



# Multiphysics Modeling

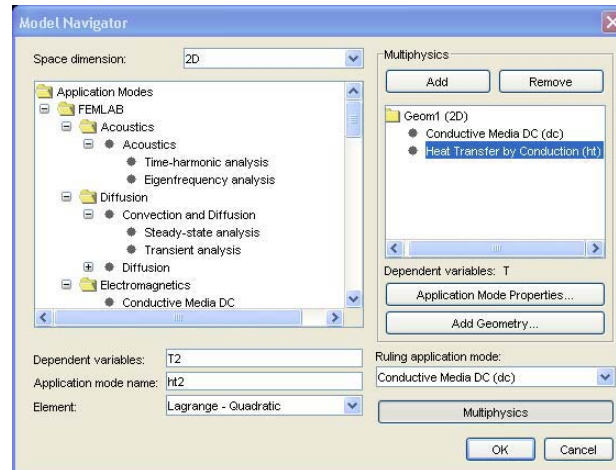
This chapter covers the use of FEMLAB for multiphysics modeling and coupled-field analyses. It first describes the various ways of building multiphysics models. Then a step-by-step example shows how to model resistive heating (also called Joule heating) taking into account thermal-electric couplings.

# Creating Multiphysics Models

The ability to create multiphysics models—those with more than one type of physics or equation such as coupled-field problems—is one of FEMLAB’s most powerful capabilities. In such a model, the software solves all the equations, taken from various areas of physics, as one fully coupled system.

## *Multiphysics Modeling Approaches*

Within FEMLAB you can choose from several ways to approach multiphysics modeling and coupled-field analysis. They all use the **Model Navigator** to specify the application modes and their properties.



### **ADDING PHYSICS SEQUENTIALLY**

With this approach you can verify that each type of physics or PDE gives the expected results before adding more complexity to the model by adding another physics or coupling fields.

Use these steps to add one application mode at the time:

- 1 Select an application mode.
- 2 Draw the geometry.
- 3 Define the physics settings.

4 Solve and visualize the results.

Then add another application mode:

5 Go to the **Multiphysics** menu and choose **Model Navigator**.

6 Select a new application mode.

7 In the **Multiphysics** area, click the **Add** button.

8 Click **OK**.

9 Add and modify settings in the **Physics** menu, including couplings between the physics.

#### **BUILDING A COUPLED MULTIPHYSICS MODEL DIRECTLY**

Another approach is to include multiple application modes when you start creating the model in the **Model Navigator**:

1 Select an application mode for the model.

2 Click the **Multiphysics** button.

3 In the **Multiphysics** area, click the **Add** button.

4 Continue selecting application modes and adding them to the model by clicking **Add**.

5 Click **OK**.

#### **REMOVING APPLICATION MODES**

To remove an application mode from a model:

1 Choose **Model Navigator** from the **Multiphysics** menu.

2 Select the application mode from the list under **Multiphysics**. You can also select a geometry in the list to remove the entire geometry and all application modes that it contains.

3 Click **Remove**.

4 Click **OK**.

#### **MULTIPHYSICS MODEL PROPERTIES**

When you add or remove application modes, FEMLAB updates some model properties to:

- Include all dependent variables in the added application modes in the list of variables to solve for.
- Update all application-dependent dialog boxes.

#### *Active Application Mode*

The application mode selected under **Multiphysics** becomes the active application mode when you begin modeling.

To shift between the different application modes during modeling, select one from the **Multiphysics** menu to make it active.

#### *Ruling Application Mode*

The *ruling application mode* determines the global settings for the multiphysics model from its application mode properties. For example, if the analysis type is different for two application modes in the same model, the software takes the analysis type from the ruling application mode. You select the ruling application mode for a multiphysics model from the Model Navigator.

### *Managing the Solver and Solution Components*

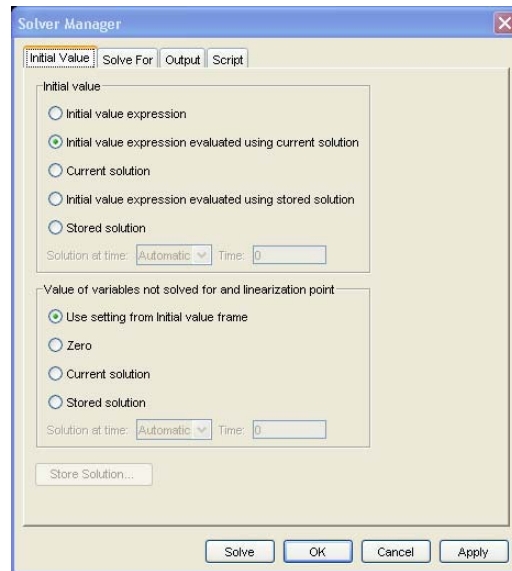
---

#### **RESTARTING THE SOLVER AND USING INITIAL VALUES**

When adding equations and physics to an existing model, you can restart the model using the current solution as an initial value. Doing so can accelerate the convergence for a coupled multiphysics model. To restart the solver, click the **Restart** button in the main toolbar.

