

COMSOL MULTIPHYSICS®

Example—Resistive Heating

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

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 electric resistance. This is called resistive heating or Joule heating. There is also a coupling working in the opposite direction: the material's electric resistance varies with the temperature, increasing as the material heats up.

Model Definition

Imagine a copper plate measuring $1\text{ m} \times 1\text{ m}$ 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.

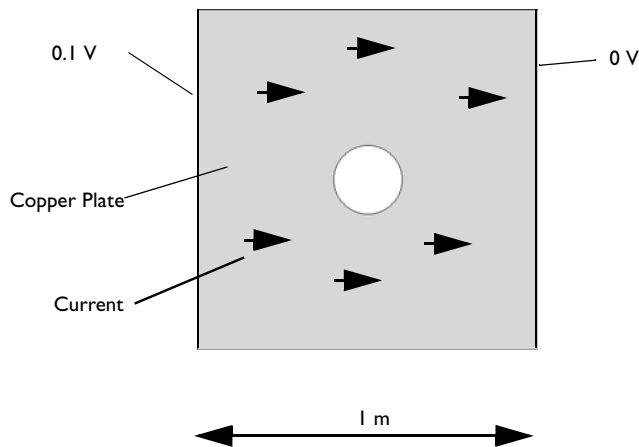


Figure 1: The model geometry and electric boundary conditions.

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

Modeling in COMSOL Multiphysics

This example sets up the coupled thermal-electric analysis using the predefined multiphysics coupling for Joule Heating, which combines a Conductive Media DC application mode with a Heat Transfer by Conduction application mode. The potential distribution in the conductive media occurs almost instantly, but because unsteady heat transfer is a transient phenomenon, the full multiphysics model uses a transient analysis.

ADDITIONAL MULTIPHYSICS COUPLINGS

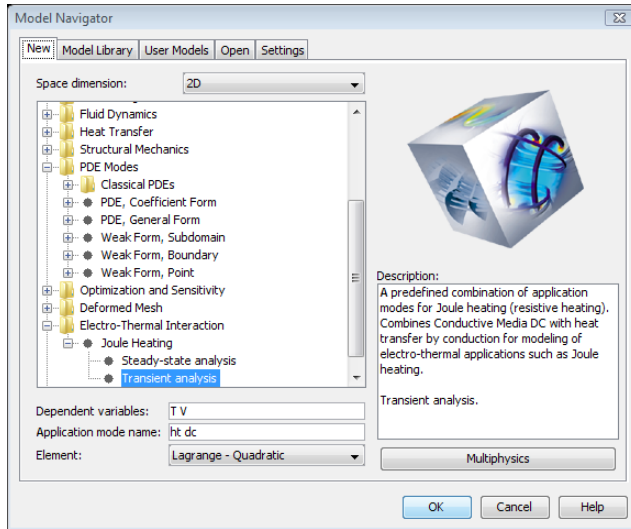
By adding a structural mechanics application mode to this model you could represent a thermomechanical coupling with a body force proportional to the temperature gradient. For more information on thermomechanical multiphysics, see the model “Simulation of a Microrobot” on page 403 in the *COMSOL Multiphysics Model Library* and the documentation for the Structural Mechanics Module.

Model Library path: COMSOL_Multiphysics/Multiphysics/
resistive_heating

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

- 1 On the **New** page, select **2D** from the **Space dimension** list.
- 2 In the list of application modes, open the **COMSOL Multiphysics>Electro-Thermal Interaction** folder and then the **Joule Heating** folder. Select **Transient analysis**.
- 3 Click **OK**.



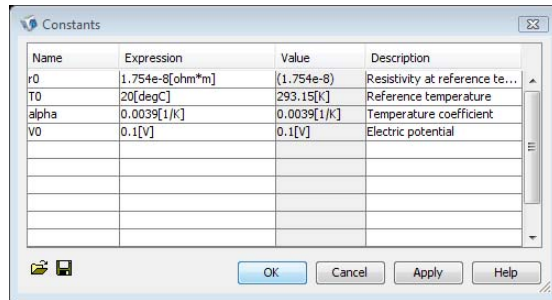
OPTIONS AND SETTINGS

- 1 From the **Options** menu choose **Constants**.

