Excerpt from the Proceedings of the COMSOL Conference 2009 Milan

A Model for High Temperature Inductive Heating

S. A. Halvorsen
Teknova AS
Gimlemoen 19, N-4630 Kristiansand, Norway, sah@teknova.no

Abstract: COMSOL Multiphysics has been applied to develop a model for inductive heating. A coarse, lumped model of the interior of a high temperature reactor is coupled to finite element models for the electromagnetic field, the temperature distribution outside the reactor, and mechanical stresses in the crucible. The model can be applied to study operational conditions, thermal stresses, or design details for a high temperature reactor.

Testing showed that special treatment of the numerical solution procedure is required for this type of highly non-linear model.

Case studies revealed further that the present estimate of non-linear iteration errors during time integration can be inaccurate by orders of magnitude. It is suggested that the overall fundamental balances (energy, material, etc.) are used as additional parameters to check the quality of the non-linear solution.

Keywords: multiphysics, inductive heating, time integration, convergence problems

1 Introduction

High temperature reactors heated by electromagnetic induction are used for various purposes like metal melting, refining, alloying, degassing, etc. This type of reactor is in principle a container with a liquid, surrounded by a thermally and electrically insulating material, c.f. figure 1. An electric coil is located outside the insulation. A brief Internet search revealed that industrial furnaces are available with maximum power input in the range of 50-20,000 kW. Typical frequencies are found in the range 50-1000 Hz. Due to high electric currents, the coil is made from water-cooled copper tubes.

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. Here, we will consider the case where the liquid is non-conductive; and only the crucible is directly heated by electric currents.

The conditions in high temperature reactors are not easily measured, and models are therefore often applied to get improved insight into operational conditions and design issues. The current study focuses on the main thermal conditions, including thermally induced mechanical stresses. The model only takes into account what is required for improved insight into these matters. Construction details, like tubes in the lid, pouring spout, steel casing, etc., are omitted. Further, the model assumes axial symmetry.

The model is implemented in COMSOL Multiphysics version 3.5a, applying three application modes for the electromagnetic field, the temperature distribution, and the thermal stresses. The inner part of the reactor is described by three discrete (lumped) state variables: the mass of the liquid, and the temperatures in the liquid and the void.

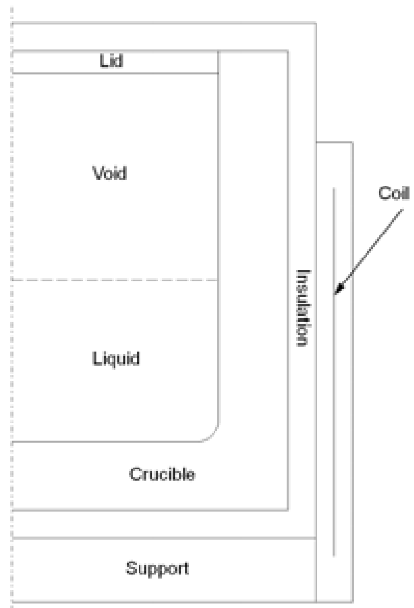

Figure 1: Axially symmetric model geometry

2 Model

2.1 AC Power Electromagnetics

Maxwell's equations need to be solved to find the electric power induced in the furnace. For an axially symmetric problem COMSOL Multiphysics version 3.5a solves for the non-zero angular component of the magnetic vector potential, c.f. the COMSOL documentation for equations and other details [1].

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 small space just outside the coil. It is further limited by the steel casing of the furnace.

The magnetic field will be parallel to the steel casing and the field guides. Hence, "magnetic insulation" is the appropriate boundary condition, except along the center line, where there is axial symmetry.

The coil is described in the model 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 showed, however, that such details would not significantly improve the accuracy of the computed power distribution in the crucible.

