

Induction Heating Simulation in EMS

Induction heating explained

Induction heating is an accurate, fast, repeatable, efficient, non-contact technique for heating metals or any other electrically-conductive materials. By applying a high-frequency alternating current to an induction coil, a time-varying magnetic field is generated. The material to be heated is placed inside the magnetic field, without any contact to the coil. The alternating electromagnetic field induces eddy currents in the workpiece (material to be heated), resulting in resistive losses, which will be converted to heat inside the workpiece. Thus, material heated by induction heating should have high electrical conductivity. The material of the workpiece may be a metal such as steel, copper, aluminum or brass or it can be also a semiconductor such as carbon, graphite or silicon carbide. Ferrous metals are heated by induction more easily than other materials. That is so because, in addition to the eddy currents, another heating mechanism occurs. The material's iron crystals are magnetized and demagnetized over and over by the alternating magnetic field. This causes the magnetic domains to flip rapidly back and forth, leading to hysteresis losses, which result in heat. Figure 1 shows a typical induction heating setup.



Figure 1 - Induction heating setup



Applications

- Industrial applications such as induction hardening, induction brazing, induction tempering, induction welding ...
- Domestic application: the main domestic application is induction cooking.
- Medical applications: annealing steel tubes, brazing miniature medical parts, soldering surgical tools, heating catheter tipping dies

Mathematical modeling of the induction heating process

In the past three decades, heating by induction has become the preferred technique in metalworking applications. This is one of the most powerful and promising methods in modern electromagnetic processing of materials because heating by induction provides a lot of advantages. Many mathematical analysis and numerical simulation methods have been developed and applied in design of induction heating system and process.

Electromagnetic model

The magneto-thermal phenomena descriptive equations are obtained from Maxwell's equations. The basic classical relation of electromagnetic model depended on the material characteristic is:

$$\nabla \times \left(\frac{1}{\mu_0 \mu_r} \nabla \times A \right) + \sigma \frac{\partial A}{\partial t} = J_s$$

Where, A is magnetic vector potential, J_s is the source current density in the induction coil, μ_r is relative magnetic permeability of the material, σ is electrical conductivity of the material. $\rho = 1/\sigma$, where ρ is electrical resistivity. The constant $\mu_0 = 4\pi \times 10^{-7}$ H/m is the permeability of free space. When the excitation current in the coil is assumed to be sinusoidal with a frequency ω :

$$j\omega\sigma A - \frac{1}{\mu_0 \mu_r} \nabla^2 \times A = J_s$$

Where A is now the phasor of the magnetic vector potential A, and J_s is now the phasor of the source current density.

Thermal model

Temperature field Eddy current produce heat by the Joule effect. The equation can be written as below:

$$Q = \frac{1}{2\sigma} |J_e|^2$$

Where J_e is the induced current density in the workpiece. Q is the rate of heat generation within the material. Therefore, the transient heat conduction can be described by the equation below:

$$\nabla(\lambda \nabla T) + Q = \gamma C_p \frac{\partial T}{\partial t}$$

Where λ , γ , and C_p are, respectively, the thermal conductivity, density, and specific heat of the workpiece.

