter 2.3 Temperature Dependent Material Properties</td>
<td>16</td>
</tr>
<tr>
<td>Chapter 2.3.1 Thermal Conduction Coefficient</td>
<td>17</td>
</tr>
<tr>
<td>Chapter 2.3.2 Mass Density</td>
<td>20</td>
</tr>
<tr>
<td>Chapter 2.3.3 Specific Heat, Latent Heat and Enthalpy</td>
<td>20</td>
</tr>
<tr>
<td>Chapter 2.3.4 Convection and Radiation at a Weld</td>
<td>25</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

FIGURE 2.1 : AXISYMMETRIC LASER WELD GEOMETRY 13
FIGURE 2.2 : THERMAL BOUNDARY CONDITIONS 14
FIGURE 2.3 : LASER PULSE FUNCTION 15
FIGURE 2.4 : MECHANICAL BOUNDARY CONDITIONS 15
FIGURE 2.5A : THERMAL CONDUCTIVITY OF KOVAR 18
FIGURE 2.5B : THERMAL CONDUCTIVITY OF 304 STAINLESS STEEL 19
FIGURE 2.6 : SPECIFIC HEAT & ENTHALPY 22
FIGURE 2.7A : SPECIFIC HEAT OF KOVAR 23
FIGURE 2.7B : SPECIFIC HEAT OF 304 STAINLESS STEEL 24
FIGURE 2.8A : COEFFICIENT OF THERMAL EXPANSION OF KOVAR 29
FIGURE 2.8B : COEFFICIENT OF THERMAL EXPANSION OF 304 STAINLESS STEEL 30
FIGURE 2.9A : ELASTIC MODULUS OF KOVAR 35
FIGURE 2.9B : ELASTIC MODULUS OF 304 STAINLESS STEEL 36
FIGURE 2.10A: YIELD AND ULTIMATE STRENGTH OF KOVAR 37
FIGURE 3.10A: RADIAL RESIDUAL STRESS 73
FIGURE 3.10B: AXIAL RESIDUAL STRESS 73
FIGURE 3.10C: TANGENTIAL RESIDUAL STRESS 74
FIGURE 3.10D: VON MISSES RESIDUAL STRESS 74
FIGURE 3.10E: FRACTURE SAFETY FOR KOVAR 76
FIGURE 3.11A: SURFACE RADIAL STRAIN HISTORY 77
FIGURE 3.11B: SURFACE AXIAL STRAIN HISTORY 78
FIGURE 3.11C: SURFACE TANGENTIAL STRAIN HISTORY 79
FIGURE 3.11D: THROUGH THICKNESS RADIAL STRAIN HISTORY 80
FIGURE 3.11E: THROUGH THICKNESS AXIAL STRAIN HISTORY 81
FIGURE 3.12A: RADIAL RESIDUAL STRESS 82
FIGURE 3.12B: AXIAL RESIDUAL STRESS 82
FIGURE 3.12C: TANGENTIAL RESIDUAL STRESS 83
FIGURE 3.12D: VON MISSES RESIDUAL STRESS 83
FIGURE 3.12E: FRACTURE SAFETY FOR 304 STAINLESS STEEL 84
LASER WELD INDUCED THERMAL DEFORMATIONS MAY CAUSE A CHANGE IN TRANSMITTED LIGHT POWER IN PHOTONIC DEVICES DUE TO THE RELATIVE MOTION OF SUB-COMPONENTS. A FINITE ELEMENT MODEL WAS DEVELOPED TO SIMULATE THE LASER WELDING OF A FLAT PLATE IN ORDER TO CHARACTERIZE THE INDUCED THERMAL STRAIN. TEMPERATURE DEPENDENT MATERIAL PROPERTIES WERE INCORPORATED TO ACCURATELY MODEL THE WELDED PART OVER THE WIDE TEMPERATURE RANGE. THE EFFECTS OF THE MATERIAL LOSS DUE TO MELTING AND VAPORIZATION WERE ACCOMPLISHED BY EXCLUDING THE EFFECTS OF MOLTEN AND OBLATED ELEMENTS FROM THE ITERATIVE CALCULATIONS. MOLTEN ELEMENTS RE-ENTERED THE CALCULATION WHEN THEIR TEMPERATURES FELL BELOW THE MATERIAL'S SOLIDUS TEMPERATURE.

IT WAS SHOWN, BOTH THERMALLY AND MECHANICALLY, THAT THE FINITE ELEMENT MODEL DEVELOPED IN THIS STUDY AGREES WITH OTHER THEORETICAL AND
experimental models with an acceptable error. Calculations indicate that approximately 50% of the laser energy is absorbed by the work piece during welding. The residual strain effects are contained within a region only three to four times the weld diameter. Calculations suggest that the material, Kovar, may be stressed beyond its ultimate strength directly below the welded region, which may result in crack formation. This is not the case for 304 stainless steel, a possible substitute for Kovar in laser weld applications in the photonics industry, however stainless steel exhibits approximately 20% higher radial strain than Kovar.
Chapter 1: Introduction

Lightwave devices require precise alignment of internal optical components for optimum light signal transmission, or coupling. Examples include devices such as analog, digital and pump laser packages, light detectors, optical isolators, optical amplifiers, and wave guides. To optimize the coupled power, the light emerging from the source component, laser chip or incoming fiber, must be focused on the destination component, optical fiber or light detector. Single mode fibers have 8 μm diameter cores which transmit the majority of light [1]. Misalignment of the components in the order of micrometers could cause a large power degradation in the transmitted light.

The optical components are joined in the precisely aligned position in order to create a marketable device. Solders [2], adhesives [3] and welds are the most popular methods used to join the source and
destination components. Quality of the device is greatly dependent on the joining technique, therefore, careful consideration of the joining technique used in a design must be incorporated into the lightwave device.

Securing optical components with solder requires heating the device to high temperatures. These elevated temperatures give rise to thermal expansions resulting in relative motion of the two optical components. Adhesives do not require the same amount of heating, however, they still shrink significantly while curing, causing relative motion of optical components. Organic adhesives may undergo long-term outgassing or redeposition onto active elements in the device causing reliability problems. These, and other deficiencies of solders and adhesives may be eliminated through the use of laser welding techniques.

Pulsed laser welding is a highly reliable material joining technique for precision applications. This method delivers high strength bonds with only local heating [4]. Active alignment techniques [5] are usually employed to adjust the relative positions of the two optical components to control the power of the transmitted light. When the desired light output power is achieved, a laser weld is placed at the interface of the two optical component subassemblies.
The power of the transmitted light randomly changes after laser welding [6] due to relative motions between the two optical components. This motion is caused by the thermal strains produced during the rapid heating and cooling around the laser weld. Prior to welding the two subassemblies, the optical components in their individual subassemblies may not be symmetrically located with respect to their bonding surfaces. For this reason the bonding surfaces of the two optical subassemblies may have a relative offset when the optical components are properly aligned. During the welding operation, these offsets may cause a non-symmetric loading on the welding joint resulting in relative motion of the optical components. This relative motion is not easy to predict. Also, the behavior of the thermal strain is not completely understood.

There is a need to analyze the laser weld induced thermal strain in order to understand the effects of material and geometry. The strain can be measured near the laser weld [7], but a mathematical model is required to determine the mechanism controlling this thermal strain. The laser welding technique can then be used more effectively in the manufacture of lightwave devices.