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 line element for the coil, perpendicular to the plane in figure 1, is part of standard COMSOL input. But for practical applications, the *total* induced *power* is the critical physical input. As the electromagnetic problem is linear with respect to the dependent variable and the input current density, the *distribution* of the induced power does not depend on the current density. It is then only necessary to solve the electromagnetic problem for an arbitrarily chosen "coil current". We define a Subdomain Integration Variable and let COMSOL compute the total induced power [W], W_{sc}, due to the chosen current:

$$W_{sc} = \iint 2\pi r\, Q_{av}(r,z)\, dr\, dz \qquad (1)$$

$Q_{av}(r,z)$ is the distributed time average resistive heating [W/m^3], one of the standard fields that can be derived from the COMSOL solution. If W_{total} is the total power input [W], the actual (distributed) resistive heating [W/m^3] is given by:

$$Q_{induced} = \frac{W_{total}}{W_{sc}} Q_{av} \qquad (2)$$

The electromagnetic solution is coupled to the heat transfer through the variable $Q_{induced}$. In the opposite direction, the electromagnetic solution will depend on the temperature distribution due to temperature dependent electrical conductivity.

2.2 Heat Transfer by Conduction

The governing equation is

$$\rho c_p \frac{\partial T}{\partial t} + \nabla(-k\nabla T) = Q_{induced} \qquad (3)$$

where ρ is the density [kg/m^3], 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/m^3] given by equation (2).

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)

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

2.3 Inner part of container - Discrete State Variables

The flow parameters for the hot liquid in the crucible are rather uncertain. A flow model for the liquid was therefore not considered, and the liquid has been represented by a lumped model. Above the liquid, heat is transferred mainly by radiation. A simplified model has been adopted where the inner surfaces are assumed to exchange heat with a fictitious body of average (radiation) temperature. When only heating and cooling are studied, the inner part of the container is described by the following three state variables:

- M_l, mass of liquid in the container [kg]
- T_l, temperature of the liquid [K]
- T_v, average radiation temperature above the liquid [K]

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 can contain more than one component.

The equations for the three state variables are:

$$\frac{dM_l}{dt} = m_l \tag{4}$$

$$M_l c_{pl} \frac{dT_l}{dt} = Q_{vl} - Q_{lw} - m_l c_{pl}(T_l - T_{feed}) \tag{5}$$

$$-Q_{vl} - Q_{vw} = 0 \tag{6}$$

where m_l is the filling rate of inflowing liquid [kg/s], c_{pl} is the heat capacity of the liquid [J/kg K], T_{feed} is the feed (input) temperature of inflowing liquid [K], Q_{vl} is the heat flow from the void to the liquid [W], Q_{lw} is the heat flow from the liquid to the crucible wall and bottom [W], and Q_{vw} is the heat flow from the void to the crucible and the lid [W].

Equation (5) describes the conditions when more liquid is added to the crucible. If the matter is charged as solid particles, the heating of the solids and the latent heat (for melting) must also be taken into account.

The heat flows are given by:

$$Q_{vl} = \pi R_l^2 \epsilon_l \sigma (T_v^4 - T_l^4) \tag{7}$$

$$Q_{lw} = \int 2\pi r h_{lw}(T_l - T)ds \tag{8}$$

$$Q_{vw} = \int 2\pi r \epsilon_w \sigma (T_v^4 - T^4)ds \tag{9}$$

where R_l is the inner radius of the crucible at the liquid level (the inner wall is not necessarily vertical) [m], ϵ_l is the effective emissivity of the liquid, ϵ_w is the effective emissivity of the inner wall and lid, $\sigma = 5.67 \cdot 10^{-8}\text{W/m}^2\text{K}^4$ is the Stefan-Boltzmann constant, h_{lw} is the heat transfer coefficient between the crucible and the liquid [W/m^{2}K], and ds is the line differential along the inner boundary of the crucible and the lid. The boundary integral in (8) is taken along the inner crucible boundary from the center at the bottom to the liquid level, while the integral in (9) is taken along the remaining part of the inner boundary.

Q_{lw} and Q_{vw} are implemented as Integration Coupling Boundary Variables in COMSOL Multiphysics, where the integrand is evaluated at a limited number of *integration points* within each boundary element. When the liquid level is gradually raised, this discretization will introduce discontinuities in the integrands each time an integration point is passed. The integrals were therefore smoothed across the liquid level.

