

Tumor Removal

Introduction

One method for removing cancerous tumors from healthy tissue is to heat the malignant tissue to a critical temperature that kills the cancer cells. This example accomplishes the localized heating by inserting a four-armed electric probe through which an electric current runs. Equations for the electric field for this case appear in the Conductive Media DC application mode, and this example couples them into the bioheat equation, which models the temperature field in the tissue. The heat source resulting from the electric field is also known as *resistive heating* or *Joule heating*. The original model comes from S. Tungjatkusolmun and others (Ref. 1), but we have made some simplifications. For instance, while the original uses RF heating (with AC currents), the COMSOL Multiphysics model approximates the energy with DC currents.

This medical procedure removes the tumorous tissue by heating it above 45 °C to 50 °C. Doing so requires a local heat source, which physicians create by inserting a small electric probe. The probe is made of a trocar (the main rod) and four electrode arms as shown in Figure 4-1. The trocar is electrically insulated except near the electrode arms.

An electric current through the probe creates an electric field in the tissue. The field is strongest in the immediate vicinity of the probe and generates resistive heating, which dominates around the probe's electrode arms because of the strong electric field.

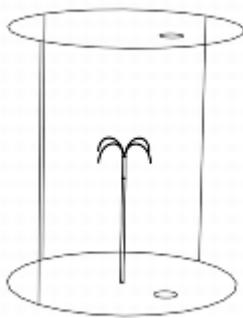


Figure 4-1: Cylindrical modeling domain with the four-armed electric probe in the middle, which is located next to a large blood vessel.

Model Definition

This model uses the Bioheat Equation and the Conductive Media DC application modes to implement a transient analysis.

The standard temperature unit in COMSOL Multiphysics is kelvin (K). This model uses the Celsius temperature scale, which is more convenient for models involving the Bioheat Equation.

The model approximates the body tissue with a large cylinder and assumes that its boundary temperature remains at 37 °C during the entire procedure. The tumor is located near the center of the cylinder and has the same thermal properties as the surrounding tissue. The model locates the probe along the cylinder's center line such that its electrodes span the region where the tumor is located. The geometry also includes a large blood vessel.

The bio-heat equation governs heat transfer in the tissue

$$\delta_{ts}\rho C \frac{\partial T}{\partial t} + \nabla \cdot (-k\nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext}$$

where δ_{ts} is a time-scaling coefficient; ρ is the tissue density (kg/m^3); C is the tissue's specific heat ($\text{J}/(\text{kg}\cdot\text{K})$); and k is its thermal conductivity ($\text{W}/(\text{m}\cdot\text{K})$). On the right side of the equality, ρ_b gives the blood's density (kg/m^3); C_b is the blood's specific heat ($\text{J}/(\text{kg}\cdot\text{K})$); ω_b is its perfusion rate ($1/\text{s}$); T_b is the arterial blood temperature ($^{\circ}\text{C}$); while Q_{met} and Q_{ext} are the heat sources from metabolism and spatial heating, respectively (W/m^3).

In this model, the bioheat equation also models heat transfer in various parts of the probe with the appropriate values for the specific heat, C (J/(kg·K)), and thermal conductivity, k (W/(m·K)). For these parts, all terms on the right-hand side are zero.

The model next sets the boundary conditions at the outer boundaries of the cylinder and at the walls of the blood vessel to a temperature of 37 °C. Assume heat flux continuity on all other boundaries.

The initial temperature equals 37 °C in all domains.

The governing equation for the Conductive Media DC application mode is

$$-\nabla \cdot (\sigma \nabla V - \mathbf{J}^e) = Q_j$$

where V is the potential (V), σ the electric conductivity (S/m), \mathbf{J}^e an externally generated current density (A/m²), Q_j the current source (A/m³).

In this model both \mathbf{J}^e and Q_j are zero. The governing equation therefore simplifies into:

$$-\nabla \cdot (\sigma \nabla V) = 0$$

The boundary conditions at the cylinder's outer boundaries is ground (0 V potential). At the electrode boundaries the potential equals 23 V. Assume continuity for all other boundaries.

