UPPSALA
UNIVERSITET

Magnetic diffusion in a conductor

Author: Guanwen Ying and Mikael Dahlqvist
Supervisor: Anders Larsson

Report in scientific computing

June 2008

DEPARTMENT OF INFORMATION TECHNOLOGY

Abstract

This report will show a simulation of the magnetic field diffusion in a conductive rod between two plates together with external circuit. It will go into some depth on the theoretical background and describe how to use COMSOL Multiphysics to implement the model. Three models are used electrical, heating and an external circuit model. The results will show distribution of current and temperature in the model.

Content

1 Introduction

Electrical power is used in every day life for cooking, communication, computers, sparkplugs, welding etc. In many of these applications the rapid potential change in the discharge will give rise to non-uniformed currents e.g. sparkplugs. The non-uniformly distributed current will lead to a non-uniformly distributed heating effect, and the efficiency of the electrical appliance will be degraded if these effects are not accounted for when designing electrical devices. These kinds of physical effects will be studied in COMSOL by using a very simple geometry with two plates and a rod between the two plates. An external circuit with a capacitance and an inductance in serial is placed between the two plates (Fig. 1). The non-uniformly distributed current density is caused by the well known 'skin effect'. Temperature distribution may also be non-uniformed due to the joule heating phenomena. The electrical conductivity which is both temperature and space dependent will also contribute to non-uniformly distribution of the current density, further more the temperature increase of the rod is not monotone due to the heat conduction and energy absorbing when temperature reaches either melting or vaporization point. The paper will discuss what is happening in the whole process with some motivation of formulas, as well as the methods to implement such behavior into the model.

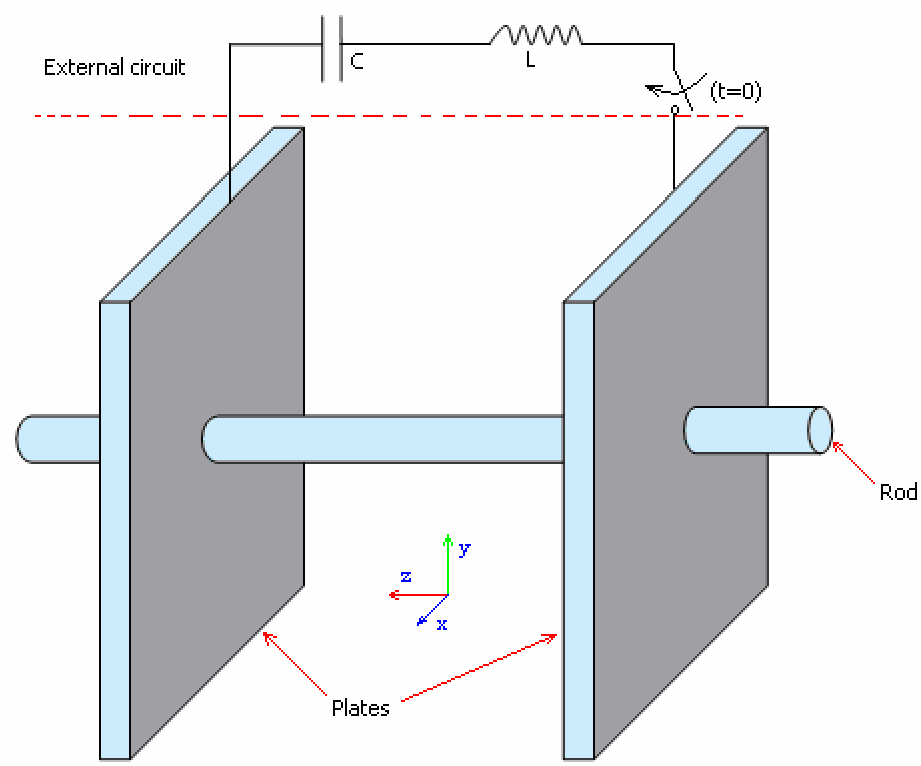


Figure 1: A simple description of geometry together with circuit, the drawing is not accurate in scale

2 Geometry

The model is assembled with two Copper plates and one Copper rod penetrating both plates. The plates are 20×20 mm in x and y direction and has thickness of 1.0 mm in the z direction. The distance between the plates is 4.5 mm. The rod itself has a length of 7.0 mm and a radius of 1.0 mm (Fig. 2).

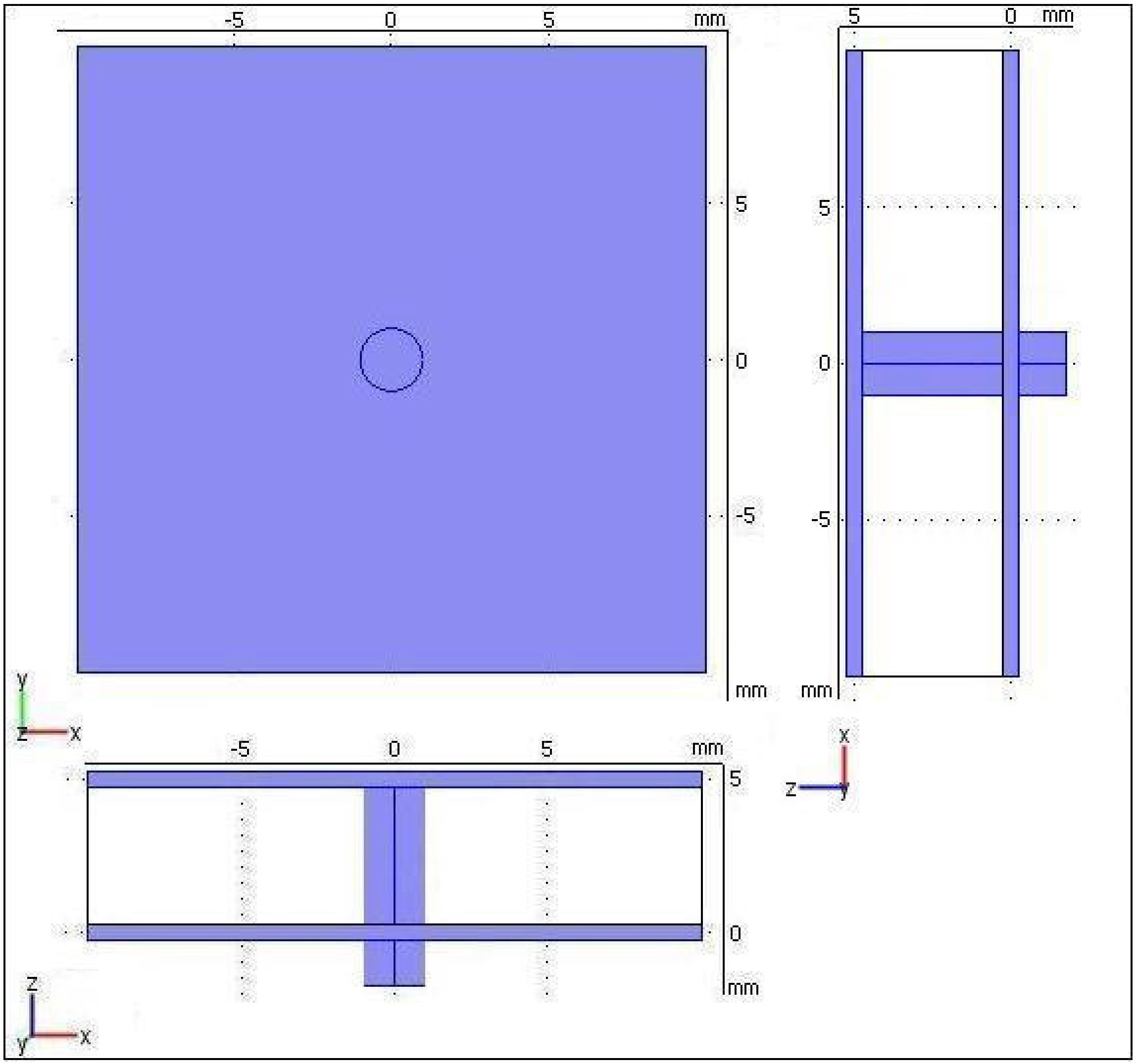


Figure 2: Geometry description of rod and plates from the front, above and side.

3 Induction current model

3.1 Basic model equation

Maxwell equation in Copper and Air:
$$\begin{cases} \nabla \cdot \vec{E} = \dfrac{\rho}{\varepsilon} \approx 0 & (1) \\ \nabla \cdot \vec{H} = 0 & (2) \\ \nabla \times \vec{E} = -\mu \dfrac{\partial \vec{H}}{\partial t} & (3) \\ \nabla \times \vec{H} = \vec{J} + \varepsilon \dfrac{\partial \vec{E}}{\partial t} \approx \vec{J} & (4) \end{cases}$$

Take curl on both sides of equation (4)

$$\nabla \times \left(\nabla \times \vec{H} \right) = \nabla \times \vec{J} \tag{5}$$

Then put Ohm's law $\vec{J} = \sigma \vec{E}$ into equation (5)

$$\nabla \times \left(\nabla \times \vec{H} \right) = \nabla \times \left(\sigma \vec{E} \right) \tag{6}$$

Together with equation (3)

$$\nabla \times \left(\nabla \times \vec{H} \right) = \nabla \times \left(\sigma \vec{E} \right) = -\mu_0 \mu_r \sigma \frac{\partial \vec{H}}{\partial t} \tag{7}$$

where as σ is both temperature and space dependent. Since the equation of magnetic force

$$\mu_0 \mu_r \vec{H} = \nabla \times \vec{A} \tag{8}$$

where A is the magnetic potential, and μ_0, μ_r is the permeability in vacuum and relative permeability respectively. Then put equation (8) into equation (7).

$$\nabla \times \left(\nabla \times \mu_0^{-1} \mu_r^{-1} \left(\nabla \times \vec{A} \right) \right) = -\sigma \frac{\partial \nabla \times \vec{A}}{\partial t} \tag{9}$$

Cancel curl on both sides, and we get final model equation, together with electrical conductivity as a function of temperature

$$\begin{cases} \sigma \dfrac{\partial \vec{A}}{\partial t} + \nabla \times \left(\mu_0^{-1} \mu_r^{-1} \nabla \times \vec{A} \right) = 0 \\ \sigma = \dfrac{\sigma_0}{1 + \alpha (T - T_0)} \quad T < 1356K \\ \sigma = \dfrac{10^6}{0.11031 + 7.83066 \cdot 10^{-5} \cdot T} \quad 1356K < T < 2855K \end{cases} \tag{10}$$

3.2 Boundary condition

1. Boundary between copper and air

$$\vec{n}\times\vec{H}=\vec{J}_s \quad (11)$$

Let $\vec{H}_1$ and $\vec{H}_2$ to be the magnetic field in copper and air respectively, from equation (11)

$$\vec{n}\times\vec{H}_1=\vec{J}_s=\vec{n}\times\vec{H}_2 \quad (12)$$

This simplified as:

$$\vec{n}\times\left(\vec{H}_1-\vec{H}_2\right)=0 \quad (13)$$

2. Boundary on the air between plates where the electric potential is applied

$$\vec{n}\times\vec{H}=\vec{J}_s \quad (14)$$

3. Boundary between simulation domain and outside area

For simulation of this problem, a simple cut-off method is applied, and this results the all rest boundaries to be electric insulation, which satisfies

$$\vec{n}\times\vec{H}=0 \quad (15)$$

This means no surface current on the rest of boundaries, thus electric insulated.

4 Heat conduction model

4.1 Basic model equation

$$\begin{cases} \sigma=\dfrac{\sigma_0}{1+\alpha(T-T_0)} & T<1356K \\ \sigma=\dfrac{10^6}{0.11031+7.83066\cdot 10^{-5}\cdot T} & 1356K<T<2855K \\ c_{p(s,l)}m\dfrac{\partial T}{\partial t}-\nabla\cdot\left(\lambda\nabla T\right)=\dfrac{\vec{J}^2}{\sigma} & \\ \lambda=L_o\cdot T\cdot\sigma & \end{cases} \quad (16)$$

Note: both temperature and electric conductivity are time and space dependent, different heat capacity for solid and liquid state of rod is used.

4.2 Material parameters

When the temperature reaches the melting and vaporization point, the energy term $\frac{\vec{J}^2}{\sigma}$ which is generated by circuit is absorbed by heat of fusion and vaporization, so that temperature should remain constant, as soon as enough energy is absorbed, the temperature begin to rise. Thus the original equation becomes four individual parts:

$$\begin{cases} \sigma = \dfrac{\sigma_0}{1+\alpha(T-T_0)} \\ \lambda = L_o \cdot T \cdot \sigma \qquad \textit{such that} \quad T < 1356K \\ c_{ps} m \dfrac{\partial T}{\partial t} - \nabla \cdot (\lambda \nabla T) = \dfrac{\vec{J}^2}{\sigma} \end{cases} \tag{17}$$

$$\begin{cases} \sigma = \dfrac{\sigma_0}{1+\alpha(T-T_0)} \\ \lambda = L_o \cdot T \cdot \sigma \qquad \textit{such that} \quad T = 1356K \;\&\& \int_{t_1}^{t} \dfrac{\vec{J}^2}{\sigma} dt < w_m \cdot m \\ \dfrac{\partial T}{\partial t} = 0 \end{cases} \tag{18}$$

$$\begin{cases} \sigma = \dfrac{10^6}{0.11031 + 7.83066 \cdot 10^{-5} \cdot T} \\ \lambda = L_o \cdot T \cdot \sigma \qquad \textit{such that} \quad 1356K < T < 2855K \\ c_{pl} m \dfrac{\partial T}{\partial t} - \nabla \cdot (\lambda \nabla T) = \dfrac{\vec{J}^2}{\sigma} \end{cases} \tag{19}$$

$$\begin{cases} \sigma = \dfrac{10^6}{0.11031 + 7.83066 \cdot 10^{-5} \cdot 2855} \\ \lambda = L_o \cdot T \cdot \sigma \qquad \textit{such that} \quad T = 2855K \;\&\& \int_{t_2}^{t} \dfrac{\vec{J}^2}{\sigma} dt < w_v \cdot m \\ \dfrac{\partial T}{\partial t} = 0 \end{cases} \tag{20}$$

Note: t_1 and t_2 is the time that copper reaches the melting and vaporization point, different points in the geometry will have different value of t_1 and t_2, COMSOL will select one of the equations from (17) (18) (19) and (20) to solve the problem for every mesh even at the same time step.

5 External circuit models

5.1 Basic model equations

When the rod and plates are serially connected with the external circuit, it can be regarded as impedance which may have inductance in some cases (Fig. 3).

