

Induction Heating Applications



Induction heating uses alternating current in driver coils to induce currents in an object and create heat through precise placement and transient excitation pattern. The thermal and electrical properties are locally functions of temperature, creating a transient electrothermal response. Induction heating is used in heat treatment of materials, seals, and joints and in many other applications.

Keywords

Induced currents, induction heating, eddy currents, Joule losses

Products Used

ANSYS® Maxwell® 15.0, ANSYS Workbench™, ANSYS thermal products

Description

Whether desired or undesired, currents flowing in solid conductors create Joule losses that contribute heat to a thermal system. Concurrently, the electrical and thermal properties of materials often are temperature dependent, which affects current distribution and heat flow. These loss and temperature variations can be monitored and controlled through proper engineering and simulated within ANSYS software.

Typical induction heating applications involve a drive coil with high currents at some kilohertz (kHz) frequency and a conductive target object placed inside or near the drive coil to induce currents and create heat (Figure 1). For high-power induction heaters, the drive coils are hollow with liquid cooling. Additional thermal/structural parts are used to support the heated object and to properly direct heat flow and duration. The target object takes the form of either the object of desired temperature increase or a plate on which the heat-treated object sits (such as with a hot plate or induction cooking).

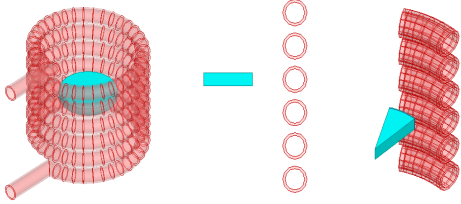


Figure 1. (left to right) Full 3-D, axisymmetric, and 1/12 wedge models for simple induction heating example with hollow, liquid-cooled drive coil and a conducting target that experiences eddy currents and Joule heating. This is an overly simplified model in which the target is thermally insulated except for convective heat transfer on all outer surface.

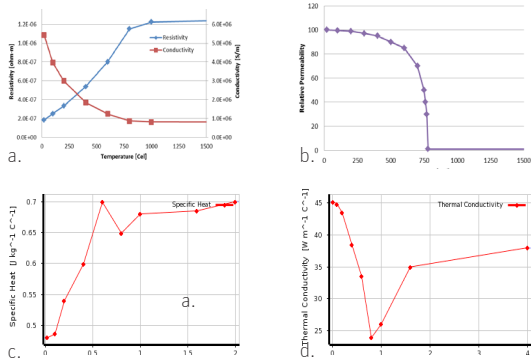


Figure 2. Temperature-dependent data for target object that experiences heating: a) electrical conductivity/resistivity, b) relative permeability, c) specific heat, and d) thermal conductivity. Such temperature dependencies make this a very complicated nonlinear analysis, which is easily accomplished through simulation.

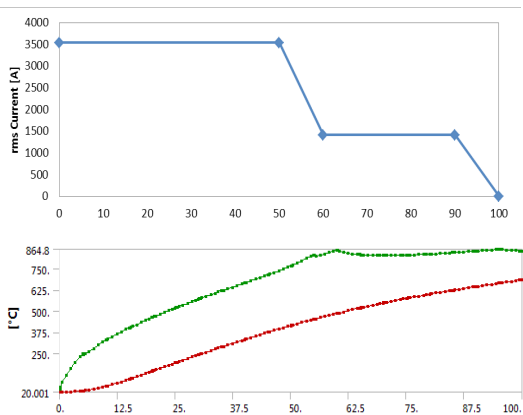


Figure 3. Transient input and output data for combined electrothermal simulation. Upper plot shows transient rms amplitude of input current through AC drive coil, which has been tapered to keep temperature constant in transient profile; lower plot provides transient results of minimum and maximum temperatures on target object.

The kHz region of frequencies is a very interesting simulation space. The kHz electrical time scale is measured in fractions of a millisecond, while the signal amplitude is often varied in thermal and structural responses with time scales that are measured in seconds and greater. You can easily separate the electrical and thermal simulations, enabling use of a high-accuracy AC simulation of the electromagnetic system with updated temperatures from the transient thermal simulation. This produces an efficient and accurate electrothermal response for this class of problems.

