

An Improved Model for High Temperature Inductive Heating

S. A. Halvorsen^{* 1} and N. Kleinknecht¹

¹Teknova AS, Kristiansand, Norway

^{*}Corresponding author: sah@teknova.no

Abstract: A multiphysics model for industrial induction furnaces has successfully been converted from COMSOL version 3.5a, via versions 4.0a and 4.1, to version 4.2. The model combines computation of magnetic fields, heat transfer and thermal stresses. The inner part of the furnace is described by a few discrete state variables. While the current in the induction coil is input for COMSOL, the model applies an equation for power control, temperature control or a combined power/temperature control. A few case studies have been selected to show differences between the control alternatives. The model can be applied to study operational conditions, thermal stresses, or design details. Compared to version 3.5a the current model has been extended to handle quasi-stationary computations, conductive or non-conductive liquid content, far more flexible dynamics, and temperature or combined control.

Keywords: multiphysics, inductive heating, control

1. Introduction

High temperature reactors heated by electromagnetic induction are used for various purposes like metal melting, refining, alloying, degassing, etc. Such a reactor consists of a crucible containing a liquid, surrounded by suitable insulating materials. An electric coil is located outside the insulation, c.f. Figure 1. Typical industrial furnaces have maximum power input in the range of 50-20,000 kW and frequency in the range of 50-1,000 Hz.

An alternating electric current in the coil sets up an alternating magnetic field. This field induces electric currents in the liquid and/or the crucible to supply the required heating. When a metal is processed, the heat can be delivered directly into the metal, while a non-conductive liquid will require sufficient electric conductivity in the crucible.

A simulation model has previously been implemented in COMSOL Multiphysics 3.5a. The model is axially symmetric and takes into account electromagnetic induction, temperature distribution, and thermal stresses. The inner part of the reactor is simplified and described by three discrete (lumped) state variables:

mass and temperature of the liquid, and (average) radiation temperature of the void [1].

The model has been improved and is now converted to COMSOL Multiphysics 4.2. While Matlab scripts and Matlab interface were required to run the solution procedure for the original model, the solver sequence feature introduced in COMSOL 4.0 can now be utilized.

Our models are intended for conceptual studies of design and operation of high temperature reactors. Construction details like tubes in the lid, pouring spout, steel casing, and other aspects not important for the studies are intentionally omitted, or included by coarse descriptions. Further, axial symmetry is assumed.

2. Model

The COMSOL model for version 4.2 includes the following application modes:

- Magnetic Fields (mf)
- Heat Transfer (ht)
- ODEs and DAEs (ge)
- Solid Mechanics (solid)

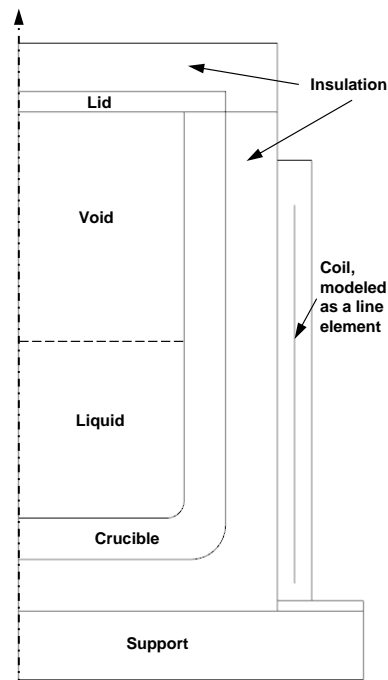


Figure 1 Axially symmetric model geometry

2.1 Magnetic Fields

Maxwell's equations need to be solved to find the electric power induced in the furnace. For an axially symmetric model COMSOL Multiphysics solves for the non-zero angular component of the magnetic vector potential. Equations and other details are described in COMSOL's documentations [2]. The equation form must be set to "Frequency domain" (not "Study controlled" or "Stationary").