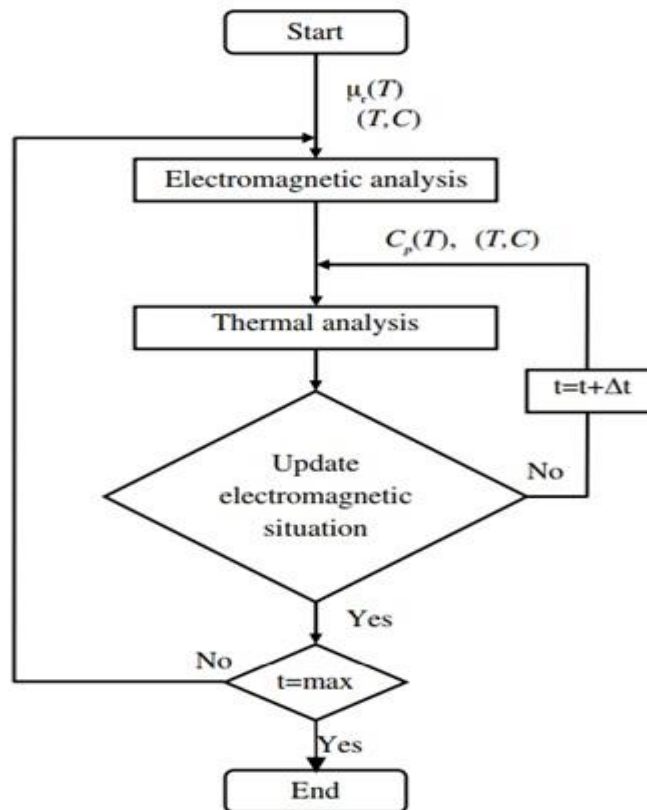


Figure 2 - Procedure of numerical simulation



Electromagnet-Thermal coupling capability of EMS

EMS ensures a Multi-physics simulation by the capability of coupling between electro-magnetic- structural-thermal field. In our case, an electro-magnetic- thermal simulation was carried on. To solve an induction heating problem in time varying domain, a transient magnetic study coupled to thermal analysis in EMS is needed.

The Transient Magnetic analysis belongs to the low-frequency electromagnetic domain or regime; i.e. displacement currents are neglected. It calculates magnetic fields that vary over time. These fields are typically caused by surges in currents or voltages. This type of analysis can be linear or non-linear. It also addresses eddy currents, power losses and magnetic forces.

The pertinent Maxwell's equations for this type of analysis:

$$\nabla \times E = -\frac{dB}{dt}$$

$$\nabla \cdot B = 0$$

where E is the electric field, and B is the magnetic flux density.

The constitutive relation that connects B and H :

$$B = \mu(H + H_c)$$

where σ is the magnetic permeability, in general a function of H . H_c is the coercive force or coercivity

The constitutive relation that connects J and E :

$$J = \sigma E$$

Problem description

In this example, a sphere inside a coil will be heated by induction heating phenomenon. Figure 3 shows a 3D CAD of simulated model. The excitation waveform is sinusoidal and the frequency is 60 Hz.

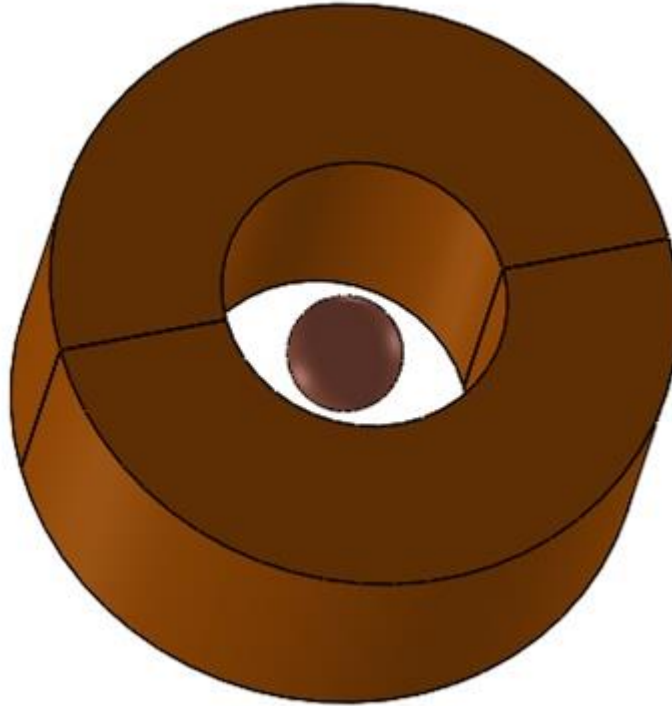


Figure 3 - The simulated model

Simulation setup

After creating an Transient Magnetic study coupled to thermal analysis in EMS, four important steps shall always be followed:

1. apply the proper material for all solid bodies
2. apply the necessary electromagnetic inputs
3. apply the necessary thermal inputs
4. mesh the entire model and run the solver

Materials

Both coil and sphere are made of copper.

Table1 - Material properties

Components / Bodies	Relative Permeability	Electrical Conductivity	Thermal Conductivity (W/m*K)	Specific Heat (J/Kg*K)	Masse Density (Kg/m ³)
Sphere/ Coil	1	5.840e+7	401	384	8960

Electromagnetic inputs

In this study, only a wound coil is assigned as electromagnetic input.

