

Induction Heating Process Design Using Comsol[®] Multiphysics Software Version 4.2a

K. Djellabi, M. E. H. Latreche

Abstract—Induction heating computer simulation is a powerful tool for process design and optimization, induction coil design, equipment selection, as well as education and business presentations. The authors share their vast experience in the practical use of computer simulation for different induction heating and heat treating processes. In this paper treated with mathematical modeling and numerical simulation of induction heating furnaces with axisymmetric geometries for the numerical solution, we propose finite element methods combined with boundary (FEM) for the electromagnetic model using COMSOL[®] Multiphysics Software. Some numerical results for an industrial furnace are shown with high frequency.

Keywords—Numerical methods, Induction furnaces, Induction Heating, Finite element method, Comsol Multiphysics software.

I. INTRODUCTION

IN the past years, induction heating techniques have been widely applied in the metallurgical and semiconductor industry. Induction heating is a method by which electrically conducting materials (generally metals) are heated by a non-contact method in an alternating electromagnetic field.

An induction heating system consists basically of one or several inductors and metallic workpieces to be heated. The inductors are supplied with alternating current which induces eddy currents inside the component being heated due to Faraday's law. This technique is widely used in the metallurgical industry in an important number of applications such as metal smelting, preheating for operations of welding.

In general, those process needing a high speed of heating in located zones of a piece of a conductive material. The overall process is very complex and involves different physical phenomena: electromagnetic, thermal with phase change. Induction heating is a non-contact heating process. It uses high frequency electricity to heat materials that are electrically conductive [1]. Since it is non-contact, the heating process does not contaminate the material being heated.

It is also very efficient since the heat is actually generated inside the work piece. This can be contrasted with other heating methods where heat is generated in a flame or heating element, which is then applied to the work piece.

A source of high frequency electricity is used to drive a large alternating current through a coil [3]. This coil is known

as the work coil. The passage of current through this coil generates a very intense and rapidly changing magnetic field in the space within the work coil. The work piece to be heated is placed within this intense alternating magnetic field [7], [3].

The alternating magnetic field induces a current flow in the conductive work piece. The arrangement of the work coil and the work piece can be thought of as an electrical transformer. The work coil is like the primary where electrical energy is fed in, and the work piece is like a single turn secondary that is short-circuited. This causes tremendous currents to flow through the work piece. These are known as eddy currents.

This work concerns an induction furnace, according to this reasons, the design of induction heating is simulated and analyzed using COMSOL[®] Multiphysics software that have a graphical and interactive simulation [2]-[5]. In this paper, the properties of the induction heating are considered such as the effect of frequency.

II. SYSTEM DESCRIPTION

This work concerns an induction furnace like the one presented in Fig. 1. It consists of a cylindrical vessel (usually called the crucible) made from a material, surrounded by an inductor coil made of a very conductive material (copper, for instance). The main idea of the process is quite simple: the coil is supplied with alternating current that produces a rapidly oscillating magnetic field which, in its turn, induces eddy currents. These eddy currents, due to the Joule effect, cause heat losses and consequently the electrically conducting materials of the work piece are heated.

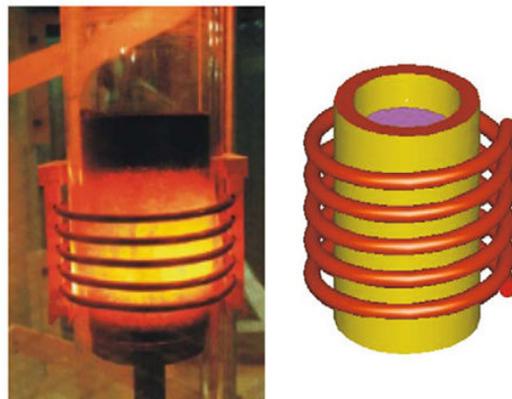


Fig. 1 Induction furnace device study

K. Djellabi is graduate student electrical engineering at the University of Constantine, Algeria (phone: 213-776-63-05-63; e-mail: rym_ima@hotmail.fr).

M. E. H. Latreche is electrical engineering professor at the University of Constantine, Algeria and president of Electrotechnical Laboratory Constantine Mentouri University (e-mail: latreche_med@yahoo.fr).