2 Enter the following constant names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
r0	1.754e-8[ohm*m]	Resistivity at reference temperature
T0	20[degC]	Reference temperature
alpha	0.0039[1/K]	Temperature coefficient
V0	0.1[V]	Electric potential

Enter the constant's name in the **Name** edit field, then place its value and unit in the **Expression** field, and finally add a description in the **Description** edit field:



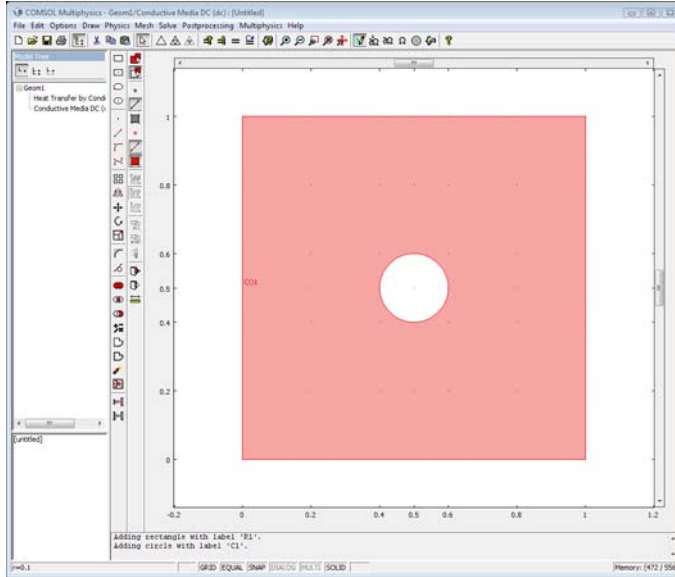
3 Click **OK**.

GEOMETRY MODELING

Define the model geometry:

- 1 Shift-click the **Rectangle/Square (Centered)** toolbar button.
- 2 In the **Square** dialog box, click **OK** to use the default values and create a unit square with corners at (0, 0) and (1, 1).
- 3 Click the **Zoom Extents** toolbar button (on the Main toolbar).
- 4 Shift-click the **Ellipse/Circle (Centered)** toolbar button.
- 5 In the **Circle** dialog box, type 0.1 in the **Radius** edit field and then type 0.5 in the **x** and **y** edit fields in the **Position** area. Click **OK**.
- 6 To create a hole, select both geometry objects by pressing Ctrl+A.

7 Click the **Difference** button on the Draw toolbar.



PHYSICS SETTINGS

The electric boundary conditions appear in Figure 1 on page 1. For the thermal boundary conditions, an air stream at 300 K (27 °C) cools the plate except on the thermally insulated upper and lower edges:

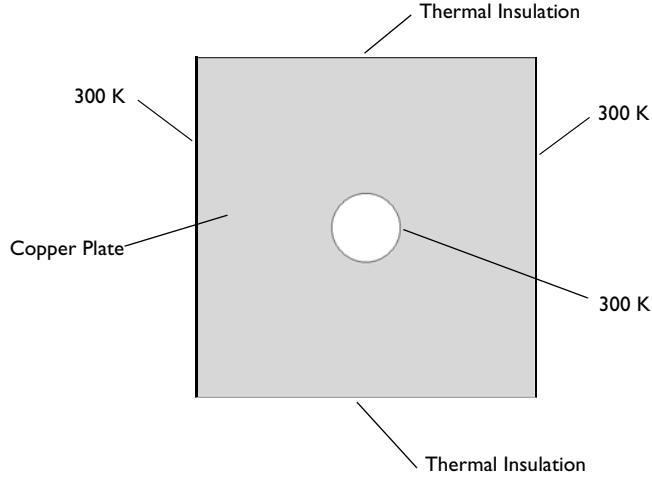


Figure 2: The model geometry and the thermal boundary conditions.

In Joule heating, the temperature increases due to the resistive heating from the electric current. The electric potential V is the solution variable in the Conductive Media DC application mode. The generated resistive 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 electric 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

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

This resistive heating source term is directly available as the variable Q_{dc} (Q_{emdc} if you use the AC/DC Module) and is predefined as the source term in the heat transfer application mode when using the Joule Heating predefined multiphysics coupling.

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

Over a range of temperatures the electric conductivity σ is a function of temperature T according to

$$\sigma = \frac{\sigma_0}{1 + \alpha(T - T_0)}$$

where σ_0 is the conductivity at the reference temperature T_0 . α is the temperature coefficient of resistivity, which describes how the resistivity varies with temperature. A typical value for copper is 0.0039 per kelvin.

In the Conductive Media DC application mode you can specify the electric conductivity for Joule heating in terms of this equation. The predefined multiphysics coupling sets up this specification of sigma with the variable for temperature in the **T** edit field and default values suitable for copper. The only thing you have to add is the reference temperature, T_0 .