For an industrial furnace the electromagnetic field should not spread outwards. Field guides are therefore located immediately outside the coil. The spread of the field is also limited by a steel casing and possibly also by one or more copper plates. The computational domain for the electromagnetic model must include the coil, and some space outside the coil. It is further limited by the steel casing of the furnace. The field guides can be represented by a "perfect magnetic conductor" boundary condition, while the steel casing and copper plates are considered as "magnetic insulation", c.f. [2]. Axial symmetry is the appropriate boundary condition along the center line.

The coil is described by a line element through the center of the turns from the bottom to the top of the coil, c.f. Figure 1. The individual turns might have been represented, which would have made the geometry considerably more complex. Preliminary computations have shown that such details would not significantly improve the accuracy of the computed power distribution.

Immediately above and below the electric coil there can be a few turns that are not connected to external current. These turns are used for additional water cooling. Preliminary calculations showed that induced electric currents in such turns will limit the vertical spread of the magnetic field. In the model they can be appropriately represented by horizontal line elements with a "magnetic insulation" condition.

The surface current density in the angular direction for the line element representing the coil is the appropriate model input parameter. But for the simulations studies the value of this current is not given a priori. Instead, either the total induced power or the temperature in some location can be specified. Hence, an appropriate constraint should be included and the current density in the coil should be adapted to fulfill the constraint.

In the previous model it was assumed that the magnetic field lines would be parallel to

the field guides outside the coil. Hence, magnetic insulation was applied as a boundary condition also for this region [1]. This idealization can be valid to compute a reasonable approximation for the distribution of the induced heating. But when the boundary is close to the coil, the model will compute high electric boundary currents. It follows from Ampere's law that these boundary currents will be counteracted by currents in the coil. Hence, the computed coil current will not be realistic.

For "frequency domain" computations the induced heating is the real part of a complex valued power. $\text{Re}(z)$ is a non-analytical function, i.e. its derivative is not defined. Previously, COMSOL has provided a "pseudo Jacobian" for derivative information to the solver for the heating terms. This has caused computational problems as temperatures often were "infected" with small imaginary components. From version 4.1 such "impurities" were eliminated by removing the pseudo Jacobian contributions. The solver had then no longer information on how the coil current would influence the temperatures and the computations would not start due to one missing equation.¹

By observing that the heating term depends quadratically on the current amplitude, an elegant solution was finally found. The induced heat source can be written as:

$$(1) \quad Q_{\text{induced}} = I_{\text{coil}}^2 Q_{\text{th}}$$

where Q_{th} is the heat source [W/m^3] due to a coil current of 1 A/m (as computed by COMSOL) and I_{coil} is the amplitude of the actual current in the coil [A/m] (to be adjusted to fulfill the constraint, c.f. chapter 2.3 below).

If the liquid is a metal, the magnetic field will induce high electric currents. The currents will interact with the magnetic field to create a force, the Lorentz force. This force will create a "magnetic pressure" that will modify the shape of the free metal surface. In addition, flow will be caused [4, 5]. The influence of a modified surface is neglected in the model, while the effect of the flow can be included by choosing a suitable heat transfer coefficient between the wall and the liquid.

2.2 Heat Transfer

The governing equation is:

¹ According to COMSOL, AC problems can be handled by a superposition of real-valued sine and cosine terms in version 4.2a, which should solve the problem.

$$(2) \quad \rho c_p \frac{\partial T}{\partial t} + \nabla(-k\nabla T) = Q_{\text{induced}}$$

where ρ is the density [kg/m³], c_p is the heat capacity [J/kg K], T is the temperature [K], k is the thermal conductivity [W/m K], and Q_{induced} is the heat source [W/m³] given by equation (1).

The boundary conditions are as follows:

- Towards the coil and the additional water-cooling immediately above and below: Heat transfer coefficient
- At the remaining exterior boundaries: Heat transfer coefficient and radiation
- Towards the inner liquid: Heat transfer coefficient
- Towards the inner void: Radiation
- Towards the center line: Radial symmetry (no flux)

The boundary conditions towards the inner part of the crucible are smoothed, as described in [1].