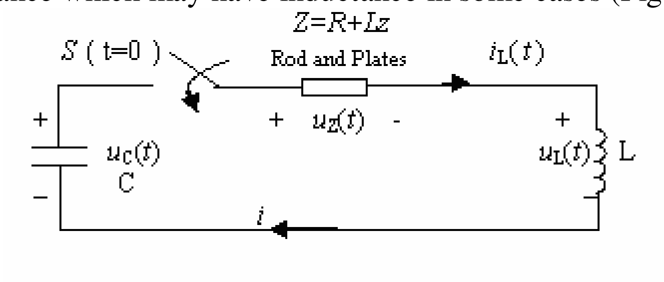


Figure 3: Simplified circuit model regarding rod and plates as an impedance

Since u_Z is the potential drop between two plates. we have the relation that:

$$u_C = u_Z + u_L \tag{21}$$

$$i = -C\frac{du_C}{dt} \tag{22}$$

$$u_L = L\frac{di}{dt} \tag{23}$$

Put equation (22) into (23) and got

$$u_L = -LC\frac{d^2u_C}{dt^2} \tag{24}$$

Put equation (24) into (21) and got

$$u_C + LC\frac{d^2u_C}{dt^2} - u_Z = 0 \tag{25}$$

In order to implement equation (25), the potential drop u_Z between two plates should be known. It can be integrated in the blue line (Fig. 4).

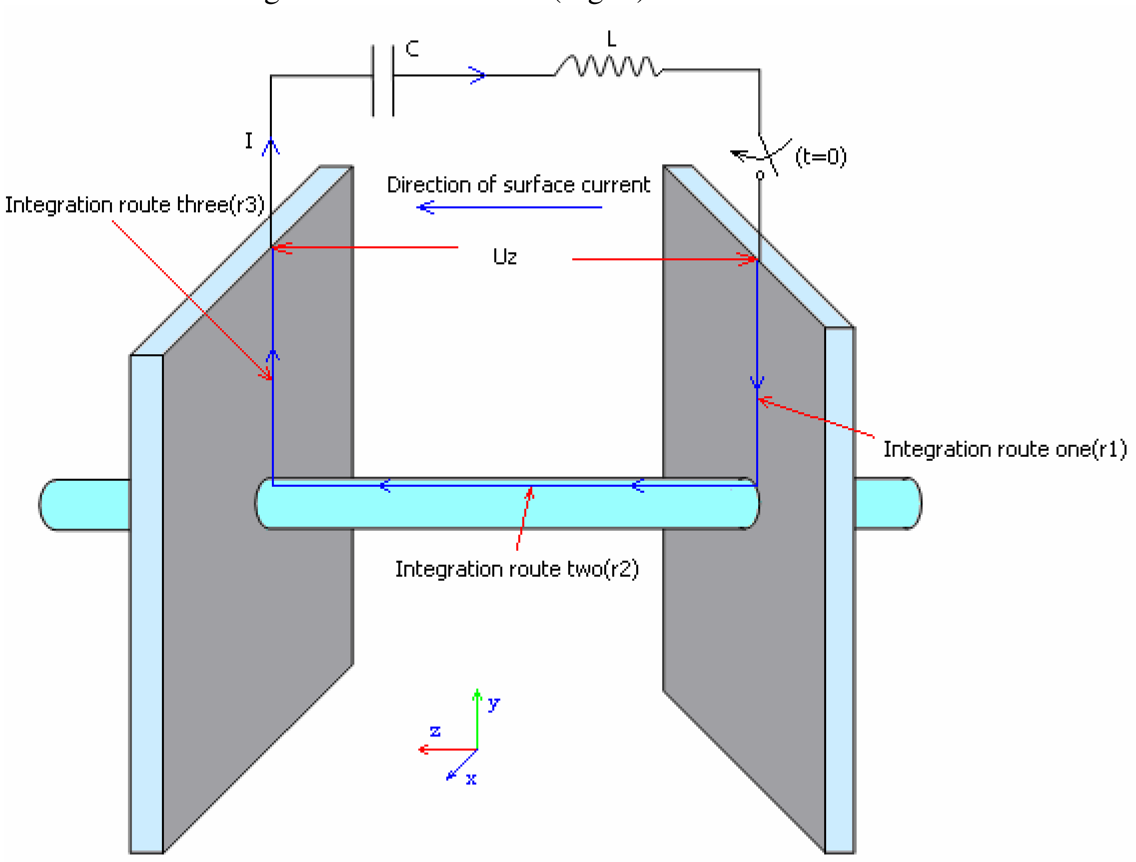


Figure 4: A brief description on how to integrate potential drop over two plates

If z direction is the positive direction for surface current in equation (14), then the electric field is going from the right plate to the left plate as shown in (Fig. 4). Since electric field is a conservative field, we can take any curve to integrate over it and get the potential drop over two plates. For simplicity, we integrate over the blue lines shown in (Fig. 4).

$$\vec{E} = -\nabla u \tag{26}$$

In a Cartesian coordinates, equation (26) can be written as

$$\vec{E} = -\frac{\partial}{\partial x} u \cdot \vec{e}_x - \frac{\partial}{\partial y} u \cdot \vec{e}_y - \frac{\partial}{\partial z} u \cdot \vec{e}_z \tag{27}$$

Whereas $\vec{e}_x$ $\vec{e}_y$ $\vec{e}_z$ are the normal vectors to x, y and z axis respectively. Thus, the potential can be written as the integral of electric field

$$u = -\int E_x dx - \int E_y dy - \int E_z dz \tag{28}$$

Therefore, the integral over route one as shown in (Fig. 4) can be written as

$$u_1 = -\int_{r1} E_x dx - \int_{r1} E_y dy - \int_{r1} E_z dz \tag{29}$$

Since route one is going only in y direction, the first and last term of (29) is zero, thus

$$u_1 = -\int_{r1} E_y dy \tag{30}$$

Similarly, we can get integral of route two and three

$$u_2 = -\int_{r2} E_z dz \qquad (31)$$

$$u_3 = -\int_{r3} E_y dy \qquad (32)$$

Then, the potential drop u_Z is u_1 u_2 *and* u_3 times the direction of axis respectively and sum up together: $u_z = u_1 \cdot negtiveY + u_2 \cdot positiveZ + u_3 \cdot positiveY$ in other words,

$$u_z = -U_1 - U_2 + U_3 \qquad (33)$$

After putting equation 31, 32 and 33 into 34, we finally get

$$u_z = \int_{r1} E_y dy - \int_{r2} E_z dz - \int_{r3} E_y dy \qquad (34)$$

So equation (25) becomes

$$u_C + LC\frac{d^2 u_C}{dt^2} - \left(\int_{r1} E_y dy - \int_{r2} E_z dz - \int_{r3} E_y dy\right) = 0 \qquad (35)$$

5.2 Initial condition

At the very beginning, the capacitance was charged to 4600 V and there are no current in the circuit before switch is closed as shown in (Fig. 3), this lead to

$$u_C\big|_{t=0} = 4600 \qquad (36)$$

$$i\big|_{t=0} = 0 \qquad (37)$$

It can be derive from equation (22) that the initial time derivative of potential over capacitance is

$$\left.\frac{du_C}{dt}\right|_{t=0} = -\frac{i\big|_{t=0}}{C} = 0 \qquad (38)$$

6 Using a prescribed current

6.1 Modeling condition

All results in section 6 are got from implementation of the following models:

1. Induction current model
2. Heat conduction model with heat of fusion and vaporization
3. A prescribed current

A simple current impulse is used as prescribed current; it alternates from zero to 200 kA within 40 μs, and drop to zero gradually as shown in (Fig. 5) it has almost the same behavior as the external circuit which is going to be implemented in section 7.

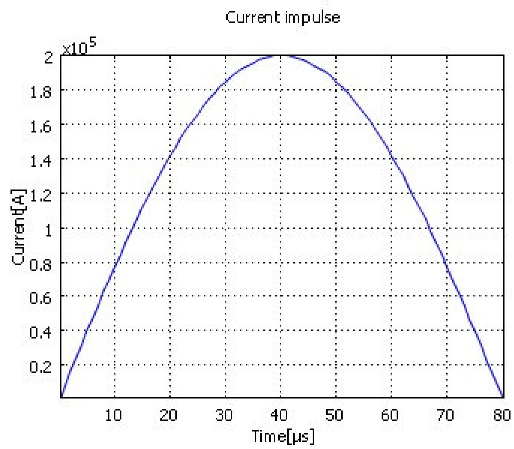


Figure 5: Original current impulse used as a prescribed current

In section 6, all slice cut in the rod is made by a plane orthogonal to z axis, at z=1 mm. The cut plane is indicated by red area in (Fig.6).

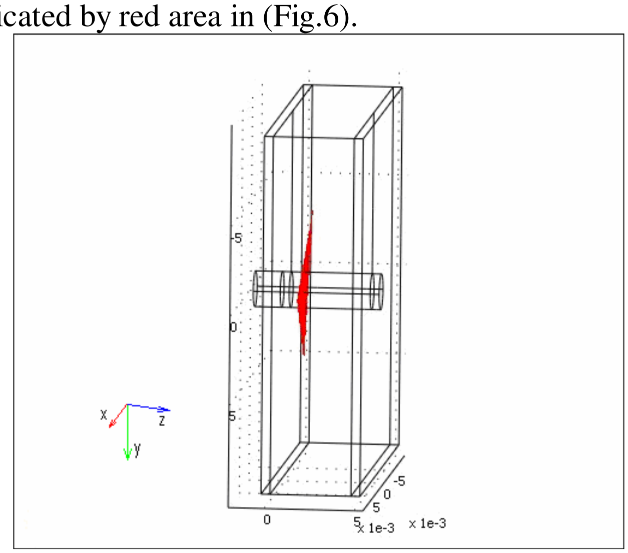


Figure 6: The exact place of slice cut in rod throughout section 6

6.2 Results for induction current model

6.2.1 Skin effect

The magnetic field diffusion in a conductor resulted in non-uniformly distributed current density, which is famously known as skin effect.

The skin effect is the tendency that the current density near the surface of the conductor is greater than that at its core. That is, the electric current tends to flow at the "skin" of the conductor. The slice cut of the rod perfectly shows the skin effect (Fig. 7), the line plot which is taken from the diameter line of circle (Fig. 8) shows how current density differs from center to surface within the rod.

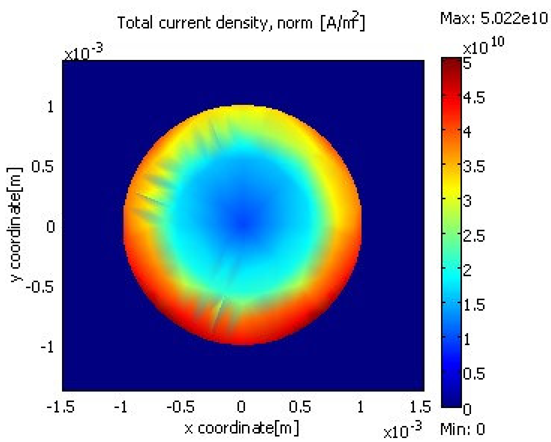


Figure 7: Slice cut of current density in rod at t=15 µs

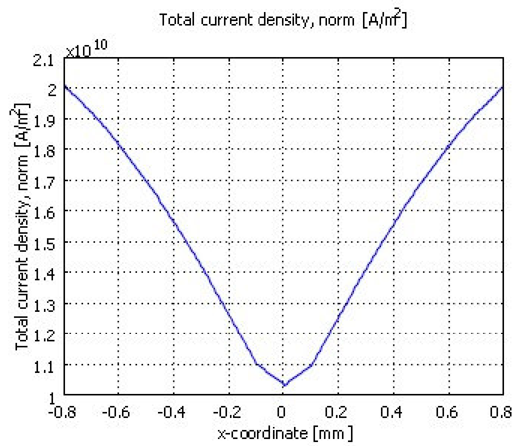


Figure 8: Diameter line of current density in rod at t=15 µs

6.2.2 Resistance heating

The resistance heating has the same behavior as the current density since the current is generating heat. The (Fig. 9) the slice cut of rod and (Fig. 10) diameter line of circle which is parallel to x coordinate show how the resistance heating distributed.

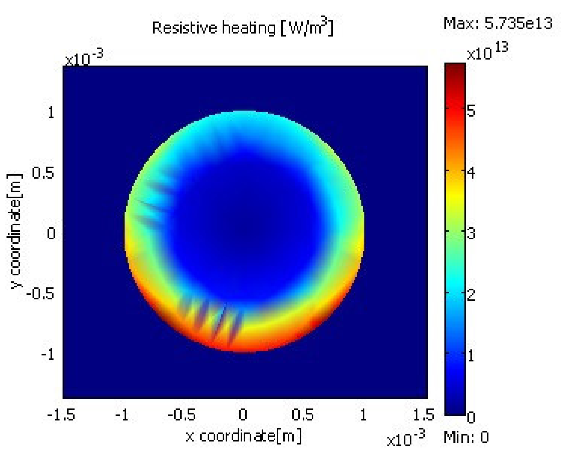


Figure 9: Slice cut of resistance heating in rod at t=15 µs

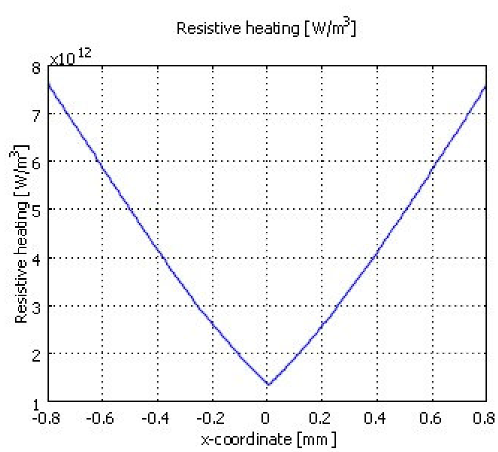


Figure 10: Diameter line of resistance heating in rod at t=15 µs

6.2.3 Current distribution in relation with time

Although the skin effect takes place and the current distribution is non-uniformed at the beginning, it won't always be the case. Judging from the current distribution in the rod at different time steps (Fig. 11~15), the current density is almost uniformly distributed after 30 μs.