III. GOVERNING EQUATIONS OF THE COMSOL MODEL MODELING APPROACH

Induction heating can be estimated using modern finite element methods, as indicated previously in Fig. 1. A 2D model of the induction heating experiments was created using the commercial code of COMSOL 4.2® [3]. This model directly solves Maxwell's equations in the frequency domain, starting from the magnetic vector potential, the formulation of the problem requires the statement of the electromagnetic field we can write:

$$\vec{B} = \vec{\nabla} \times \vec{A} \quad (1)$$

where

B flux density [T]
A vector potential [Wb/m]

$$\vec{B} = \mu_0 \mu_r \mu \vec{H} \quad (2)$$

where H is the magnetic field intensity [A/m].

$$\vec{\nabla} \times \vec{H} = \vec{J} \quad (3)$$

where J is the current density [$\frac{A}{m^2}$] and the displacement current $\frac{\partial \vec{D}}{\partial t}$ is neglected

$$\vec{J} = \sigma \vec{E} + \vec{J}^e \quad (4)$$

where

Σ The electrical conductivity [S/m],
E The electric field [V/m],
 J^e The external current density [A/m²].

$$\vec{E} = \nabla V - \frac{\partial \vec{A}}{\partial t} \quad (5)$$

where

V The electric potential [V]
T Time [t]

$$\vec{\nabla} \times \vec{E} = -\frac{\partial \vec{B}}{\partial t} \quad (6)$$

In the present 2D axisymmetric model, the mathematical model for a eddy current problem with excitation \vec{J} at frequency f is described by the well known equation of the magnetic vector potential \vec{A}

❖ For the workpiece

$$\nabla^2 \vec{A} - j\omega\mu\gamma\vec{A} = 0 \quad (7)$$

❖ For the inductor

$$\nabla^2 \vec{A} - j\omega\mu\gamma\vec{A} = -\mu\vec{J} \quad (8)$$

The current density is described by the formula:

$$\vec{J} = -j\omega\mu\gamma\vec{A} \quad (9)$$

Eddy currents derived by the electromagnetic model manifest themselves through heat production due to the Joule effect. The heat is then distributed throughout the workpiece. The process is described by a classical heat equation

$$\rho C \frac{\partial T}{\partial t} - \nabla(\lambda \nabla T) = Q \quad (10)$$

IV. ANALYSIS METHODS

Several numerical methods are more important today for the analysis of electromagnetic problems, but the solution of 2D electromagnetic system eddy current is still the fastest and most effective if fem codes are used for the solutions of the forward problem. For this reason the actual design of the inductor has been carried out with the Comsol Multiphysics software. A simplified version of the induction coil is used for the finite element simulations [6], [9]. The two-dimensional axisymmetric geometry with 10 coil imported into COMSOL is shown in Fig. 2.

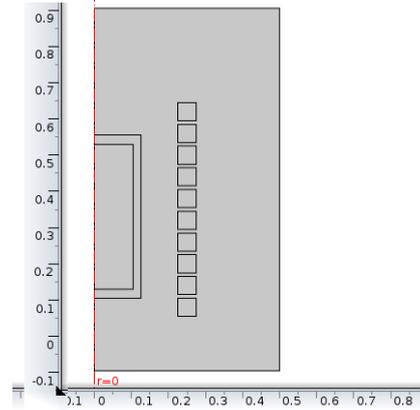


Fig. 2 2D geometry

TABLE I
PARAMETER OF SIMULATION

Symbol	Quantity	Value
I	Current	2000 [A]
f	Frequency	5 [kHz]
t	Times	1000 [s]
d	Diameter coil	10 [mm]
	Turn coil	10

The predefined transient analysis, induction heating, electromagnetic interaction mode in the AC/DC module is used for the models. The induction heating simulations use the quasi-static, induction currents, time-harmonic application mode (azimuthal induction currents in 2-D) to solve for the magnetic vector potential. Heating from the induction currents application mode as a source term in the energy equation for a transient heat transfer simulation, in the general heat transfer application mode [9], [8].