Non-linearities are included due to radiation boundary conditions and temperature dependent thermal and electrical conductivities of the crucible.

2.3 ODEs and DAEs

The constraint for the current in the coil is implemented as an equation for I_{coil} . Principally there are two alternatives:

$$(3) \quad W_{\text{Total}}(t) = W_{\text{SetPt}}(t)$$

$$(4) \quad T_{\text{Chosen}}(t) = T_{\text{SetPt}}(t)$$

where t is the time [s], W_{Total} the total induced power [W], and T_{Chosen} the computed temperature in a chosen location [K], for instance the liquid temperature or the temperature at a point where it can be measured. The corresponding set point values, W_{SetPt} [W] and T_{SetPt} [K], are input to the simulation. For dynamic computations the set point values can be given as functions of the time.

To find the total power COMSOL must evaluate the integral:

$$(5) \quad W_{\text{Total}}(t) = \iiint 2\pi r Q_{\text{induced}}(r, z, t) dr dz$$

For case studies of melting, the inner part of the container can be described by four discrete state variables:

- M_{liq} , amount of liquid [kg]
- M_{sol} , apparent amount of solid (unmelted) material [kg]
- T_{liq} , temperature in the liquid [K]
- T_{void} , average (radiation) temperature in the void [K]

As simplifications it has been assumed that the temperature of the liquid is uniform and that the liquid and the inner surfaces facing the void exchange heat with a fictitious body of average (radiation) temperature.

Depending on the process to be studied, a few more state variables can be relevant. There can, for instance, be two liquid phases, or the liquid(s) can contain more than one component.

The equations for the state variables are:

$$(6) \quad \frac{dM_{\text{sol}}}{dt} = m_{\text{input}}(t) - m_{\text{melt}}$$

$$(7) \quad \frac{dM_{\text{liq}}}{dt} = m_{\text{melt}} - m_{\text{tap}}(t)$$

$$(8) \quad M_{\text{liq}} c_{\text{pl}} \frac{dT_{\text{liq}}}{dt} = W_{\text{liq}} + Q_{\text{void_liq}} - Q_{\text{liq_w}} - m_{\text{melt}}(c_{\text{ps}}(T_{\text{melt}} - T_0) + L_{\text{melt}} + c_{\text{pl}}(T_{\text{liq}} - T_{\text{melt}}))$$

$$(9) \quad Q_{\text{void_liq}} + Q_{\text{void_w}} = 0$$

where m_{input} is the charging rate of cold, solid material [kg/s], m_{melt} is the effective melting rate [kg/s], m_{tap} is the tapping rate [kg/s], c_{ps} and c_{pl} the heat capacity of the solid and the liquid material [J/kg K], L_{melt} the heat of fusion (melting heat) [J/kg], T_0 the temperature of the charged solid material [K], W_{liq} the induced heat in the liquid [W], $Q_{\text{void_liq}}$ the heat flow from the void to the liquid [W], $Q_{\text{liq_w}}$ the heat flow from the liquid to the crucible bottom and walls [W], and $Q_{\text{void_w}}$ the heat flow from the void to the crucible wall and lid [W].

When cold, solid material is charged, some liquid will instantly freeze at the solid/liquid interface. At this interface, the temperature will be equal to the melting point. Due to the temperature differences there will be a heat flow from the bulk liquid to the solid/liquid

interface and from the interface to the interior of each unmelted particle. For a short time after a particle has been charged, the latter will be greater than the former, and liquid will freeze on the particle. Then the heat flow from the bulk will be the greater one, and the particle will melt. A comprehensive model for this process would be rather complex.

We have chosen a simplified approach, where M_{sol} is the apparent amount of solid material in the container, as if it had stayed at the input temperature until melted. When the solid material melts, the model assumes that the temperature is instantly raised to the liquid temperature.