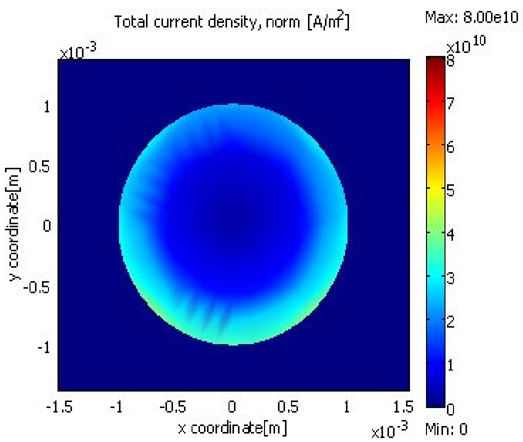


Figure 11: Slice cut of Current density in rod at t=10 μs

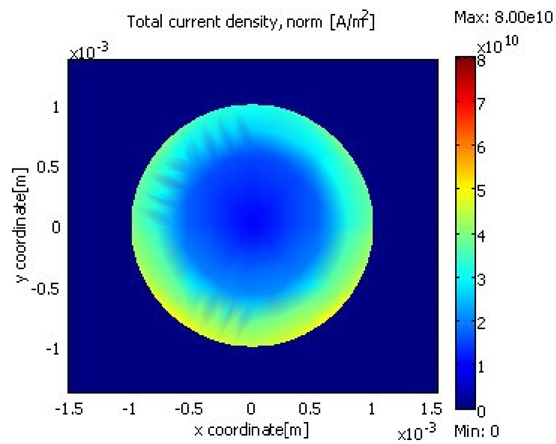


Figure 12: Slice cut of Current density in rod at t=15 μs

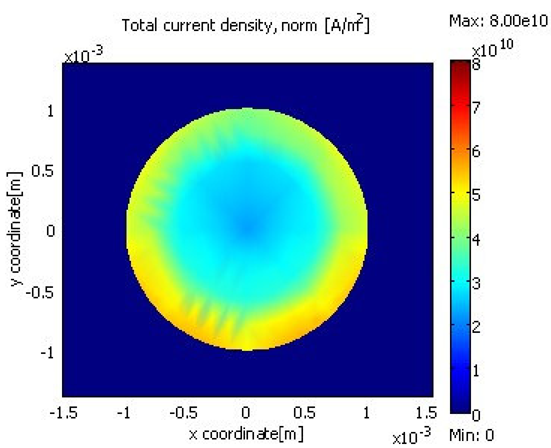


Figure 13: Slice cut of Current density in rod at t=20 μs

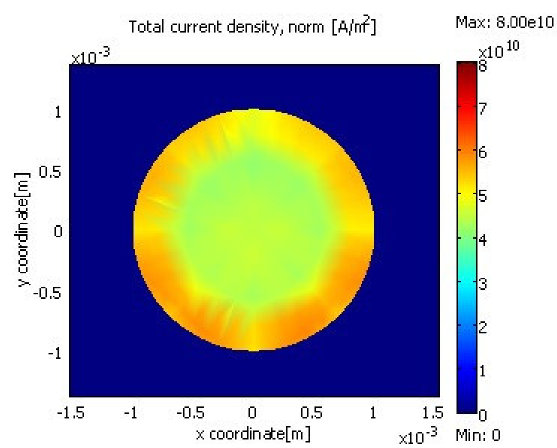


Figure 14: Slice cut of Current density in rod at t=25 μs

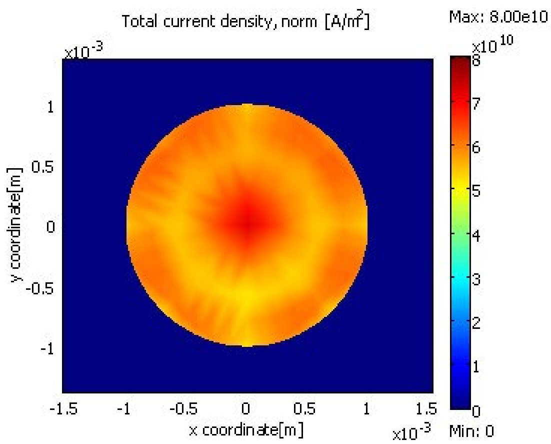


Figure 15: Slice cut of Current density in rod at t=10 µs

6.3 Results for heat conduction model

6.3.1 Temperature behavior over time

Since the material parameter is implemented, the temperature will not be monotone as a function of time; the curve will be rather flat at some period of time during melting and vaporization point. This is so called 'heat of fusion' and 'heat of vaporization'. (Fig. 16) and (Fig. 17) showing temperature of different points in rod are perfectly corresponding to what we are expecting.

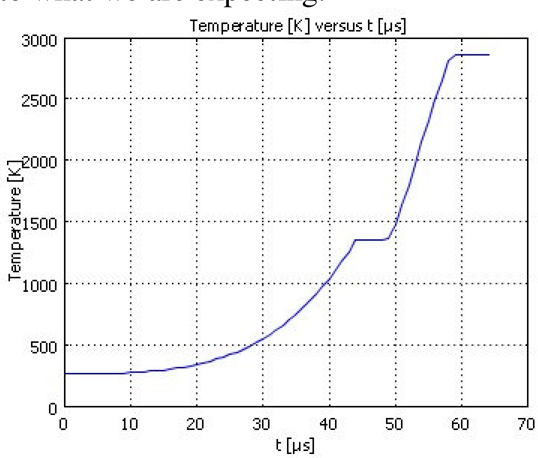


Figure 16: Temperature over time at point (0.1 mm, 0.8 mm, 2 mm)

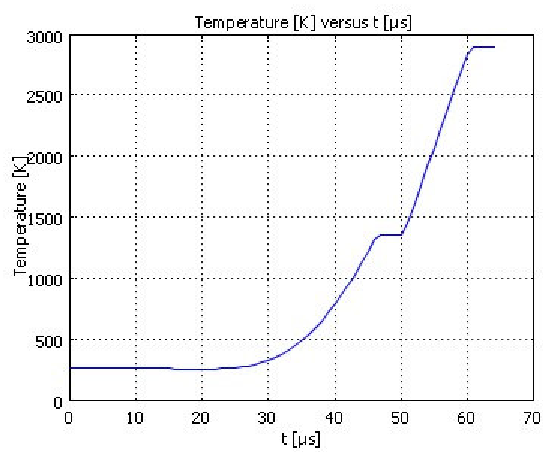
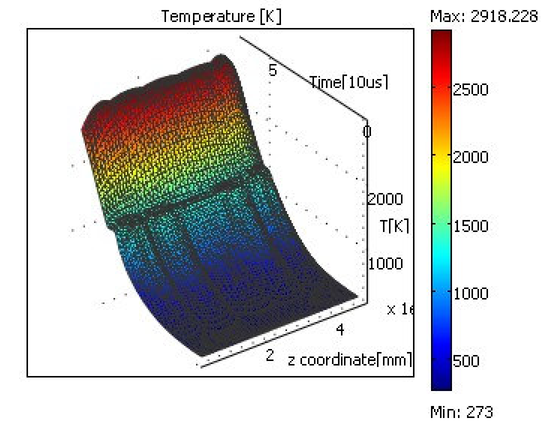


Figure 17: Temperature over time at point (0.1 mm, 0.1 mm, 4 mm)

The temperature in rod does not vary a lot on the line in parallel with z axis within the rod (for z axis see Fig. 1) since the rod is really small. (Fig. 18) and (Fig. 19) are taken from two different lines parallel with z coordinate on the rod surface. (Fig.18) is taken from the top surface of the rod, while (Fig. 19) is taken from the bottom surface of the rod.



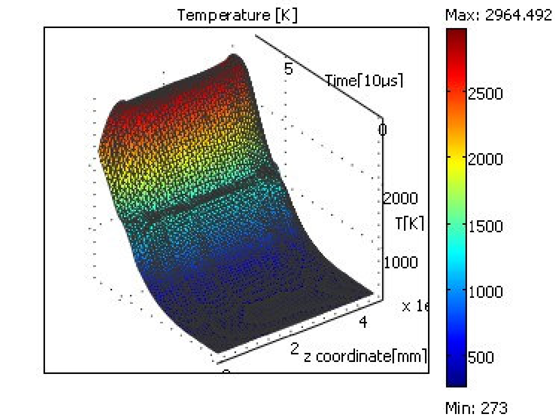


Figure 18: Temperature on top surface line parallel with z coordinates over time

Figure 19: Temperature on bottom surface line parallel with z coordinates over time

6.3.2 Temperature distribution

The temperature distribution is uniformed at most of the time, but disturbance can be seen in distribution at some time step (e.g. at 30 μs is almost non-uniform distribution). (Fig. 20~26) show temperature distribution at different time steps.

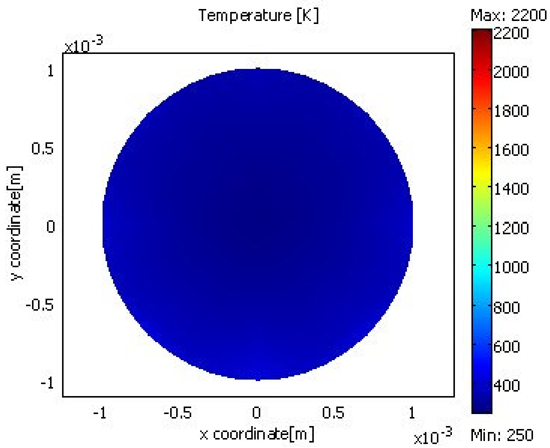


Figure 20: Slice cut of temperature distribution in rod at t=20 μs

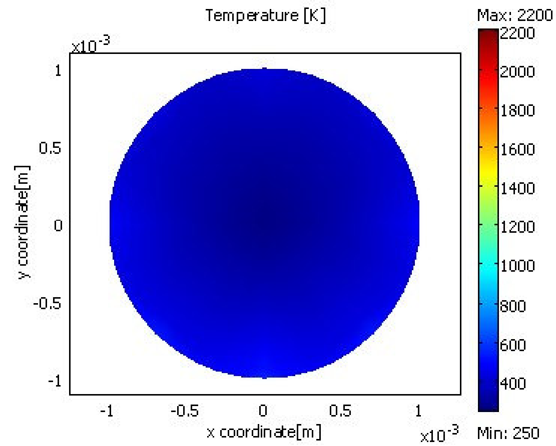


Figure 21: Slice cut of temperature distribution in rod at t=25 μs

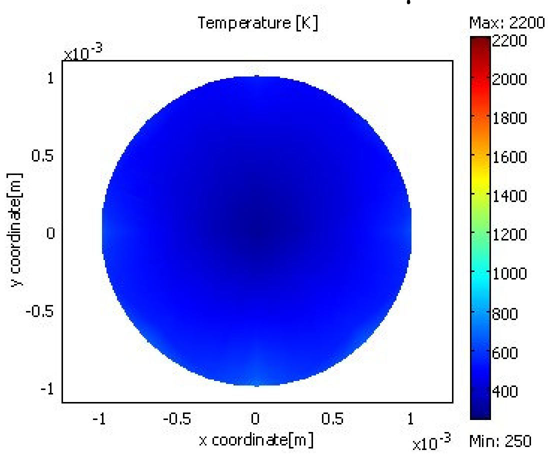


Figure 22: Slice cut of temperature distribution in rod at t=30 μs

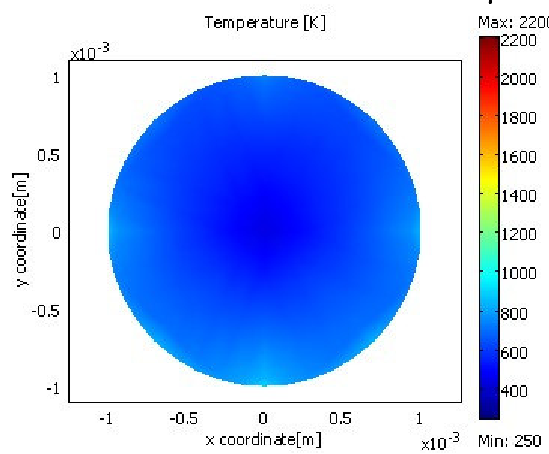


Figure 23: Slice cut of temperature distribution in rod at t=35 μs

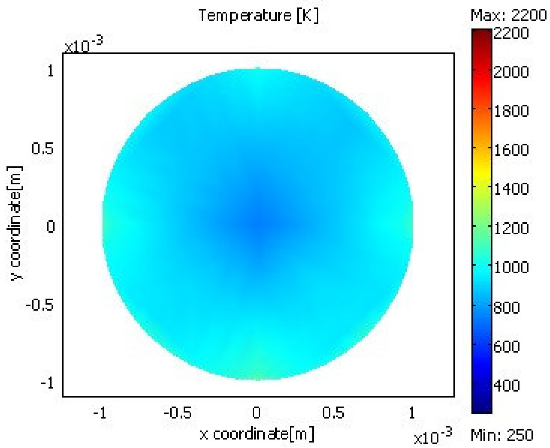


Figure 24: Slice cut of temperature distribution in rod at t=40 µs

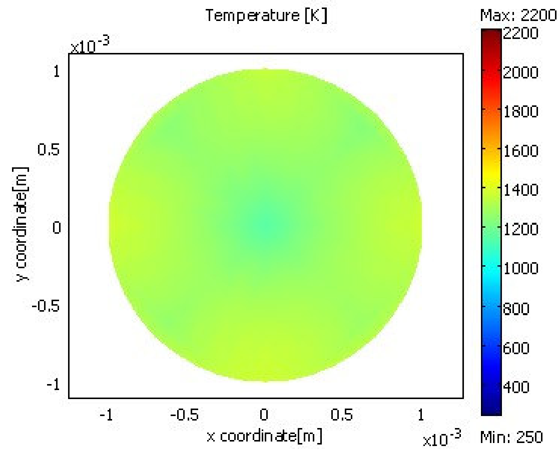


Figure 25: Slice cut of temperature distribution in rod at t=45 µs

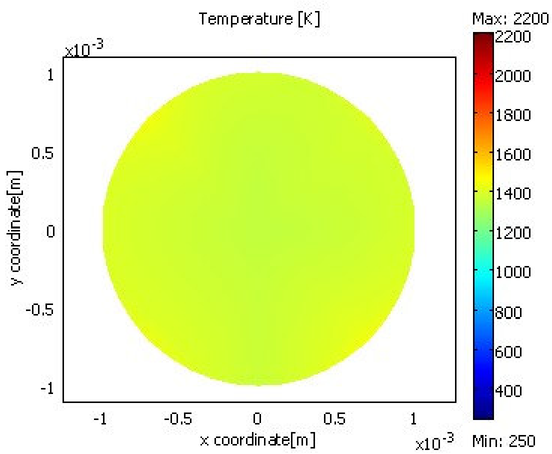


Figure 26: Slice cut of temperature distribution in rod at t=50µs

7 Using external circuit

7.1 Modeling condition

The current in external circuit behaves like an impulse rising from zero to more than 200 kA while the capacitance discharges from the very beginning. All results in section 7 are got from implementation of the following models:

1. Induction current model
2. Heat conduction model without heat of fusion and vaporization
3. External circuit model

7.2 Current and potential behavior

Since the external circuit is in serial with the rod and plates, the current through the rod is the same as the total current in external circuit. The total current in external circuit raises form zero to peak value in about 40 µs, while the potential over the capacitance drops from 4600 V. Furthermore, the potential over two plates rises from zero to about 180 V. These are shown in (Fig. 27) (Fig. 28) and (Fig. 29).