Chapter 2 of this report will discuss the model implementation, such as the boundary conditions, material properties, and other important assumptions which were used in the calculation. Chapter 3
will report the results of the finite element analysis. Chapter 4 will discuss the meaning of the results and sort out the important findings. Chapter 5 will summarize and draw conclusions as well as suggest paths for future study.
Chapter 2: Model Implementation

A finite element model was constructed, using ANSYS [8][9][10], which incorporates the major thermal-mechanical factors involved during a laser welding process.

Chapter 2.1 General Procedure

For completeness, a brief discussion of the events which take place during a laser weld operation will be given here. The thermally related influences will be discussed, and then the mechanical reactions.

After the incident laser beam strikes the material, only a small percentage of the laser's energy is absorbed by the material, the rest is reflected away. The amount of energy that is absorbed is dependent on
many variables including the material's surface temperature and roughness. The energy that is absorbed is used to raise the material's temperature, and is spread, by conduction, through the material [11]. The heat of diffusion and the heat of vaporization store energy while heating up, but then release that energy later as the material cools down. Heat within the material is lost to the surrounding environment through both radiation and convection. The material that vaporizes carries energy out of the system through mass transfer. The violence of the event may also cause energy to be lost through the splattering of the molten material. The amount of heat lost through convection is further enhanced by the swirling of the molten material within the weld pool [12]. The material eventually cools to ambient temperature.

While this is happening, the material undergoes mechanical stresses due to the thermally induced expansion. These stresses are initially compressive and large enough to exceed the elastic limit of the material, thus, plastic flow occurs. The amount of plastic flow which occurs is further increased by the reduction of the material's yield strength and stiffness in the locally heated areas. The loss of material, due to splattering and vaporization, will create new free surfaces which may further influence the plastic flow. As the material begins to cool off, the now over-compressed material tries to return to its initial
condition, but the plastic deformation it has sustained requires the plastic flow to reverse from compression to tension. When the material reaches room temperature, it will be shown that the material is primarily in tension, and the resultant strain is comprised of both elastic and plastic strain.

The finite element model must be capable of solving the system of non-linear, thermal-mechanical, coupled field equations shown below:

\[
\frac{\partial}{\partial t} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) + q = \rho C_p \frac{\partial T}{\partial t}
\]

\[\{\sigma\} = [B] \{\varepsilon\}_s + \{\varepsilon\}_e + \{\varepsilon\}_p\]

\[
[B] = \begin{bmatrix}
1/E & -\nu/E & -\nu/E & 0 & 0 & 0 \\
-\nu/E & 1/E & -\nu/E & 0 & 0 & 0 \\
-\nu/E & -\nu/E & 1/E & 0 & 0 & 0 \\
0 & 0 & 0 & 1/G_{xy} & 0 & 0 \\
0 & 0 & 0 & 0 & 1/G_{yx} & 0 \\
0 & 0 & 0 & 0 & 0 & 1/G_{zz}
\end{bmatrix}^{-1}
\]

\[
\{\sigma\} = \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\sigma_z \\
\tau_{xy} \\
\tau_{yz} \\
\tau_{xz}
\end{bmatrix}
\]
\[
\{\varepsilon\}_x = \begin{bmatrix} 
\varepsilon_x \\
\varepsilon_y \\
\varepsilon_z \\
\gamma_{xy} \\
\gamma_{xz} \\
\gamma_{yz} 
\end{bmatrix} 
\]

\[
\{\varepsilon\}_p = \begin{bmatrix} 
\varepsilon_x \\
\varepsilon_y \\
\varepsilon_z \\
\gamma_{xy} \\
\gamma_{xz} \\
\gamma_{yz} 
\end{bmatrix}_p 
\]

\[
\{\varepsilon\}_n = \Delta T \alpha_x \begin{bmatrix} 
1 \\
1 \\
0 \\
0 \\
0 
\end{bmatrix} 
\]

\[
E = \begin{cases} 
E_s(T) & \text{if } \sigma_s \leq \sigma_y(T) \\
E_p(T) & \text{if } \sigma_s > \sigma_y(T) 
\end{cases} 
\]

\[
E_p = \frac{\sigma_{\text{ult}} - \sigma_y}{\varepsilon_{\text{max}}} - \frac{\sigma_y}{E_s} 
\]

\[
\sigma_s = \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_1 - \sigma_3)^2} 
\]

Where:

- \(T\) is temperature
- \(t\) is time
- \(q\) is the heat input
- \(\{\sigma\}\) is the stress vector
• $\sigma$, is the von Mises equivalent stress
• $\sigma_{1,2,3}$ are principal stresses
• $[B]$ is the bulk modulus matrix
• $\{e\}_e$ is the elastic strain vector
• $\{e\}_p$ is the incremental plastic strain vector
• $\{e\}_h$ is the thermal strain vector
• $k$ is the thermal conduction coefficient
• $\rho$ is the mass density
• $C_p$ is the specific heat
• $\alpha$ is the coefficient of thermal expansion
• $E$ is the effective modulus
• $G$ is the shear modulus
• $E_p$ is the plastic modulus
• $E_*$ is the elastic modulus
• $\sigma_y$ is the yield strength
• $\sigma_{ul}$ is the ultimate strength
• $\varepsilon_{max}$ is the maximum strain

The non-linear portion of the model is characterized by the temperature dependent material properties, as well as the onset of plastic behavior of the welded material. It is this plastic behavior which needs to be characterized, since it is responsible for the permanent displacement near the weld.

The problem is solved in two parts: the thermal solution and the mechanical solution. The first part is a solution for a full field temperature for the three dimensional domain over time. The output, $T = f(x,y,z,t)$, can then be used as the input for the second part of the
problem in order to obtain a mechanical solution \( \varepsilon = f(x,y,z,t) \) or \( \sigma = f(x,y,z,t) \).

Since plasticity is path dependent, the time-temperature history of the material needs to be determined reasonably accurately. The Theory of Plasticity requires load inputs to be applied gradually so as to accurately depict the plastic flow [13]. This is accomplished by calculating the stress state in small time increments, thus, a time history of the plastic stress state as well as a thermal history will be determined.

Chapter 2.2 Model Setup Conditions

Chapter 2.2.1 Model Geometry

The model is axisymmetric, \( f(x,y,z,t) = f(r,z,t) \), and approximates a disk shaped material of finite thickness (Figure 2.1). The diameter of the disk was chosen to be similar to the diameter of the plastically affected area of laser welds that were measured by Kowalski [7].
FIGURE 2.1: AXISYMMETRIC LASER WELD GEOMETRY
Chapter 2.2.2 Thermal Boundary and Loading Conditions

The thermal boundary conditions (Figure 2.2) assume that the top and bottom surfaces are perfect insulators, i.e. \( \frac{\partial T}{\partial n} = 0 \), where \( T \) is temperature, and \( n \) is a surface normal. The outer diameter cylindrical surface parallel to the symmetry axis is chosen to be large enough so as not to see any temperature variations, i.e. \( T = \text{Constant} \). A diameter of 2mm was used on the basis of previous strain measurements [7]. Energy is inputted to the system by applying a heat flux over the focused laser spot; \( \frac{\partial Q}{\partial t} = q(t) \). The function, \( q(t) \), used in this study is shown in Figure 2.3. This function approximates a 5 ms laser pulse with a total output of 3.5 J, the same laser parameters as used in [7].