The corresponding smooth boundary condition for the heat transfer problem can be written as:

$$\begin{aligned} q = &\text{flc2hs}(z_l - z, \epsilon_z)\, h_{lw}(T_l - T) \\ &+\text{flc2hs}(z - z_l, \epsilon_z)\, \epsilon_w \sigma (T_v^4 - T^4) \end{aligned} \tag{10}$$

where q is the normal heat flux from the liquid or void to the crucible or the lid [W/m^2], and z_l is the liquid level [m]. flc2hs(z, ϵ_z) is a smoothed version of the Heaviside step function [2]. The function is zero when $z < -\epsilon_z$, one when $z > \epsilon_z$, and smoothed in the interval $-\epsilon_z < z < \epsilon_z$. The parameter ϵ_z [m] should be comparable to the linear size of the elements involved.

2.4 Thermal Stresses - Axial Symmetry, Stress-Strain

The crucible will expand considerably when heated from room temperature to operating conditions, while there is no significant expansion of the water-cooled coil. To prevent damage, it is required that the insulation be soft.

Thermal expansion of the lid must also be limited to prevent a significant force on the crucible.

Uneven temperature distribution can cause significant thermal stresses. An Axial Symmetry, Stress-Strain computation is therefore included for the crucible. The model assumes that the crucible can expand freely in all directions, except along the center line where only vertical deformations can take place. One point on the center line must be kept fixed to ensure a unique solution of the equations.

COMSOL's application mode for Axial Symmetry, Stress-Strain (smaxi) is applied with thermal expansion as a load. The strain temperature is taken from the Heat Transfer solution for the crucible, and the strain reference temperature (for zero strain) is set to room temperature. Mechanical load due to pressure from the liquid will be small, and has therefore been neglected. The equations, input parameters, etc., are described in the COMSOL documentation [3].

The equations are linear with respect to the mechanical field variables. They depend on the computed temperatures, but there are no couplings *from* the mechanical problem to other equations. The problem is quasi-static, i.e. the temperatures change with time, while the mechanical equations are stationary.

3 Case Studies

Artificial, but reasonable, material data was chosen for the case studies. The electrical frequency was set to 100 Hz.

First a stationary solution was computed following these steps:

1. Solve the electromagnetic equation
2. Solve the electromagnetic *and* the temperature equations
3. Solve the mechanical equations

Step 2 involves non-linearities and requires several iterations, while step 1 and 3 only involve linear equations (one iteration). Initial testing proved that step 1 is required to provide a reasonable initial value for the power input, $Q_{induced}$, in the temperature problem.

The stationary computations were followed by a dynamic simulation for two different cases.

The model was set up in COMSOL's graphical user interface. Matlab scripts and the Matlab interface were then applied to perform the computations.

As the electrical conductivity in the crucible depends on the temperature, the formulation in equations 2 and 3 imply that all temperatures in the crucible are directly coupled. The formulation worked fine in version 3.4 of COMSOL Multiphysics, but the computational time increased some 20-30 times after upgrading to version 3.5. The problem for version 3.5 was solved by excluding $Q_{induced}$ in the evaluation of the system's Jacobian matrix.

3.1 Instantaneous Filling

A stationary computation was performed with 400 kg liquid in the container and 42.3 kW power. The power had then been adjusted to make the liquid temperature approximately equal to 1000 °C. Then the amount of the liquid was instantaneously changed to 1200 kg with a temperature of 600 °C, by solving fictitious equations for these two state variables. 150 kW was added to the power released in the crucible, and a dynamic simulation was started to simulate the development for two hours. The following numerical parameters were chosen:

```
epsz = 1.e-6;  % Smoothing parameter
rtol = 0.001;  % Relative tolerance
atol = 0.0001; % Absolute tolerance
```

where `epsz` is ϵ_z, the smoothing parameter in equation 10, and `rtol` and `atol` are standard parameters for COMSOL's time integration. The maximum time step was set to 240 s.