To specify initial values:

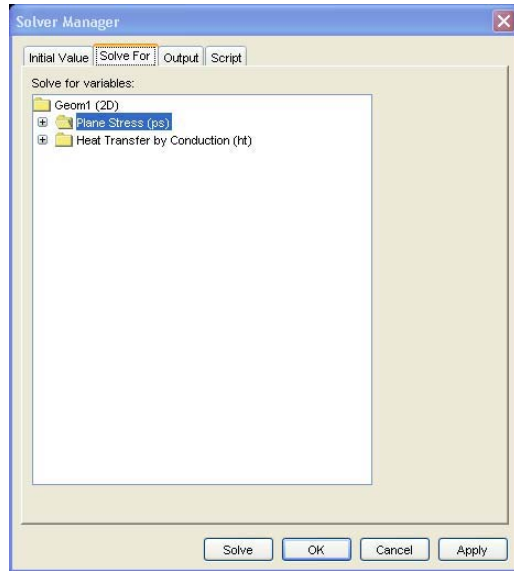
- 1** Go to the **Solve** menu and choose **Solver Manager**.
- 2** In the **Solver Manager** dialog box click the **Initial Value** tab.
- 3** Click the **Initial value expression using current solution** button (the default setting) to evaluate the expressions for the initial values as specified for each application mode using the current solution.
- 4** Click **OK**.



### SOLVING FOR A SUBSET OF DEPENDENT VARIABLES

The solver normally solves for all dependent variables. It is also possible to have it solve for a subset of the dependent variables. The initial conditions then define the value of the other variables. To specify which variables to solve for:

- 1 Go to the **Solve** menu and choose the **Solver Manager**.
- 2 In the **Solver Manager** dialog box click the **Solve For** tab.
- 3 Within the **Solve for variables** list the software selects all application modes by default. Clear the selections for those application modes whose dependent variables you do not want to solve for.
- 4 To select individual dependent variables for an application mode, click the plus sign to open the application mode folder and select the dependent variables to include.
- 5 Click **OK**.



Use this subset approach to reduce the computational load when there is only a one-way coupling in the multiphysics model:

- 1 Solve for the variables that are not affected by the coupling.
- 2 Solve for the remaining variables, using the solution from the initial analysis by restarting the solver using the previous solution data.

For a fully coupled multiphysics model with bidirectional couplings you can also use this approach to reach convergence in an iterative manner. You solve for one type of physics at a time and then use that solution as the initial value when solving for the other type of physics. This method provides a “weakly coupled” way of solving a coupled multiphysics model. Using the standard FEMLAB approach to solving fully coupled equations simultaneously normally provides better convergence.

#### USING VARIABLES DURING ANALYSIS ONLY

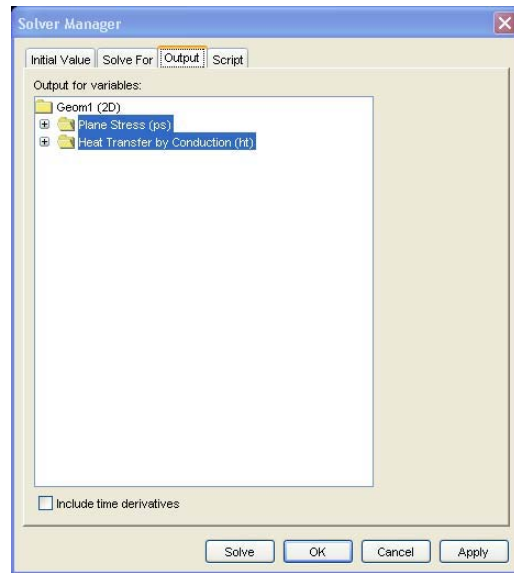
In a similar way you can choose to output the results for a subset of the dependent variables:

- 1 In the **Solver Manager** dialog box click the **Output** tab.
- 2 In the **Output for variables** list select from the application modes and their dependent variables in the same way as for the variables to solve for.

**3** Click **OK**.

This method can be useful if you want to include variables for multiphysics couplings during the analysis but are not interested in using them for postprocessing.

If you include nonsolved variables for output they take on the specified initial value as their solution data in the output from the solver.



### *Deactivating Physics or Equations*

---

Sometimes subdomains have different types of physics associated with them, and an application mode might not be meaningful in a subdomain. For example, fluid flow can be present only in some subdomains of a model whereas heat transfer takes place in the entire geometry. By default all application modes are active in all subdomains. To deactivate an application mode, clear the **Active in this subdomain** check box in the **Subdomain Settings** dialog box. Some interior boundaries then become exterior boundaries, and some of the original boundaries are absent for certain application modes.

## Using the Model Navigator for Multiphysics Models

---

To add and remove application modes and geometries for a multiphysics model, go to the **Multiphysics** menu and then open the **Model Navigator**. Use its **Multiphysics** area to make changes to the current model's multiphysics properties.