Table 2 - Coil information

	Number of turns	RMS total current
Wound coil	1000	6.468 A

Meshing

Meshing is a very crucial step in any FEA simulation. EMS estimates a global element size for the model taking into consideration its volume, surface area, and other geometric details. The size of the generated mesh (number of nodes and elements) depends on the geometry and dimensions of the model, element size, mesh tolerance, and mesh control. In the early stages of design analysis where approximate results may suffice, you can specify a larger element size for a faster solution. For a more accurate solution, a smaller element size may be required.

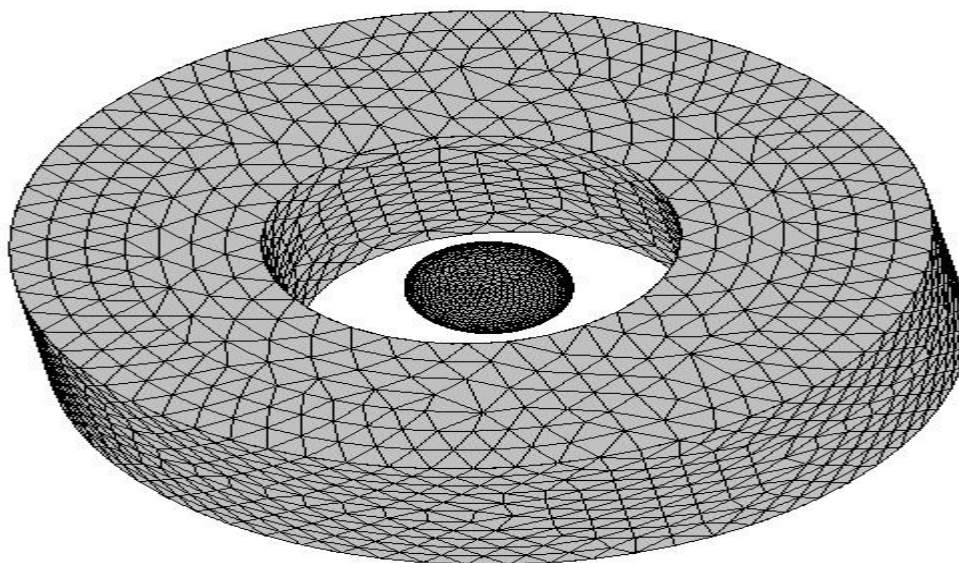


Figure 4 - Meshed model

Electromagnetic-thermal results

After finishing the setup, a successful run of Transient Magnetic simulation coupled to thermal analysis generates both magnetic and thermal transient results.

In the figure below is a 3D plot of magnetic flux density in the sphere at 50 ms.

