Use of Coupled Field FE Modeling in Study of Resistive Heating in a 6061-T6511 Aluminum Specimen

Russell Warley, Amir Khalilollahi, and John Roth
Penn State Erie, The Behrend College

Abstract

The effect of electric current on material mechanical properties has been of interest since it can reduce the mechanical energy associated with cutting/machining a material. An ANSYS model is used to evaluate the distribution of temperature resulting from the dissipation of electrical energy in a cylindrical tensile bar of 6061-T6511 aluminum while the specimen is carrying a large DC current. The simulation results are compared to surface infrared temperature measurements in order to (1) refine and verify the FE model, and (2) offer more qualitative insight into the effects of electric field.

Introduction

When metallic parts are created via deformation-based manufacturing processes, the force required to initiate and maintain the flow of the workpiece controls many aspects of the operation. Since materials flow with greater ease at elevated temperatures, primarily due to the increase in the energy within the material, today’s common practice is to work the material hot in order reduce these forces. However, recent studies have investigated using alternative energy sources in order to provide a comparable increase in the material flow without increasing the workpiece temperature significantly. Along those lines, in 1969, Troitskii found that electric current pulses could be used to temporarily reduce the yield strength in metal. Since then, further research has been conducted to investigate the effect of electricity on material properties. Investigations by Xu et al. demonstrated that continuous current flow can enhance the recrystallization rate and grain size in select materials. Chen et al. have linked electrical flow to the formation and growth of intermetallic compounds, and Conrad reported in several publications that very short duration high density electrical pulses affect the plasticity and phase transformations of metals and ceramics. Based on these studies there is a strong indication that an electric current running through a workpiece may reduce the flow stress of a material (the stress at any given strain required to maintain plastic deformation). This would consequently reduce both the force and energy necessary to deform the part. This effect should be similar to hot working without the drawbacks of elevated temperature methods.

In 2004, as part of the preliminary investigation into the possibility of using an electrical current in this manner, Andrawes et al. reported on the effects of DC current on the stress-strain behavior of aged 6061-T6511 aluminum. From this study, it was found that the current flow brought about significant reductions in both the force and energy required to perform tensile deformations. These changes in behavior occurred at relatively low workpiece temperatures (i.e., the workpiece temperature stayed well below the recrystallization temperature of the material). Following this investigation, a secondary study was conducted by Heigel et al. to determine whether the effects of the electricity in the 6061-T6511 aluminum samples were due to microstructural changes in the specimens. It was shown in this work that microstructural changes in the material such as recrystallization were not the major contributors to the electrical effects.

In the prior work presented by Andrawes et al., the temperature field in a circular cross section tensile specimen carrying large DC currents was approximated with an axisymmetric heat transfer model with constant material properties and a uniform heat generation rate computed from I^2R. In this study it is desired to more accurately estimate the effects of the resistive heating on the resulting transient temperature field. Therefore, the coupled thermal-electrical field problem was modeled using variable thermal and electrical properties to render the transient temperature profiles of an undeformed specimen. Since a mathematical or FE model can represent the electric heating effects exclusive of the effect of current on the material properties, it is reasonable to generate this type of model in order to compare the results to experimental data under the same conditions.
The ultimate goal of this work is to have a representative model of the thermal effects in the tensile specimen so that the simple effect of temperature can be separated from any mechanical material property changes due to the electric current. In this work we refine the ANSYS FE model and compare the results to experimental data on surface temperatures in a representative test. Comparison with the experimental results on transient temperature profiles of an undeformed specimen can offer confidence in further use of FEA as a significant tool in understanding the electrical effects on material properties. The present model includes the coupled thermal-electrical field problem using variable thermal and electrical properties with consideration of convection/radiation effects for an un-deformed specimen. Favorable comparisons between the FEA and experimental results in this study justify highly the use of ANSYS in completion of this problem.

**Modeling Issues-Material Properties**

The temperature dependent thermal conductivity\(^{10}\), specific heat, thermal expansion coefficient and electrical resistivity\(^{11}\) data of 6061-T6511 aluminum are shown in Figures 1 and 2. The electrical resistivity data were fit with a linear relation vs. temperature over the range 250 – 500 °C and the linear relation was used to extrapolate to lower temperatures based on similar behavior of pure aluminum in the same paper. The linear fit was then scaled to have the typical value of electrical resistivity at room temperature for this heat treat condition (3.66×10^(-6) Ω cm).

![Figure 1. Thermal conductivity and specific heat of 6061-T6511 aluminum](image_url)
To investigate the heating effect of an electron wind on Aluminum 6061-T6511, a current density of 32.4 A/mm² (1600 Amps) was applied to an Aluminum tensile specimen. The specimen was installed in the fixture as if it was going to be uniaxially stretched. However, the specimen was not actually deformed during the test. The current was then applied for 17.5 seconds (the time duration of the tensile tests under normal conditions) and the temperature rise in the material was recorded.

The current was generated by a Lincoln Electric Idealarc R3S welder. To control the current, both the welder's built-in control systems and a variable resistor with cooling were employed (Figure 3).

The test samples used for the experiment were circular cross section tensile specimens with an effective length (between filets) of 78.7 mm (3.1 in) and a diameter of 7.925 mm ± 0.025 mm (0.312 in ± 0.001 in). The specimens were threaded on each end for attachment to the fixture. To isolate the machine and the operator from the electricity, the tensile specimen was threaded into steel fixtures which were insulated with Haysite Reinforced Polyester Thermoset and PVC electrical piping. The Reinforced Polyester was incorporated in areas which encountered significant loading, and was selected for its high strength in compression (the fixture design allowed all of the loading of the plastic to remain compressive). The PVC
piping was only used to insulate the areas of the fixture that would not be loaded during a deformation test (Figure 4).