### ADDING GEOMETRIES TO A MODEL

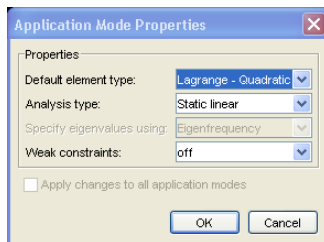
A model always has at least one geometry. To add another geometry to a model, click the **Add Geometry** button in the **Multiphysics** area of the **Model Navigator**. In such an *extended multiphysics* model the geometries can have different space dimensions. You can also choose the names of the independent variables (spatial coordinates).

### CHANGING APPLICATION MODE PROPERTIES

Before adding an application mode, you can edit its name and dependent variables. FEMLAB provides a unique name for each application mode in the model to identify the origin of the variables. You can change the name to any other name that is unique within the model.

You can also specify the element type, although the default is usually a good choice.

To set the analysis type and other application mode properties for the active application mode in a multiphysics model, click the **Application Mode Properties** button. These properties vary depending on the application mode.





# Example: Resistive Heating

For an example of a multiphysics model with thermal-electric couplings, look at resistive heating in a copper plate.

## *Introduction*

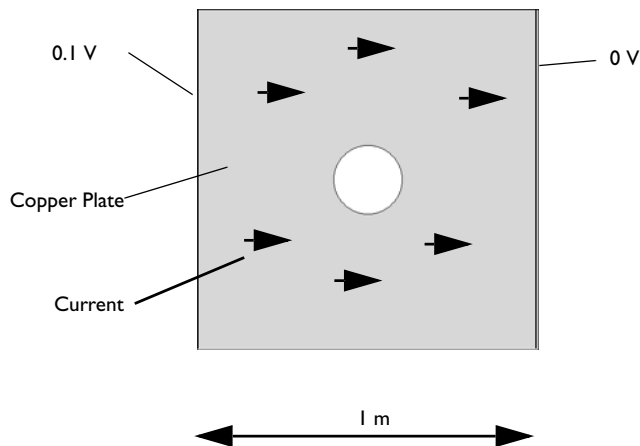
---

The material heats up when an electric current passes through it due to electrical resistance. This is called resistive heating or Joule heating. The coupling also works in the opposite direction—the material’s electric resistance varies with temperature and increases as the material heats up.

## *Model Definition*

---

Imagine a copper plate measuring 1 x 1 m that also contains a small hole. The plate’s thickness has no effect on the model. Suppose that you subject the plate to an electric potential difference across two opposite sides (all other sides are insulated). The potential difference induces a current that heats the plate.



In the 2D model you view the plate from above.

## *Modeling in FEMLAB*

---

This example first sets up the model's electric portion using the Conductive Media DC application mode. The potential distribution in the conductive media occurs almost instantly, and you can use a stationary analysis. Then add a heat transfer application mode to create a coupled multiphysics model. Because heat transfer is a transient phenomenon, the full multiphysics model uses a time-dependent solver.

### **ADDITIONAL MULTIPHYSICS COUPLINGS**

By adding a structural mechanics application mode to this model you could represent thermomechanical coupling with a body force proportional to the temperature gradient. For more information on thermomechanical multiphysics see the model “Stresses in Cracked Heat Exchanger Tubes” on page 265 in the *FEMLAB Model Library*.

---

**Model Library path:** FEMLAB/Multiphysics/resistive\_heating

---

## *Modeling Using the Graphical User Interface*

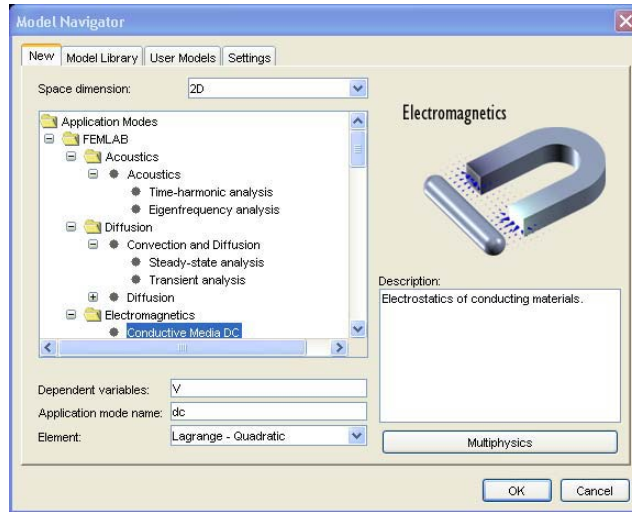
---

To start making a stationary model of the electric effects, use the Conductive Media DC application mode.

### **MODEL NAVIGATOR**

- 1 On the **New** page, select **2D** from the **Space dimension** list.
- 2 In the list of application modes, open the **Electromagnetics** folder (in the **FEMLAB** folder if your license also include modules) and select **Conductive Media DC**. By default the element type is set to **Lagrange - Quadratic**.