A number of assumptions are used in defining the model. Firstly, the current in the induction coil are treated as a boundary condition of induction current on the surface and on the inner coil. The maximum temperature of the test article is limited to 600°C during the experiments. Finally, the resistive losses in the copper are treated as a boundary condition in the heat transfer [6].

In this case we build the mesh shown in Fig. 3 Very dense mesh was applied especially to the surface of workpiece and between coils of inductor, because of proper calculation of electromagnetic phenomena.

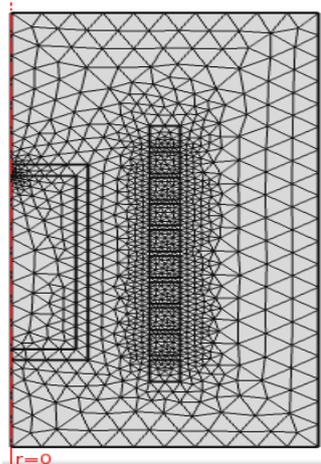


Fig. 3 Element finite model

V. SIMULATION RESULT AND DISCUSSION

A. Frequency

It is possible to use the COMSOL® with high frequency simulations of induction processes, but only if work piece data is the sole desired output, the work piece remains surrounded by the same magneto-motive force, regardless of frequency [1].

In this part of the paper, the frequency has been changed, for five different frequencies. First of all, the skin effect which has been introduced before should be presented using the software. The following example compares the power and inductance for the same configuration; only the frequency is changed as can be seen in Table II.

TABLE II
INFLUENCE OF FREQUENCY ON THE PARAMETERS OF MATERIAL

Frequency [HZ]	Power [w]	Inductance [μ H]	Skin thickness [mm]
50	8600	4.3	9.19
500	4.95×10^4	0.116	2.9
5000	5.49×10^5	0.0105	0.91
50000	5.3×10^6	1.15×10^{-3}	0.29
500000	5.5×10^7	6.43×10^{-7}	0.09

It is easy to see the change and effects of the skin effect. With a higher frequency, the Skin effect only depends on frequency. The current density is increased near the boundary,

power, and inductance. The distribution of current density in a workpiece is shown in Fig. 4. We also see a coil (with 10 turns) covered by grid computing [8], [2].

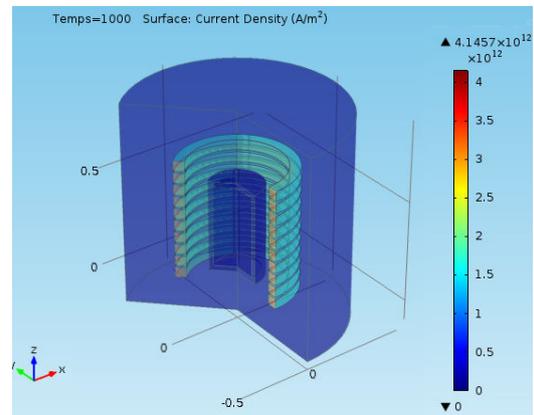


Fig. 4 The distribution of current density in a cylindrical work piece

Note that at low frequencies the skin-depth is higher, and so the electromagnetic field penetrates deeper into the workpiece. At higher frequencies, Lorentz force concentrates near the graphite crucible, and so buoyancy forces become more important.

B. Heat Transfer

This paper is treated with the finite element method (FEM) software. Comsol is 2D finite element method software devoted to compute electromagnetic fields. In Fig. 5 we have presented the electric field lines, at the end of heating process. [4]:

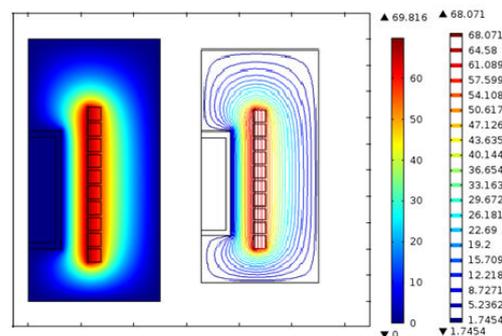


Fig. 5 Electric field lines, end of heating process

In the coil study, the temperature distribution obtained at the end of 1000 s when a steady current of 2000 A at 5 kHz frequency flows through it is shown in Fig. 8. We can see that the heating is concentrated inside the coil and around the slot. The maximum temperature is reached at 600°C.

