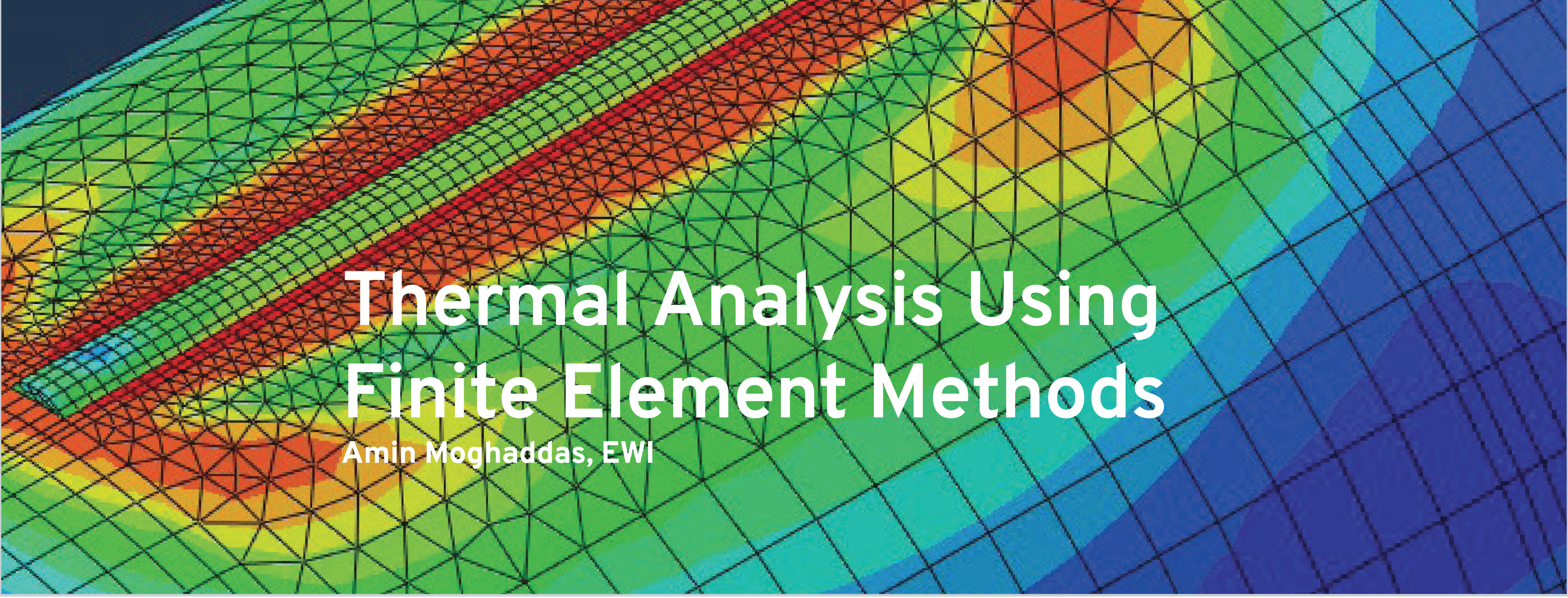


Thermal Analysis Using Finite Element Methods

Amin Moghaddas, EWI

Integrated computational materials engineering (ICME) as applied to welding is a framework combining an understanding materials and processes to predict the performance of created structures. This emerging discipline can accelerate product development and unify design and manufacturing. The basic structure of ICME for joining processes was outlined in a previous paper, *Overview of Integrated Computational Materials Engineering (ICME) Tools Used at EWI.*[1] Elements of this structure include tools for predicting thermal excursions from processing conditions, microstructures from thermal excursions, and properties from the developed microstructures.

As has been the theme throughout EWI's recent ICME paper series,[1-4] a key element in any welding-related ICME construct is predicting thermal response from processing inputs. Available tools for predicting thermal response range from analytical relationships to complex numerical simulations. Note that the selection of an appropriate tool is based on the constraints of the problem and the necessary fidelity of the prediction. As discussed in *Analytical Methods for Predicting Thermal Excursions for Continuous Joining Processes*,[2] closed-form thermal modeling tools are appropriate for localized propagating heat sources at far-field locations in the workpiece.

These approaches can be adopted to a wide range of welding and joining processes where the mechanisms of heat generation are understood. In *AM Applications of Analytical Methods to Predict Thermal Excursions for Continuous Joining Processes*,[3] analytical thermal models based the 3D Rosenthal equation[5] were used for the selection of start and stop parameters during electron-beam directed energy deposition (DED). For this application, the underlying Rosenthal solution was modified through a Gaussian heat-source approximation and applied to predict melt pool temperatures as a function of processing condition.