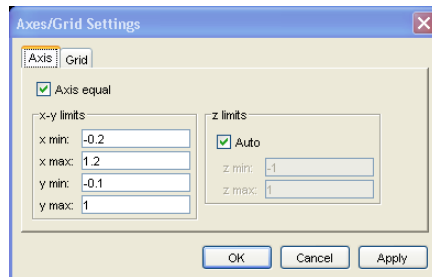
3 Click **OK**.



#### OPTIONS AND SETTINGS

1 From the **Options** menu choose **Axes/Grid Settings**.

2 In the **Axes/Grid Settings** dialog box click the **Axis** tab and make these entries: -0.2 for **x min**, 1.2 for **x max**, -0.1 for **y min**, and 1.1 for **y max**.



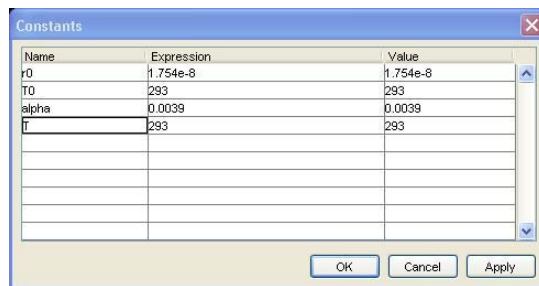
3 Click **OK**.

4 From the **Options** menu choose **Constants** to define constants to use as parameters for the model.

5 Enter the following constant names and values:

NAME OF CONSTANT	EXPRESSION
r0	1.754e-8
T0	293
alpha	0.0039
T	293

Enter the constants name in the **Name** edit field and then place its value in the **Expression** field:



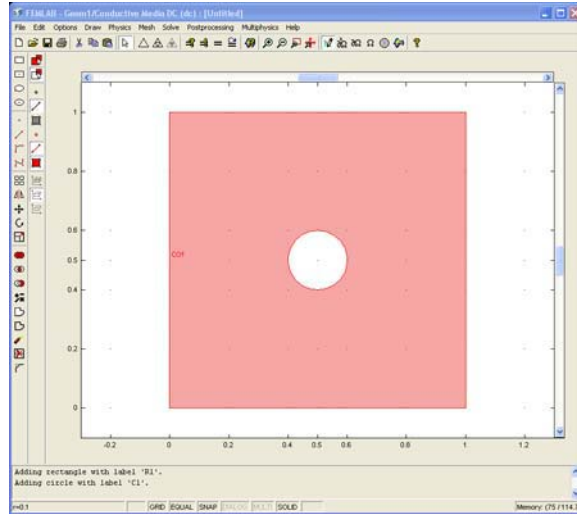
6 Click **OK**.

## GEOMETRY MODELING

Define the model geometry:

- 1 Click the **Rectangle/Square** toolbar button.
- 2 Using the right mouse button, draw a square with corners at (0, 0) and (1, 1).
- 3 Open the **Axes/Grid Settings** dialog box and click the **Grid** tab.
- 4 Clear the **Auto** check box. Leave the *x*- and *y*-axis grid spacing at 0.2, and add extra *x*- and *y*-tick marks at  $x = 0.5$  and  $y = 0.5$ .
- 5 Click **OK**.
- 6 Draw a small circle centered at (0.5, 0.5) by first clicking the **Ellipse/Circle (Centered)** toolbar button. Right click at (0.5, 0.5) and drag the pointer to create a circle with a radius of 0.1.
- 7 To create a hole, select both geometry objects by pressing **Ctrl+A**.

8 Click the **Difference** button on the draw toolbar.



## PHYSICS SETTINGS

### Boundary Conditions

1 Open the **Boundary Settings** dialog box by choosing **Boundary Settings** from the **Physics** menu. The boundary conditions for the Conductive Media DC application mode in this example are:

BOUNDARY	1	4	2,3,5-8
Type	Electric potential	Electric potential	Insulation
V	0.1	0	

2 Press **Ctrl+A** to select all boundaries.

3 In the **Boundary condition** list select **Electric insulation**.

Change the conditions of boundaries 1 and 4:

4 In the **Boundary selection** list select boundary 1, the left boundary.

5 In the **Boundary condition** list select **Electric potential** and type 0.1 in the  $V_0$  edit field.

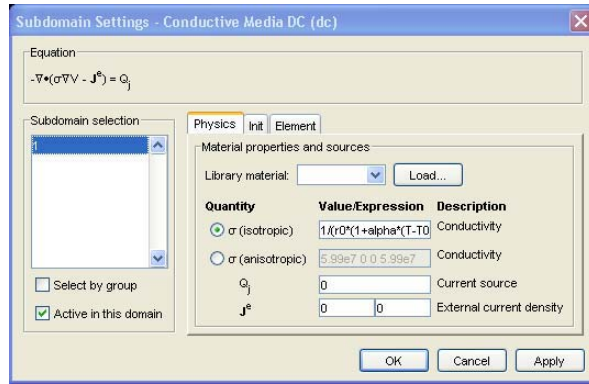
6 Select boundary 4.