Some computed temperatures are shown in figure 2: T_v, T_l, and the maximum temperature in the crucible (Max T). The average radiation temperature in the void, T_v, shows irregular behavior after some 95 minutes. The case was therefore rerun with the optional parameter `ntolfact = 0.1`. The tolerance for errors due to non-linear equation solving will then be reduced by this factor, compared to the time stepping tolerance. The solution still showed some irregularities, and `ntolfact = 0.01` was therefore tested. The temperatures for this run showed smooth time evolution, c.f. figure 3. The corresponding maximum stresses in the cylindrical coordinate directions, φ, z, and r, are plotted in figure 4.

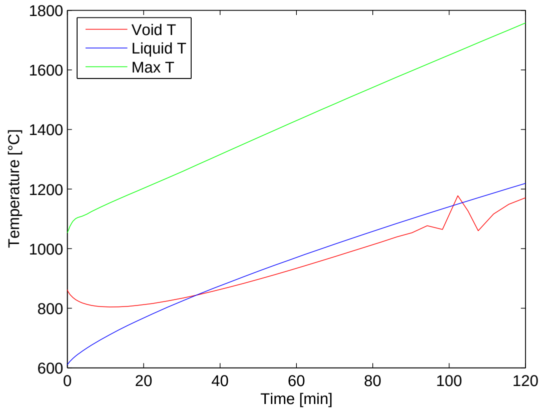

Figure 2: Case 1 - First run.

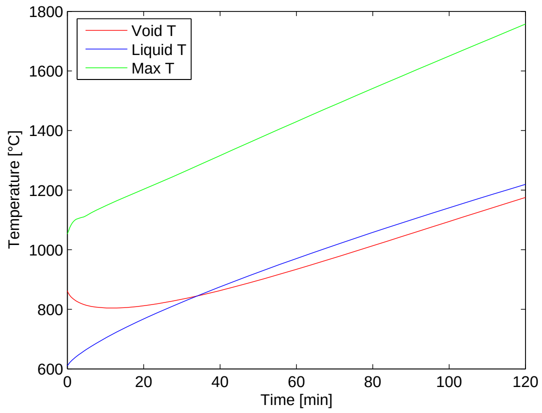

Figure 3: Case 1 - `ntolfact = 0.01`.

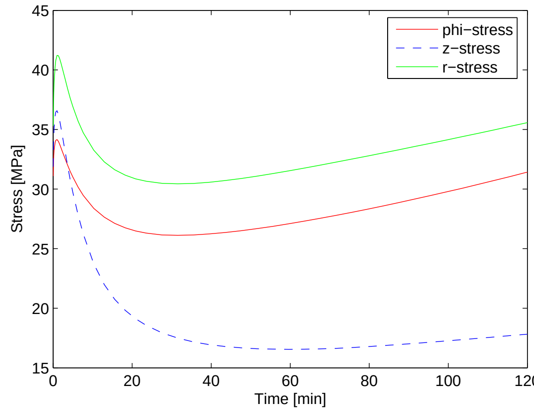

Figure 4: Case 1 - Maximum stresses in crucible.

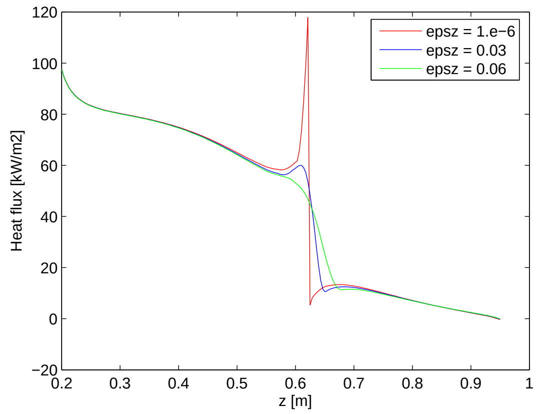

Figure 5: Case 1 - Normal heat flux from the crucible wall to the inner parts after 20 minutes.

Finally, a few runs were made for different values of the smoothing parameter in equation 10. The influence on the heat flux from the crucible wall to the liquid and the void is illustrated in figure 5, where the liquid level is located at $z = 0.624$ m, and the simulation time is 20 minutes.