As described in *Analytical Tools for Assessing Thermal/ Mechanical Response During Single-shot Welding Processes*,[4] when heating is not continuous or propagating (so described single-shot processes), different mathematical constructs are required. For these applications, quasi one-dimensional analytical approaches for single-shot welding processes have proved effective.

It is often the case, however, that joining applications are either too complex, or the fidelity of the solution is too great for the use of such analytical models. In such cases, the use of numerical methods-based solutions may be more appropriate. Advanced computational tools (such *finite element analysis*) can be employed for more robust predictions. Highly flexible both in both sophistication and resolution, these tools can be used to achieve solutions with a range of complexity and allow the inclusion of temperature-dependent material properties, complex geometry, element manipulation for changing mass conditions, etc. They can also be used to assess residual stresses, final component distortion, etc. This paper provides both a description of finite element methods and how EWI uses this tool to predict the thermal excursions to address two specific applications: 1) Battery module electrical interconnects, a wire's fusing potential based on material, diameter, and length, and 2) prediction of residual stress and distortion in arc welding processes.

What is Finite Element Analysis (FEA)?

FEA is the process of discretizing complicated problems, both in the geometry and time domains, allowing local simplification of the governing equations defining the process response of interest. Generally, finite element methods provide solutions to the various governing equations describing the physics for space- and time-dependent problems. This methodology allows problems to be addressed that cannot be solved with analytical methods. Of note, FEA methods have been highly effective for simulating physical phenomena under given conditions, thereby reducing the need for physical prototypes.

The basis of finite element methods is the discretization of complex structures into smaller elements as shown in Figure 1. Then the elements assemble at the nodes and the contacts, initial and boundary conditions between the different components of the model are applied to form a system of governing equations. These unique sets of equations are then solved at each of these points by iterating the solution away from the boundary system, creating a solution addressing the desired response (e.g., temperature) for the entire domain. Note that since the discretization mesh is dense, the combined model is enough to accurately simulate the reality.

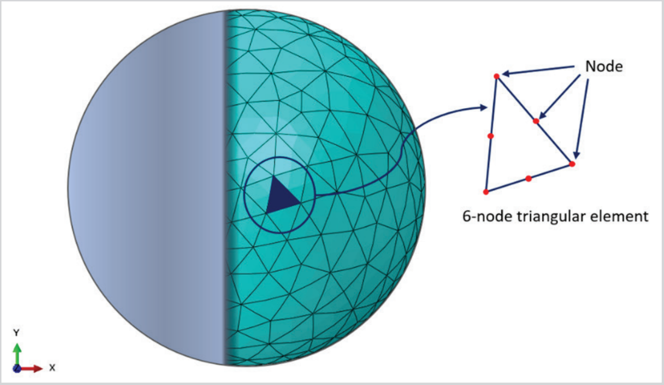


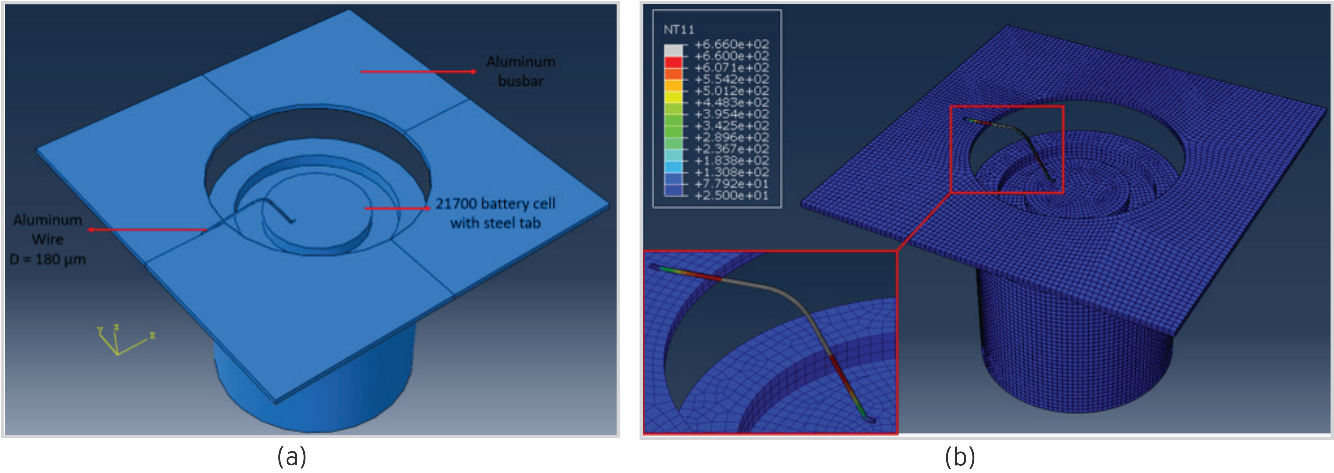
Figure 1. Triangular finite element discretization on the surface of a complex body

FEA is a powerful tool for examining heat transfer problems because it allows for the simulation of complex systems and geometries with relatively low investment. Therefore, in complicated problems, there is no need to sacrifice the accuracy of the results by oversimplifying the problems, as might be the case with analytical approaches. By creating such a detailed 3D model of the system, FEA can predict heat flow through the system, estimating responses such as thermal excursions, stress, and deformation. Note that prediction of some of these responses (such as stress) are impossible or very expensive using other approaches. Furthermore, user-subroutines such DFLUX can be employed by FEA software to add more flexibility and capability.