The boundary conditions for the Conductive Media DC application mode are:

$$\begin{aligned} V &= 0 && \text{on the cylinder wall} \\ V &= V_0 && \text{on the electrode surfaces} \\ \mathbf{n} \cdot (\mathbf{J}_1 - \mathbf{J}_2) &= 0 && \text{on all other boundaries} \end{aligned}$$

The boundary conditions for the bioheat equation are:

$$\begin{aligned} T &= T_b && \text{on the cylinder wall and blood-vessel wall} \\ \mathbf{n} \cdot (k_1 \nabla T_1 - k_2 \nabla T_2) &= 0 && \text{on all interior boundaries} \end{aligned}$$

The model solves the above equations with the given boundary conditions to obtain the temperature field as a function time.

Results and Discussion

The model shows how the temperature increases with time in the tissue around the electrode.

The slice plot in [Figure 4-2](#) illustrates the temperature field 60 seconds after starting the procedure.

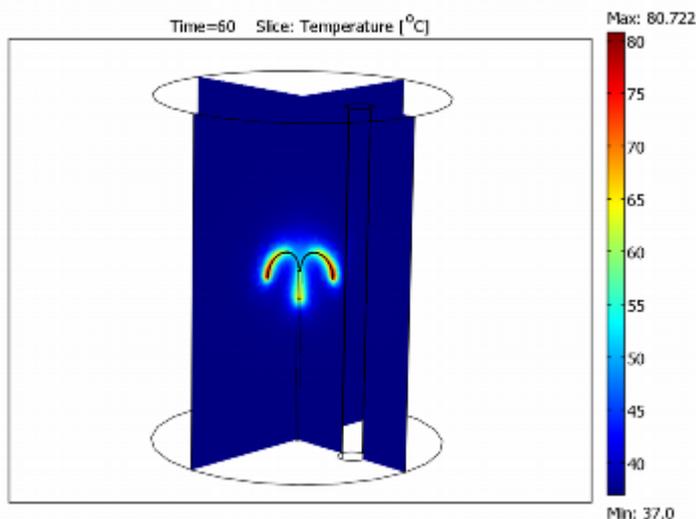


Figure 4-2: Temperature field at time = 60 seconds.

[Figure 4-3](#) shows the temperature at the tip of one of the electrode arms. The temperature rises quickly until it reaches a steady-state temperature of about 90 °C.

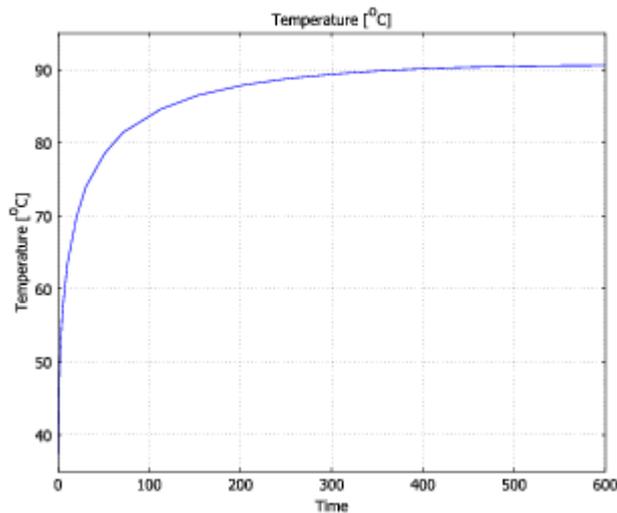


Figure 4-3: Temperature versus time at the tip of one of the electrode arms.

It is also interesting to visualize the region where cancer cells die, that is, where the temperature has reached at least 50 °C. You can visualize this area with an isosurface for that temperature; [Figure 4-4](#) shows one after 8 minutes.

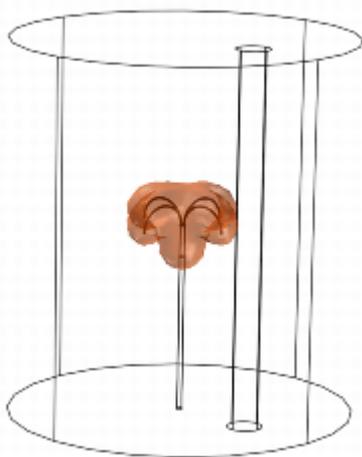


Figure 4-4: Visualization of the region that has reached 50 °C after 8 minutes.