7 In the **Boundary condition** list select **Ground**.

8 Click **OK**.

### Subdomain Settings

1 From the **Physics** menu choose **Subdomain Settings**.



The domain parameters are:

SUBDOMAIN	1
$\sigma$	$1 / (r0 * (1 + \alpha * (T - T0)))$
$Q_j$	0

In the Conductive Media DC application mode you specify the electrical conductivity,  $\sigma$ , rather than electrical resistivity,  $\rho$ . Recall the relation between these quantities is  $\sigma = 1/\rho$ .

Over a range of temperatures the conductivity  $\sigma$  is a function of temperature  $T$  according to

$$\sigma = \frac{1}{[\rho_0(1 + \alpha(T - T_0))]}$$

where  $\rho_0$  is the resistivity at the reference temperature  $T_0$ . Because  $T = T_0 = 293$  in the **Constants** dialog box, the actual relation in this first part of the coupled model is

$$\sigma = \frac{1}{\rho_0}.$$

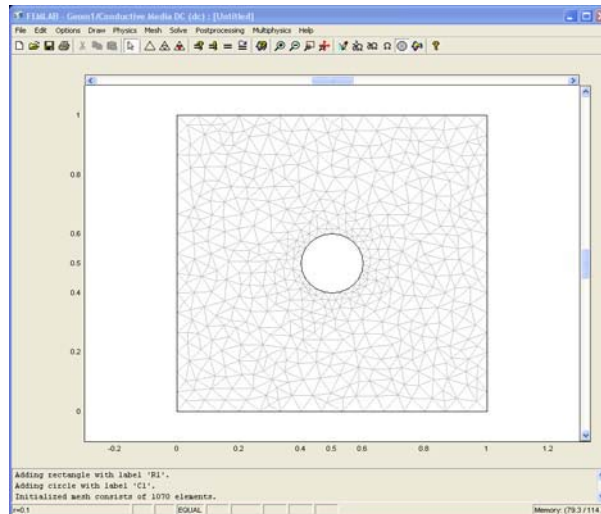
Later,  $T$  will be the temperature from the multiphysics coupling to the chosen heat transfer application mode.

2 Select subdomain 1 from the **Subdomain selection** list.

- 3 Enter the expression  $1 / (r0 * (1 + \alpha * (T - T0)))$  in the **Conductivity (isotropic)** edit field.
- 4 Click **OK**.

### MESH GENERATION

Create a mesh by clicking the **Initialize Mesh** button on the main toolbar.



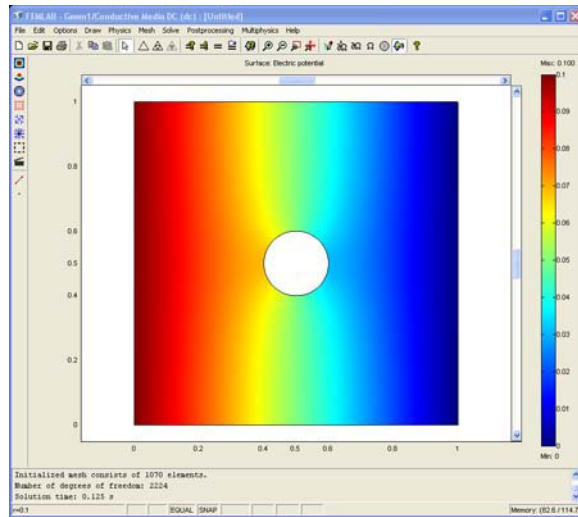
### COMPUTING THE SOLUTION

To start the solution process:

- 1 Click the **Solve** button.
- 2 When the plot appears click the **Zoom Extents** button.

## POSTPROCESSING AND VISUALIZATION

The default plot is a surface plot of the electric potential (V) in the domain

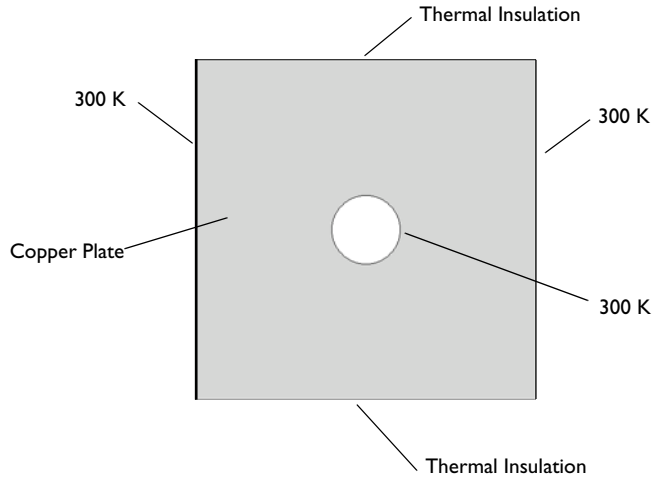


## HEAT TRANSFER AND CONDUCTIVE MEDIA DC COUPLING

The electric current generates heat in the copper plate. To model this effect, add the Conduction application mode to account for heat transfer by conduction. You can then introduce couplings using FEMLAB's multiphysics capabilities.



