COUPLED ELECTROMAGNETIC AND HEAT TRANSFER SIMULATION FOR MODELING INDUCTION HEATING

A. Kürkchübasche[†], K. Gundu[†], V. Belsky[†], Peter Hammes[*], Dmytro Farina[*], Frank Weiand[*]

† Dassault Systemes Simulia Corp.
1301 Atwood Ave., Suite 101W, Johnston, RI 02919, US
Email: Albert.KURKCHUBASCHE@3ds.com
Krishna.GUNDU@3ds.com
Vladimir.BELSKY@3ds.com

[*]CST – Computer Simulation Technologies
Bad Nauheimer Str. 19, 64289 Darmstadt, Germany
Email: Peter.HAMMES@cst.com
Dmytro.FARINA@cst.com
Frank.WEIAND@cst.com

KEYWORDS

Induction Heating, Co-Simulation, Multiphysics, Electromagnetic, Heat Transfer, Finite Elements

ABSTRACT

Induction heating is an important heat treatment process utilized in many industrial applications including welding, heat treatment of metals such as hardening and annealing, aseptic packaging of food and pharmaceutical products, and can also be found in appliances such as induction cookers and laser printers. In the paper we simulate the physics of induction heating which can be summarized as follows: an alternating current in an electromagnet generates a rapid varying magnetic field; this magnetic field penetrates a conductive body placed next to it, thus generating eddy currents within the body. The currents flowing through the body release heat due its resistance. The key advantage is that the heat is generated within the conductive body, and

does not rely on an external heating element. The heating process is clean, efficient and can be applied in a controllable manner.

Simulation of induction heating processes requires the ability to simultaneously model multiple coupled physics. To accurately model the induction heating process one needs to account for the temperature dependent material parameters such as the mechanical properties (e.g. density, thermal expansion, modulus), the thermal properties (e.g. conductivity, specific heat, expansion coefficient) and electrical properties (e.g. resistivity, permeability). We employ the Co-Simulation Services of the 3DEXPERIENCE® Platform between an electromagnetic solver and a heat transfer solver. The electromagnetic solver simulates the full coupling between time-varying electric and magnetic fields governed by Maxwell's equations, and the heat transfer solver models the thermal energy due to conduction, convection and/or radiation. Due to the associated time-scales being very different the co-simulation capability allows coupling of a steady-state time harmonic electromagnetic analysis with a transient thermal analysis.

We present two examples; the first example, an Abaqus-to-CST Studio co-simulation, illustrating the metal hardening process of a work piece shown in Figure 1. We discuss the handling of the different time scales, and proper mesh refinements to accurately capture the critical skin effects.

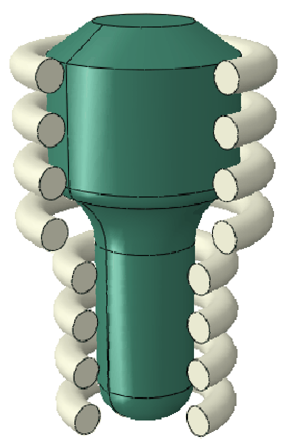

Figure 1: Work coil and work piece for metal hardening example

The second example is a model of an induction heating fuser (shown in Figure 2) such as used by laser printers. The industry employs induction heating fusers to reduce warm-up time and power consumption. The co-simulation is performed between two Abaqus models (electromagnetic and a heat transfer simulations). The Abaqus heat transfer model uses convection-diffusion heat transfer elements to account for the motion of the roller.

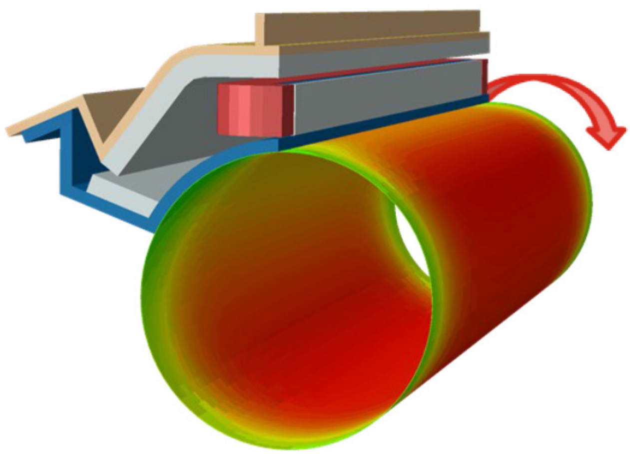

Figure 2: Temperature contour of induction heated fuser