FIGURE 2.2 : THERMAL BOUNDARY CONDITIONS
FIGURE 2.3: LASER PULSE FUNCTION

FIGURE 2.4: MECHANICAL BOUNDARY CONDITIONS
Chapter 2.2.3 Mechanical Boundary and Loading Conditions

The mechanical boundary conditions (Figure 2.4) assume that all surfaces are free from any restraints. The symmetry axis remains radially fixed. One node is fixed axially opposite the welding surface. The input forcing function is the temperature function calculated in the thermal part of the problem: $T(r,z,t)$.

Chapter 2.3 Temperature Dependent Material Properties

The materials considered in this analysis are Kovar [14], and 304 stainless steel. These materials were chosen because of their wide use in the photonics industry and the need to understand how they are affected by laser weld processes.

Since the temperatures reached in this analysis span from room temperature to the melting and vaporization temperatures of the materials in question, the materials need to be characterized over a wide temperature range. Some of the relevant material properties required in Equations 2.1 through 2.10 have been documented [14][15][16], but others have not. Even those that have been documented do not span the
necessary temperature range. These undocumented material properties have been estimated based on the slope of the material property at a known temperature and on the material property of other similar materials.

Chapter 2.3.1 Thermal Conduction Coefficient

The thermal conduction coefficient is a heat transport material property. It is described as the power per distance per temperature degree, or more specifically, W/m°C. Graphs of the temperature dependent conduction coefficients for Kovar, and 304 stainless steel, compared with those which were input to the finite element model, can be found in Figures 2.5a and 2.5b.

Notice how the conduction coefficient curves used in the finite element model are flat above the melting temperature of the material. This was done for computational convenience. Molten material’s effective conduction probably increases dramatically due to convective currents in the weld pool, however, these effects impose only a small error on the remaining solid material and are not considered here.
FIGURE 2.5A: THERMAL CONDUCTIVITY OF KOVAR
FIGURE 2.5B: THERMAL CONDUCTIVITY OF 304 STAINLESS STEEL
Chapter 2.3.2 Mass Density

The temperature dependent mass density of a solid material is related to the coefficient of thermal expansion (CTE) of that material. The average CTE, of the materials in question, over the temperature range bounded by room temperature and the material's melting temperature ($\Delta T \approx 1400^\circ$C), are on the order of $15 \times 10^{-6}$ /°C. This would indicate a mass density reduction of approximately 6% over the above temperature range. Because this discrepancy is small, and since the mass density only enters into the thermal half of the calculation, the mass density is assumed to be constant over the above temperature range.

Once the material undergoes a phase change to liquid or vapor, the mass density changes more drastically, however, this has little effect on the strains present in the solid material.

Chapter 2.3.3 Specific Heat, Latent Heat and Enthalpy

The specific heat of a material is the amount of energy required per unit mass to raise the temperature of a material by one temperature degree. The units of specific heat are J/Kg-°C. The specific heat of a
material changes drastically with temperature. It typically will have an increasing spike at the temperature where the material undergoes a phase change, and returns to near the same value after the phase change. This large increase of the specific heat near a material's phase change is known as a latent heat [17].

Originally, the latent heat of a material was modeled by inputting a very large slope in the specific heat curve about the phase change temperature (Figure 2.6a). The difficulty with this approach is that when the time step used for the analysis is too large, it is possible to step over the spike in the specific heat curve. This difficulty does not occur with an enthalpy formulation (Figure 2.6b).

Enthalpy is defined by the following equation:

\[ \text{enth} = \int C_p \rho \, dT \]  

2.11

Enthalpy is not an absolute value, it is relative to an arbitrary reference point. Its value at any given temperature has no meaning, but its slope over temperature is a measure of a material's density and specific heat. If density is known, or constant as in this case, specific heat can be directly assessed. Since specific heat is a more accepted form for characterizing this phenomena, enthalpy will not be used any further to describe the modeling in this report.
Graphs of the temperature dependent specific heat for Kovar, and 304 stainless steel, compared with those which were input to the finite element model, can be found in Figures 2.7a and 2.7b. Heat of fusion information was only available for Kovar, and no heat of vaporization information was found for either. Therefore, the heat of vaporization was assumed to be twice that of the heat of fusion and these properties were used for both Kovar and 304 stainless steel. Again, the use of these assumptions will only effect the behavior of the molten material and will have little effect on the solid material.

FIGURE 2.6 : SPECIFIC HEAT & ENTHALPY
Heat of Fusion = 270000 J/kg

\[ 270000 \text{ (J/kg)} \]

\[ 13500 \text{ (J/kg}^\circ \text{C)} \]

**FIGURE 2.7A: SPECIFIC HEAT OF KOVAR**
FIGURE 2.7B : SPECIFIC HEAT OF 304 STAINLESS STEEL
Chapter 2.3.4 Convection and Radiation at a Weld

The only form of heat transfer accounted for in this model is conduction through the material. It is clear that the effects of radiation/reflection and convection/mass transport play a very major role in determining the time dependent temperature profile of the material, but it is not the purpose of this study to model all these individual phenomena. Instead, a more experimental, rather than theoretical, approach was used to correlate the model to real life events.

In order to explain this experimental approach, one must realize that heat input at a laser welding station is usually controlled by the beam's focus, pulse time, and intensity. The beam's focus can be modeled with a surface heat flux where the beam is focused. Pulse time determines how long the heat flux is active. The unknown values for absorption coefficients make modeling the beam intensity difficult. Therefore, heat input to the model is not controlled by the intensity of the laser beam, but instead is determined by matching the size of the weld pools of the model with that of the experimental work. This means that the model does not describe a welding phenomena caused by using a 3.5 J laser pulse, but instead, models a welding phenomena intense enough to create a 0.6 mm diameter weld pool [7].
The advantages to this approach become more desirable because the model no longer needs to accurately depict the events which take place after the material exceeds its melting temperature. Also, the amount of heat necessary to create a useful weld varies with material, and thus it becomes more relevant to compare the strains in different materials with an equal weld diameter than those with an equal heat input.

Chapter 2.3.5 Melting and Vaporization (Thermal Influence)

The material which melts will eventually, upon cooling, re-solidify back on the material and form the weld [18]. However, it will not necessarily solidify in the same location from which it melted. In order to accurately model the new solidification location, the model would need to keep track of the fluid flow through the weld pool [12]. The material which vaporizes is assumed to be removed forever from the material sample. Along with the vaporized material, heat energy is lost from the system.

Since the thermal model calculations are performed before any mechanical calculations are done, and since it is not as critical to
accurately model the material thermally once its melting point is exceeded, as discussed in Section 2.3.4, the current model assumes that the melted material does not move and only transfers heat through conduction. If the vaporized material, along with its heat energy, was removed from the calculations (element death), no heat energy would be transferred beyond the first time increment, and no useful information would be gained. Therefore, in the thermal model, the material which vaporizes is not removed from the model and continues to transfer heat through conduction.

Chapter 2.3.6 Thermal Coefficient of thermal expansion

The coefficient of thermal expansion is a material property defining the amount a material changes size due to a temperature change. It is described as expansion distance per unit distance per temperature degree, or, since the values are small, more commonly as micro-strain per temperature degree (µε/°C).