An air stream at 300 K (27° C) cools the plate except on the thermally insulated upper and lower edges



To set up the multiphysics model, you need to know how the electric current affects temperature. The electric potential  $V$  is the solution variable in the Conductive Media DC application mode. The generated heat  $Q$  is proportional to the square of the magnitude of the electric current density  $J$ . Current density, in turn, is proportional to the electric field, which equals the negative of the gradient of the potential  $V$ :

$$Q \propto |J|^2$$

The coefficient of proportionality is the electrical resistivity  $\rho = 1/\sigma$ , which is also the reciprocal of the temperature-dependent electric conductivity  $\sigma = \sigma(T)$ . Combining these facts gives the fully coupled relation

$$Q = \frac{1}{\sigma} |J|^2 = \frac{1}{\sigma} |\sigma E|^2 = \sigma |\nabla V|^2$$

Quantities other than  $\sigma$  also vary with temperature. For example, the thermal conductivity is temperature dependent, and a refined model would take this into account.

### OPTIONS AND SETTINGS

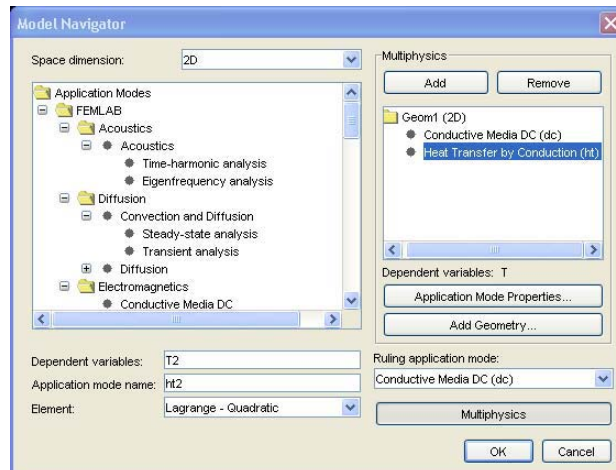
To include thermal coupling, add a heat transfer application mode. It includes a dependent variable  $T$  for temperature. However, the model you just created already has a model parameter with that name, so you must delete the constant  $T$ :

- 1 Go to the **Options** menu and choose **Constants**.
- 2 Select the constant  $T$  and click **Delete**.
- 3 Click **OK**.

### MODEL NAVIGATOR

Add the desired heat transfer application mode:

- 1 From the **Multiphysics** menu choose **Model Navigator**.
- 2 In the list of application modes on the left open the **Heat Transfer** folder (in the **FEMLAB** folder, if your license includes additional modules).
- 3 Select **Conduction**.
- 4 Click the **Add** button.
- 5 Click **OK**.



**Heat Transfer by Conduction** now appears in the FEMLAB title bar.

### PHYSICS SETTINGS

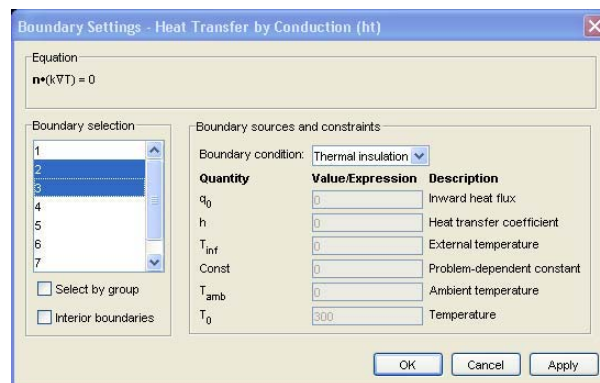
#### *Boundary Conditions*

- 1 Once again open the **Boundary Settings** dialog box from the **Physics** menu.

The boundary conditions for this model are:

BOUNDARY	1,4-8	2,3
Type	Temperature	Insulation
$T$	300	

- 2 Select all boundaries.
- 3 Select **Temperature** from the **Boundary conditions** list.
- 4 Type 300 in the  $T_0$  (temperature) edit field. This value corresponds to holding all boundaries at 300 K by positioning them in an air stream at room temperature.
- 5 Select boundaries 2 and 3, the top and bottom boundaries.
- 6 Select **Thermal insulation** from the **Boundary conditions** list.
- 7 Click **OK**.



### Subdomain Settings

The material properties for heat transfer are:

SUBDOMAIN	1
$\rho$	8930
$C$	340
$k$	384
$Q$	$1 / (r0 * (1 + a1pha * (T - T0))) * (Vx^2 + Vy^2)$

- 8 From the **Physics** menu choose **Subdomain Settings**.
- 9 Select subdomain 1 from the **Subdomain selection** list.
- 10 Type 8930 in the **Density** edit field (the density of copper expressed in  $\text{kg}/\text{m}^3$ ).