Boundary Conditions—Heat Transfer

- 1 Make sure that **Heat Transfer by Conduction (ht)** or **General Heat Transfer (htgh)** (if you use the Heat Transfer Module) is the selected application mode in the **Multiphysics** menu.
- 2 Open the **Boundary Settings** dialog box from the **Physics** menu.

The thermal boundary conditions are:

SETTINGS	BOUNDARIES 1, 4-8	BOUNDARIES 2, 3
Type	Temperature	Thermal insulation
T_0	300	

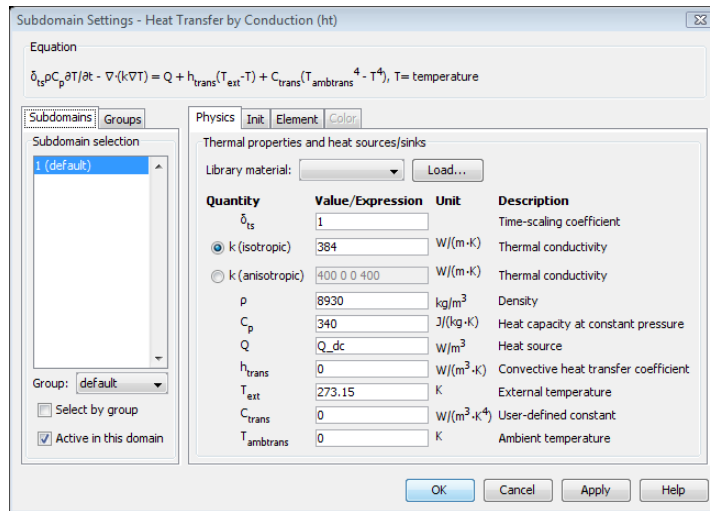
- 3 Select all boundaries.
- 4 Select **Temperature** from the **Boundary condition** list.
- 5 Type 300 in the **T_0** (temperature) edit field. This value corresponds to holding all boundaries at 300 K by positioning them in an air stream at room temperature.
- 6 Select Boundaries 2 and 3, the top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

Subdomain Settings—Heat Transfer

The material properties for heat transfer are:

PROPERTY	VALUE
ρ	8930
C_p	340
k (isotropic)	384
Q	Q_dc

- 1 From the **Physics** menu choose **Subdomain Settings**.
- 2 Select Subdomain 1 from the **Subdomain selection** list.
- 3 Type 8930 in the **Density** edit field (the density of copper in kg/m^3).
- 4 Type 340 in the **Heat capacity at constant pressure** edit field. The unit for heat capacity in this model is $\text{J}/(\text{kg}\cdot\text{K})$, the SI unit.
- 5 Type 384 in the **k (isotropic)** edit field for the thermal conductivity.
This coefficient has the SI unit $\text{W}/(\text{m}\cdot\text{K})$ and represents the material's ability to conduct heat per unit time.
- 6 The predefined setting in the **Heat source** edit field is Q_dc (or Q_emdc). This is a predefined variable for the resistive heating from the Conductive Media DC application mode and includes the temperature-dependent conductivity and the gradient of the electric potential.



Initial Conditions—Heat Transfer

Also enter the initial value for the temperature, which is 300 K just as the boundary condition:

- 1 Click the **Init** tab in the **Subdomain Settings** dialog box.
- 2 Type 300 as the initial value in the edit field for **T(t₀)**.
- 3 Click **OK**.

Application Mode Selection

To define the settings for the copper plate's electric properties, switch to the Conductive Media DC (dc) application mode, selecting it from the **Multiphysics** menu or using the Model Tree.

Boundary Conditions—Conductive Media DC

- 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:

SETTINGS	BOUNDARY 1	BOUNDARY 4	BOUNDARIES 2, 3, 5–8
Type	Electric potential	Ground	Electric insulation
V	V0		

- 2 Press Ctrl+A to select all boundaries.
- 3 From the **Boundary condition** list select **Electric insulation**.
Change the conditions on Boundaries 1 and 4:
- 4 From the **Boundary selection** list select Boundary 1, the left boundary.
- 5 From the **Boundary condition** list select **Electric potential** and type V0 in the **V₀** edit field.
- 6 Select Boundary 4.
- 7 From the **Boundary condition** list select **Ground**.
- 8 Click **OK**.

Subdomain Settings—Conductive Media DC

- 1 From the **Physics** menu choose **Subdomain Settings**.
- 2 Select Subdomain 1 from the **Subdomain selection** list.
- 3 From the **Conductivity relation** list, select **Linear temperature relation**.
- 4 Make sure that the **T** edit field contains the field variable for temperature, T.
- 5 Type r0 in the ρ_0 edit field for the resistivity at the reference temperature.