3.2 Gradual Filling

800 kg "cold" liquid at 400 °C was charged at a rate of 1600 kg/h in case 2 . The filling rate was gradually increased during two minutes at the start and gradually diminished at the end by applying the function flc2hs; implying 32 minutes for the total charging period.

For an initial test with the same numerical parameters as for case 1 the error in the final computed amount of liquid was several kg. The relative tolerance, `rtol`, was then set to zero and individual tolerances, `atol`, were set for each of the state variables: 0.01 kg for the mass, 0.1 K for temperatures, and values corresponding to `rtol = 0.001` for the remaining ones. The error in the amount of liquid was then reduced to about 1 kg, or less.

Several cases were run for different values of `epsz`, the smoothing parameter in equation 10. The values correspond to almost zero, and $\frac{1}{2}$, 1, 2 and 3 times the maximum mesh size. A summary of the main time integration results are shown in table 1. *CPU* is elapsed CPU time in seconds, *Steps* are the number of time steps, *Jac* is the number of evaluations of the Jacobian matrix, and *Sol* is the number of linear solutions.

In order to increase the accuracy of the thermal boundary condition towards the inner part of the crucible, the integration order for the boundary elements was increased from 4 (standard) to 10 along the critical (vertical) boundary. The main time integration results are shown in table 2

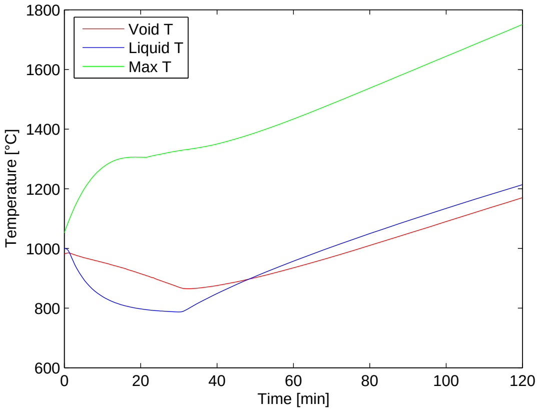

Figure 6: Case 2 - Main temperatures.

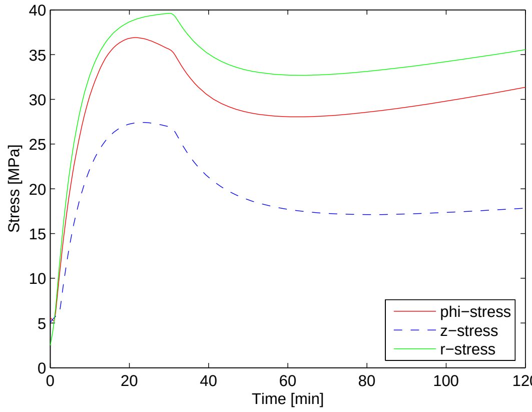

Figure 7: Case 2 - Maximum stresses in crucible.

Only minor variations were found for the main results, shown in figures 6 and 7 for `epsz = 0.03 m`. The maximum void temperature varied for instance within 2 K, while the maximum stress in the r-direction varied within 0.2 MPa.

Figures 6 and 7 show smooth behavior. However, the computed heat flux along the inner vertical boundary varied irregularly, especially for low values of `epsz`, c.f. figure 8. The irregular oscillations disappeared when `ntolfact` was lowered from 0.01 to 0.001 (not shown in the figure).

4 Discussion

4.1 Case Studies

The model can be applied to study operational conditions, thermal stresses, and design issues for the reactor. Main results have been presented for two simple cases in figures 3, 4, 6, and 7.

For case 1 the liquid temperature was suddenly dropped to 600 °C after adding more liquid instantaneously. At the same time, the power was increased and the temperatures started to grow steadily. Figure 3 shows that the liquid temperature returned to the stationary value, 1000 °C, after some 65 minutes. During this time period, the maximum temperature (in the crucible) was increased by some 400 K, i.e. a lot of power was spent to increase the temperature of the relatively thick crucible far beyond the stationary state.