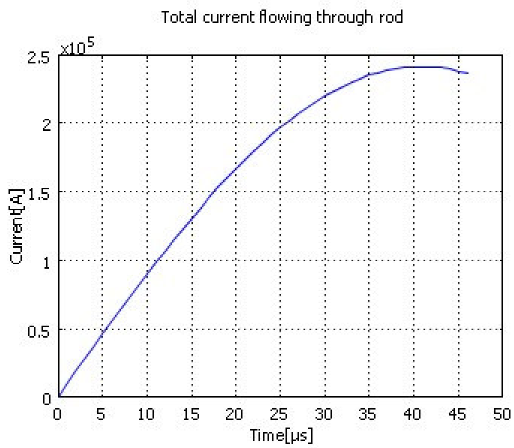


Figure 27: Current flowing through the rod over time

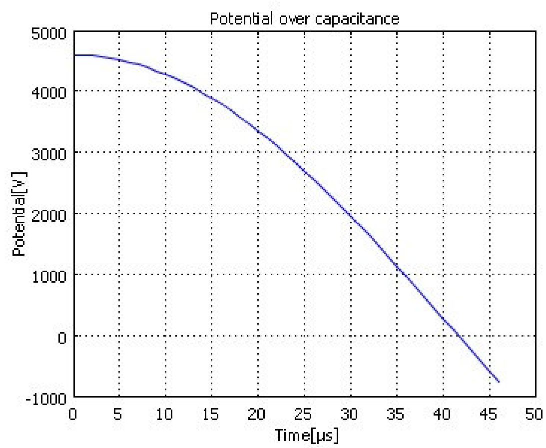


Figure 28: Potential of capacitance over time

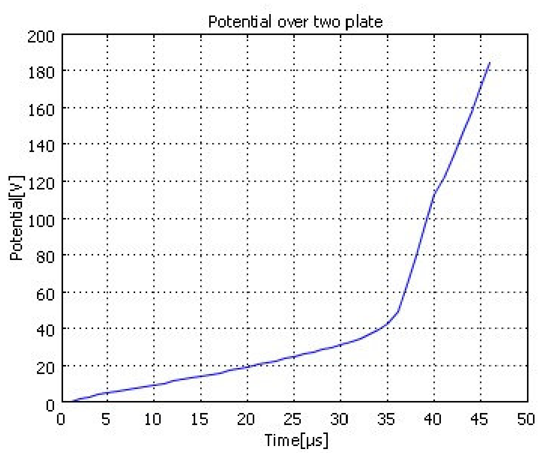


Figure 29: Potential between two plates over time

7.3 Energy behavior

7.3.1 Energy as a function of time

As for the capacitance, the energy can be measured as

$$W_C = \int u_C \cdot i_C dt \tag{39}$$

Since the current is defined as

$$i_C = \frac{dQ}{dt} \tag{40}$$

Also the capacitance is defined as

$$C = \frac{Q}{u_C} \tag{41}$$

Put equation (40) and (41) into equation (39), thus we obtain

$$W_C = C\int u_C du_C = \frac{1}{2} C \cdot u_C^2 \tag{42}$$

Similarly, for the inductance

$$W_L = \int u_L \cdot i_L dt \tag{43}$$

Since the potential over inductance is defined as

$$u_L = L\frac{di_L}{dt} \tag{44}$$

Combining equation (43) and (44) together and we get

$$W_L = L\int I_L dI_L = \frac{1}{2} L I_L^2 \tag{45}$$

Thus, the total energy in external circuit is calculated by adding (42) and (45) together

$$W = \frac{1}{2}\left(C \cdot u_C^2 + L \cdot i_L^2\right) \tag{46}$$

Since the circuit is serial, this means $i_C = i_L = i$, thus

$$W = \frac{1}{2}\left(C \cdot u_C^2 + L \cdot i^2\right) \tag{47}$$

The initial energy in the system, is measured by equation (47)

$$W(0) = \frac{1}{2}\left(C \cdot u_C^2\Big|_{t=0} + L \cdot i^2\Big|_{t=0}\right) \tag{48}$$

Put in the initial condition of the circuit (see equation (36))

$$W(0) = \frac{1}{2} \cdot 1.4 \times 10^{-3} \cdot 4600^2 = 14812\ J \tag{49}$$

Then the energy absorbed by the whole rod is measured as

$$W' = W(0) - W \tag{50}$$

It seems the rod slowly absorbs energy at the beginning and suddenly increases dramatically after 35 µs as shown in (Fig. 31) while the external circuit slowly releasing energy before 35 µs and drop suddenly then as shown in (Fig. 30), mainly because the current is reaching peak value. The relationship between energy and current will be discussed in section 7.3.2

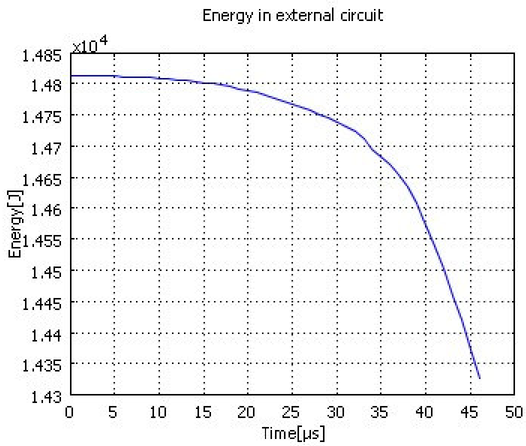


Figure 30: Total energy in external circuit over time

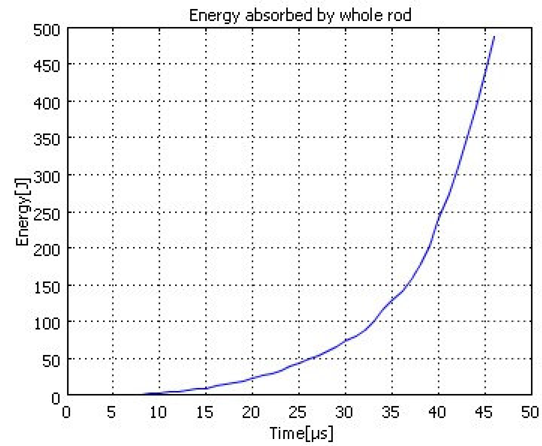


Figure 31: Energy absorbed by whole rod over time

7.3.2 Energy absorbed in whole rod versus current flowing though and potential drop over two plates

The energy absorbed by the whole rod is increasing slowly for low current, but it increases dramatically as soon as the current reaches the peak value. The energy in relationship with potential drop between two plates increases quite steadily. (Fig. 32) shows the energy versus the current flowing through rod while (Fig. 33) shows energy versus potential over two plates.

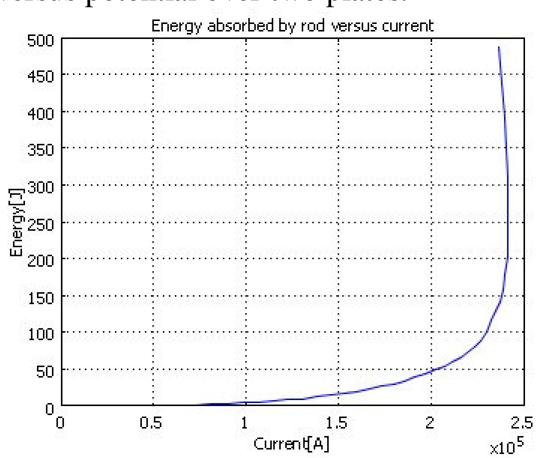


Figure 32: Energy absorbed by whole rod over current

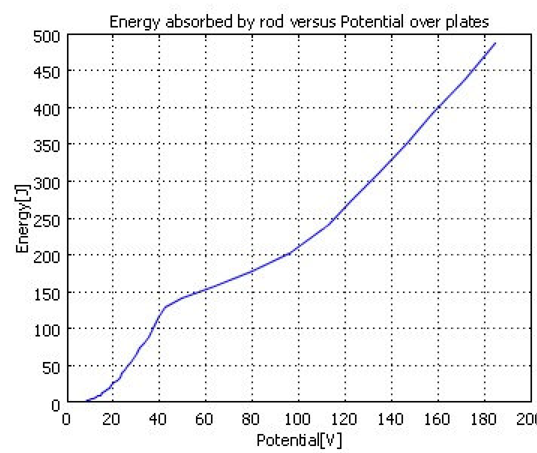


Figure 33: Energy absorbed over potential drop of rod

8 Discussion

From the very first beginning COMSOL showed it self to be a powerful tool for modeling physical problems. By sharing variables[1] much the same way as global or external variables is shared in common programming languages. Different physical models can be combined to perform even more complex models. However this makes it very hard to construct a model for this project with the necessary numerical stability and reliability. For example, the external circuit model can not exist with heat conduction model with heat of fusion and vaporization. Mainly because when trying to implement heat of fusion and vaporization together with the external circuit the thermal conductivity and heat source needs to be zero. This will result in zero time derivatives and is needed to stop temperature from rising when melting or vaporization point is reached. In these cases numerical problem arises from zero rows in the stiffness matrix, thus make it to be singular. Care should also be taken when constructing logical functions that evaluates to 1 if true and 0 if false. Discontinuous functions can be smoothed with a switch function to improve convergence. If the temperature could be fixed during the melting and vaporization point, some of the above problems could probably be resolved.

9 Conclusion

It is concluded that the current distribution is very much relevant to the total current flowing into the rod. This is concluded when a current impulse with top value of about 200 kA and duration of about 80 µs is applied to a 2 mm radius rod. The current distribution in rod is almost uniform when the impulse reaches 1/5 of its peak value. However, temperature distribution is uniform for most of the time mainly because the rod is rather small. The energy absorbed by whole rod in relationship with current impulse also confirms that the rod absorbs most energy to heat up when the impulse is about to reach the peak value.

To heat up a rod with larger radius, a current impulse with higher peak value is needed. Moreover, when a current impulse with higher peak value is applied to the same rod will only increase its speed to heat up.

[1] In COMSOL they are called coupling variables

10 Appendix

10.1 Constant parameter list

- Geometry data

Rod diameter	$2.0\ mm$	
Plate size	$20\ mm \times 20\ mm$	
Plate distance	$4.5\ mm$	

- Material data (copper)

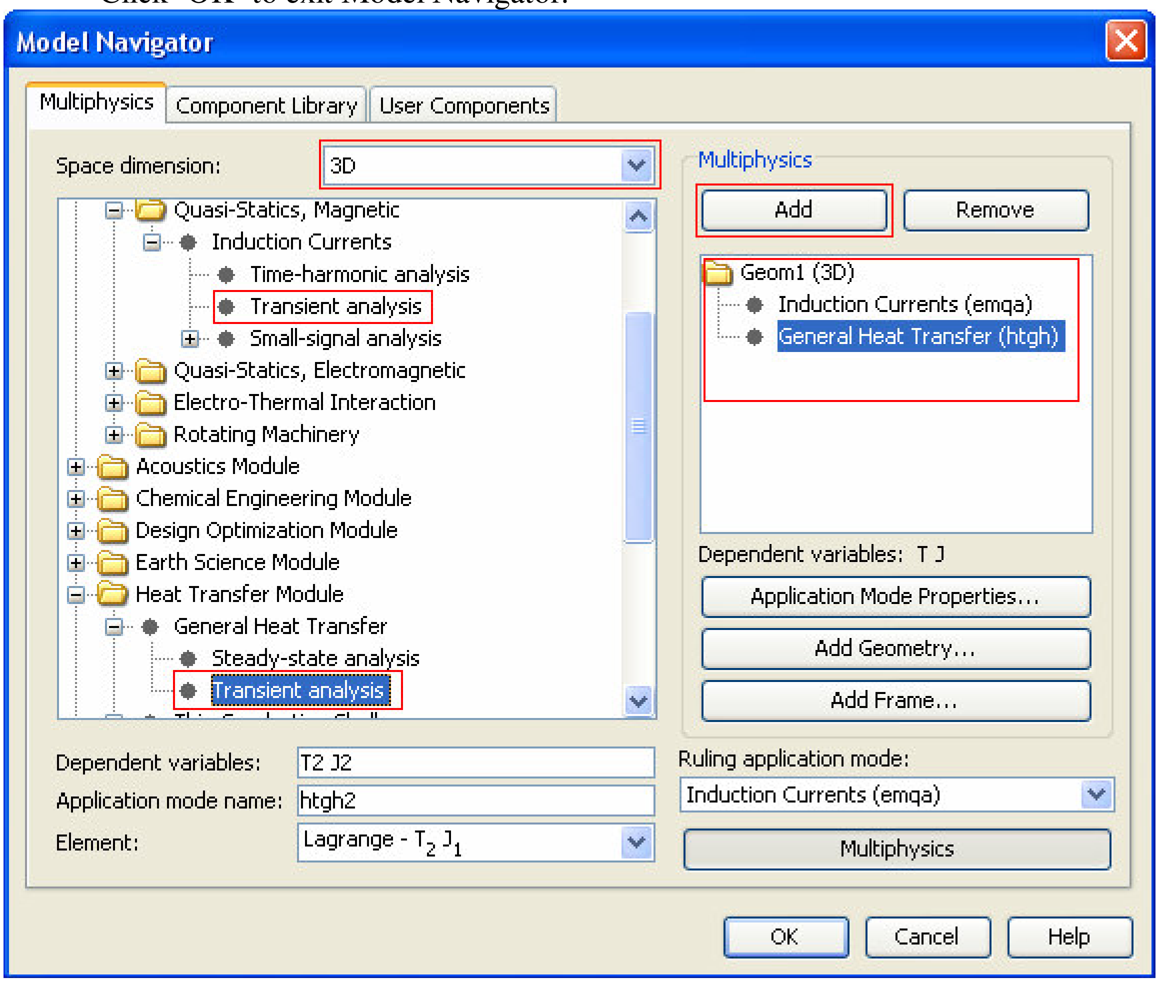
σ_0	$5.961\times10^{7}\ S/m$	Initial conductivity (at 273K)
μ_r	1	Relative permeability
μ_0	$1.2566\times10^{-6}\ N/A^2$	Permeability in vacuum
T_0	$273\ K$	Initial temperatures
α	$4.33\times10^{-3}\ K^{-1}$	Temperature coefficient
c_{ps}	$481\ J/(kg\cdot K)$	Heat capacity for solid
c_{pl}	$531\ J/(kg\cdot K)$	Heat capacity of liquid
T_m	$1356\ K$	Melting temperature
T_v	$2855\ K$	Vaporization temperature
m	$8960\ kg/m^3$	Density
w_m	$2.05\times10^{5}\ J/kg$	Heat of fusion
w_v	$4.75\times10^{6}\ J/kg$	Heat of vaporization
L_o	$2.54\times10^{-8}\ V^2/K^2$	Lorenz parameter
C	$1.4\times10^{-3}\ F$	Capacitance in circuit
L	$5.00\times10^{-11}\ H$	Inductance in circuit
$u_C\big\|_{t=0}$	$4600\ V$	Initial potential over capacitance