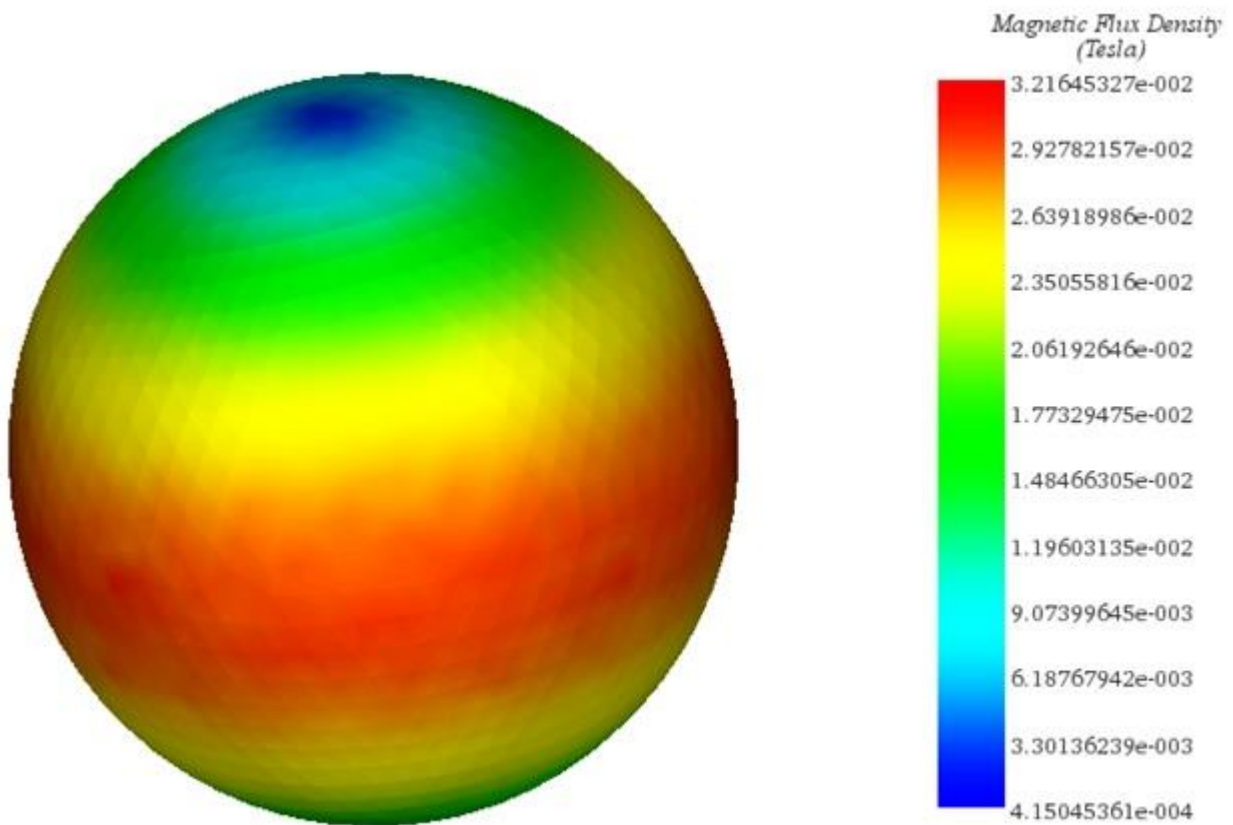


Figure 5 - Magnetic flux density in the sphere at 50 ms.

In the figure 6, 3D plot of current density distribution shows the circulation of eddy current in the surface of sphere.