The melting rate is given by the following linear relation:

$$(10) \quad m_{\text{melt}} = k_k M_{\text{sol}} (T_{\text{liq}} - T_{\text{melt}})$$

where k_k is a kinetic constant that can be adapted to experimental data.

Furnace tapping can be included by specifying a time dependent tapping. An induction furnace can be tilted during tapping, but this effect is neglected in the model.

The induced power in the liquid, W_{liq} , is found by integration in COMSOL. To ensure numerical stability a small, non-zero value is chosen for the electrical conductivity in the void and a gradual transition (smoothing) is applied between the high conductivity in a metal liquid and the void.

The height of the liquid is easily computed when M_{liq} and M_{sol} are known.

The heat fluxes in equations (8) and (9) are given by:

$$(11) \quad Q_{\text{void_liq}} = \pi R_{\text{liq}}^2 \varepsilon_{\text{liq}} \sigma (T_{\text{void}}^4 - T_{\text{liq}}^4)$$

$$(12) \quad Q_{\text{liq_w}} = \int 2\pi r h_{\text{liq_w}} (T_{\text{liq}} - T) ds$$

$$(13) \quad Q_{\text{void_w}} = \int 2\pi r \varepsilon_w \sigma (T_{\text{void}}^4 - T^4) ds$$

where R_{liq} is the inner radius of the crucible at the liquid level, ε_{liq} and ε_w the effective emissivities of the liquid and the wall, σ the Stefan-Boltzmann constant, $h_{\text{liq_w}}$ the heat transfer coefficient between the liquid and the crucible, and ds the line differential along the inner boundary of the crucible and the lid. The boundary integral in equation (12) is taken along the inner crucible boundary from the center at the bottom to the liquid level, while the integral in (13) is taken along the remain-

ing part of the boundary. Both integrals are smoothed at the liquid level [1].

Processes in an industrial furnace can be comparatively slow and a *quasi-stationary* state can be observed. For simulations, the time dependent terms in equations (2), (6), and (8) shall then be dropped. The thermal effect of changing the temperatures in the liquid and the crucible will be neglected, while the power required for heating and melting the charged material is retained. The latter is a major thermal effect that can not be omitted.

For COMSOL computations a stationary study is performed, without applying equation (7). The computations assume that the melting rate is equal to the charging rate, c.f. equation (6). Hence, if there is some buildup or removal of the apparent amount of solid, the (presumed) melting rate must be applied for m_{input} , instead of the actual charging rate.

2.4 Solid Mechanics

For an industrial furnace the insulation needs to be soft to enable (approximately) stress-free expansion of the crucible from room temperature to operating conditions. Correspondingly, thermal expansion of the lid must be limited to prevent a significant force on the crucible.

Uneven temperature distribution can cause significant thermal stresses. A stress-strain computation is therefore included, with thermal expansion as a load. The behavior is quasi-static, i.e. the temperatures change with time, while the mechanical equations are stationary. The equations, input parameters, etc., are described in COMSOL's documentation [2, 3].

When the model was tested in COMSOL Multiphysics version 4.2, we unexpectedly discovered error in the heat balance when the Solid Mechanics application mode was included. It turned out that COMSOL now would assume that the Solid Mechanics mode should deform the mesh. But it will only modify the mesh that is included in the stress-strain calculations. Hence, on the boundary between the crucible and the insulation, deformed element sizes would be applied for the crucible and undeformed ones for the insulation. Such inconsistencies will necessarily violate the heat balance. The solution is to make sure that the mesh is not modified, i.e. the check-box for "Displacements control spatial frame" must be unchecked (Solid Mechanics, Advanced Settings).