10.2 Implement the model in COMSOL

10.2.1 To begin

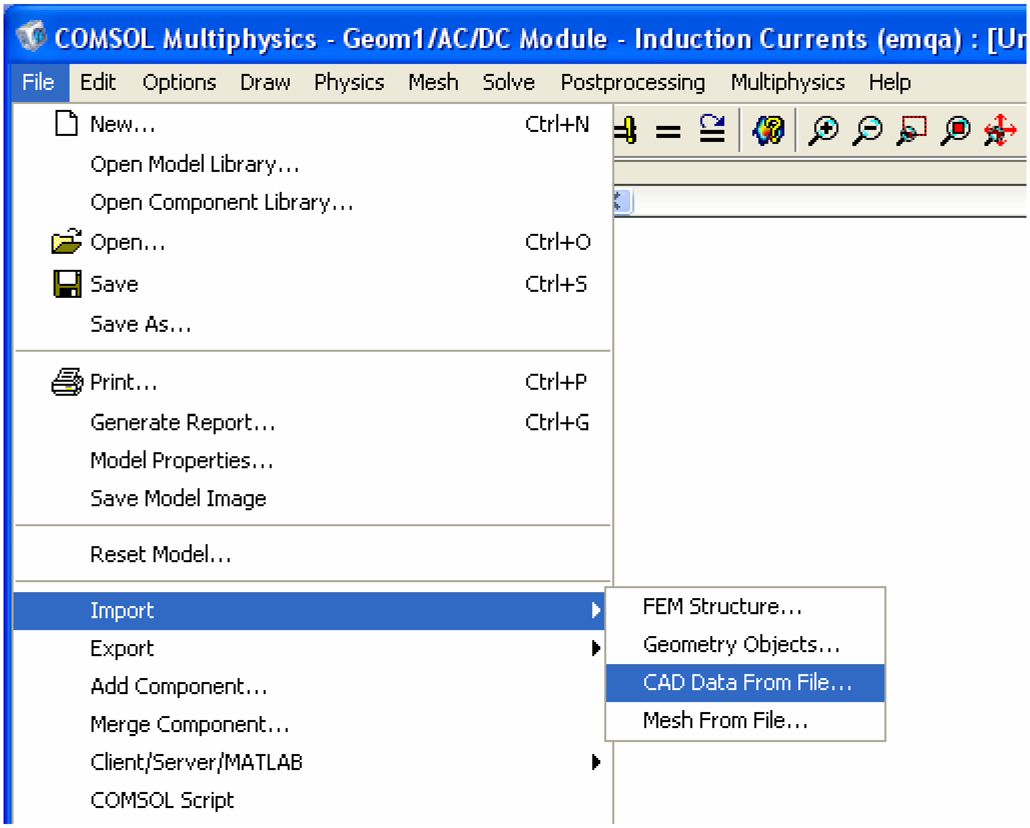
In order to begin, you need COMSOL 3.3 or later with AC/DC module installed on your computer. You also need to draw the model or download it from the web[2]. and click COMSOL to open the model navigator.

- In 'space dimension' , choose '3D'
- Choose 'Application Modes'→ 'AC/DC module'→ 'Quasi-Statics, Magnetic'→ 'Induction currents'→ 'Transient analysis', then click 'Add' button to add.
- Choose 'Application Modes'→ 'Heat Transfer module'→ 'General Heat transfer'→ 'transient analysis', then click 'Add' button to add.
- Click 'OK' to exit Model Navigator.



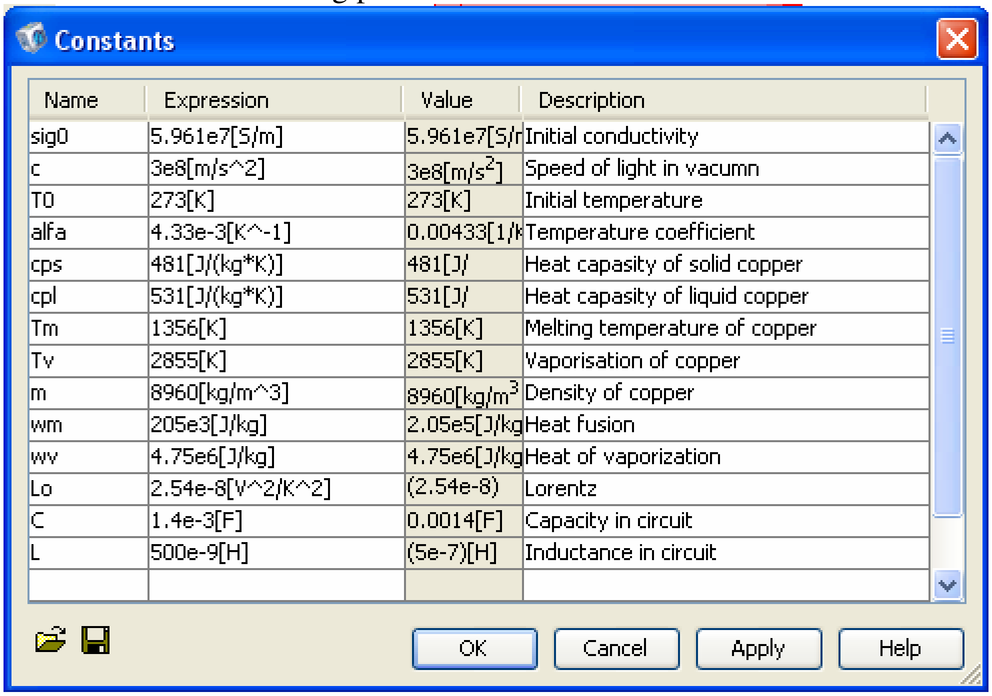
[2]The geometry object model is fairly easy to draw see (Fig. 2) for specifications.
The model file can also be found on the web.
http://www.it.uu.se/edu/course/homepage/projektTDB/vt08/project2/project2_geometry.mphbin

- Click 'File'→ 'Import'→ 'CAD data from file' to import the data you have just downloaded.



10.2.2 Set the constant value

- Select 'Options'→ 'Constants' on the menu bar to open the constant data
- Enter the following parameters



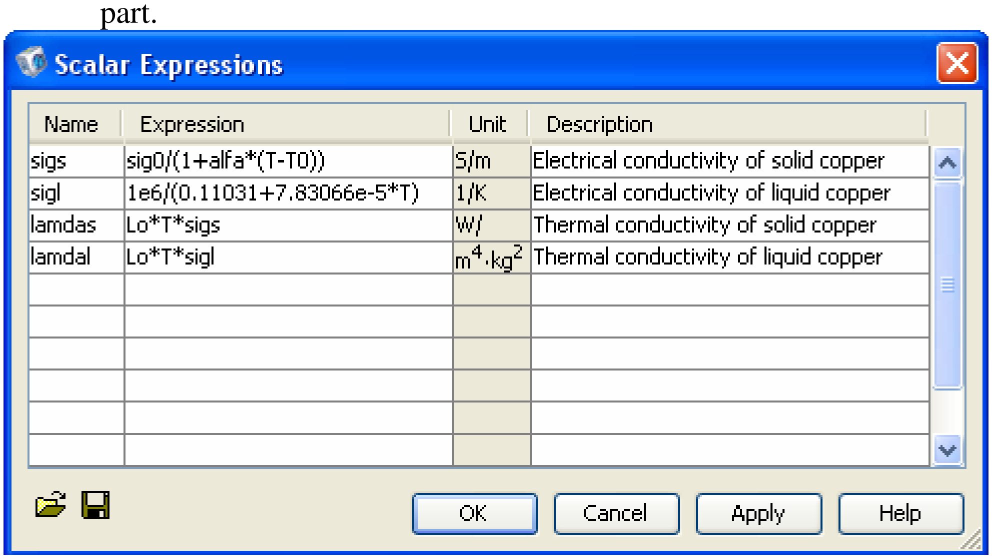
Constants

Name	Expression	Value	Description
sig0	5.961e7[S/m]	5.961e7[S/r	Initial conductivity
c	3e8[m/s^2]	3e8[m/s^2]	Speed of light in vacumn
T0	273[K]	273[K]	Initial temperature
alfa	4.33e-3[K^-1]	0.00433[1/k	Temperature coefficient
cps	481[J/(kg*K)]	481[J/	Heat capacity of solid copper
cpl	531[J/(kg*K)]	531[J/	Heat capacity of liquid copper
Tm	1356[K]	1356[K]	Melting temperature of copper
Tv	2855[K]	2855[K]	Vaporisation of copper
m	8960[kg/m^3]	8960[kg/m^3	Density of copper
wm	205e3[J/kg]	2.05e5[J/kg	Heat fusion
wv	4.75e6[J/kg]	4.75e6[J/kg	Heat of vaporization
Lo	2.54e-8[V^2/K^2]	(2.54e-8)	Lorentz
C	1.4e-3[F]	0.0014[F]	Capacity in circuit
L	500e-9[H]	(5e-7)[H]	Inductance in circuit

OK Cancel Apply Help

10.2.3 Enter scalar expressions

- Select 'Options'→ 'Expressions'→ 'Scalar expressions' from the menu bar
- Enter the following formula to implement solid and liquid expression for both electrical conductivity and thermal conductivity. For more details, see theoretical part.



Scalar Expressions

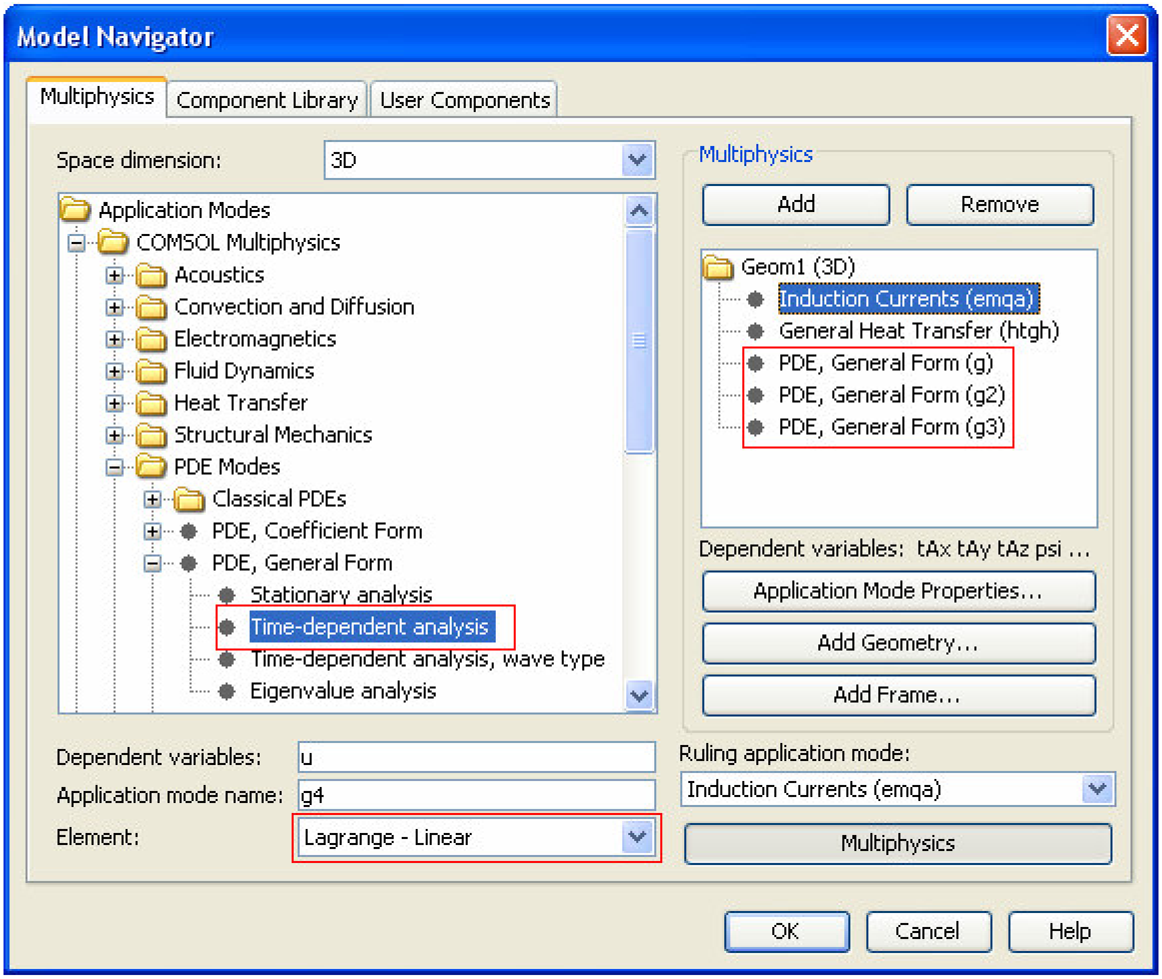
Name	Expression	Unit	Description
sigs	sig0/(1+alfa*(T-T0))	S/m	Electrical conductivity of solid copper
sigl	1e6/(0.11031+7.83066e-5*T)	1/K	Electrical conductivity of liquid copper
lamdas	Lo*T*sigs	W/	Thermal conductivity of solid copper
lamdal	Lo*T*sigl	$m^4.kg^2$	Thermal conductivity of liquid copper

OK Cancel Apply Help

10.2.4 Energy estimation[3]

In order to calculate the heat absorbed at melting and vaporization point, we need three additional models, one calculating resistance heating until melting point, and the other calculating until vaporization and the third until the end.

