SAND--98-8455C
CONF-9707100--

FINITE ELEMENT MODELING OF ULTRASONIC WAVES PRODUCED BY A PULSED LASER

J. J. Dike
Solid & Material Mechanics Department
Sandia National Laboratories
Livermore, CA 94551-0969

RECEIVED
MAR 27 1998
OSTI

INTRODUCTION

As part of an effort to apply laser ultrasonics to stress evaluation, sequential thermal and mechanical finite element analyses were used to simulate heating a region of an aluminum surface by a laser pulse and the stress waves that result. As residual or applied stresses can be related to changes in wave velocities, time-of-flight measurements may be used to determine the stresses. The goal of the effort is to improve time-of-flight measurements, and therefore resolution of the calculated stresses, using calculated waveform shapes in model-based signal processing techniques [1]. Detailed finite element simulations of laser ultrasonics may also be used to aid development of techniques that can generate narrow band ultrasound. Because penetration of Rayleigh waves is frequency dependent, they can be used to obtain information about gradients near a surface. If the frequency of the laser generated Rayleigh waves can be controlled, laser ultrasound becomes a more useful tool for examining gradients in material properties or stresses at the surface of a part.

Presented here are some preliminary finite element simulations of laser generation of ultrasound waves. Techniques for using commercial finite element codes are discussed and calculated displacement histories are presented for epicentral and same-surface locations. These displacement histories are compared with results from the literature.

PREVIOUS WORK

DTIC QUALITY INSPECTED 2

Brief literature reviews of laser ultrasonics modeling are presented in [2, 3]. Much work has focused on representing the laser source by an equivalent mechanical loading and then using elasticity theory to predict the resulting waveforms. The problem has also been formulated in terms of fully coupled thermoelasticity. Generalized theories of thermoelasticity have been formulated to model problems where very short duration and high heat fluxes are present [e.g. 4]. These formulations differ from those of classical thermoelasticity by introducing one or more relaxation time constants to account for thermal diffusion into the material. Semi-analytical approaches have been used to calculate waveforms where various transforms are applied to the equations involved and then the

19980507 088
MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

inverse transforms are solved numerically. These methods are limited in the geometries and boundary conditions to which they may be applied.

The finite element method is useful for obtaining full field solutions for arbitrary geometries, boundary conditions, and general material models. Finite element models have already been used to examine some aspects of generalized thermoelasticity [3, 5, 6]. In [3], the generalized equations of thermoelasticity, as formulated by Green and Lindsey [4], are solved using finite element techniques. Good agreement was obtained overall between waveforms for several plate thicknesses, laser source energy densities, and excitation spot sizes.

MODELING TECHNIQUE

The work presented in this paper is a direct application of uncoupled thermal and mechanical finite element analyses. A thermal analysis using a heat flux with Gaussian variations in time and space is used to simulate heating of an aluminum surface due to a laser pulse. The region modeled is large enough such that the boundaries away from the heat source do not change temperature during the time span of interest. This model is small in spatial extent compared to the mechanical model. Two types of models have been developed. One uses all solid elements, while the other uses a combination of shell and solid elements. For the case when the models are constructed only of solid elements, the thermal and mechanical meshes are different, even in the regions where the two overlap, so the temperatures must be mapped from the mesh used in the thermal analysis to the mesh used in the mechanical analysis. Mechanical analyses are performed using explicit dynamic analysis codes [7, 8]. For mechanical analyses with both shell and solid elements, the thermal analyses are performed using only shell elements. When mechanical analyses combine shell and solid elements, there is a one-to-one correspondence of shell elements and so mapping is not necessary. All analyses discussed here were 2D axisymmetric, but it is straightforward to extend the technique to 3D.

Thermal Analyses

Thermal analysis procedures were first verified using theoretically predicted temperature histories and distributions for a continuous wave laser heating a semi-infinite body [9]. A thermal analysis was performed using a Sandia-developed finite element code for multidimensional nonlinear heat conduction analyses [10]. The finite element mesh was constructed of four-node quadrilateral elements. A spatially Gaussian surface flux was used to model heat input by the laser. The finite element mesh for the 2D axisymmetric problem had many elements along the radius of the laser source in order to accurately capture the spatial variation in the laser source's energy deposition. Steady state and transient finite element results were nearly identical to calculations using the analytical model.