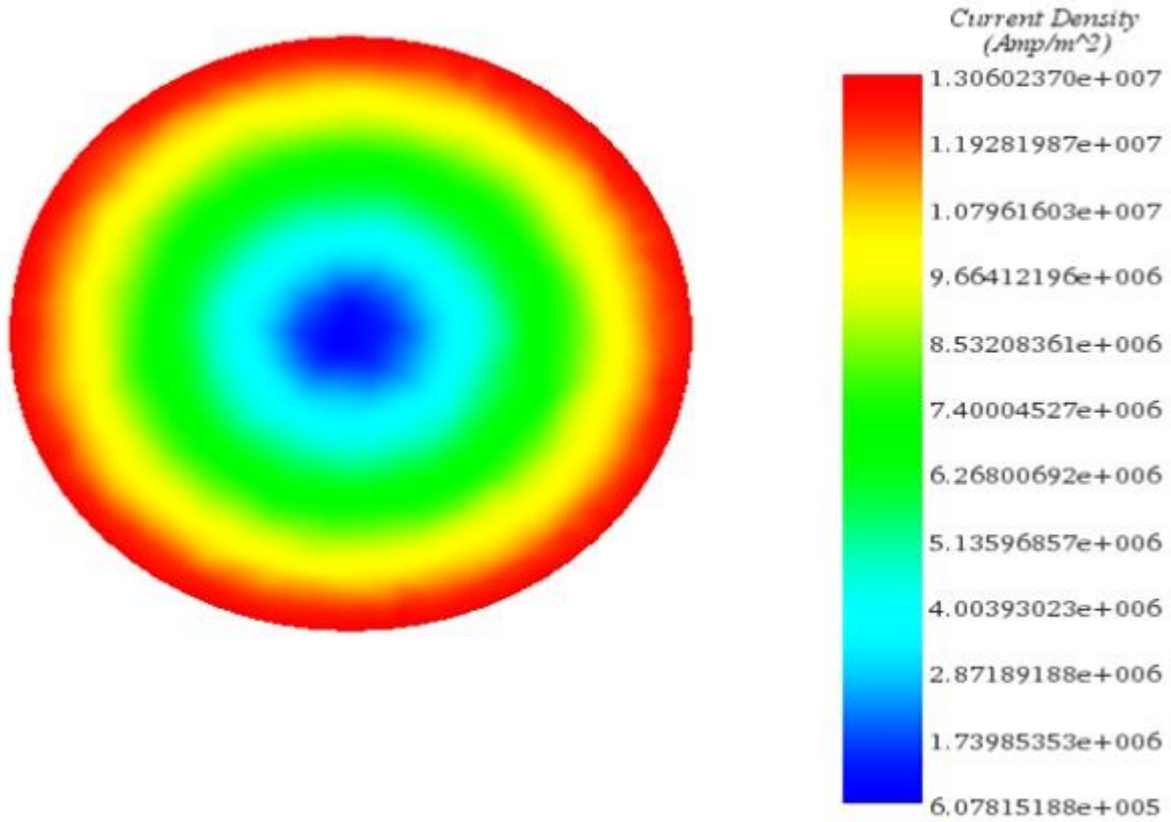


Figure 6 - Current density distribution in the sphere, section plot

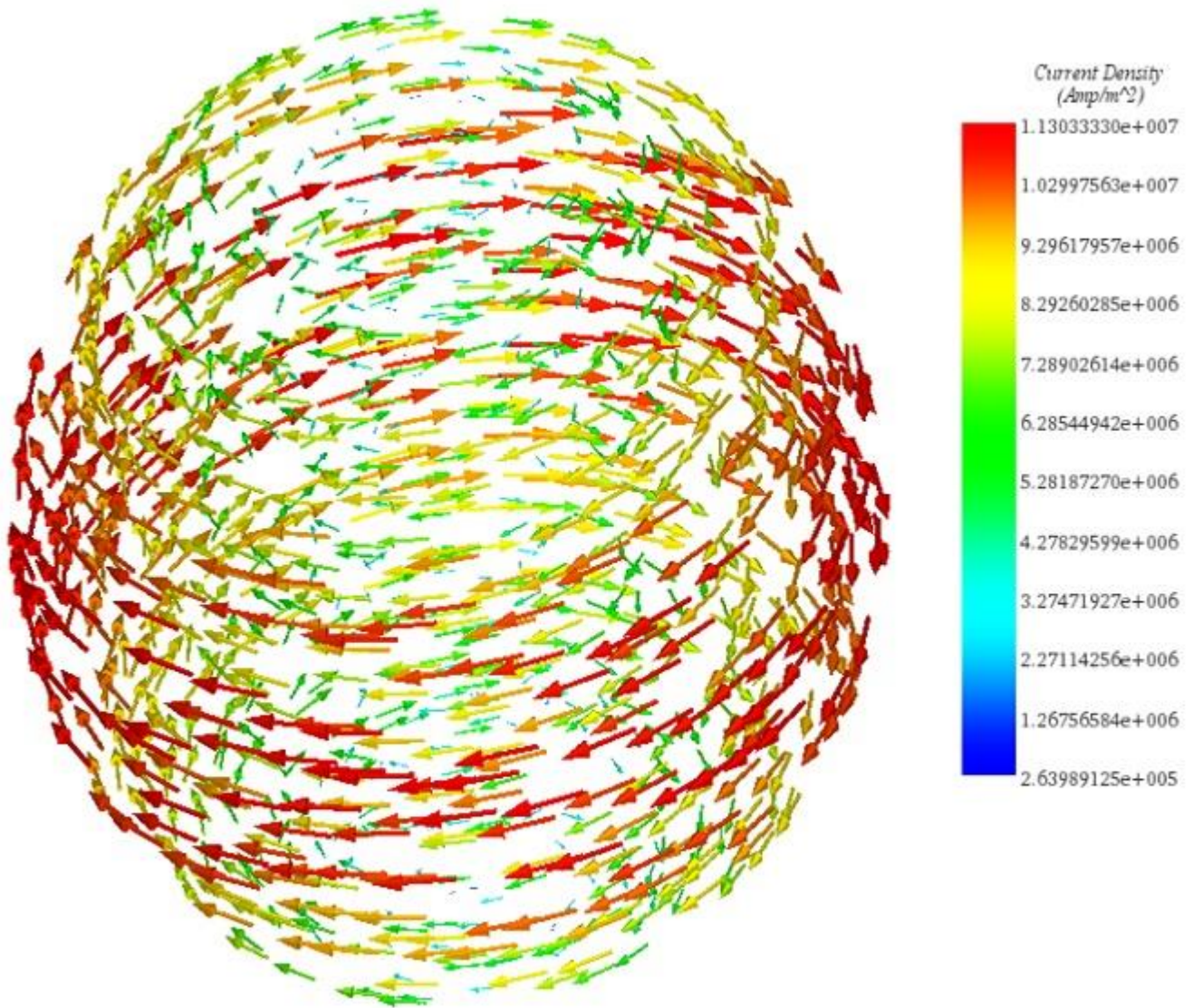


Figure 7 - Eddy current animation in the sphere at 0.044 second

Figure 8 shows the temperature distribution in the sphere after 1.6 hour.