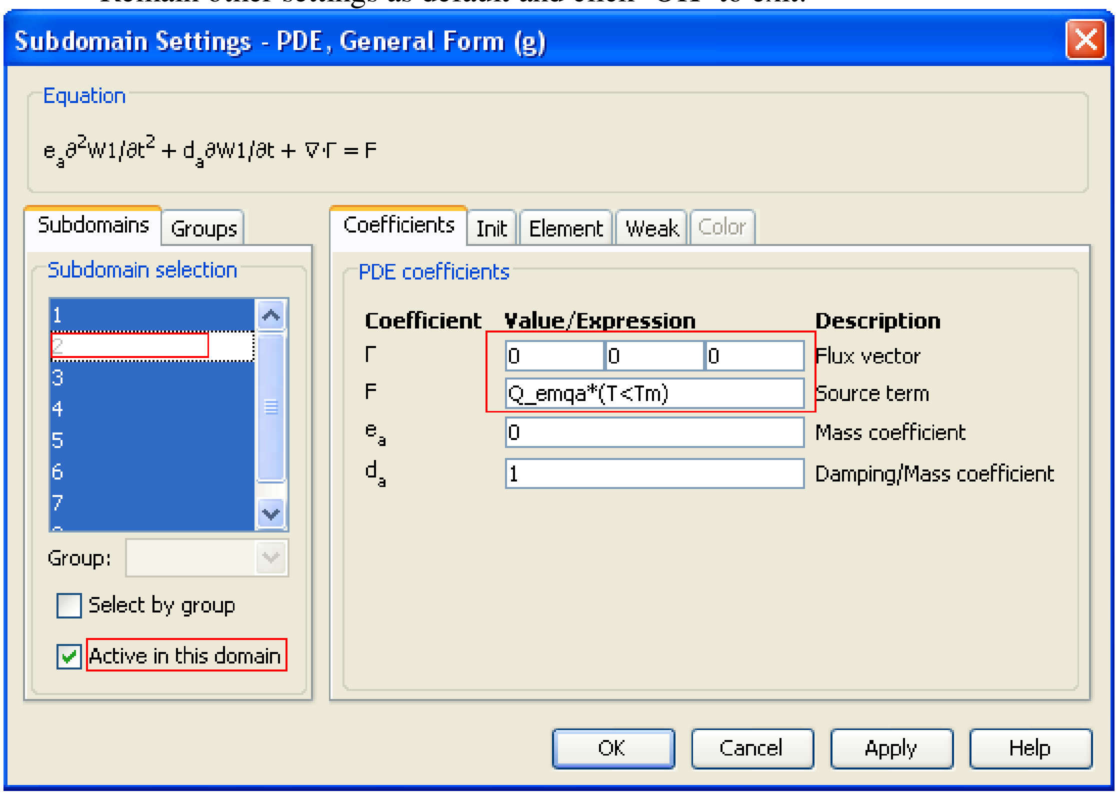
- Choose 'Multiphysics'→ 'Model Navigator' in menu bar.
- Choose 'Application modes'→ 'Comsol Multiphysics'→ 'PDE Mode'→ 'PDE, General Form'→ 'Time dependent analysis', and set 'Element' to linear in order to save computation.
- Add three models named as 'W1' 'W2' and 'W3' respectively.
- Click 'OK' to exit.



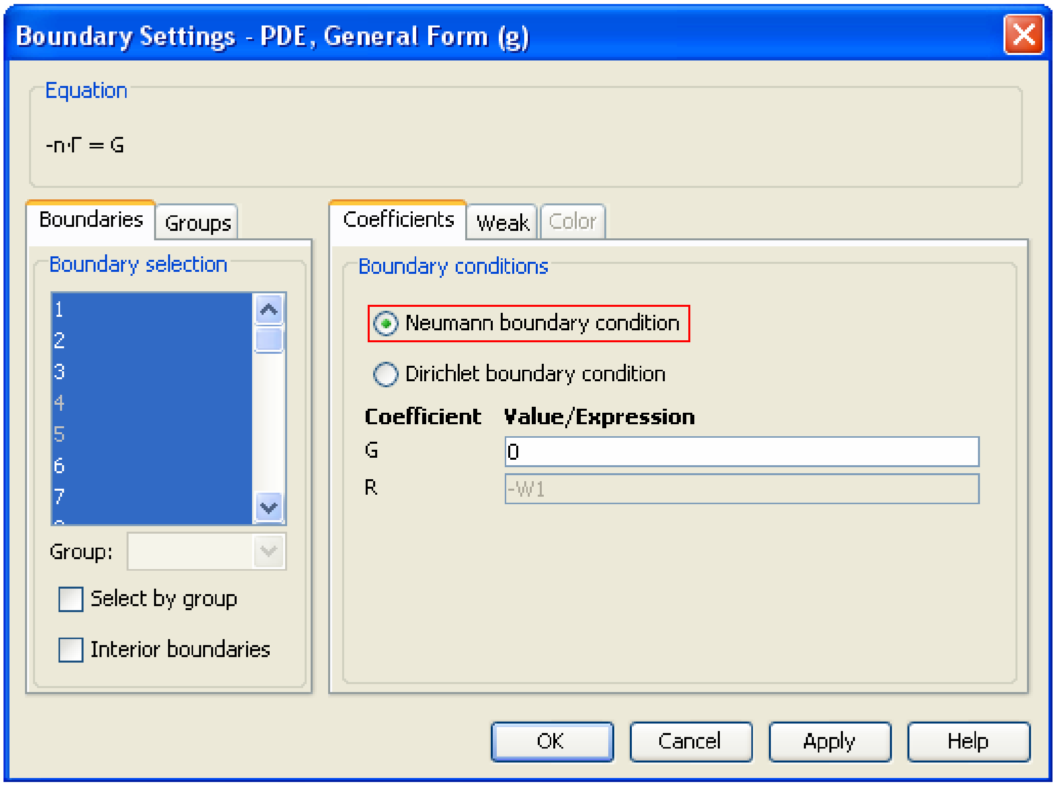
..

[3] If you do not want to implement heat of fusion and vaporization, you can skip this section.

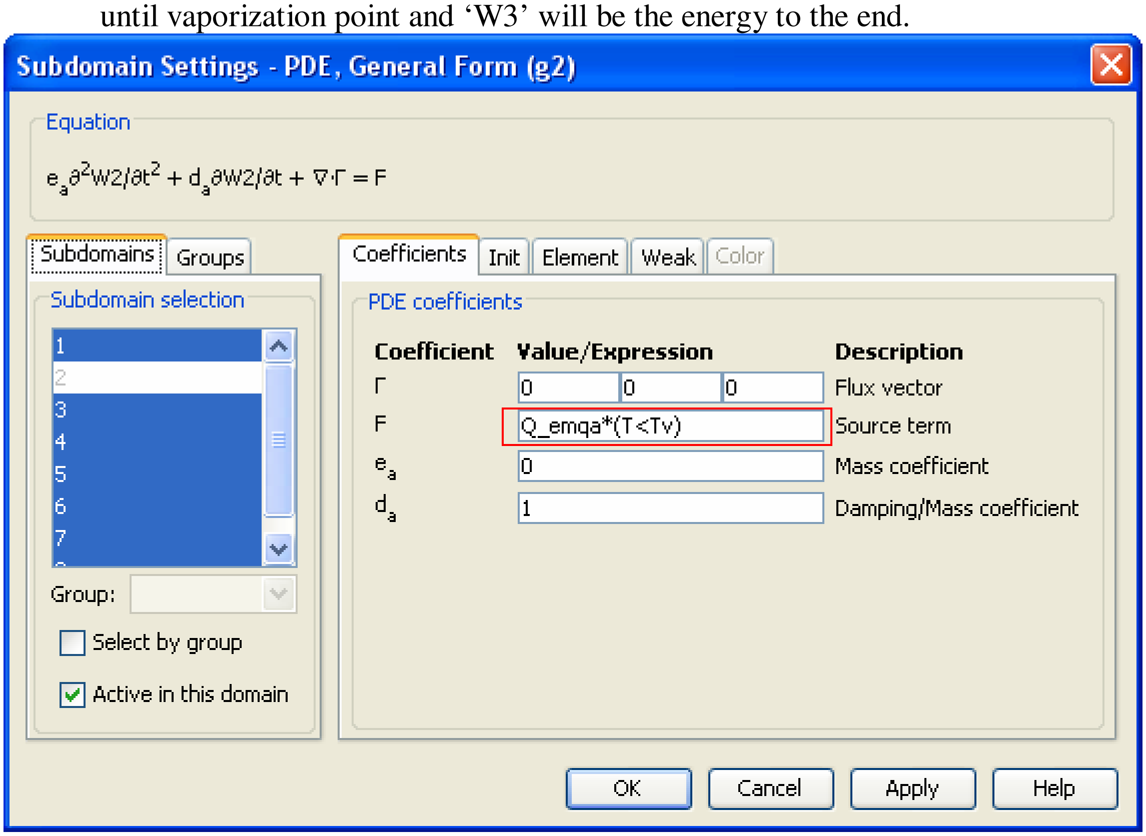
- In the model tree at left hand side, right click mouse on 'PDE, General form (g)' and select 'Subdomain settings'.
- In the dialog box, disable the Subdomain 2 by uncheck the box beside the 'Active in this domain', since there will be no current in the air.
- Select the rest subdomains, enter three zeros in 'Flux vector' and 'Q_emqa*(T<Tm)' so that the time derivative of 'W1' is resistance heating source until the temperature reaches melting point, which makes it to be the resistance heating energy until melting point. Moreover, 'W1' should remain constant after temperature exceeds the melting point.
- Remain other settings as default and click 'OK' to exit.

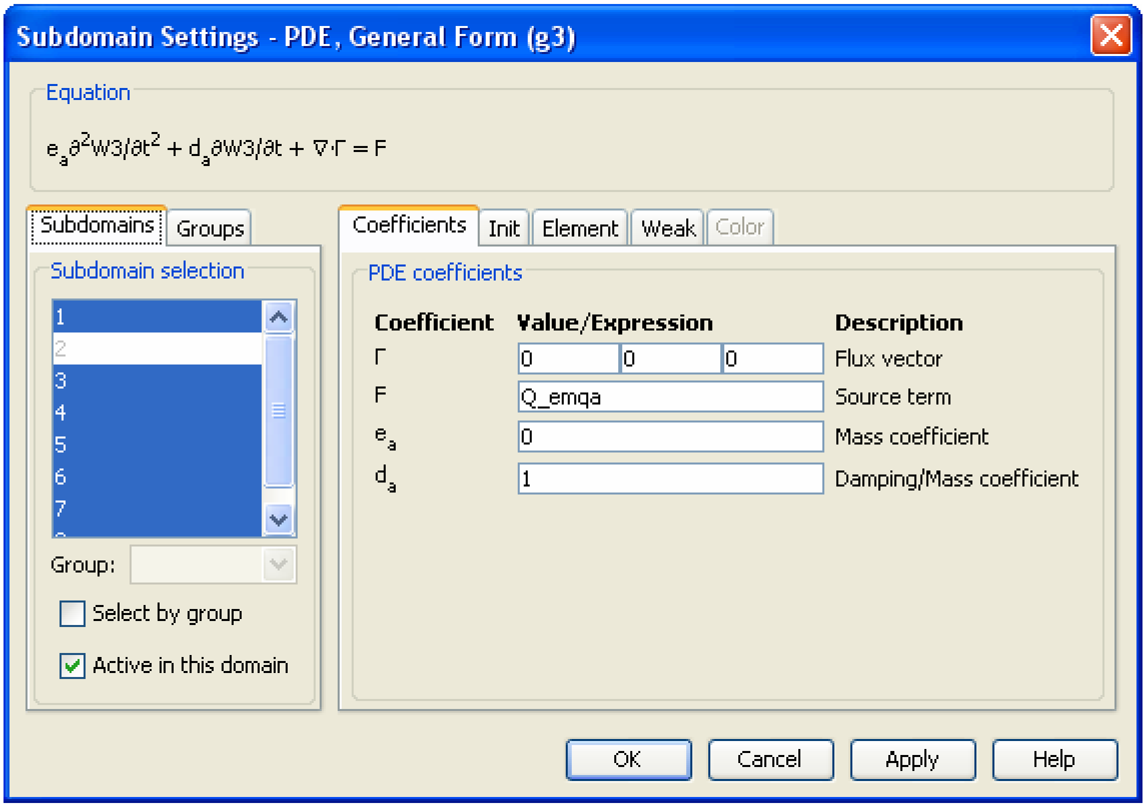


- In the model tree at left hand side, right click mouse on 'PDE, General form (g)' and select 'Boundary settings'.
- Select all boundaries and choose 'Neumann boundary condition' so that 'W1' will be free on all boundaries.



- Do the same settings for the rest two models except 'Q_emqa*(T<Tv)' for 'W2' where as 'Q_emqa' for 'W3', so that 'W2' will be the resistance heating energy until vaporization point and 'W3' will be the energy to the end.




Subdomain Settings - PDE, General Form (g3)
Equation
e_a∂²w3/∂t² + d_a∂w3/∂t + ∇·Γ = F
Subdomains
Groups
Subdomain selection
1
2
3
4
5
6
7
Group:
Select by group
Active in this domain
Coefficients
Init
Element
Weak
Color
PDE coefficients
Coefficient
Value/Expression
Description
Γ
0
0
0
Flux vector
F
Q_emqa
Source term
e_a
0
Mass coefficient
d_a
1
Damping/Mass coefficient
OK
Cancel
Apply
Help

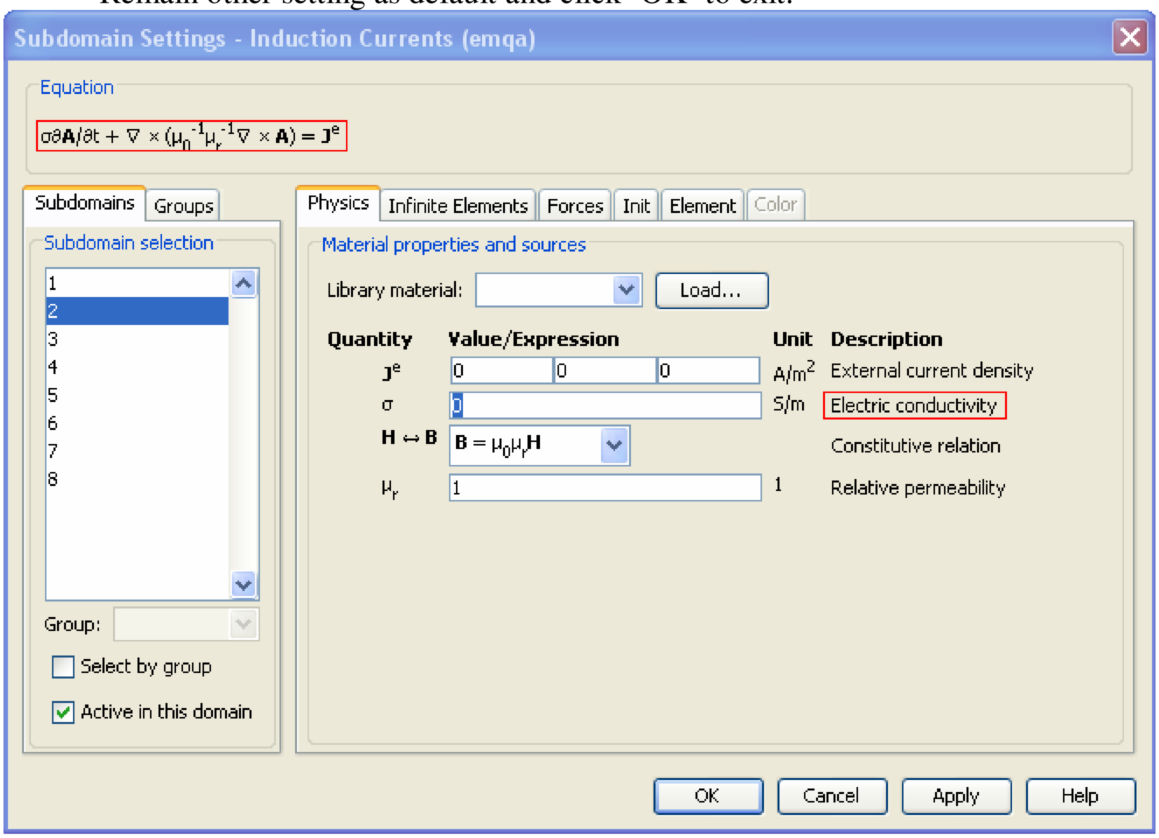
10.2.5 Configure 'Induction Currents' model

- In the 'Model tree' at the left side, right click your mouse on 'Induction Currents' model and choose 'Subdomain Settings', and check your equation in this model which should consistent with equation (10) we motivated in the theoretical part.
- Set different parameters in 'Electric conductivity field'

Sub domain	Electric conductivity[4]
2 (Air)	0
1, 3, 4, 5, 6, 7, 8 (Copper)	sigs*(T<Tm)+sigs*(T>Tm)*((W2-W1)<wm*m)+sigl*(T>Tm)*(T<Tv)+sigl*(T>Tv)*((W3-W2)<wv*m)

The electric conductivity of copper is a discontinuous function it should be consistent to the two individual functions described in equation (10).