Thermal expansion can be described in many ways. Two of these ways are the instantaneous coefficient of thermal expansion and the average coefficient of thermal expansion. Consider a one unit length bar
of a given material at an initial temperature. Now increase the
temperature, and record the displacement as a function of temperature.
The instantaneous coefficient of thermal expansion at a given
temperature would be the slope of the curve at that temperature
\(\frac{dx}{dT}\). The average coefficient of thermal expansion at a given
temperature would be the slope of the secant passing through the curve
at both the given temperature and some arbitrary reference temperature,
\(T_{ref}\).

\[
\alpha_{avg} = \frac{x(T) - x(T_{ref})}{T - T_{ref}}.
\]

Eq. 2.12

Typically, the reference temperature is the temperature at which there is
zero thermal strain.

Difficulties exist with programming the instantaneous coefficient
of thermal expansion definition. This method requires an integration of
coefficient values over temperature load steps, forcing accuracy to be
dependent on the temperature load increment size. With the average
coefficient of thermal expansion definition, the amount of thermal strain
present is only dependent on the difference between the current
temperature and the reference temperature. The ANSYS software uses
the average coefficient of thermal expansion definition.
FIGURE 2.8A: COEFFICIENT OF THERMAL EXPANSION OF KOVAR
FIGURE 2.8B: COEFFICIENT OF THERMAL EXPANSION OF 304 STAINLESS STEEL
The average coefficient of thermal expansion is dependent on the value of the reference temperature. This means that all coefficient of thermal expansion reference data must be converted from its given reference temperature to the reference temperature being used in the current problem. This can be accomplished for any temperature, T, with the following equation:

\[
\alpha_{\text{eff}}(T) = \alpha_o(T) + \frac{T_{\text{ref}} - T_0}{T - T_{\text{ref}}} \left[ \alpha_o(T) - \alpha_o(T_{\text{ref}}) \right]
\]

where \( \alpha_{\text{eff}}(T) \) is the average coefficient of thermal expansion based on \( T_{\text{ref}} \), the new reference temperature, and \( \alpha_o(T) \) is the average coefficient of thermal expansion based on \( T_0 \), the given reference temperature.

Graphs of the temperature dependent coefficient of thermal expansions for Kovar [14], and 304 stainless steel [15][16], compared with those which were input to the finite element model, can be found in Figures 2.8a and 2.8b. For computational convenience, the coefficient of thermal expansion was assumed to be constant at elevated
temperatures, where information on the coefficient of thermal expansion was not available.

**Chapter 2.3.7 Stress - Strain Relation**

The stress-strain relationship of the modeled material is, by far, the most critical group of material properties governing these calculations. Since the model undergoes a cyclic type loading condition (initially compressive and then tensile), bi-linear kinematic strain hardening will be used with von Mises yield criterion. The plastic stiffness, $E_p$, can be determined by the following equation:

$$E_p = \frac{\sigma_{ult} - \sigma_y}{\varepsilon_{max} - \frac{\sigma_y}{E}}$$  \hspace{1cm} 2.14

where $\sigma_{ult}$ is the ultimate strength, $\sigma_y$ is the yield strength, $E$ is the elastic modulus and $\varepsilon_{max}$ is the maximum elongation. Graphs of the temperature dependent elastic stiffness, yield strength, ultimate strength, and maximum elongation for Kovar, and 304 stainless steel, compared with those which were input to the finite element models, are shown in Figures 2.9a through 2.11b. Very little information exists for these properties at high temperatures, therefore estimates were made.
The elastic stiffness for 304 stainless steel (Figure 2.9b) was unknown above 800°C, therefore estimates were based on a linear relation from the measured value at 800°C and a somewhat arbitrary value of 80GPa at the material's melting temperature. This value may be larger than the actual elastic stiffness at that temperature, however, it reduces computational difficulties associated with having large changes in the elastic stiffness. Temperature dependent elastic stiffness information was unavailable for Kovar (Figure 2.9a). Kovar was assumed to possess a similar percentage elastic stiffness degradation over temperature as 304 stainless steel from room temperature to 800°C. Above 800°C, a similar linear relationship, as was used for the 304 stainless steel, was employed using a somewhat arbitrary value of 40GPa at the material's melting temperature.

The yield and ultimate strength for Kovar converged upon each other at 800°C, which was the maximum temperature that these properties were measured (Figure 2.10a). This indicates that the material would fail before yielding at this elevated temperature. Since a positive plastic stiffness is required in this model, the yield strength must be non-zero, and less than the ultimate strength for all temperatures below melting. An arbitrary value for the ultimate strength
was chosen to be 60MPa, just below the melting temperature, which is one order of magnitude less than the maximum ultimate strength (a computational convenience). The yield strength was chosen to be 10MPa less than the ultimate strength at the same temperature. A linear relationship was used to bridge the gap from the known values to the assumed values at the melting temperature. Similar assumptions were made for the 304 stainless steel in Figure 2.10b. Here, an extra value for the ultimate strength at approximately $1000^\circ$C was ignored because it was too low, and may have caused convergence difficulties.

The maximum elongation values for Kovar were piece-wise linearly approximated over their known values, and were assumed to be low at higher temperatures where the materials began failing before yielding. Its value must be greater than the maximum possible elastic strain $\sigma_y/E$, and should be greater than twice that value in order for the plastic modulus in Equation 2.14 to be less than the elastic modulus. Typically, the elastic modulus is over one order of magnitude greater than the plastic modulus which would put the lower limit for the maximum elongation for Kovar to be approximately 2%. A value of 5% was used as the melting temperature maximum elongation for Kovar and 10% for 304 stainless steel.
FIGURE 2.9A: ELASTIC MODULUS OF KOVAR
FIGURE 2.9B: ELASTIC MODULUS OF 304 STAINLESS STEEL
FIGURE 2.10A: YIELD AND ULTIMATE STRENGTH OF KOVAR
FIGURE 2.10B: YIELD AND ULTIMATE STRENGTH OF 304 STAINLESS STEEL
FIGURE 2.11A: MAXIMUM ELONGATION OF KOVAR
FIGURE 2.11B: MAXIMUM ELONGATION OF 304 STAINLESS STEEL
Chapter 2.3.8 Melting and Vaporization (Mechanical Influence)

When material melts or vaporizes, its mechanical influences on the rest of the material become negligible. The removal of this material from the calculations for a given time increment is accomplished by setting the resultant forces of the finite elements of that material equal to zero. This is known as element death. The elements for the material which exceed their vaporization temperature are "killed", never to return to the calculation. However, the elements for the material which only exceed their melting temperature, and not their vaporization temperature, are "killed" for the time increments that they are liquid, but are "revived" for the time increments after they return to the solid state.

The reactivated elements have no record of strain history, however, an element can experience thermal strains during the first load step after being reactivated. This strain is based on the reference temperature of the element which is initially defined as room temperature. In order to correctly model the strain in an element after solidification, the element's reference temperature must be redefined as the solidus temperature, and the material's coefficient of thermal
expansion must be converted to the new reference temperature, as discussed in Section 2.3.6.

One drawback with reactivating elements is that the elements return to their originally pre-melted geometric positions. The large deflections and large strain of the elements can be accounted for by using large strain theory, however, the flow of the material was not accounted for in the melted state. This flow is dependent on the direction of external body forces, such as gravity and ambient air flow, that are present during the welding process. These external forces could alter the shape of the molten material before solidification, influencing the stress state of the surrounding material. By allowing the material to return to its original geometric position, the model may impose an improper condition which would lead to inaccurate results. However, not allowing the material to return at all, would guarantee inaccurate results. This dilemma will be handled by comparing the results of both scenarios, which will, at least, give an idea of the amount of error in the calculation.
Chapter 2.4 Model Precision

The results of a finite element analysis are only as good as the model generated. When dealing with a transient coupled-field analysis, both element size and time step increment become critical to the final results. This section will relate the effects these variables have on the final results, and what can be done to correct any difficulties which may arise.