Figure 4 shows that the crucible experienced a thermal shock just after the “cold” liquid had been added. The mechanical stresses increased to maximum values after about one minute. Then the stresses were reduced, before they started to increase again. The final stress growth was due to increasing temperature differences from the power input to the crucible.

For case 2, gradual filling during $\frac{1}{2}$ hour, the liquid temperature fell gradually to about 790 °C while the maximum temperature was increased to some 1330 °C during the filling period, c.f. figure 6. Temperature differences in the crucible were caused by both cooling from the liquid and heating by the released power, and the maximum stress was gradually raised to almost the same level as for instantaneous filling.

epsz	CPU	Steps	Jac	Sol
1e-6	583	370	257	1022
0.015	107	99	22	308
0.03	79	93	17	210
0.06	72	80	16	196
0.09	73	72	15	210

Table 1: Time integration summary, case 2. Standard integration order along all boundaries.

epsz	CPU	Steps	Jac	Sol
1e-6	301	210	75	834
0.015	94	99	21	245
0.03	79	89	17	214
0.06	73	83	14	204
0.09	74	72	15	210

Table 2: Time integration summary, case 2. High integration order along the critical boundary.

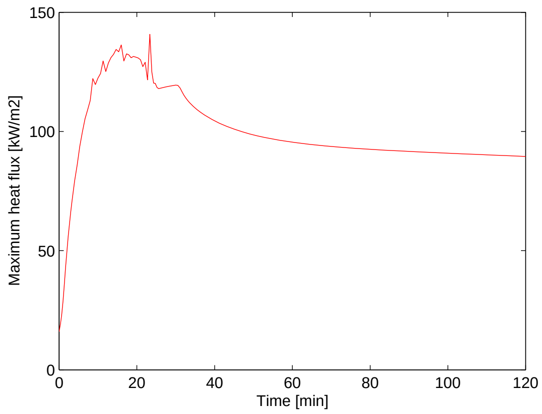

Figure 8: Case 2 - Maximum normal heat flux on the vertical inner boundary, `epsz = 0.03 m`.

After the filling period the temperatures grew. The stresses diminished for some period before they started to increase due to increasing temperature differences.

Additional output from COMSOL can be used to improve the understanding of the two cases.

Further simulations can be run in order to find: improved time dependent power input to minimize the variations in the liquid temperature, operational conditions or design modifications to reduce the mechanical stresses, etc. The model can easily be modified to include more effects, like induction and/or reactions in the liquid, or some induction in the lining.

4.2 Time Integration Issues

The COMSOL code discretizes the partial differential equations involved and combines them with the equations for the model's discrete state variables to form a system of non-linear differential algebraic equations (DAEs). This system is then solved by a code that should keep the errors below specified tolerances [2]. There are two kinds of errors to be considered:

1. Time integration error - Difference from the exact solution due to the time stepping method. This type of error can be made arbitrarily small by reducing the time step. If the estimated error is smaller than the tolerance, the time step is increased; and vice versa.

2. Non-linear iteration error - Error due to limited number of iterations and rate of convergence. Can be reduced by more iterations and new evaluation(s) of the system's Jacobian matrix. Can be close to, but not less than, the number precision of the computer.

The evaluation and subsequent decomposition of the Jacobian matrix is comparatively time consuming. The code will therefore try to avoid new evaluations until it is required due to slow convergence of the non-linear iterations.

For case 1, the relative tolerance for both types of errors was set to 0.001, and the errors for one time step should be around 1 K or less for the temperatures. Figure 2 shows, however, some irregular short time variations of magnitude around 100 K. These variations disappeared when the non-linear tolerance was reduced by a factor of 0.01. They were therefore caused by high non-linear iteration errors.

For case 2, such variations can not be spotted for the plotted temperatures. However, irregular variations were found for the maximum heat flux towards the inner part of the crucible, some 10 kW when `epsz = 0.03 m`. Such irregularities correspond to temperature variations around 5 K, as the boundary condition in this region was heat flux with a heat transfer coefficient of 2000 W/m^2 K. This is far more than the tolerance for non-linear iteration errors, which was `atol*ntolfact = 0.001 K`. The irregularities disappeared after `ntolfact` was sufficiently reduced.