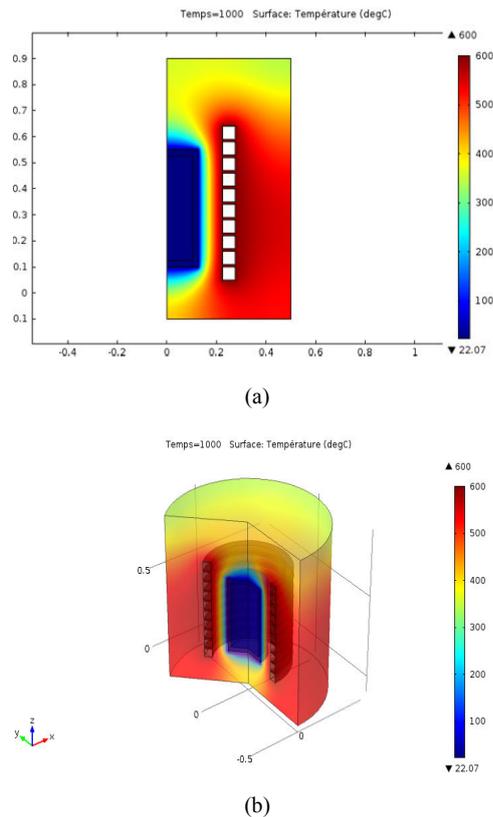


Fig. 6 Temperature distribution in mock-up induction furnace
(a) 2D temperature distribution (b) 3D temperature distribution

VI. CONCLUSION

This paper has focused on the pre-design of induction heating using COMSOL® Multiphysics software because was very quick, the solving of model takes about 20min on computer based on core i3 equipped with 64GB RAM. On the basis of the obtained results we can come to following conclusions:

- ❖ The variation of the width inductor has no effect on the current density distribution, but it affects the efficiency of the induction heating process. Therefore, the good design of induction heating has a small inductor that also takes care about the skin effect that will be happened in high frequency.
- ❖ At the end of heating process when we reach the maximum temperature (600°C), the surface of piece is characterized with temperature above the Curie point, the piece is non-magnetic, and the electromagnetic field has a better penetration in the piece.

REFERENCES

- [1] C. Snow, Formula for the inductance of a helix made with wire of any section, Ed. US Govt. Print. Off, 1926, p. 91.
- [2] M. W. Kennedy, S. Akhtar, J. A. Bakken, and R. E. Aune, *Analytical and Experimental validation of electromagnetic simulations using COMSOL re Inductance induction heating and magnetic field*, Comsol Conference, Stuttgart, 2011.

- [3] M. W. Kennedy, S. Akhtar, J. A. Bakken, and R. E. Aune, *Review of Classical Design Methods as Applied to Aluminum Billet Heating with Induction Coils*, EPD Congress, San Diego, California, February 27 - March 3, (2011), 707-722.
- [4] T. A. Jankowski, D.P. Johnson, J. D. Journey, E. Freer, L. M. Dougherty, and S. A. Stout, *Experimental Observation and Numerical Prediction of Induction Heating in a Graphite Test Article*, *Expert from the proceedings of the Comsol Conference*, Boston, 2009.
- [5] U. Ludtke and D. Schulze, *FEM software for simulation of heating by internal sources*, *Proc. of HIS-01 Int. Seminar*, Padua, Italy, Sept. 12-14, 2001.
- [6] A. Bermúdez, D. Gómez, M.C. Muñiz and P. Salgado, *Transient numerical simulation of a thermo electrical problem in cylindrical induction heating furnaces*, *Advances in Computational Mathematics* (2007) 26: 39–62.
- [7] S. Clain, J. Rappaz, M. Swierkosz and R. Touzani, *Numerical modeling of induction heating for two-dimensional geometries*, *Math. Models Methods Appl. Sci.* 3 (1993) 805–822.
- [8] D. Istardi, A. Triwinarko, *Induction Heating Process Design Using COMSOL® Multiphysics Software*, ISSN: 1693-6930, *TELKOMNIKA, Vol.9, No.2, August 2011*, pp. 327–334.
- [9] COMSOL Multiphysics Version 4.2a : www.comsol.fr.