Thermal analyses of a pulsed laser heating a surface of silver were then performed and the results compared with experimental data from [11]. In this experiment surface temperature changes due to the application of a 1068 nm laser with 1.8 mm diameter source spot size, 16 ns pulse duration, and 0.23 mJ energy deposition were measured. When reporting the data, an analytical solution was also presented for the temperature history at the center of the laser source. The model assumed a pulse uniform in space (radially) and triangular in time. Finite element simulations were performed using two laser pulse time histories. Both were Gaussian in space but one history was Gaussian in

Figure 1. Comparison of temperature histories on silver surface at center of laser source. Data and analytical solution from [11].

time while the other history had a triangular pulse as a function of time. The finite element results compared quite well with the experimental and analytical results reported in [11] and are shown in Figure 1. The Gaussian pulse shape in time gives a peak temperature change approximately 10% higher than a pulse in which intensity does not vary with radius. The peak temperature change is slightly higher for the pulse Gaussian in time and space than for the pulse triangular in time and Gaussian in space.

When thermal analyses were performed using solid element models, very small elements were used near the surface in order to capture the thermal gradients early in time. Element dimensions were typically on the order of 10^{-4} mm near the surface in the region of the laser source. The domain of the meshes extended to 4 mm radially and 0.5 mm through the thickness. These distances were larger than the distance heat could diffuse in the maximum 6 microsecond duration of any of the analyses. The solid element models consisted of as many as 50,000 elements.

Shell element meshes for the thermal analyses consisted of a line of axisymmetric shell elements extending to a radius of 4 mm. Elements were 0.05 mm long to match

Figure 2. Shaded domain in upper left shows extent of thermal model relative to size of mechanical model for solid element models. Thermal region is 4 mm x 0.5 mm while mechanical region is 18 mm x 5 mm. Finite element analyses were 2D axisymmetric.

elements used in the mechanical analyses. Shell elements were specified to have a thickness of 0.001 mm to 0.05 mm with 15 evenly spaced temperature points through the shell thickness. These meshes consisted of 80 shell elements. The analysis code ABAQUS/Standard [8] was used with element type DSAX1.

Time steps were kept on the order of 0.1 ns during the duration of the pulse. Time steps were allowed to increase automatically after termination of the pulse. Thermal analyses were run to several microseconds to provide temperature histories for the full duration of the mechanical analyses.

Thermal analyses used temperature independent material properties. Energy fluxes were kept low enough so that melting and ablation would not occur, consistent with the experimental procedure described in [2]. Material properties used were density = 2.77×10^{-3} g/mm³, specific heat = 0.875 J/g-K, and thermal conductivity = 0.177 W/mm-K. Figure 2 shows the size of the thermal and mechanical domains used in generating the waveforms from meshes constructed only of solid elements.

Mechanical Analyses

When solid element thermal models were used, temperatures from the thermal analyses were mapped onto the mesh used for the mechanical analyses. Elements in the mechanical mesh that are outside of the domain of the thermal model were kept at constant temperature during the duration of the mechanical analysis. Thermal and mechanical meshes were different, with elements in the thermal model being much smaller than those in the mechanical model near the surface. Meshes for the mechanical analyses were typically meshed with uniformly sized square elements throughout the mesh. Element sizes were selected by requiring several elements over the wavelength of the highest frequency wave component thought to be important in the model. Typically, mechanical meshes consisted of uniformly sized square elements to avoid causing inadvertent reflections due to poorly shaped elements and regions of the mesh where elements do not smoothly transition in size. The uniform size and shape of the elements also reduces inadvertent filtering of waves that could occur when waves travel from finely to coarsely meshed regions. For a region 18 mm x 5 mm, 36,000 0.05 mm square elements were required. Although the finite element mesh used in the mechanical analyses was able to capture waves of relatively high frequency and had many elements within the laser source radius, it was too coarse to accurately capture the temperature gradient at the surface. In the thermal analysis, heat diffused to approximately 10^{-3} mm by 4 ns and approximately 0.07 mm by 6 microseconds in 2024 aluminum for a 4 ns duration, 2.25 mm diameter pulse. In the mechanical mesh

Figure 3. Schematic illustrating concept of shell and solid model. Temperature gradient is captured within the shell elements.

just described, the temperature gradient is always spread over at least one 0.05 mm element. To better capture the thermal gradient but still keep the model size relatively small, the temperatures from the thermal analyses were applied to a layer of shell elements which were attached to a solid element model of the plate. Figure 3 shows an example of this arrangement.