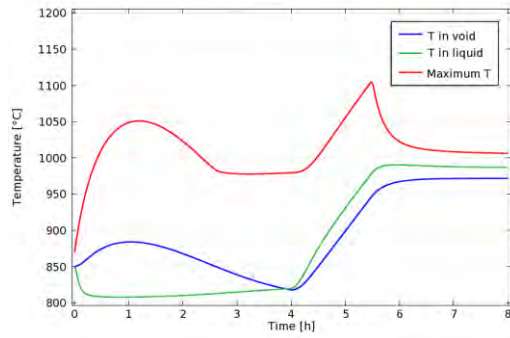


Figure 2 Case 3.1: Temperature development

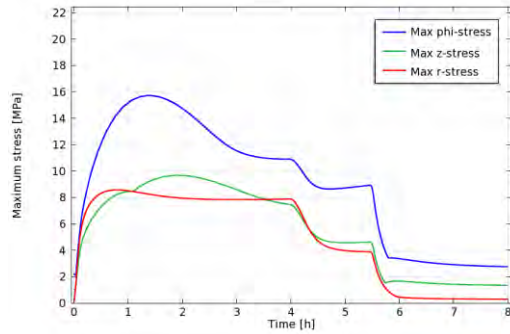


Figure 3 Case 3.1: Maximum stress in the three main directions

3. Case Studies

For the case studies we have set the frequency to 150 Hz and chosen data for artificial materials with reasonable properties. The liquid in the crucible is initially at 850 °C, 50 °C above its melting point. Then 1000 kg bulk solid shall be charged at a constant rate during 4 hours, where after the liquid temperature shall be raised to 1000 °C. Smooth changes in dynamic input data was chosen to ensure numerical stability. The input feed rate was raised from zero to a constant value during 30 s at the start and then lowered for 5 min after 4 hours. The longer period at the end was chosen as 2.5 times the maximum time step. Higher accuracy was demanded for the non-linear iterations than for the time stepping, c.f. recommendations in [1].

3.1 Non-conductive Liquid, Power Control

Power control applies equation (3) to compute the electric current in the coil. First a proper stationary state was computed. Then the power set point was raised during 30 s to a level sufficient to melt the input material and heat it to 850 °C. Heat calculations showed that the liquid temperature would be raised to

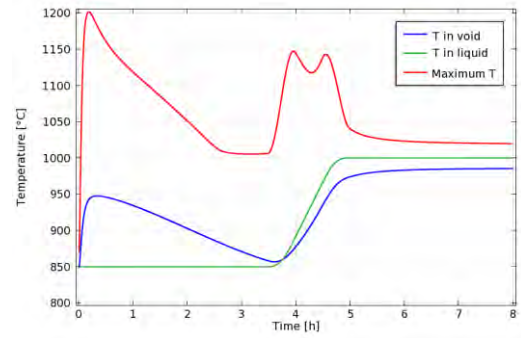


Figure 4 Case 3.2: Temperature development

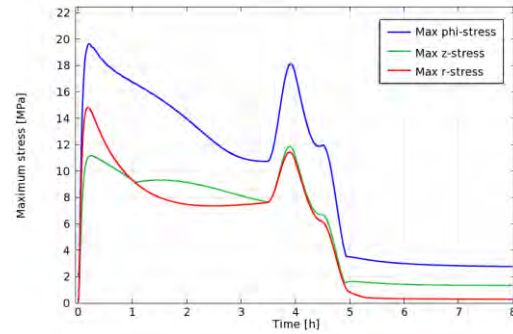


Figure 5 Case 3.2: Maximum stress in the three main directions

1000 °C if the power level was kept 31 min longer than the charging period. But it is required to heat the crucible and the insulation at the same time and the high power was maintained for another 59 min (after a few trial runs). Some computed temperatures and maximum stresses are shown in Figure 2 and Figure 3.

3.2 Non-conductive Liquid, Temperature Control

Now equation (4) was solved with T_{liq} as T_{Chosen} . First a stationary state was computed, followed by a dynamic computation. Some test simulations revealed that temperature control according to equation (4) is rather “stiff” and more smooth transitions were chosen. The charging rate was now raised during 120 s. Reducing the charging rate after some 4 hours was difficult. For the first tests the computed power would drop to zero and the computations would then stop due to an error condition. After some trial and error one hour diminishing period, partly overlapped by the temperature increase to 1000 °C, worked reasonably well. Some computed temperatures and maximum stresses are shown in Figure 4 and Figure 5.