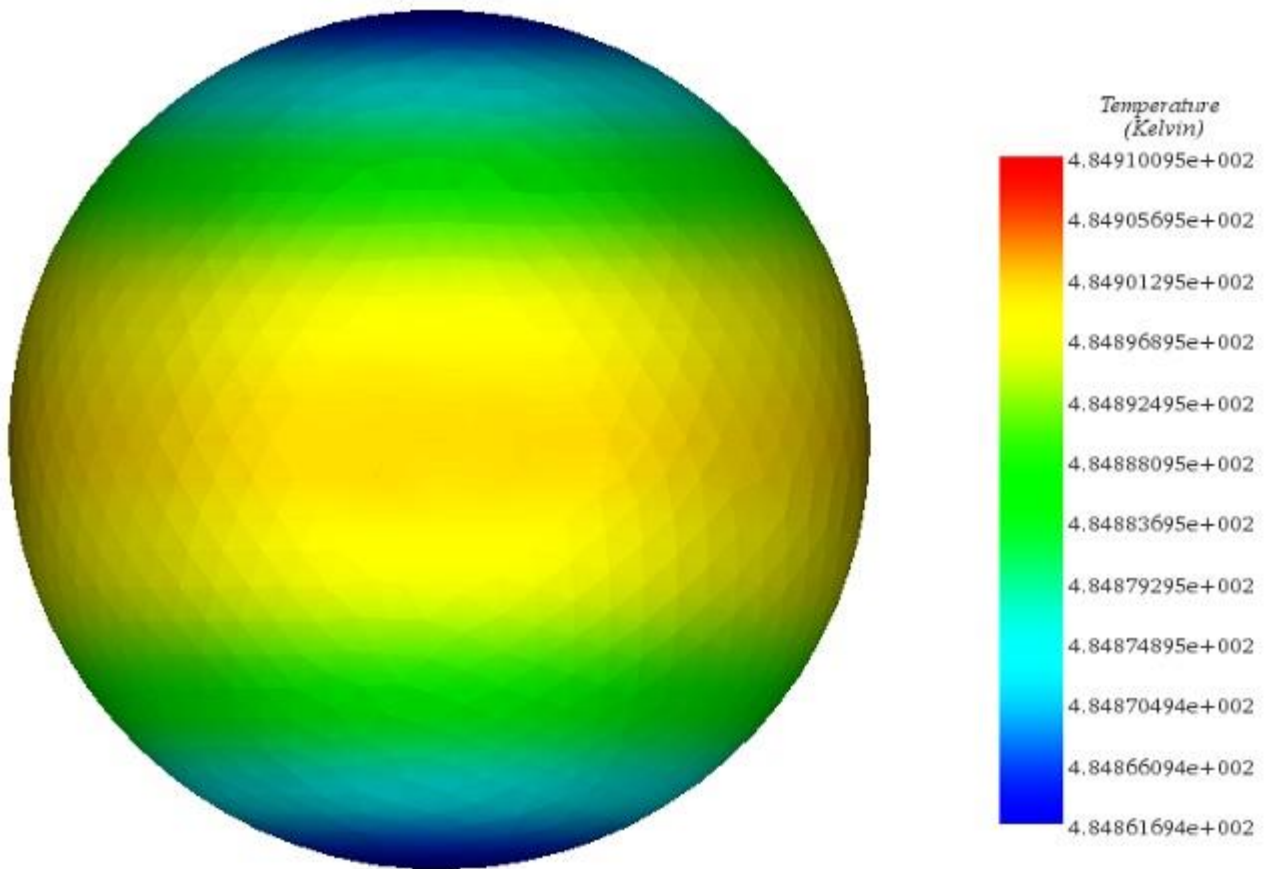


Figure 8 - Temperature in the sphere at 1.6 hour

Temperature - 5

Study Name: Multi physical simulation of induction heating process

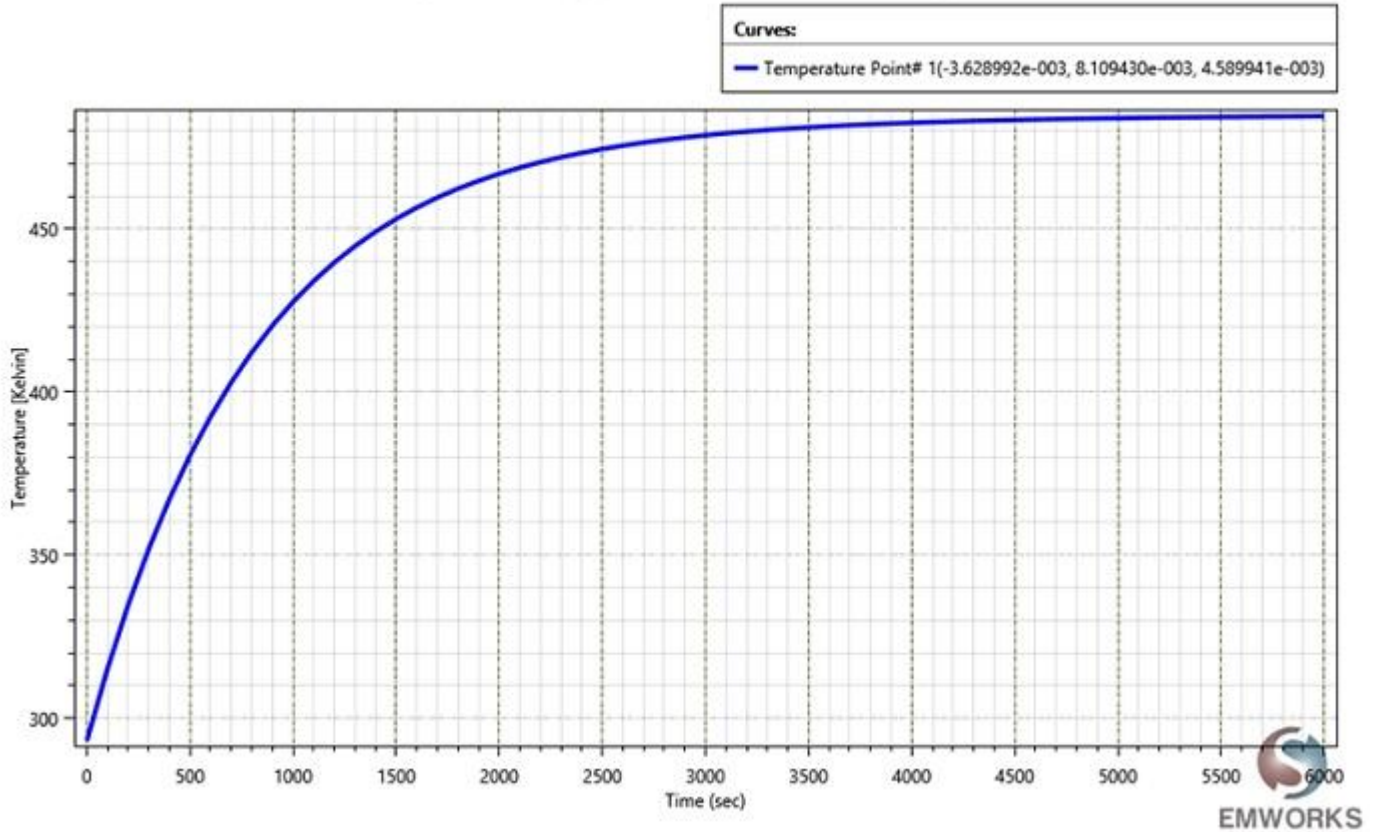


Figure 9 - Temperature evolution

Conclusion

The electro-thermal simulation capability in EMS enables users to effectively test their coil designs for induction heating application. Engineers can now simulate multiple designs of their coils and choose the most effective coil based on the component to be heated. By studying the temperature distribution inside the heated component, one can complete understand the heating profile.