The commercial finite element analysis code ABAQUS [8] was used to experiment with the shell/solid element model. Shell elements were tied to the base structure using contact surfaces, while the base structure was discretized using solid elements. The user may specify temperatures at a number of equally spaced points through the thickness at each node of the shell element. Heat transfer as well as thermal stress analysis problems may be modeled this way, with temperature histories from the thermal analysis used as the loading in the mechanical analyses. Thermal analyses required much less computer time when shell models were used than solid element models of the same domain. In addition, no mapping of the temperatures between different thermal and mechanical meshes was required.

Comparison of Calculated Surface Waveforms

Waveforms on the surface upon which the laser is incident were calculated for one laser source diameter (2.25 mm) and two plate thicknesses (5 mm and 25 mm) for models constructed of only solid elements. The waveform for a 5 mm thick plate was also computed using the shell/solid model. For both solid and shell/solid models, temperature histories were calculated for 6 microseconds. For the solid models the temperatures were mapped to the mechanical mesh consisting of 36,000 0.05 mm x 0.05 mm elements. The nodal displacements approximately 9 mm from the center of the laser source are compared with results from Spicer [2]. Spicer formulated fully coupled equations of classical and temperature rate dependent thermoelasticity. The laser source was represented by an equivalent elastic boundary source to predict displacements in infinite isotropic plates of finite thickness. Displacements are calculated using Laplace-Hankel transform solutions

Figure 4. Comparison of surface waveforms as calculated by Spicer [2] and finite element models. Waveforms are for point on same surface as laser source, 9 mm from source center for 5 mm thick 2024-T6 aluminum plate for a 2.25 mm diameter source.

Figure 5. Comparison of surface waveforms as calculated by Spicer [2] and solid element finite element model. Waveforms are for point on same surface as laser source, 9 mm from source center for 25 mm thick 2024-T6 aluminum plate for a 2.25 mm diameter laser source.

inverted numerically. Figures 4 and 5 compare the surface waveforms calculated using the finite element models described here with Spicer's calculated waveforms for 5 mm and 25 mm thick plates, respectively. The curves representing Spicer's calculations were digitized from figures in the literature and so are somewhat less smooth than the curves he reported.

The mechanical finite element analyses for solid element models represented in Figures 4 and 5 were performed using a Sandia-developed explicit transient dynamics code [7]. Material properties were Young's modulus = 70 GPa, Poisson's ratio = .3, and coefficient of linear thermal expansion = $23 \times 10^{-6} \text{ C}^{-1}$. As the analysis was elastic and temperature independent properties were used in both thermal and mechanical analyses, displacements from the mechanical analyses were scaled to match peak displacements from Spicer's data.

The features displayed in Spicer's data, which agreed very well with his experimental measurements, are also exhibited in the finite element displacement curves. In order to remove numerical oscillations in the finite element displacement histories, a small amount of stiffness proportional damping was specified. For the mesh with 0.05 mm square elements, time steps in the mechanical analyses were on the order of 5 ns. It was observed that a stiffness proportional material damping factor (which has units of time) of less than 2.5 ns removed the oscillations from the displacement histories in the ABAQUS analyses. It has not yet been determined how little material damping can be added and still produce acceptable results. Figures 4 and 5 show that the finite element curves are somewhat smoother than the semi-analytical curves. Addition of material damping does remove the unwanted numerical oscillations, but it also tends to smear many of the features. It may possible to reduce stiffness proportional damping and still control the numerical oscillations and minimize smearing of the wave features.

The surface waveforms do not appear to be particularly sensitive to the steepness of the temperature gradient in the material under the laser source. The meshes used to generate the results in Figures 4 and 5 used 0.05 mm square elements. This is quite large com-

pared to the depth of heat penetration during the first 100 ns which is approximately 7 μm . The temperature gradient in the mechanical analyses was much smaller than in the thermal analyses due to the difference in element sizes at the surface. The shell/solid model results shown in Figure 4 were generated using 0.01 mm thick shell elements with 15 temperature points through the thickness, providing a much larger thermal gradient than the solid element models. The shell/solid waveform is in somewhat better agreement with Spicer's results than the solid model waveform.