The ANSYS Maxwell eddy current solver is used for the AC simulation in the kHz region to resolve skin depths within objects. The electromagnetic properties in Maxwell can be defined as a function of temperature (Figure 2), for which the temperature can be defined either per object in Maxwell or mapped directly from ANSYS thermal tools. Maxwell’s automatic adaptive meshing allows efficient and accurate solution of losses due to induced currents for arbitrary geometries. The automated mesh process assures that the loss calculation is accurate for arbitrary geometry changes; it robustly captures the skin depth of induced currents, which would not be possible with manual mesh generation.

The ANSYS thermal solver is used for either static or transient thermal analysis. The average AC losses from Maxwell are mapped as a spatially distributed load directly to either the static or transient thermal solver. The thermal simulation provides a solution to the nonlinear heat flow response based off the Joule heat losses and any additional loads and boundary conditions. The transient electrothermal simulation is accomplished with great flexibility through robust scripting within the ANSYS Workbench framework, allowing time varying inputs and boundary conditions. Example results are provided in Figures 3 and 4.

Working within Workbench allows coupling of solution data from the respective high-fidelity simulations, but it also defines a simplified workflow and expanded capabilities. The two simulation tools are linked within Workbench by definition of the project schematic (Figure 5), and the geometry and solutions are transferred through this link. The geometry is shared in such a way that only the necessary pieces are used in either the electromagnetic or thermal simulations, but all the geometry is transferred, available, and updated whenever geometry changes are applied. This allows easy geometry variation and design changes; it also makes available the geometry handling capabilities of ANSYS tools. Further, a structural simulation can be connected directly to the thermal simulation for thermally induced stress analysis and fatigue due to thermal cycling.

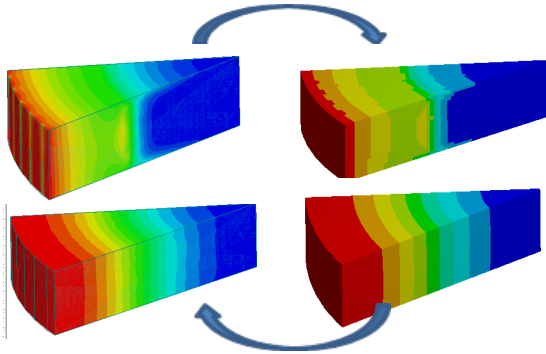


Figure 4. Finite element simulations for AC electromagnetics and transient thermal response produce distributed losses and temperatures – data that is shared between the two simulation tools. The distributed electromagnetic loss calculated in Maxwell is transferred as an imported load to ANSYS thermal tools, which calculate the temperature and map the distributed temperature data back to Maxwell. The visualizations allow rapid identification of heat flow paths and local hot spots.

Conclusion

ANSYS provides accurate, flexible, and robust solutions for electromagnetic and thermal simulations, brought together within the ANSYS Workbench framework.

Authors

Paul Larsen, paul.larsen@ansys.com
Tomoya Horiuchi, tomoya.horiuchi@ansys.com

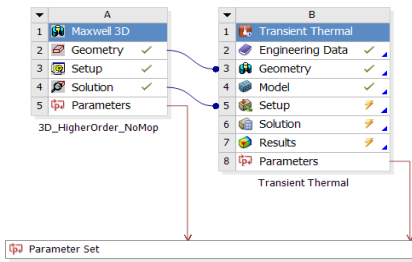


Figure 5. ANSYS Workbench enables transfer of geometry and solution data. This allows mapping of losses from an electromagnetic simulation to a static and/or transient thermal simulation along with feedback mapping of temperature for electromagnetic properties.

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.

724.746.3304
ansysinfo@ansys.com

© 2013 ANSYS, Inc. All Rights Reserved.

ANSYS, Inc. is one of the world’s leading engineering simulation software providers. Its technology has enabled customers to predict with accuracy that their product designs will thrive in the real world. The company offers a common platform of fully integrated multiphysics software tools designed to optimize product development processes for a wide range of industries, including aerospace, automotive, civil engineering, consumer products, chemical process, electronics, environmental, healthcare, marine, power, sports and others. Applied to design concept, final-stage testing, validation and trouble-shooting existing designs, software from ANSYS can significantly speed design and development times, reduce costs, and provide insight and understanding into product and process performance. Visit www.ansys.com for more information.