- Remain other setting as default and click 'OK' to exit.

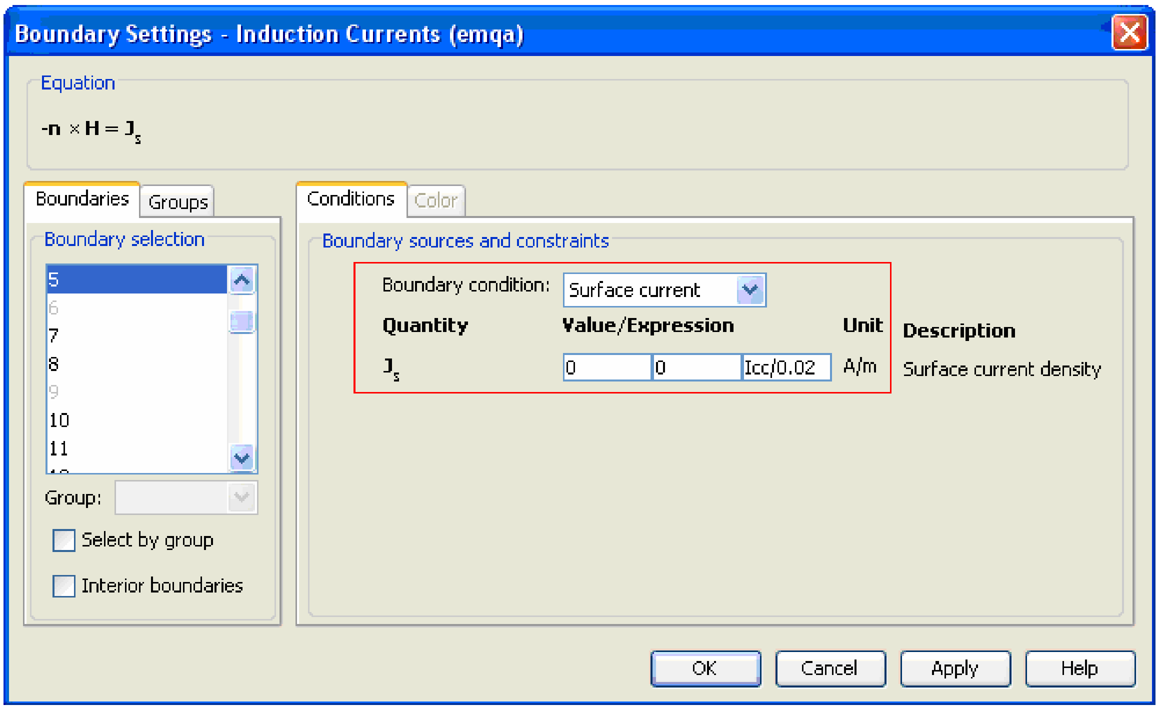


[4] Use 'sigs*(T<Tm)+sigl*(T>Tm)*(T<Tv)' for modeling without heat of fusion and vaporization

- In the model tree at left hand side, right click mouse on 'Induction Currents' and select 'Boundary settings'.
- Select different boundary conditions in 'Conditions' tab

Boundary	Boundary condition	value
1, 2, 3, 4, 7, 8, 10, 11, 12, 13, 14, 15, 16, 26,27, 29, 30, 34, 35, 39, 40, 41, 42	Electrical insulation	N/A
6, 9, 20, 21, 17, 18, 19, 20, 21, 22, 23, 24, 25, 28, 31, 32, 33, 36, 37, 38	Continuity	N/A
5	Surface current	Icc/0.02

- Click 'OK' to exit.

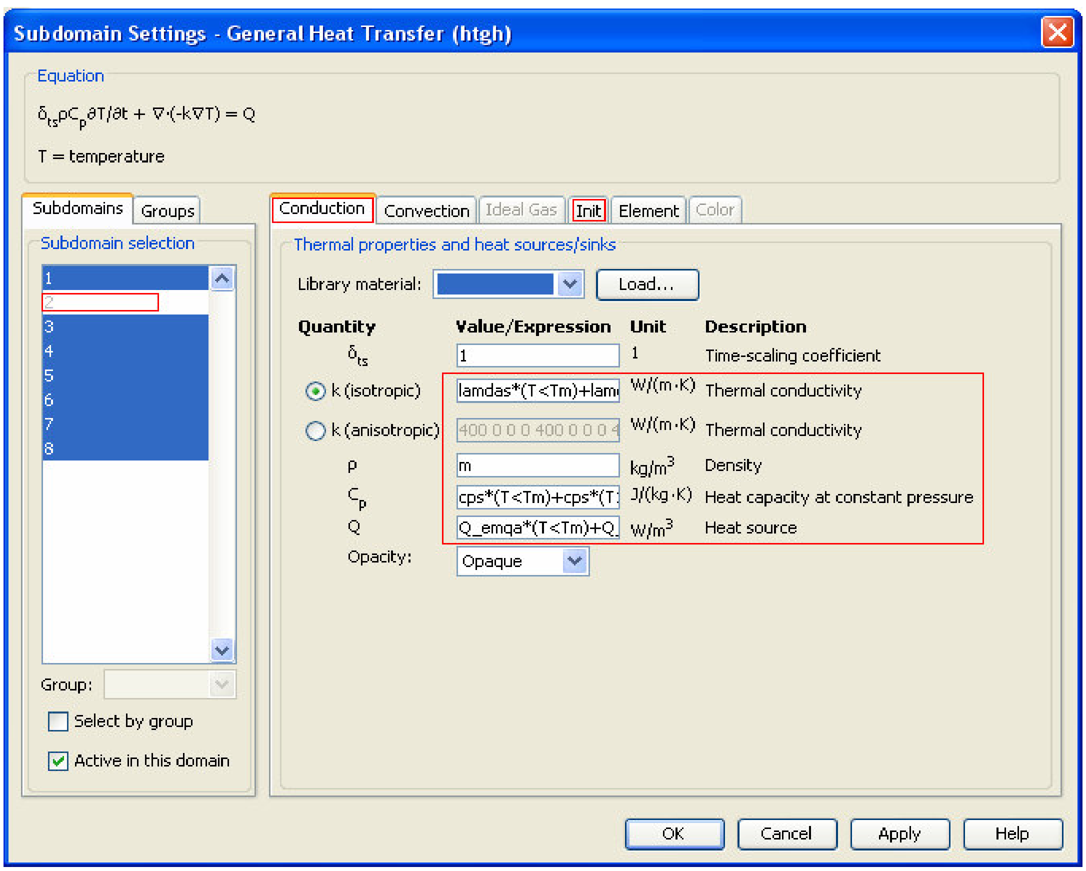


10.2.6 Configure the ‘General heat transfer’ model

- In the ‘Model tree’ at the left side, right click your mouse on ‘General heat transfer’ model and choose ‘Subdomain Settings’.
- Select subdomain 2 and uncheck the box beside ‘Active in this domain’ to disable it, since the air is ideal which is thermal insulation and without any current.
- Select the rest subdomains and set the following parameters in the ‘Conduction’ tab

Name	expression
Thermal conductivity[5]	lamdas*(T<Tm)+lamdal*(T>Tm)*(T<Tv)*((W2-W1)>wm*m)
Density	m
Heat capacity at constant pressure[6]	cps*(T<Tm)+cps*(T>Tm)*((W2-W1)<wm*m)+cpl*(T>Tm)*(T<Tv)+cpl*(T>Tv)*((W3-W2)<wv*m)
Heat source[7]	Q_emqa*(T<Tm)+Q_emqa*(T>Tm)*(T<Tv)*((W2-W1)>wm*m)

- In the ‘Init’ tab, set initial temperature to ‘T0’.
- Remain others to be default and exit.

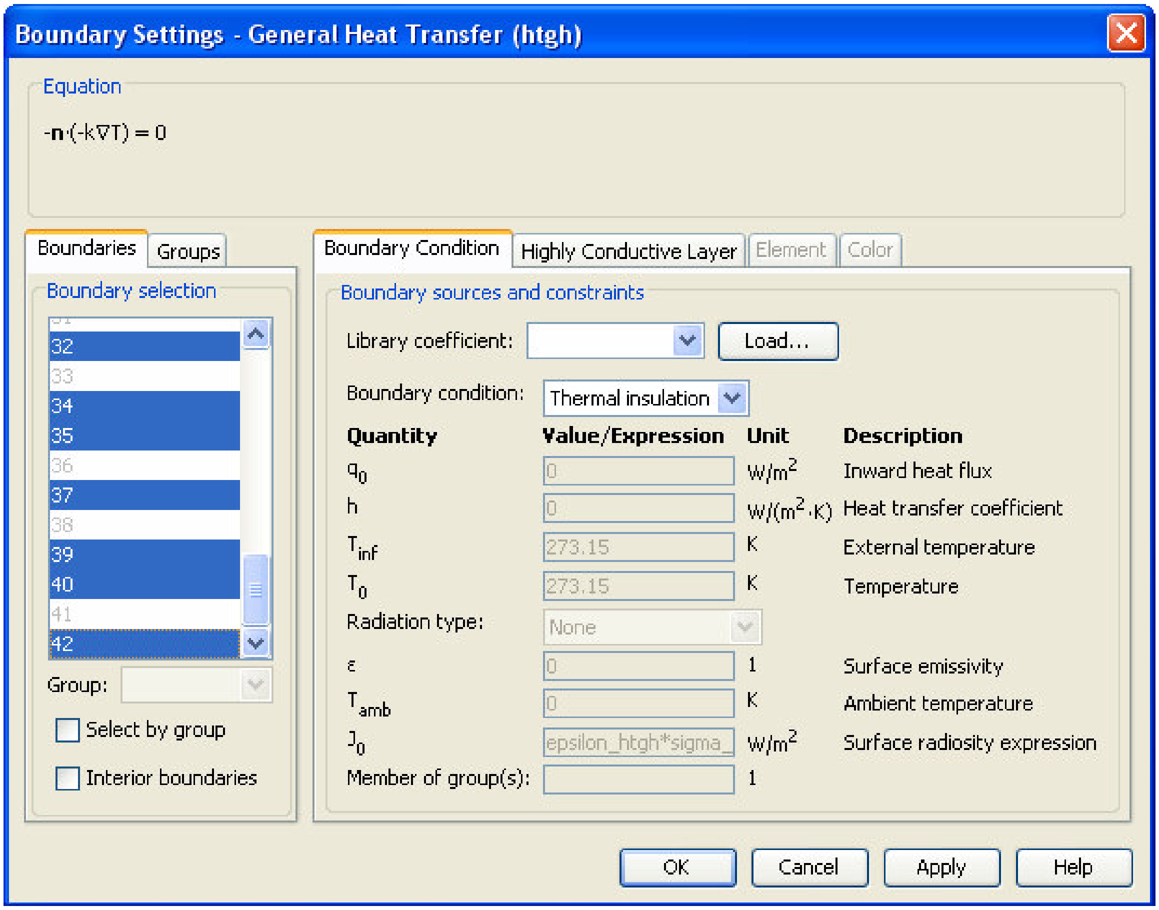


[5] Use ‘lamdas*(T<Tm)+lamdal*(T>Tm)*(T<Tv)’ for modeling without heat of fusion and vaporization
[6] Use ‘cps*(T<Tm)+cpl*(T>Tm)*(T<Tv)’ for modeling without heat of fusion and vaporization
[7] Use ‘Q_eqma’ for modeling without heat of fusion and vaporization

- In the ‘Model tree’ at the left side, right click your mouse on ‘General heat transfer’ model and choose ‘Boundary settings’.
- Set the parameters in ‘Boundary condition’ tab

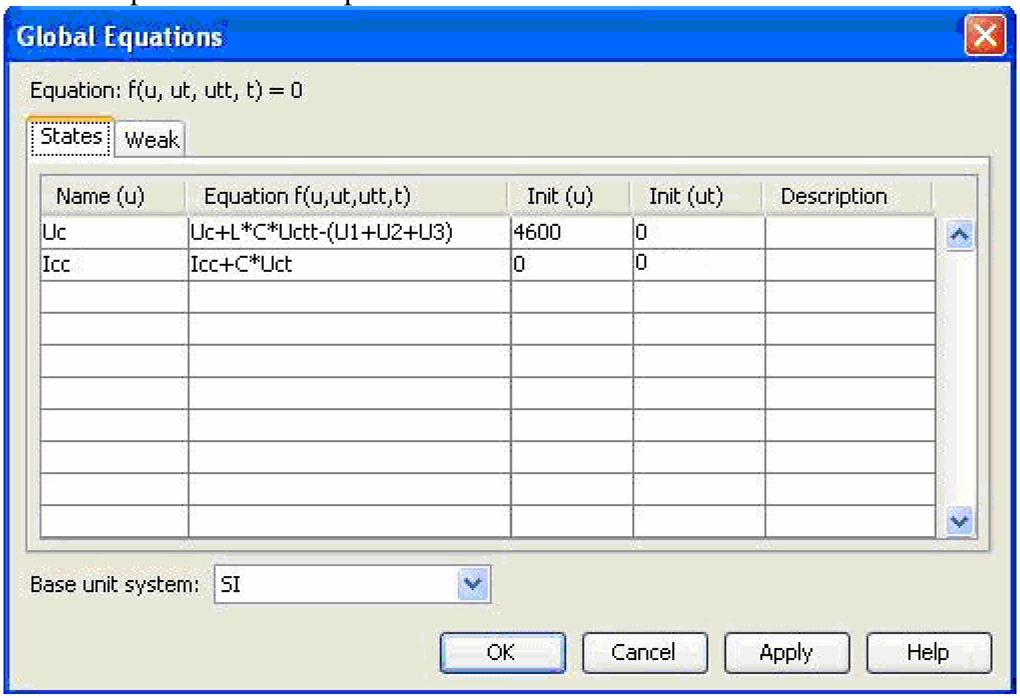
Boundary	Boundary condition
4, 5, 12, 17, 18, 19, 22, 23, 24, 25, 28, 31, 33, 36, 38, 41	Continuity
1, 2, 3, 6, 7, 8, 9, 10, 11, 13, 14, 15, 16, 20, 21, 26, 27, 29, 30, 32, 34, 35, 37, 39, 40, 42	Thermal insulation

- Remain the rest as default and click ‘OK’ to exit.