Chapter 2.4.1 Element Size

Smaller elements will yield more accurate results, but will also increase the solution matrix, and thus solution time and required computer resources. It then becomes advantageous to use the largest elements that will still yield a result with an acceptable error.

In order to further maximize run efficiency, variable element sizes can be used. Thus, the smaller elements only need to be placed where the largest temperature and stress/strain gradients are expected to be. These high gradient locations can be found, in this model, around the
heat input region representing the laser beam incidence location. Figure 2.12 shows an example element mesh used in the analysis.

Chapter 2.4.2 Time Step Size

As with element size, time step size, or time increment, affects the outcome of a finite element analysis. Every time increment solved for in a non-linear transient finite element analysis requires the equivalent computer resources of a non-linear static finite element
analysis. Also, in non-linear static analyses, equilibrium iterations may be required to obtain a converged solution for a given time step. Each iteration requires the equivalent computer resources of a linear static finite element analysis. Thus, both non-linear effects, and transient analyses, geometrically increase the number of iterations needed to be solved.

Smaller time increments will yield more accurate results, but will also increase the total number of time steps to be solved for. This may or may not increase the solution time for a non-linear analysis, since when a smaller time increment is used, fewer equilibrium iterations may be required to converge on a solution for a given time step. If a time step is too large, it may not be possible to converge on a solution for a given time step. Many times this condition is characterized by a cyclic behavior from one iteration to the next. This characteristic is a typical condition found when modeling material going through a phase change where the material's stiffness or specific heat suddenly undergoes a large change.

The time step size may also vary through the analysis, with the smaller time steps being used during the most dynamic time zones, and larger time steps being used during the calmer time periods. The
software being used also has built-in time step changers when a non-convergence situation arises.

There is an optimum time step size where the time step is as large as it can be, yet small enough in order not to sacrifice accuracy, or the number of equilibrium iterations required per time step. Unfortunately, this optimum time step varies through the solution, and can only be found through trial and error.

Chapter 2.4.3 Modeling Within Computer Capabilities

In an attempt to reduce computer run time, without sacrificing accuracy or element size, a thinner and smaller diameter specimen, when compared to the specimen used in [7], was analyzed. This reduction of the analyzed domain rather than the increase of element size, conserves model accuracy while reducing the required computer resources (300 nodes and elements yield 30-70 hours run time rather than several weeks on an HP 720 workstation). It will be demonstrated that the top surface results, both thermal and mechanical, are independent of thickness for thicknesses greater than the weld diameter. It will also be shown that it
is sufficient to consider a diametral domain only three to four times the weld diameter.

Chapter 2.4.4 Verification of Results

There are many techniques that may be used to verify that a finite element model is yielding meaningful results. The results of an analysis do not necessarily have to be absolutely correct in order to reveal trends that may be very useful. The following two sections will deal with the various techniques used in this analysis to assure reliable solutions.

Chapter 2.4.4.1 Verification with the Finite Element Model

A "rule of thumb" to follow to determine if a model's mesh or time step is refined enough is to assure that there are no elements in any converged time step solution which have a temperature or stress/strain change across an element, or time step, that is greater than 5-10% of the total temperature or stress/strain range in the model. This rule will not guarantee an adequate mesh, or time step, but it will point out the areas which may require fine tuning. These areas can then be further
developed by reducing the element sizes, or time increments, in that zone and re-running the calculation. If a significant change takes place in the new solution, the above process should be repeated until an acceptable result is converged upon.

The above processes were carried out for one of the material cases with a goal to keep the total error under 5%. The resulting mesh was then used for all the runs carried out in this material study. All of these precautions will verify that, if the above simplifying assumptions are valid, the software is converging on an accurate solution.

Chapter 2.4.4.2 Verification with Experimental Results

Laser weld induced thermal strains were measured using a moire' interferometry technique [7][19]. One can determine if the above mentioned simplifying assumptions are valid by matching the finite element model's inputs to the conditions of these experiments, and verifying that the results are supportive of each other.
Chapter 3: Results

In this chapter, the results of the finite element analysis will be presented with other available theoretical models and experimental results. The thermal results of the analysis will be discussed first, followed by the mechanical results.

Most of the analysis deals with the material Kovar, since it is the preferred material used in laser weld assembled microelectronic devices. It is also the only material that was used in previous experiments where laser weld induced thermal strain was measured [7]. Therefore, throughout this chapter, unless otherwise specified the material being discussed will be Kovar. Stainless steel, type 304, is also a preferred material for laser welding applications, and is analyzed in this report.
The results discussed below are for a 0.6 mm weld diameter, administered by a 5 ms laser pulse. This size weld is typical of a 3.5 J power dissipation for the given pulse duration.

Chapter 3.1 Thermal Calculation

Chapter 3.1.1 Thermal Thickness Independence

As discussed in Section 2.4.3, it is advantageous to model a smaller specimen than the one used by Kowalski [7], therefore, this section will present a comparison of a larger/thicker model with a smaller/thinner model.

Figures 3.1a and 3.1b show the ANSYS calculated temperature from a 4 mm diameter model with a thickness of 1.27 mm. Figure 3.1a shows the temperature of the model's top surface as a function of radial position from the weld center for multiple time instances. Figure 3.1b shows the temperature of the model's center axis as a function of time and the position from the top surface through the thickness of the specimen. Figures 3.2a and 3.2b show the same temperature plots for a 2 mm diameter model with a 0.5 mm thickness.
FIGURE 3.1A: THICK SPECIMEN SURFACE TEMPERATURE HISTORY

Material: Kovar
FIGURE 3.1B: THICK SPECIMEN AXIAL TEMPERATURE HISTORY
FIGURE 3.2A : THIN SPECIMEN SURFACE TEMPERATURE HISTORY
FIGURE 3.2B: THIN SPECIMEN AXIAL TEMPERATURE HISTORY
Chapter 3.1.2 Thermal Comparison of Finite Element and Closed Form Solution

The governing equation for heat conduction through a solid is:

$$\frac{\partial}{\partial x} \left( k_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k_y \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k_z \frac{\partial T}{\partial z} \right) + q = \rho C_p \frac{\partial T}{\partial t}$$  \hspace{1cm} 2.1

where $q$ is the heat input to the system, $\rho$ is the material density, $C_p$ is the specific heat, $k$ is the conduction coefficient and $T(x,y,z,t)$ is the temperature at any time ($t$) and position ($x,y,z$). If material properties are constant over temperature, then the equation reduces to:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{q}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$  \hspace{1cm} 3.1

where $\alpha = \frac{k}{\rho C_p}$, is the thermal diffusivity. For the case of an instantaneous point source in an infinite body: [20]

$$T(r,t) = \frac{q / \rho C_p}{8(\pi \alpha)^{3/2}} e^{-\frac{r^2}{4\alpha t}} + T_0$$  \hspace{1cm} 3.2

satisfies the governing equation, with $T_0$ being the initial uniform temperature of the system. Assuming the point source is on the surface of a semi-infinite body, the equation becomes:

$$T(r,t) = \frac{q / \rho C_p}{4(\pi \alpha)^{3/2}} e^{-\frac{r^2}{4\alpha t}} + T_0$$  \hspace{1cm} 3.3
Figure 3.3 presents this curve along with the finite element calculation, at a radial position of 0.4 mm from the heat source.