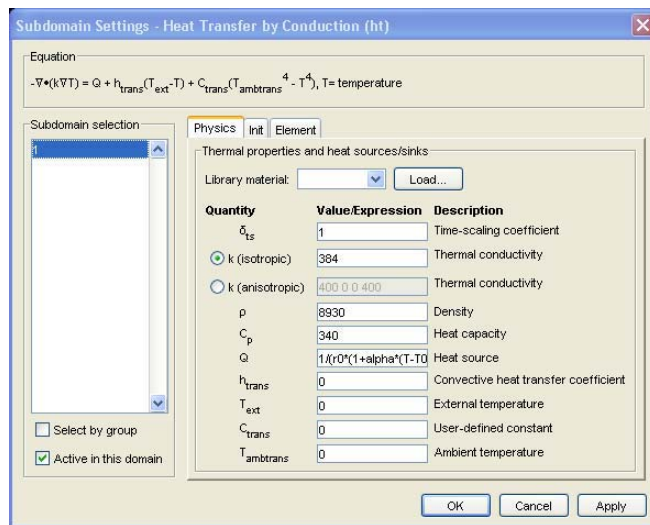
**11** Type 340 in the **Heat capacity** edit field. The unit for heat capacity in this model is Ws/kgK, the SI unit.

**12** Type 384 in the **Thermal conductivity (isotropic)** edit field.

This coefficient has the unit W/mK and represents the material's ability to conduct heat per unit time.

**13** Type  $1 / (r_0 * (1 + \alpha * (T - T_0))) * (V_x^2 + V_y^2)$  in the **Heat source** edit field.

Doing so makes this a nonlinear multiphysics model because a function of the gradient of the electric potential ( $V_x^2 + V_y^2$ ) occurs in the heat source.



Also enter the following initial values:

SUBDOMAIN	I
$V$	$0.1 * (1 - x)$
$T$	300

**1** Click the **Init** tab in the **Subdomain Settings** dialog box.

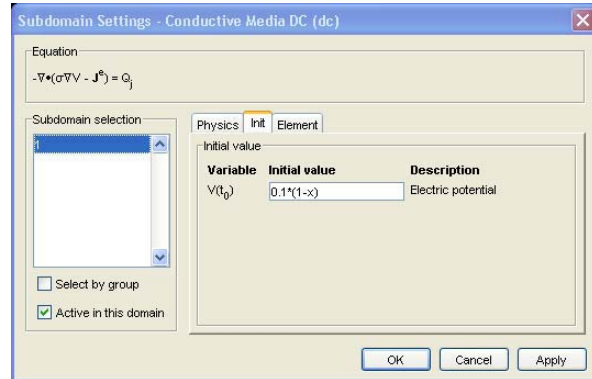
**2** Type 300 as the initial value in the edit field for **T(t<sub>0</sub>)**.

**3** Click **OK**.

**4** Switch to the Conductive Media DC application mode, selecting it from the **Multiphysics** menu.

**5** From the **Physics** menu again choose **Subdomains Settings**.

- 6 Click the **Init** tab of the **Subdomain Settings** dialog box for the Conductive Media DC application mode
- 7 Type  $0.1 * (1 - x)$  in the edit field for  $V(t_0)$ . This expression means that the initial potential distribution varies linearly from 0.1 V at the left boundary ( $x = 0$ ) to 0 V at the right boundary ( $x = 1$ ).



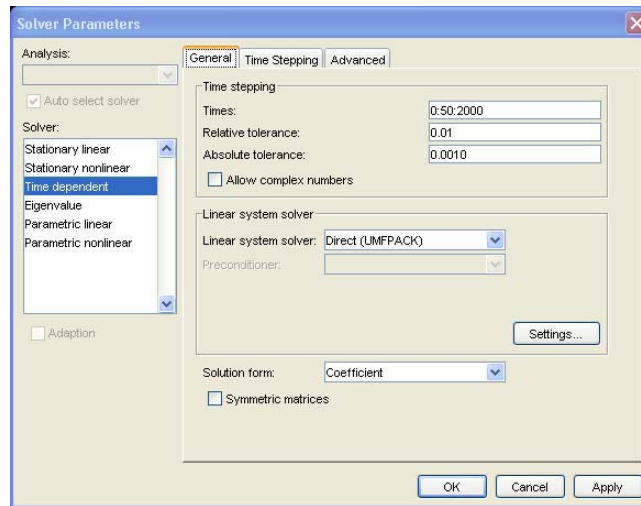
- 8 Click **OK**.

### SOLVING THE MODEL

Heat transfer is a transient process, so the model must use a time-dependent solver for a transient analysis. You must specify the simulation's time interval and the points in time to present solution data.

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu.
- 2 Select **Time dependent** in the **Solver** list.
- 3 Type  $0:50:2000$  in the **Times** edit field. Doing so produces a vector of 41 output times, linearly distributed every 50 seconds from 0 to 2000 seconds, for which the solution is accessible for postprocessing.