- 6 Type α in the α edit field for the temperature coefficient.
- 7 Type T_0 in the T_0 edit field for the reference temperature.

Initial Conditions—Conductive Media DC

- 1 In the **Subdomain Settings** dialog box, click the **Init** tab.
- 2 Type $V_0*(1-x[1/m])$ in the edit field for $V(t_0)$. This expression means that the initial potential distribution varies linearly from V_0 (0.1 V) at the left boundary ($x = 0$) to 0 V at the right boundary ($x = 1$). This is an initial condition that matches the boundary conditions. The reason for the multiplying x with the unit [1/m] is to make it dimensionless.
- 3 Click **OK**.

MESH GENERATION

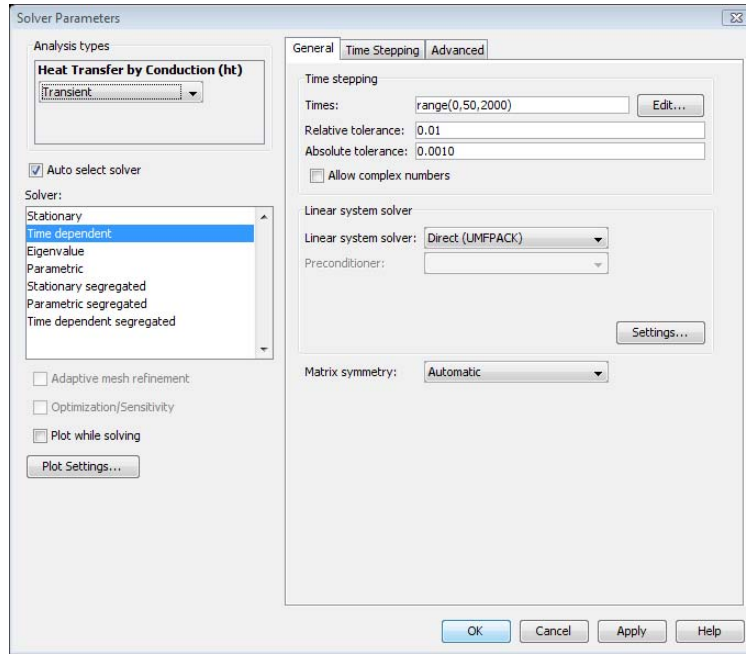
Click the **Initialize Mesh** button on the Main toolbar to create a mesh using the default settings.

COMPUTING THE SOLUTION

The heat transfer in this model is a transient process, so the model uses a time-dependent solver for a transient analysis. First 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 Type range (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 available for postprocessing.

3 Click **OK**.



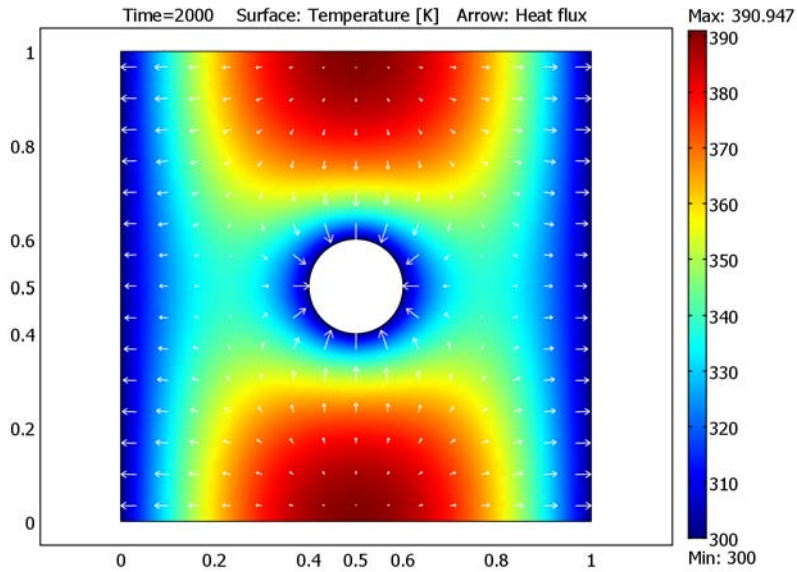
COMPUTING THE SOLUTION

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

POSTPROCESSING AND VISUALIZATION

The default plot from this analysis shows the temperature or 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 **Postprocessing** menu.
- 2 Click the **Arrow** tab.
- 3 Select the **Arrow plot** check box.
- 4 Select **Heat Transfer by Conduction (ht)>Heat flux** from 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 use** list on the **General** page.
- 3 Click the **Point plot** button.
- 4 Click the **Point** tab.
- 5 Select **Heat Transfer by Conduction (ht)>Temperature** in 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 hardly increases at all if you extend the simulation over a longer time span.