Chapter 3.1.3 Energy and Mass Phase Changes

The controlling input to the model, as outlined in Section 2.3.4, is weld diameter and laser pulse duration. The model can then predict the energy required to achieve the input weld diameter. The energy
absorption coefficient can be calculated by dividing the finite element prediction of how much energy is required to produce a 0.6 mm weld by 3.5J, the laser energy actually required to create a 0.6 mm weld diameter in Kovar [7]. The energy absorption coefficient was calculated to be 0.486 for Kovar, and assuming the same amount of laser energy is required in 304 stainless steel, its energy absorption coefficient was calculated to be 0.413.

Figure 3.4 shows the cross-section of the weld region modeled in this analysis. The thermal analysis suggests that there is a region surrounding the weld center that vaporizes. This region is 0.43 mm in diameter, and 0.21 mm deep. The thermal section of the model was continuously re-run with ever increasing heat input values until the calculated weld diameter agreed with the specimen's measured weld diameter of 0.6 mm. The amount of heat required to be input to the model in order to result in a 0.6 mm weld diameter was then assumed to be the amount of heat absorbed by the given material. No controls were placed on the weld depth, which was calculated by ANSYS to be 0.3 mm, a half sphere. Figure 3.5 shows three views of an actual welded piece. The cross-section indicates a weld depth of 0.33 mm.
FIGURE 3.4: CALCULATED MOLTEN AND VAPOR REGIONS

FIGURE 3.5: WELD PHOTOGRAPH

Top left: perpendicular view of weld, top right: cross section view of weld, bottom: 70° off perpendicular axis view of weld.
FIGURE 3.4: CALCULATED MOLTEN AND VAPOR REGIONS

FIGURE 3.5: WELD PHOTOGRAPH

Top left: perpendicular view of weld, top right: cross section view of weld, bottom: 70° off perpendicular axis view of weld.
Figures 3.6a and 3.6b show the surface and axis temperature histories respectively for a thin-specimen of 304 stainless steel as were calculated by ANSYS.
Material: 304 Stainless Steel

FIGURE 3.6B : THIN SPECIMEN AXIAL TEMPERATURE HISTORY
Chapter 3.2 Mechanical Calculation

Chapter 3.2.1 Mechanical Thickness Independence

As discussed in Section 2.4.3, it is advantageous to model a smaller specimen than the one used by Kowalski [7]. This section will present a thermal strain comparison of a larger/thicker model with a smaller/thinner model.

Figures 3.7a through 3.7e compare the radial, axial and tangential thermal strains of a Kovar specimen's top surface as well as through its thickness along the laser-focus axis for the large model (a 4 mm diameter model with a thickness of 1.27 mm), and the small model (a 2 mm diameter model with a 0.5 mm thickness).
FIGURE 3.7A: FINAL SURFACE RADIAL STRAIN
FIGURE 3.7B: FINAL SURFACE AXIAL STRAIN

Material: Kovar

Micro-Strain

Small Specimen

Large Specimen

Radial Position (mm)
FIGURE 3.7C: FINAL SURFACE TANGENTIAL STRAIN
FIGURE 3.7D: FINAL THROUGH THICKNESS RADIAL STRAIN at central laser focus axis
Chapter 3.2.2 Calculated vs. Measured Strain

As mentioned in Section 2.3.8, the method of treatment for the return of the molten material will affect the outcome of the analysis.
Therefore two following conditions have been addressed for molten material that had never vaporized: 1) the mechanical influences of melted material are discounted, but it is allowed to reenter the mechanical calculations after its temperature falls below the solidus temperature, and 2) the mechanical influences of melted material are discounted, even after it returns to a temperature below its solidus temperature. The results of these two models should suggest the influence of the re solidification of molten material on thermal strain about the weld. Figure 3.8 compares the results of these two models with the averaged experimental results reported in [7].

![Figure 3.8: Surface Radial Strain, Calculated vs. Measured](image)

**Figure 3.8: Surface Radial Strain, Calculated vs. Measured**
Chapter 3.2.3 Strain History

One of the advantages of calculating the strain near a weld over experimentally measuring the strain is the ability of viewing the strain history throughout the welding process. Another advantage is viewing this strain inside of the welded part. Figures 3.9a through 3.9e show the strain history of the specimen for the top surface radial, axial, and tangential strains and the through thickness laser-focus axis radial, and axial strain respectively.

![Figure 3.9A: Surface Radial Strain History](image)

**FIGURE 3.9A : SURFACE RADIAL STRAIN HISTORY**
Material: Kovar

FIGURE 3.9B: SURFACE AXIAL STRAIN HISTORY
FIGURE 3.9C: SURFACE TANGENTIAL STRAIN HISTORY
FIGURE 3.9D: THROUGH THICKNESS RADIAL STRAIN HISTORY at central laser focus axis

Material: Kovar
Chapter 3.2.4 Residual Stress

One of the reasons for performing this analysis is to determine the residual stress left in the material after the welding operation. The residual stress calculations can be used in predicting weld failure under...
external loading conditions. Figures 3.10a through 3.10d show respectively the radial, axial, tangential, and von Mises residual stress contours of the weld sample. The material which was determined to be vaporized has been omitted from the plots.
Figure 3.10d shows the von Mises stress condition (Equation 2.10). This is the stress criteria which was used to determine whether the plastic strain was to take place.
It is possible that the temperature dependent ultimate stress of the material was exceeded in the model at some elevated temperature during the welding process. If this occurs, cracking and voids may appear about the weld, however, such an event was not considered in this analysis. This possibility will be approached by defining the weld fracture coefficient \( F_w \) as:

\[
F_w(r,z,t) = \frac{\sigma^e(r,z,t)}{\sigma_{ult}(T(r,z,t))}
\]

where \( \sigma^e \) is the von Mises stress of the work piece, \( \sigma_{ult} \) is the ultimate strength of the material, \( T \) is temperature, \( r \) and \( z \) are the radial and axial coordinates of the model, and \( t \) is the time considered. Figure 3.10e shows the maximum weld fracture coefficient against time. A value greater than one indicates a time when the ultimate stress in the model was exceeded.
Chapter 3.2.5 Stainless Steel

The top surface radial, axial, and tangential strain histories and the interior center axis radial and axial strain histories, respectively, for a 304 stainless steel specimen are shown in Figures 3.11a through 3.11e.
FIGURE 3.11A: SURFACE RADIAL STRAIN HISTORY
Material: 304 Stainless Steel

FIGURE 3.11B: SURFACE AXIAL STRAIN HISTORY
Material: 304 Stainless Steel

FIGURE 3.11C: SURFACE TANGENTIAL STRAIN HISTORY
FIGURE 3.11D: THROUGH THICKNESS RADIAL STRAIN HISTORY at central laser focus axis
The radial, axial, tangential and von Mises residual stress contours for 304 stainless steel are shown in Figures 3.12a through 3.12d. The material which was determined to be vaporized has been omitted from the plots and calculations. This formed the jagged surface displayed in the weld cavity which is a product of the model’s element
size (Figure 2.12). High stressed regions adjacent to this artificial feature should be ignored. Figure 3.12e shows the maximum weld fracture coefficient against time for 304 stainless steel, as discussed in Section 3.2.4.