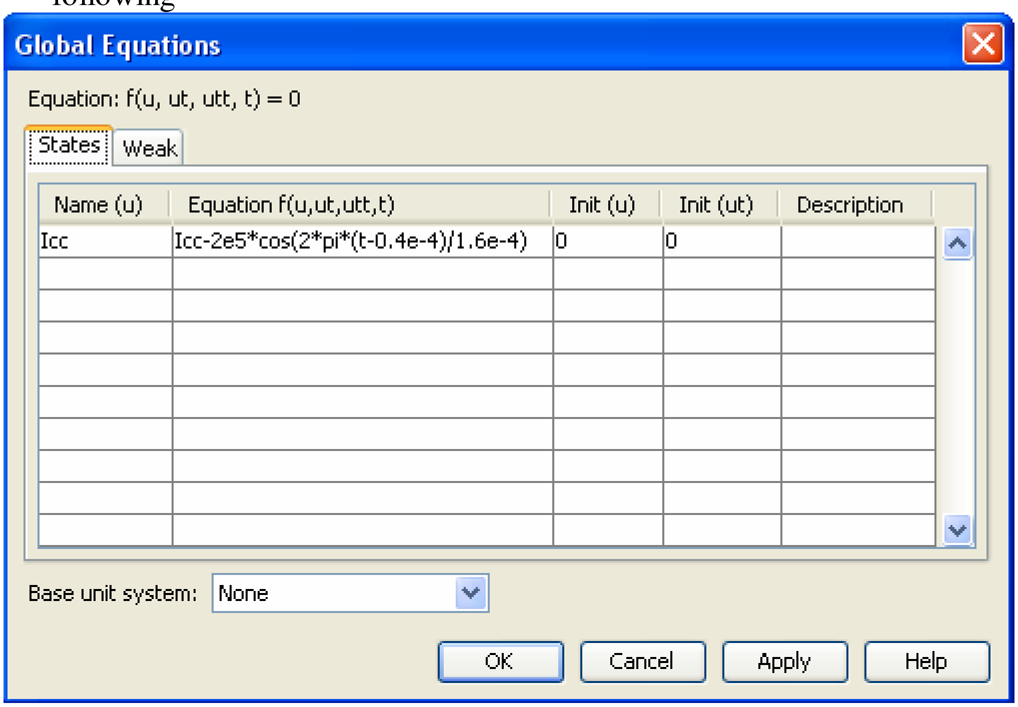


10.2.7 External circuit model

- In the menu bar, click ‘Physics’→ ‘Global Equations’
- Enter parameters to implement external circuit[8]



- If you want a prescribed current described in Fig. 5, then set the parameter as following



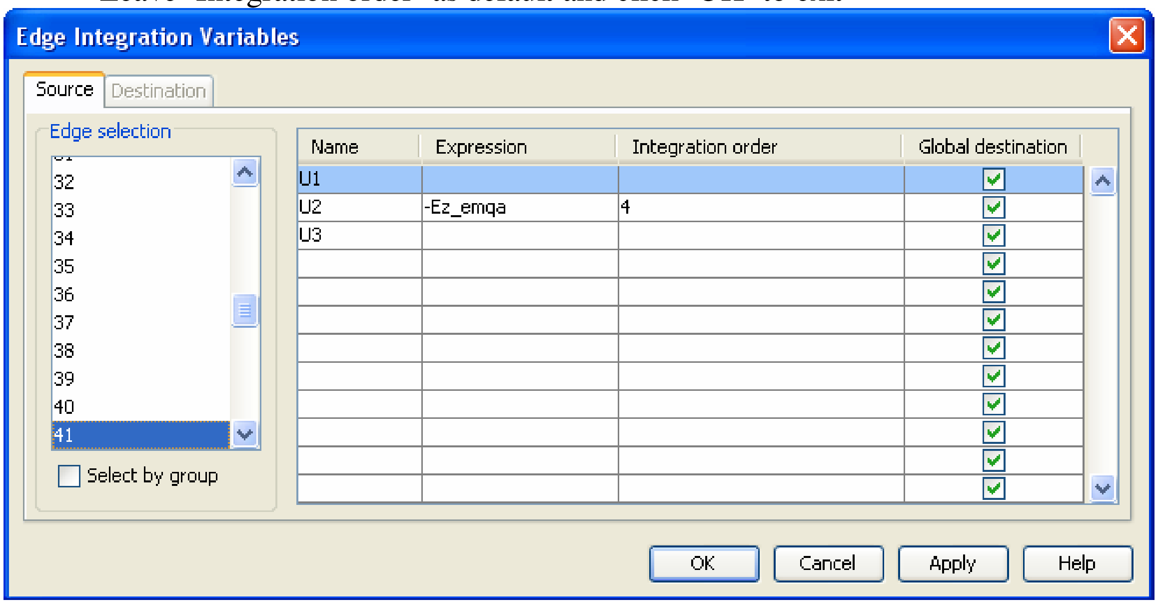
[8] Note: if you name your variable as ‘u’, then ‘ut’ and ‘utt’ are time derivative and second time derivative respectively, COMSL recognize them automatically.

10.2.8 Getting potential drop over two plates

- In the main menu bar, click 'Options'→ 'Integration coupling variables'→ 'Edge variables'
- Enter parameters as described

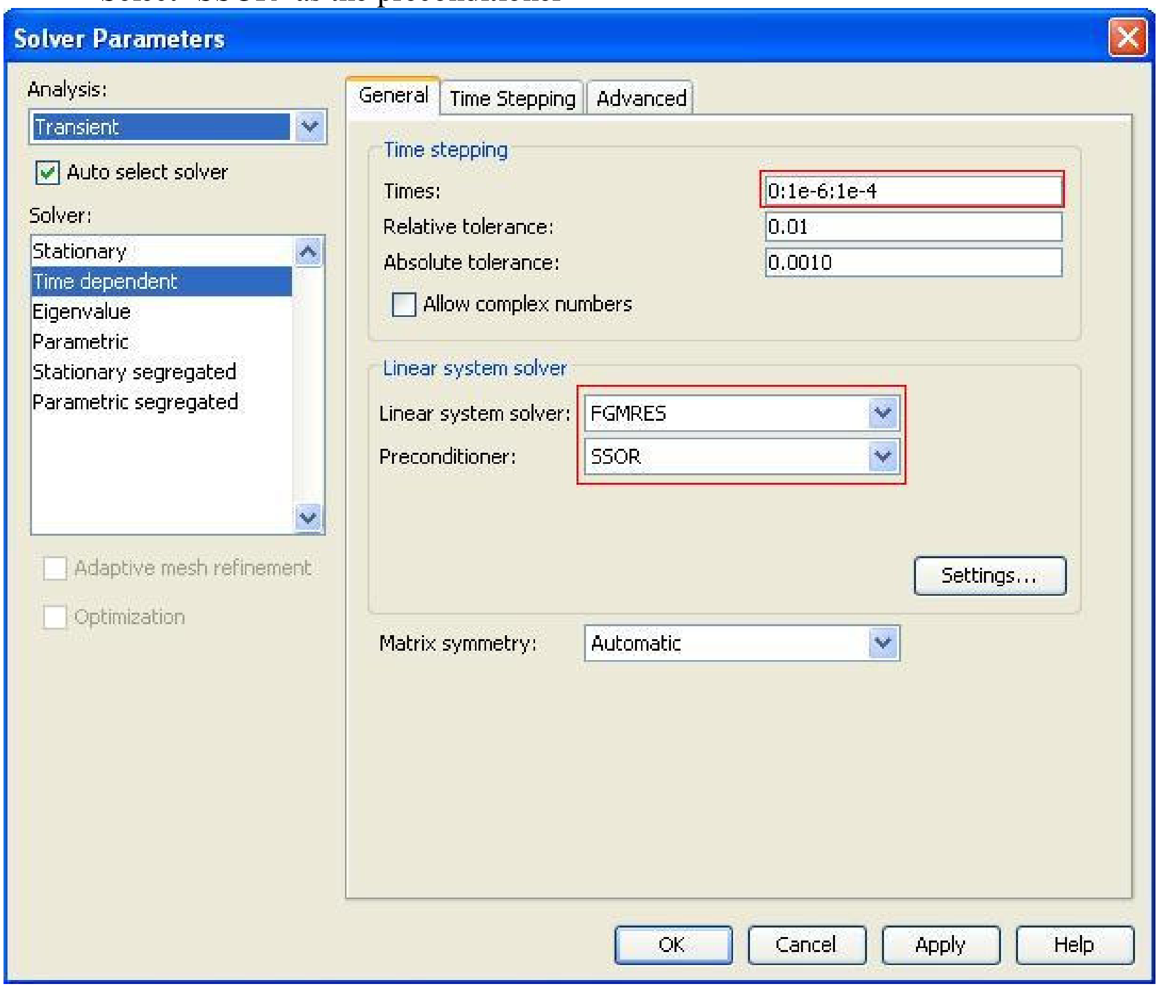
Edge	Name	Expression
33	U3	-Ey_emqa
35	U1	Ey_emqa
41	U2	-Ez_emqa

- Leave 'Integration order' as default and click 'OK' to exit

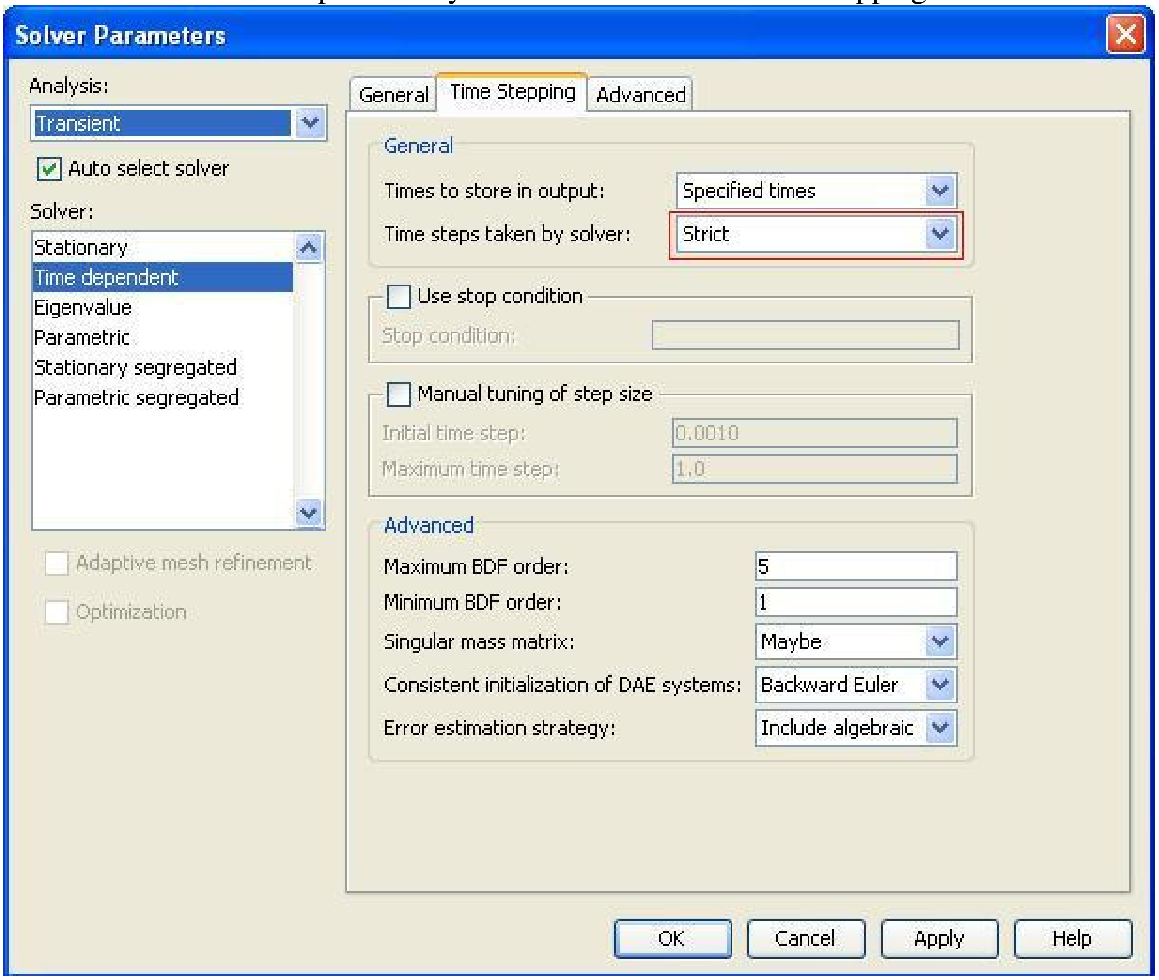


10.2.9 Solver settings

- In the main menu bar, click 'Solve'→ 'Solver Parameters' to set the solver
- Choose 'Time dependent' in the left column
- Set times as '0:1e-6:1e-4'
- Select 'FGMRES' in liner system solver
- Select 'SSOR' as the preconditioner



- Set ‘Time steps taken by solver’ to ‘Strict’ in ‘Time stepping’ tab



- Remain the rest settings as default and click ‘OK’ to exit

Note: iterative solver such as ‘FGMRES’ is suggested, choosing a better preconditioner will make it faster to compute the solution.

Acknowledgements

The authors would like to thank Anders Larsson at FOI for support and guidance during the project. Maya Neytcheva for answering questions and discussions.
Thanks is also directed to COMSOL for providing a trial license to COMSOL Multiphysics.

References

C Cagran, "Thermal conductivity and thermal diffusivity of liquid copper". Diploma Thesis, Institut für Experimentalphysik, Techniche Universität Graz, Austria (2000) http://iep.tu-graz.ac.at/thermofiles/DA-Claus_Cagran.pdf

Ch Kittel, *Introduction to Solid State Physics*, New York: John Wiley & Sons (1986)

C Nordling and J Österman, *Physics Handbook*, Lund: Studentlitteratur (1982)

COMSOL Multiphysics 3.4 manual

E. Hallén. *Elektricitetslära*. Almquist & Wiksells, Uppsala, 1953.