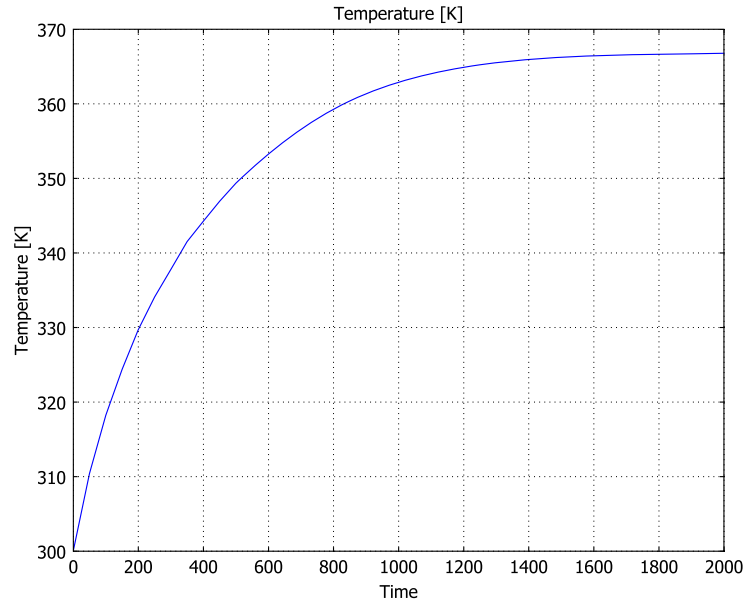


Figure 3: Heat transfer dynamics visualized as temperature versus time.

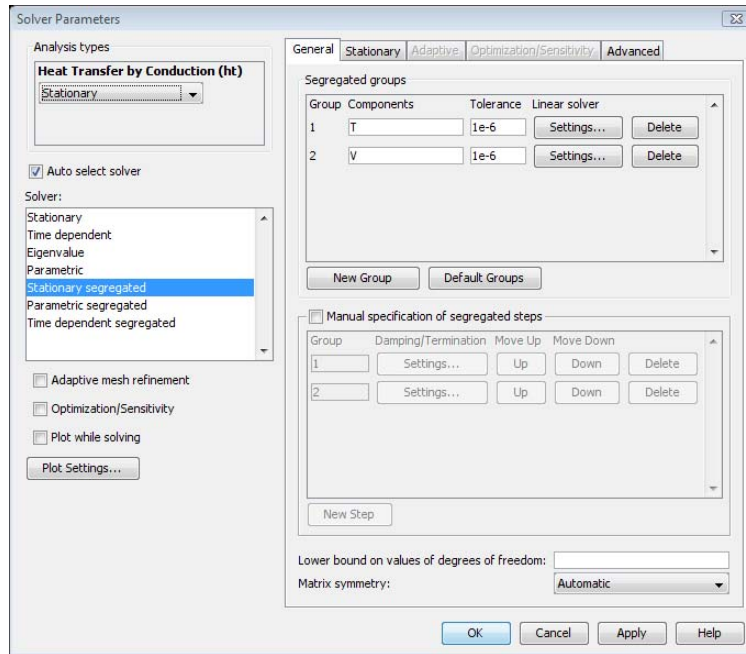
COMPUTING THE SOLUTION USING A SEGREGATED STATIONARY SOLVER

You can also solve this model for the steady-state condition. There is no problem solving this coupled multiphysics model using the standard stationary solver because the model is small. For large models, a segregated solver approach can be beneficial. This example shows how to solve a multiphysics model using the segregated stationary solver. You continue with the Resistive Heating model, adding the following steps:

- 1** From the **Solve** menu, choose **Solver Parameters**.
- 2** Select **Stationary** from the **Analysis** list.
- 3** Select **Stationary segregated** from the **Solver** list. COMSOL Multiphysics has two predefined groups, one for each application mode (the temperature T and the electric potential V). All you need to do is to specify the tolerances so that the

segregated solver scheme converges to the same solution as the standard stationary solver. Use the default linear solver settings (the direct solver UMFPACK).

- 4 Type $1e-6$ in the **Tolerance** edit field for both groups (Group 1 for T and Group 2 for V). The following figure shows the settings for the stationary segregated solver.



- 5 Click **OK**.
- 6 Click the **Solve** button on the Main toolbar.