FIGURE 3.12A: RADIAL RESIDUAL STRESS

FIGURE 3.12B: AXIAL RESIDUAL STRESS
FIGURE 3.12C: TANGENTIAL RESIDUAL STRESS

FIGURE 3.12D: VON MISES RESIDUAL STRESS
FIGURE 3.12E: FRACTURE SAFETY FOR 304 STAINLESS STEEL
Chapter 4: Discussion

The following chapter discusses the results reported in Chapter 3. The sections in this chapter are arranged to correspond with the sections in Chapter 3.

Chapter 4.1 Thermal Calculation

Chapter 4.1.1 Thermal Thickness Independence

Consider the set of surface temperature curves found in Figures 3.1a and 3.2a. The surface temperature over time of these two specimens are nearly identical. They only differ slightly in the rate of
cooling, the large specimen being quicker. In Figure 3.2a, the temperature at a radial position of 1 mm is held constant. The temperature in the larger specimen is held constant at a 2 mm radial position, however, by noticing the large and small specimen's temperature variation (Figure 3.1a and 3.2a), over time, at a 1 mm radial position, it is observed that only a relatively small error (less than 5% over all) is induced in the smaller diameter model.

When comparing the axial temperature curves in Figures 3.1b and 3.2b, there are some significant differences. In the thin specimen, the heat path is restricted and causes the material to cool more slowly. For example, 20 ms after the laser weld was started, the temperature 0.5 mm through the thickness of the larger specimen is approximately 400°C, the thin specimen is approximately 700°C at the same time and position. Also, the smaller specimen reaches a through-thickness uniform temperature more quickly. The temperature along the central axis of the smaller specimen, in Figure 3.2b, reaches a uniform temperature, within 100°C from top to bottom, before 30 ms have elapsed. The larger specimen, Figure 3.1b, requires at least 40 ms to reach the same uniform temperature.
These results show that the thermal conditions between the two models differ through the thickness of the specimen, but are very similar on the top surface.

Chapter 4.1.2 Thermal Comparison of Finite Element and Closed Form Solution

The small discrepancy of the two curves in Figure 3.3 are attributed to the assumption of constant material properties used in the closed form solution. The finite element solution considers temperature dependent material properties, including latent heat effects which would account for the temperature peaking at a later time. Even with these differences, the agreement between the two curves gives a high degree of confidence in the validity of the results.

Chapter 4.1.3 Energy and Mass Phase Change

The values of the energy absorption coefficients calculated in Section 3.1.3 are at least 3 times greater than those previously
calculated [21]. This discrepancy can be attributed to the difference in laser power, pulse duration, and materials.

The weld shown in Figure 3.5 is slightly deeper than the weld which was modeled in Figure 3.4. This error is attributed to the assumption that the laser incidence is stationary on the material's surface. During experimentation, a deeper weld is formed when the laser incidence falls beneath the material's top surface as vaporized material is oblated [22][23].

Since no data was available in regards to the heat of vaporization for Kovar, a value of twice the value of the heat of fusion was assumed. This may account for the discrepancy between the calculated vapor region and the amount of oblated material observed in Figure 3.5. If a larger value for the heat of fusion was used, less material would be able to achieve a temperature above the vaporization temperature. This would reduce the diameter of the vapor region in the finite element calculation without largely affecting the results obtained in the solid material.
Chapter 4.1.4 Stainless Steel

The surface and axial temperature histories shown in Figures 3.6a and 3.6b for 304 stainless steel are very similar to those displayed for Kovar in Figures 3.2a and 3.2b. The major difference is the rate of cooling. The higher conduction coefficient of the steel proves a better heat path; however, the accelerated cooling rate does not affect the instantaneous thermal profile. In other words, the shapes of the curves plotted in Figures 3.2a and 3.2b are nearly identical to the curves in Figures 3.6a and 3.6b, only the time at which each curve occurs is different. This difference is attributable to the ratio of the material's thermal diffusivity. As long as this ratio remains relatively constant over temperature, as is true between Kovar and 304 stainless steel, this matching of thermal profiles will occur. The value of the thermal diffusivity for Kovar and 304 stainless steel are:

\[
\kappa_{\text{Kovar}} = \frac{k}{\rho C_p} = \frac{16.5 \frac{W}{mC}}{(8359 \frac{kg}{m^3}) \cdot (440 \frac{J}{kgC})} = 4.49 \times 10^{-6} \frac{m^2}{s}
\]

4.1

\[
\kappa_{\text{Steel}} = \frac{k}{\rho C_p} = \frac{14.4 \frac{W}{mC}}{(7832 \frac{kg}{m^3}) \cdot (456 \frac{J}{kgC})} = 4.03 \times 10^{-6} \frac{m^2}{s}
\]

4.2
\[ \text{Ratio} = \frac{K_{\text{Kovar}}}{K_{\text{Steel}}} = 1.113 \]

Based on this calculation, the 18 ms thermal profile curve for 304 stainless steel in Figures 3.6a and 3.6b will coincide with the $1.113 \times 18\text{ms} = 20\text{ms}$ curve for Kovar in Figures 3.2a and 3.2b. The actual comparison of these curves supports this calculation.

Chapter 4.2 Mechanical Calculation

Chapter 4.2.1 Mechanical Thickness Independence

The surface strains displayed in Figures 3.7a through 3.7c show that the difference in surface strain is less than 20% between the thick and thin specimen over the entire strain range and less than 5% in the non-extreme strain values. As mentioned in Section 4.1.1, one difference on the top surface between the large and the small specimen is the rate of cooling. The model assumes that the elastic and plastic strain are independent of the strain rate. Since the strain rate is driven by the rate of temperature change, the thermal loading conditions should yield similar, if not identical, top surface strain results.
Another distinguishing characteristic of the two models is the effect of the boundary conditions. One can visualize the smaller, and thinner model being superimposed upon the larger model. The excess material of the larger model imposes a pseudo boundary condition about where the smaller model's outer boundary would be. This would result in the smaller model being more compliant near the outer boundary than the larger model resulting in reduced strains in the smaller model.

Notice that the larger model in Figure 3.7a, shows a slightly larger peak strain closer to the edge of the weld, but less strain further from the weld. This could be explained by realizing that the smaller model is indeed more compliant.

These results show that the surface strains of the two different sized specimens are very similar (Figures 3.7a through 3.7c), however, there is a more pronounced deviation through their thickness (Figures 3.7d and 3.7e).

Chapter 4.2.2 Calculated and Measured Strain

Since only surface radial strains are being compared in this section, the thin specimen results were substituted for the thick
specimen results, based on Section 4.2.1. This is advantageous because the thick model requires more computer resources than is convenient to work with.

In Figure 3.8, the comparison of the calculated surface radial strain results with the experimental results reported in [7] showed a very close match with the first model, which allowed material to re-solidify after melting, as discussed in Section 3.2.2. For this reason, all further discussion of calculated strain will refer to that modeling technique.

Figure 3.8 gives a view of the strain closer to the weld center than the experimental measurements were able. It shows the radial strain peaking at a radial position of 0.4 mm from the weld center, with a radial strain component of 8000με. This is almost twice the experimentally measured strain found just 0.1 mm away. The deviation of the experimental to theoretical results beyond a 0.6 mm radius may be attributed to the element size in that region (the piece-wise linearization of the theoretical curves indicate the element size).