Application of FEA Techniques for Predicting Fusing in Interconnections in Battery Modules

As an effective example of applying FEA methods, consider the engineering associated with interconnects for electric vehicles and grid storage system lithium battery packs. Such structures are key to achieving high-density electrical storage. For applications including multiple cells, interconnects are often made using aluminum wire which is lighter and less expensive than copper. To further improve both manufacturing efficiency and safety, there is interest in designing interconnects to also function as fuses, which can prevent current overload of the individual cells and potential thermal runaway. Fusing behavior is described as the locus of currents and times that the wire melts and effectively terminates current flow protecting the battery. This fusing characteristic is typically defined by the length and diameter of the wire.

Finite element (FE) methods are ideal for designing interconnect geometries to achieve specific fusing behavior. To this end, EWI developed an FEA coupled thermal-electrical model specifically for interconnect wire design. This tool saves a significant amount of time and cost over the trial-and-error approach of lab testing. The physical layout for the model is provided in Figure 2a. This includes a representative battery cell, an aluminum busbar, and a fine aluminum interconnect wire. For this application, inputs to the model included the diameter and length of the wire and physical properties of the aluminum alloy used. Boundary conditions included the initial temperature, the attachments to the base plate and cell, along with conductive effects off the surface of the wire. The desired output of the model was to be able to predict fusing behavior for representative wire geometries. Some sample results are shown in Figure 2b. Note that temperatures are largely uniform over the length of the wire with steep gradients near each attachment point. As shown in Figure 2b, the aluminum wire can get to its melting point (i.e., 660°C) and fuse after passing 250 A for 14 milliseconds using the wire diameter and length designed in this model. This sort of FE modeling can help engineers define interconnect materials and geometries to achieve differing levels of protection for production battery units.



(a) (b)

Figure 2. a) 3D model of the 21700-battery cell connected to aluminum busbar using 180 µm diameter aluminum wire; b) FEA results – wire temperature when 250 A is passed through the 180 µm aluminum wire.

Assessment of Thermal Excursions, Distortion and Residual Stress in Different Welding Processes

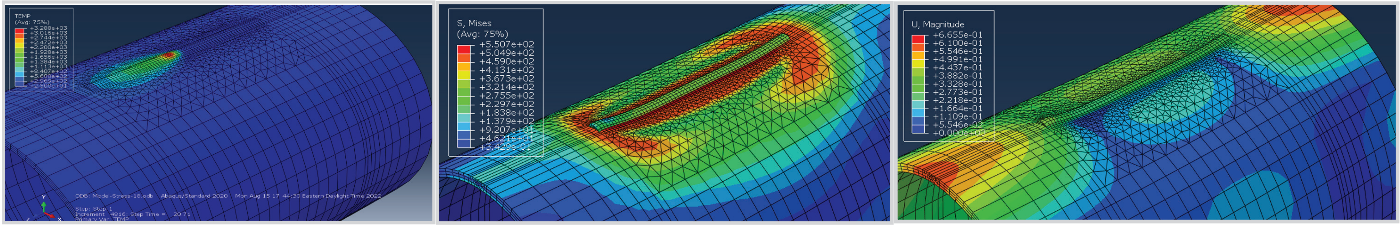
Another well-established application for finite element tools is the prediction of distortion resulting from fusion welding. Fusion welding processes (GMAW, SAW, GTAW, laser, etc.) are commonly used across the industrial spectrum for the joining metals. The thermal cycles associated with fusion welding processes have the potential to create residual stresses and distortion in the weld metal and heat-affected zone (HAZ). Residual stresses and associated distortion are due to uneven heating and weld pool formation that are inherent for such continuous processes. Residual stresses are known to have negative impacts on mechanical performance.[6,7,8] These stresses can result in fusion zone cracking and adversely impact fatigue life. Furthermore, such stresses at a weld joint can also increase hydrogen diffusion rate[9] and contribute to corrosion cracking.[10]

EWI has extensive expertise in applying FEA analysis for the purpose of estimating residual stress. An example is shown in Figure 3. Here, EWI developed a sequentially coupled thermal-mechanical model to predict temperature excursions and residual stresses during gas metal arc welding a short length section on an X52 pipe steel. The surface thermal profile (Figure 3a) demonstrates a typical elliptical shape. The stress analysis (Figure 3b) shows relatively low levels in the weld itself, but high in the heat affected zone at the ends of the length of the weld. Resulting distortion (Figure 3c) appears asymmetric about the weld line, and to achieve maximum values at the free pipe ends. The created model then can be used to achieve processing compromises between weld penetration, residual stresses, and resulting physical distortion.