Epicentral Waveforms

Epicentral waveforms were calculated for plate thicknesses of 10 mm and 25 mm. Two types of mechanical models were considered. In one model, only solid elements were used. Approximate temperature time histories were generated by using the temperature history from the COYOTE code thermal analysis directly at the laser's center and scaling it using a Gaussian profile for all heated nodes on the surface of the model. The Gaussian temperature distribution was specified on the surface such that the temperature decayed to 5% of its peak value within the 4 mm radius of the laser heat source. The remainder of the material was specified to have an initial and constant temperature of 293 K. Figure 6 compares the finite element epicentral results obtained using the approximate temperature histories with curves from Spicer for 5 mm and 10 mm thick plates. The epicentral finite element waveforms show the same features as the semi-analytical waveforms, but some features like the precursor spikes are smoother and the arrival times do not match as well. The finite element results for the 10 mm thick plate shows some overshoot just after the precursor and after the arrival of the Rayleigh wave at ~ 3.3 microseconds. This overshoot can be reduced using stiffness proportional material damping but at the cost of making the precursor less sharp. Epicentral waveforms using a shell and solid element mechanical model in which the shell elements overlay only the first 4 mm radially of the mechanical model lack the precursor spike but otherwise are similar to those of a solid element model. More manipulation of the shell/solid element mechanical model is required to determine the effect of various shell and solid element configurations.

Figure 6. Epicentral waveforms from Spicer [2] and finite element consisting of solid elements. All heated nodes on the surface are scaled by the same amplitude history. A spatially Gaussian temperature profile is specified within the 4 mm radius of the laser source.

SUMMARY AND CONCLUSIONS

Sequentially coupled finite element analyses using standard finite element codes were used to calculate same-surface and epicentral waveforms in 2024-T6 aluminum plates. Two types of finite element models were used. In one model, both thermal and mechanical models were constructed of solid finite elements. In the other model, only axisymmetric shell elements were used in the thermal analysis while the mechanical analysis consisted of axisymmetric shell elements tied to a substrate of solid elements by contact surfaces. The shell/solid model produces results that were, in general, equal in quality to the solid element models but at a significantly lower cost in computing time. However, the precursor spike in the epicentral waveform has not been observed with the shell/solid models examined so far.

The overall goal of the modeling effort is the prediction of waveforms to use in a model based signal processing technique to improve signal to noise ratios and thus the accuracy of time-of-flight measurements. Improved time-of-flight measurements will aid in obtaining more accurate determinations of applied and residual stresses in the structures being examined. Another anticipated use of these modeling techniques is to aid development of techniques to generate narrow band Rayleigh waves. This would allow improved evaluations of gradients in material properties and stresses at the surface of a part.

Future work will involve exercising the modeling techniques outlined here to study additional material thicknesses, temperature dependence of material properties, different materials, and effects of laser source diameter on epicentral and same-surface waveforms along with comparisons with semi-analytical and experimental data.

ACKNOWLEDGMENTS

This work is supported by the U.S. Department of Energy, Sandia National Laboratories under contract DE-AC04-94AL85000 and is part of the Sandia Laboratory Directed Research and Development (LDRD) program.

REFERENCES

1. J. V. Candy, G. H. Thomas, D. Chinn, and J. B. Spicer, "Laser Ultrasonic Signal Processing: A Model Reference Approach", Lawrence Livermore National Laboratory Report UCRL-JC-122414, Oct. 1995.
2. J. B. Spicer, "Laser Ultrasonics in Finite Structures: Comprehensive Modeling with Supporting Experiment", Ph.D. Thesis, The Johns Hopkins University, 1991.
3. C. Suh, "Numerical Modeling of Laser Ultrasonic Waves using Finite Element Method and Fast Integral Wavelet Transform", Ph.D. Thesis, Texas A&M University, 1997.
4. A. E. Green and K. A. Lindsay, *J. Elasticity* 2, 1 (1972).
5. H. J. Prevost and D. Tao, *J. Applied Mechanics* 50, 817 (1983).
6. H. J. Lee and C. P. Burger, *Computers & Structures* 54, 499 (1995).
7. L. M. Taylor and D. P. Flanagan, "PRONTO 2D A Two-Dimensional Transient Solid Dynamics Program", Sandia National Laboratories Report SAND86-0594, 1995.
8. ABAQUS/Explicit and ABAQUS/Standard User's Manuals, Version 5.6, Hibbitt, Karlsson & Sorensen, Inc.
9. G. T. Martin and H. F. Bowman, *Int. Comm. Heat Mass Transfer* 17, 93 (1990).
10. D. K. Gartling and R. E. Hogan, "Coyote II - A Finite Element Computer Program for Nonlinear Heat Conduction Problems, Part II - User's Manual", Sandia National Laboratories Report SAND94-1179, 1994.
11. J. M. Hicks, L. E. Urbach, E. W. Plummer, and H. Dai, *Physical Review Letters* 61(2), 2588 (1988).

M98052517

Report Number (14) SAND--98-8455C
CONF-9707100--

Publ. Date (11) 199803
Sponsor Code (18) DOE/ER, XF
UC Category (19) UC-411, DOE/ER

DOE