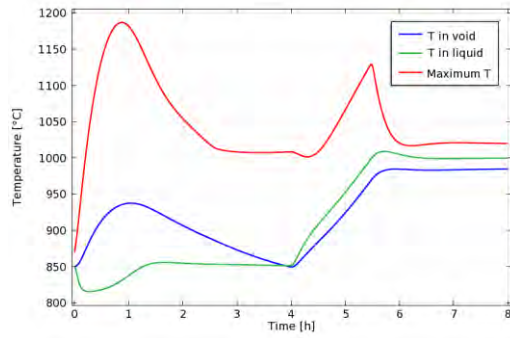


Figure 6 Case 3.3: Temperature development

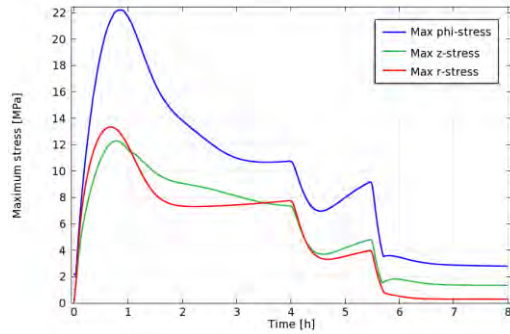


Figure 7 Case 3.3: Maximum stress in the three main directions

3.3 Non-conductive Liquid, Combined Control

A combined control was also tested. Formally power control was applied, using equation (3). Then proportional temperature control was included for additional power adjustment:

$$(14) W_{\text{SetPt}} = W_{\text{SetPt}_0} + k_T (T_{\text{SetPt}} - T_{\text{liq}})$$

where W_{SetPt_0} is the power set point that was applied for case 3.1, T_{SetPt} was set to 850 °C during the first 4 hours and then raised smoothly to 1000 °C during 1½ hours. k_T is a constant for tuning. For a very low value of k_T , the last term in the equation is negligible and the combined controller will essentially work as pure power control. For a high value of the parameter, the liquid temperature will be close to its set point and the controller will be close to pure temperature control. $k_T = 2$ [kW/K] was chosen for our case study. Some computed temperatures and maximum stresses are shown in Figure 6 and Figure 7.

3.4 Conductive Liquid, Combined Control

Combined control was chosen for a test case with a conducting liquid. The electric conduc-

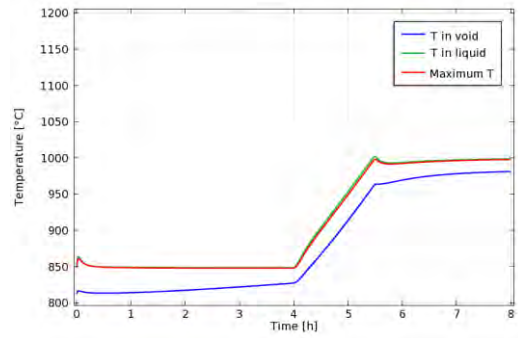


Figure 8 Case 3.4: Temperature development

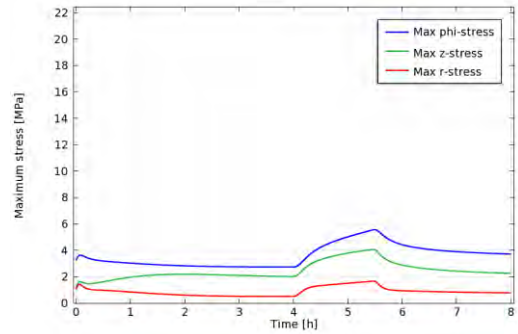


Figure 9 Case 3.4: Maximum stress in the three main directions

tivity of the liquid was raised to a level typical for metals, while the crucible was treated as an electric insulator. The remaining input was the same as in the previous case. Some computed temperatures and maximum stresses are shown in Figure 8 and Figure 9. As the power is now induced in the liquid, temperature gradients and thermal stresses in the crucible are considerably reduced compared to case 3.3.