The examples show that the estimate for the non-linear errors can be inaccurate by orders of magnitude. This is probably a general problem for time integration, i.e. not COMSOL specific.

For the case studies, it was found a variable or an expression where irregular behavior would reveal unsatisfactory convergence. Sufficient accuracy could then be obtained by lowering the tolerance for the non-linear iterations. This method has some drawbacks:

- There exist no guidelines on how to find a suitable expression.

- An expression that works well for one problem may not be sufficient for another one.

- Lower non-linear tolerances will increase the computational time during *all* parts of the simulation, i.e. also where it is not required to obtain sufficient accuracy.

What is needed is an improved error estimate or some additional, reliable expression(s) which the program can apply to detect when the estimate should not be trusted.

Some 20 years ago, the author experienced the same problem using a code for time integration of ordinary differential equations (ODEs). At that time, the problem was solved by monitoring the overall material and energy balances for the equations involved and reducing the time step whenever these balances exceeded a specified tolerance. For this ODE system it was proven that any deviation from zero balances would be due to non-linear iteration errors.

Proposal: Monitor the overall fundamental balances (energy, material, ...) and continue the non-linear iterations whenever at least one balance is above its specified tolerance.

This will require modifications in the COMSOL code and appropriate formulations by the user. The balances should be formulated such that they will be zero if the non-linear iteration errors are zero. It remains to be shown whether this is possible for the finite element method and the chosen DAE solver.

If the proposal should not be sufficient, other criteria should be tested. It is then required that the COMSOL code allows for additional, user specified tests, to check if the non-linear iterations have converged.

Figures 2 and 3 show that the liquid temperature started around 610 instead of 600 °C. The modified start temperature was due to the standard consistent initialization of the DAE system in COMSOL Multiphysics. Correct start temperature can be obtained by turning off the consistent initialization or reducing the initial time step. This will improve the resolution of the initial transient at the cost of more computational effort. The author recommends that the user should be informed about the size of the perturbations due to consistent initialization.

4.3 Smoothing

Without smoothing of the thermal boundary condition along the vertical inner part of the crucible, there will be large "overshoot" for the heat flux around the liquid level, c.f. figure 5 for `epsz = 1.e-6`. Such high overshoot is caused by the use of a rough model for the inner part of the crucible, and some smoothing is required to get more realistic results.

Generally, a model should be reasonably balanced, i.e. when one part of the model is coarse, other parts should not be too detailed. The heat flux to the crucible can therefore be somewhat coarse. The author recommends a value for `epsz` around the element size. Further, a very fine element grid will not improve the overall accuracy of the model.

Numerically, table 1 and 2 indicate that `epsz` around 1-2 times the element size should be chosen to reduce the computational time.

Increased (spatial) integration order for the boundary elements will significantly improve the accuracy of the boundary condition around the liquid level when `epsz` is lower than the size of one element. Significant improvements imply smoother variations when the liquid level changes, which will reduce the computational time, compare table 1 and 2.

5 Conclusions

COMSOL Multiphysics has successfully been applied to develop a model for inductive heating. The model can be applied to study operational conditions, thermal stresses, or design issues for a high temperature reactor.

Special treatment of the numerical solution procedure is required for this type of highly non-linear model.

The present estimate for the non-linear iteration errors during time integration can be inaccurate by orders of magnitude. It is suggested that the overall fundamental balances (energy, material, ...) are utilized as an additional criterion to check whether the non-linear iterations have converged.

Appropriate smoothing is recommended along the boundary that couples the lumped state variables to the finite element field variables.

References

[1] *COMSOL Multiphysics Modeling Guide*, Version 3.5a, 2008.

[2] *COMSOL Multiphysics Reference Guide*, Version 3.5a, 2008.

[3] *COMSOL Multiphysics, Structural Mechanics Module, User's Guide*, Version 3.5a, 2008.