The stress and strain inside the weld diameter was not attempted to be modeled accurately due to the difficulties in modeling the fluid motions in the weld pool. The material modeled there was only meant to
serve as a stiffening boundary condition for the material on the weld perimeter.

Chapter 4.2.3 Strain History

The radial strain (Figure 3.9a) is mostly compressive for the first 10 ms. This is due to the higher temperature material at the center of the weld expanding in a restrictive, cooler outer boundary. After 10 ms, the material reverses to a tensile strain due to the plastically compressed material being forced to fill its original space. The majority of straining takes place in the first 20 ms.

Notice the step in the 20 ms radial strain curve around 0.45 mm from the weld center. This step can be explained by the reduced yield strength, due to the high temperature of the material, in that region. At 20 ms, the temperature of the material surface, at a 0.45 mm radial position, is approximately 500°C (Figure 3.2a). This places the yield strength of the material at approximately 200 MPa (Figure 2.10a), nearly one half of its room temperature value. For this reason, deformation tends to take place closer to the center of the weld. This helps keep the material's deformation confined to a small region about the weld sight.
Chapter 4.2.4 Residual Stress

The radial stress (Figure 3.10a) is mostly tensile. Two tensile peaks exist, the smaller is on the surface of the material forming a tensile stress ring about the weld, and the larger is interior to the material, directly beneath the weld. There is some radial compression stress opposite the weld side. This indicates a certain amount of bending taking place.

Figure 3.10b shows very little axial stress at all. There is a compression region approximately 0.4 mm from the weld axis, and 0.2 mm beneath the top surface. An axial tensile region surrounds the weld region. This tensile stress peak exists mostly in the material which was once melted.

The tangential stress peak in the central axis of Figure 3.10c could have been predicted from the same peak present in the radial stress plot. Along the central axis of an axisymmetric model, radial and tangential stress are, by definition, the same. A conical tangential compression peak exists about the weld, surfacing 0.7 mm from the center axis.

Figure 3.10d shows an almost uniform peak von Mises stress of 275 MPa exists inside a 0.75 mm radial region about the central weld
The actual peak stress is 345 MPa directly below the weld. This stress is below the material's yield strength, 410 MPa, (Figure 2.10a) which means the specimen is in a state of unloading.

Figure 3.10e shows the fracture safety for Kovar over the time of the welding process. A value greater than unity occurs between 5 ms and 6 ms directly beneath the center of the weld. A more defined mesh about the region in question would be required to analyze this phenomena more thoroughly, however, it does suggest that a tendency exists for cracking beneath the weld to take place.

This is interesting considering the cracks present in the weld pictured in Figure 3.5. It has been suggested that the presence of impurities in Kovar, such as gold plating, may increase the likelihood of cracking [24]. Although there was no gold plating on the Kovar specimen pictured in Figure 3.5, there is the presence of the epoxy grating used to measure the thermal strains [7]. This model could be used to investigate the occurrence of cracks in laser welding.

**Chapter 4.2.5 Stainless Steel**

The strain histories displayed in Figures 3.11a through 3.11e show similar results as those outlined in Section 4.2.3 (Figures 3.9a...
through 3.9e), however, there are some interesting differences. To begin with, 304 stainless steel is softer and yields more than Kovar. This causes the amount of total strain to be larger, and increases the diameter of the strain affected zone. Also, the Kovar specimen shows the existence of axial compressive strain surrounding the weld which is not present in the 304 stainless steel specimen.

The residual stress contour plots displayed in Figures 3.12a through 3.12d show similar results as those outlined in Section 4.2.4 (Figures 3.10a through 3.12d), however, there are some interesting differences. The stresses in the steel are less than Kovar due to a lower yield strength (Figures 2.10a and 2.10b). Figure 3.12c shows parallel stress contour lines in the stainless steel, while Figure 3.10c shows conical contour lines in the Kovar, at a position 0.5 mm from the weld axis. This indicates that the steel sample is behaving more like a thin sheet, while the Kovar behaves more like a semi-infinite body. Also, the weld fracture coefficient for the 304 stainless steel stays well below a unity value in Figure 3.12e. This indicates that the 304 stainless steel has a lower tendency to crack than that of Kovar (Figure 3.10e).
Chapter 5: Summary and Conclusions

During the manufacture of lightwave devices, optical coupling tolerances require a stable and precise method of achieving alignment. Laser welding provides such a method without the difficulties associated with soldering and epoxy. It is very critical to reliability that relative motion of lightwave components does not occur during or after laser welding because it may result in a change in light power transmission. Laser weld induced thermal strain may be the cause of light power transmission changes.

A finite element model was developed using ANSYS to simulate the laser welding of a flat plate. Temperature dependent material properties were incorporated in order to accurately model the welded part over the wide temperature range. The effects of the loss of material
due to melting and vaporization was accomplished by excluding the effects of oblated elements from the iterative calculation. Molten elements re-entered the calculation when their temperatures fell below the material's solidus temperature.

It was shown both thermally and mechanically, that the finite element model developed in this study agrees with other theoretical and experimental models. The surface temperature and surface strain are virtually independent of the thickness of the welded piece when the thickness is greater than two times the weld diameter, however, a reduction in thickness in this range does affect the outcome throughout the bulk of the model. Calculations indicate that approximately 50% of the laser energy is absorbed by the work piece during welding. The residual strain effects are contained within a region only three to four times the weld diameter. The finite element model is able to predict strain in locations close to the weld center as well as in the interior of the model where it is very difficult, if not impossible, to measure experimentally. The model is able to calculate strain 0.1 mm closer to the weld center than experimental methods were able to measure, and found the strain in that region to be nearly two times greater than the maximum measured strain. Also, calculations suggest that the material, Kovar, may be stressed beyond its ultimate strength directly below the
welded region, which may result in crack formation. This is not the case for 304 stainless steel, a possible substitute for Kovar in laser weld applications in the photonics industry, however, stainless steel exhibits approximately 20% higher radial strain than Kovar.

It would be beneficial to continue studies to characterize the effects of individual material properties on weld formation. This model could accomplish this relatively easily by changing the input to one individual material property at a time (i.e. elastic modulus) and noting changes in the program’s output (i.e. maximum radial strain). This type of testing could be used to determine the model’s sensitivity to the error in the input material properties outlined in Chapter two, thus determining which material properties need to be modeled accurately, and which do not. This could possibly reduce the computer resources required to run this model. This model could also be very useful in solving the weld cracking problems in Kovar. Since the weld fracture coefficient defined in Section 3.2.4 is maximum immediately after the weld pulse shuts down (Figure 3.10e), perhaps modifying the tail end of the weld pulse (Figure 2.3) could reduce the weld’s tendency to crack. The finite element model developed in this study provides a tool for understanding material and geometric influences on thermal strain in a
pulsed laser welding operation, as well as estimating many of the characteristics of the resulting welded part.
VITA

Fred W. Warning Jr. was born on July 9, 1964 in Bethlehem, Pennsylvania. He completed his Bachelor of Science degree in Mechanical Engineering from The Pennsylvania State University in 1986. He is currently employed as a mechanical design and analysis engineer for AT&T Bell Laboratories in Breinigsville, Pennsylvania.
REFERENCES


END OF TITLE