Summary

FEA offers a powerful tool for analyzing heat transfer problems of complex systems and geometries without sacrificing the accuracy of the results by simplifying the problems to solve them with analytical approaches. It can be used for the modeling and optimization of transient heat transfer and cooling process in the applications where the temperature of a system varies over time. FEA methodologies are part of a suite of analysis approaches contained within the EWI ICME toolbox. We employ these tools for root-cause analysis, process optimization, and to reduce the development time and costs of specific products. These tools are part of a package allowing EWI to provide customer solutions that balance data resolution and integrity with levels of effort. In this way, EWI offers a range of applicable solutions that allow customers to obtain necessary results in the shortest possible time and the lowest cost.

To learn more about FEA and EWI's ICME toolbox, contact Amin Moghaddas at **amoghaddas@ewi.org**.



(a) (b) (c)

Figure 3. FEA results of GMAW Related Distortion on an X52 pipe steel; a) Thermal history (°C), b) Von-Mises stress (MPa), c) Overall distortion.

Note: ***Note:*** *Any reference to specific equipment and/or materials is for informational purposes only. Any reference made to a specific product does not constitute or imply an endorsement by EWI of the product or its producer or provider.*

Reference

1 Gould J. (2023). Overview of Integrated Computational Materials Engineering (ICME) Tools Used at EWI. EWI. https://ewi.org/overview-of-icme-tools-used-at-ewi/

2 Gould J. (2023). Analytical Methods for Predicting Thermal Excursions for Continuous Joining Processes. EWI. https://ewi.org/understanding-analytical-modeling-of-thermal-excursions-for-continuous-joining-processes/

3 Kitt A. (2023). AM Applications of Analytical Methods to Predict Thermal Excursions for Continuous Joining Processes. EWI. https://ewi.org/icme-modeling-in-additive-manufacturing-a-case-study/

4 Gould J. (2023). Analytical Tools for Assessing Thermal/Mechanical Response During Single-Shot Welding Processes. EWI. https://ewi.org/using-icme-to-predict-thermal-response-in-single-shot-welding-processes/

5 Rosenthal, D. (1941). Mathematical theory of heat distribution during welding and cutting. *Welding Journal Research Supplement*, 20(5), 220s-234s.

6 Webster G.A., Wimpory R.C. (2001). Non-destructive measurement of residual stress by neutron diffraction. *Journal of Material Process Technology* (117), 395–399. https://doi.org/10.1016/S0924-0136(01)00802-0

7 Rong Y., Zhang G., Huang Y. (2016). Study of welding distortion and residual stress considering nonlinear yield stress curves and multi-constraint equations. *Journal of Materials Engineering and Performance* 25, 4484–4494. https://doi.org/10.1007/s11665-016-2259-1

8 Diogo F., Almeida R.F., Martins J.B. Cardoso. (2017). Numerical simulation of residual stresses induced by TIG butt-welding of a thin plate made of AISI 316L stainless steel. *Procedia Structural Integrity*, 5, 633–639.

9 Mohr W., Moghaddas A. (2022). Modeling the Effect of Hydrogen Gas on Steel Pipeline Welds. EWI. https://ewi.org/modeling-the-effect-of-hydrogen-gas-on-steel-pipeline-welds/

10 Yupiter H.P., Manurung A., Robert N.L., Rahim M.R., Zakaria M.Y., Redza M.R., Sulaiman M.S., Ghalib T.S., Abas K. (2013). Welding distortion analysis of multipass joint combination with different sequences using 3D FEM and experiment. *International Journal of Pressure Vessels and Piping*, (111-112), 89–98. https://doi.org/10.1016/j.ijpvp.2013.05.002

Amin Moghaddas, *Project Engineer, is part of EWI's resistance and solid-state welding group where he works on both customer projects and internal research related to ultrasonic-assisted manufacturing processes. His is an expert in vibration analysis using Finite Element Analysis (FEA) to design and fabricate tooling for high power ultrasonic systems. Amin is also experienced in FE modeling of thermal related processes to predict temperature, residual stress, and distortion. He has applied his knowledge to model static and dynamic tests needed to develop spot-weld and arc-weld failure criteria for use in automotive crash analysis.*