3.5 Quasi-stationary computations

Quasi-stationary computations have been run and compared with corresponding dynamic simulations. For stable periods the quasi-stationary results agree very well with the corresponding dynamic ones.

3.6 Other Experience and Comments

The model was first converted from COMSOL Multiphysics version 3.5a to version 4.0a. While the Matlab interface was required for the computational procedure in version 3.5a, it was now possible to set up proper studies in the new COMSOL GUI. To perform a stationary study the magnetic equations are solved first. This solution is then applied as initial values for the magnetic field for a computation of the complete system.

The conversion revealed issues that did not work properly in the new COMSOL version. The faults have been corrected one by one by COMSOL and the model now performs well for COMSOL Multiphysics version 4.2.

Dynamic input is now easily described by functions. The functions can depend on various parameters and user functions can be applied in the description of other functions. While our previous model applied Matlab to compute discrete changes, these are now implemented as rapid, dynamic changes. The user effort is far less than before, and the simulations more realistic.

The computations are now faster. While the previous model applied some 12 iterations for a typical stationary case, only around 7 is required in version 4.2.

For the time integration we have applied the experience from the previous testing [1]:

- Very low relative tolerance: $10^{-20} \sim 0$
- Individual, suitable absolute tolerances for each group of variables
- Maximum order 2 for the time stepping
- Not too high maximum time step (2 min)
- (Non-linear) tolerance factor: 0.1 or less

For case 3.4 we found large oscillations in the computed coil current and the power during part of the simulation. The oscillations disappeared after the non-linear tolerance factor had been reduced (from 0.1 to 0.001), in accordance with our previous experience [1].

4. Discussion

The quasi-stationary option is a valuable tool for studying the effect of design issues like geometry (crucible, insulation, location of the coil, etc.), material properties, and frequency. Alternative designs can be studied for typical operational conditions and key figures for each design can easily be compared. Dynamic simulations are far more time consuming (man hours and computer time) and can be more difficult to compare.

Dynamic simulations are required to study operating procedures, especially to get improved insight in transient behavior. Our case studies show that flexible options for control can be important. For the previous version, only power control was available, but it is difficult to manually adjust the power to obtain a requested temperature evolution. The final temperature in Figure 2 is for instance lower than the target value of 1000 °C.

Temperature control ensures correct target temperature, but can involve unrealistic, even unphysical, power levels. If a rapid temperature increase is specified, the required power can be extremely high, while a too fast power decrease would need negative power. Combined power and temperature control is a suitable option than can be made even more flexible with a time dependent tuning factor, k_T . More sophisticated control, like a PID controller, can easily be included.

5. Conclusions

A multiphysics model for industrial induction furnaces has successfully been converted from COMSOL version 3.5a, via versions 4.0a and 4.1, to version 4.2.

The model has been extended to handle:

- Quasi-stationary cases
- Liquid that is either conductive or non-conductive
- Far more complex dynamic cases
- Either the total induced power, a temperature, or a combination can be specified (to determine the coil current).

The model can be applied to study operational conditions, thermal stresses, or design details for a high temperature reactor.

6. References

1. S. A. Halvorsen, A Model for High Temperature Inductive Heating, paper presented at the COMSOL Conference, Milano, Italy (2009)
2. COMSOL Multiphysics User's Guide, Version 4.2 (2011)
3. COMSOL Multiphysics, Structural Mechanics Module, User's Guide, Version 4.2 (2011).
4. P. A. Davidson, An Introduction to Magnetohydrodynamics, Cambridge University Press (2001)
5. N. Kleinknecht and S. A. Halvorsen, High Frequency Magnetohydrodynamic Calculations in COMSOL Multiphysics, poster presented at the COMSOL Conference, Stuttgart, Germany (2011)

7. Acknowledgements

The authors would like to thank Aust- and Vest-Agder counties and the Competence Development Fund of Southern Norway for their financial support.