Measurements of the current, voltage and temperature were recorded at the beginning of the test, at 5, 10, and 17.5 seconds. The current flowing through the system was monitored using an Omega HHM592D Clamp-On ammeter and the voltage drop across the entire system was measured using a BK Survivor 2870 DMM. The temperature of the test specimens was measured in three ways. The first method was based on a ground isolated Omega 871A thermocouple attached to the middle of the tensile specimen. The second temperature verification was achieved using a hand-held non-contact thermal gun (Raynger ST ProPlus). Finally, the temperature was also recorded using a FLIR thermal imaging camera (ThermoVision A20m). To correct for the emissivity of the Aluminum, the specimen was coated black using high temperature paint. The temperatures reached in this work were below the maximum operating temperature of the paint. Therefore, the emissivity of the specimens can be considered to remain relatively stable, at least for the short duration of this experiment. The paint was removed at the location where the thermocouple was attached in order to ensure sufficient thermal contact. All three temperature measurement devices recorded values within a few degrees of each other for the test. The infrared camera provides the most complete transient account of the specimen temperature profile and these results are compared to model predictions.

The finite element model uses mapped axisymmetric thermal/electric PLANE67 elements. Since the fixture (1018 steel $k = 51.9 \text{ W/(mK)}$, $\rho = 7870 \text{ kg/m}^3$, $C_p = 0.486 \text{ J/(g °C)}$) is a significant heat sink with respect to the sample, it is included in the FE model. The symmetrical model includes the top half of the specimen and fixture. Symmetrical radiation with iterative radiosity option and temperature space option were used in the model. The top boundary of the fixture is assumed to be insulated. At $t = 0$ a large current (1600 amps) starts to flow through the specimen, represented by nodal current excitations on the top boundary. The bottom boundary is the symmetry line and is kept at $voltage=0$. The initial condition included $voltage = 0$ and $T = 20 \degree C$ at $t=0$. It was found that the changing of top boundary line from insulated to isothermal, $T = 20 \degree C$, had a negligible effect on the temperature profile in the specimen.

**Results of FE Analysis**

Figure 5 shows the mesh of the entire model including the test fixture.
Figure 5. Finite element mesh of specimen and a portion of the fixture.

Figure 6 shows a contour plot of the temperature field in the half-length axisymmetric model of the specimen at $t = 17.5\ s$. It is observed that there is essentially no temperature gradient in the radial direction in agreement with prior results\(^8\). Comparison of the temperatures at the surface and the centerline of the model yields less than $1\degree C$ difference across the sample. The temperature field in the fixture is uniform except for the region very near the sample attachment point.

Figure 6. Contour plot of temperature in specimen at 17.5 s test duration.
Experimental Results

The result is a 2D coordinate map of the temperature distribution corresponding to the projected area of the specimen that is visible to the camera (Figure 7).

![Figure 7](image1.jpg)

**Figure 7.** 2D map of temperatures at 17.5s. Features of experiment setup are shown as annotations.

The temperature data files were reduced in Matlab to provide a plot of the axial temperature profile along the specimen. Figure 8 shows a plot of temperature along the length of the specimen.

![Figure 8](image2.jpg)

**Figure 8.** Plot of temperature along the length of the specimen at 17.5 s. Colors are the same as in Figure 7.
Figure 9 shows a comparison of the ANSYS model predictions compared to the thermal camera data. The left side of the plot represents the bottom of the effective length of the specimen and the right side represents the top of the specimen. The ANSYS results are symmetric due to the nature of the model and the experimental data show somewhat higher temperatures toward the top. This is almost certainly an artifact of the rising high temperature air causing higher temperature indications as opposed to a real specimen temperature difference due to non-uniform dissipation.

![Figure 9. Comparison of experimental and ANSYS model temperatures for 5 sec, 10 sec, and 17.5 sec of operation.](image)

A plot of the maximum temperature in the specimen as a function of the test duration is shown in Figure 10 along with the temperature rise that would be expected for a thermally insulated specimen. In the case of the experimental data the maximum was found from the center 60% of the specimen since the high temperatures at the upper end of the specimen are not realistic.

![Figure 10. Plot of maximum temperature in specimen vs. time of operation. The solid line represents the maximum possible temperature estimated from Equation 1.](image)
The temperature rise rate for a thermally insulated specimen is computed by assuming uniform dissipation of energy in the circular cross section specimen and perfectly insulated specimen boundaries (Equation 1).

$$\frac{dT}{dt} = \frac{16I^2 \rho_{el}}{\pi^2 D^4 \rho C_p}$$  

As anticipated the adiabatic specimen model is reasonable for short time durations but the error grows as the timescale progresses. In this simple calculation, the electrical dissipation rate is assumed to be constant using the properties at 20°C.

**Conclusion**

The results show that the FE model of the coupled thermal electrical system reasonably represents the temperature profile found experimentally. The temperature gradients are almost entirely axial in nature and for short times of operation the max temperature is nearly equal to the adiabatic sample value. Further work is needed to extend the simulation to the situation where the sample is being deformed axially in tension simultaneous with the development of the temperature field.

**References**

1) Troitskii, O.A., 1969, zhurnal eksperimental'noi teoreticheskoi kiziki/ akademii'i'a nauk sssr - pis'ma v zhurnal.eksperimental' i teoretiheskoi fiziki, 10, pp. 18.