4 Click **OK**.



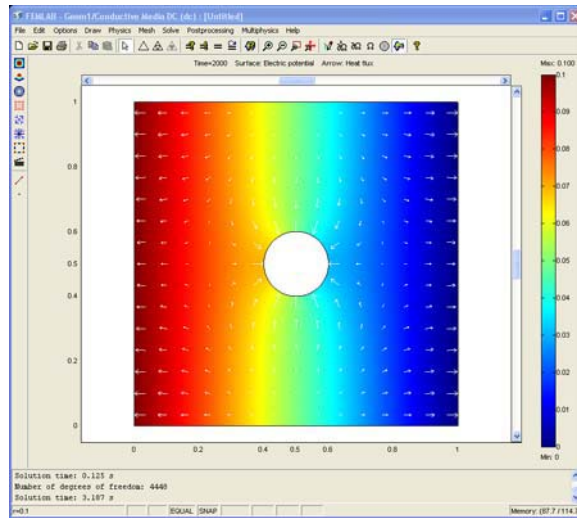
#### **SOLVING THE MODEL**

You can now run the transient analysis. To do so, simply click the **Solve** button on the main toolbar.

#### **POSTPROCESSING AND VISUALIZATION**

The plot from this analysis also shows the potential distribution.

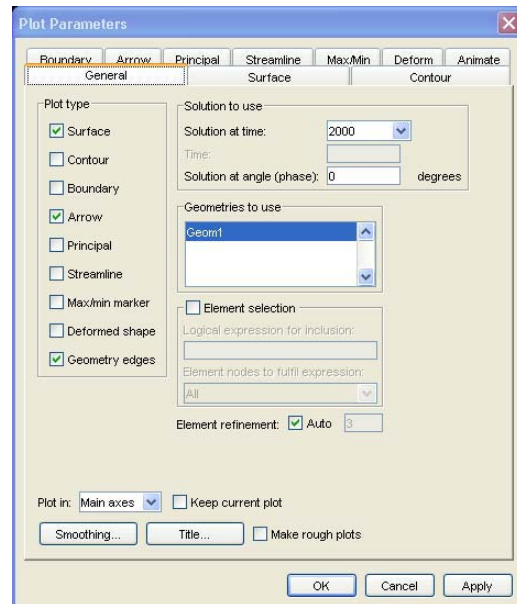
## Visualizing Heat Flux Using Arrow Plots



A quantity that is interesting to animate is the heat flux:

- 1 Open the **Plot Parameters** dialog box from the **Post** menu.
- 2 Click the **Arrow** tab.
- 3 Select the **Arrow plot** check box.
- 4 Select **Heat flux (ht)** in the **Predefined quantities** list.
- 5 Click the **Color** button and select a color for the arrows, for example, white.

6 Click **OK**.



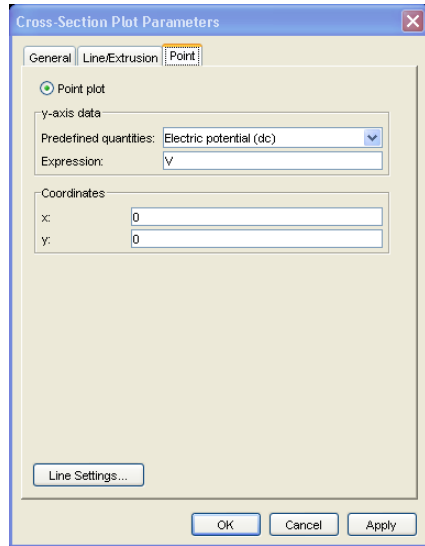
### Checking Heat Transfer Dynamics

Use a plot of temperature over time to determine if the time span for the simulation is sufficient to reach steady state:

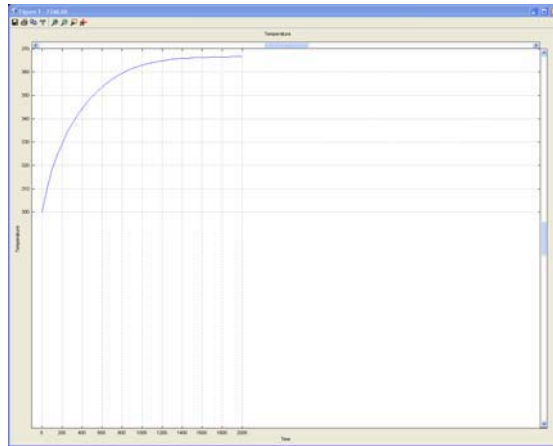
- 1 From the **Postprocessing** menu choose **Cross-Section Plot Parameters**.
- 2 Make sure to select all time steps in the **Solutions to plot** list.
- 3 Click the **Point plot** button.
- 4 Click the **Point** tab.
- 5 Select **Temperature (ht)** from the **Predefined quantities** list.
- 6 In the **Coordinates** area, type 0.5 in the **x** edit field and 0.75 in the **y** edit field.



7 Click **OK**.



The plot of temperature as a function of time shows that it has reached a steady state. The temperature does hardly increase at all if you extend the simulation over a longer time span.



*Heat transfer dynamics visualized as temperature versus time.*