Reference

1. S. Tungjitkusolmun, S. Tyler Staelin, D. Haemmerich, J.Z. Tsai, H. Cao, J.G. Webster, F.T. Lee, Jr., D.M. Mahvi, V.R. Vorperian, *Three-Dimensional Finite Element Analyses for Radio-Frequency Hepatic Tumor Ablation*, IEEE Transactions on Biomedical Engineering, Vol 49, No. 1, January 2002.

Model Library Path:

Heat_Transfer_Module/Medical_Technology/tumor_ablation

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**. On the **New** page, select **3D** in the **Space dimension** list.

- 2 Go to the **Heat Transfer Module** menu and select **Bioheat Equation>Transient analysis**.
- 3 Click the **Multiphysics** button and add the application mode to the model by clicking the **Add** button.
- 4 Similarly add the **Conductive Media DC** application mode, from the **COMSOL Multiphysics>Electromagnetics** menu.
- 5 Click **OK**.

OPTIONS AND SETTINGS

- 1 Define the following constants in **Constants** under the **Options** menu:

NAME	EXPRESSION	DESCRIPTION
rho_e	6450 [kg/m ³]	
rho_t	21500 [kg/m ³]	Density
rho_l	1060 [kg/m ³]	
rho_b	1000 [kg/m ³]	Density, blood
rho_c	70 [kg/m ³]	
c_e	840 [J/ (kg*K)]	
c_t	132 [J/ (kg*K)]	Heat capacity, tumor
c_l	3600 [J/ (kg*K)]	
c_b	4180 [J/ (kg*K)]	Heat capacity, blood
c_c	1045 [J/ (kg*K)]	
k_e	18 [W/ (m*K)]	
k_t	71 [W/ (m*K)]	Thermal conductivity, tumor
k_l	0.512 [W/ (m*K)]	
k_b	0.543 [W/ (m*K)]	Thermal conductivity, blood
k_c	0.026 [W/ (m*K)]	
sigma_e	1e8 [S/m]	
sigma_t	4e6 [S/m]	Electric conductivity, tumor
sigma_l	0.333 [S/m]	
sigma_b	0.667 [S/m]	Electric conductivity, blood
sigma_c	1e-5 [S/m]	
T_b	37 [degC]	Arterial blood temperature
omega_b	6.4e-3 [1/s]	Blood perfusion rate
T0	37 [degC]	Initial temperature
V0	22 [V]	

- 2 Click **OK**.

GEOMETRY MODELING

- 1 Go to the **Draw** menu and select **Work-Plane Settings**.
- 2 Create an **x-y plane** at the **z** coordinate 0.06.

- 3 Click **OK**.
- 4 Press the Shift key and click the **Ellipse/Circle (Centered)** button.
- 5 In the dialog box that appears, enter the following circle properties:

OBJECT DIMENSIONS	EXPRESSION
Radius	$9.144e-4$
Base	Center
x	0
y	0

- 6 Click **OK**.
- 7 Click the **Zoom Extents** button on the Main toolbar.
- 8 Repeat Steps 4–6 to create five additional circles, for smaller ones and a large one. The properties of each circle appear in the following five tables:

OBJECT DIMENSIONS	EXPRESSION
Radius	$2.667e-4$
Base	Center
x	$-5e-4$
y	0

OBJECT DIMENSIONS	EXPRESSION
Radius	$2.667e-4$
Base	Center
x	$5e-4$
y	0

OBJECT DIMENSIONS	EXPRESSION
Radius	$2.667e-4$
Base	Center
x	0
y	$-5e-4$

OBJECT DIMENSIONS	EXPRESSION
Radius	$2.667e-4$
Base	Center
x	0
y	$5e-4$

OBJECT DIMENSIONS	EXPRESSION
Radius	
Base	
x	
y	

Radius	5e-3
Base	Center
x	2.6e-2
y	0

9 Click the **Zoom Extents** button on the Main toolbar.

This completes the geometry needed in the 2D work plane, and the result appears in [Figure 4-5](#).

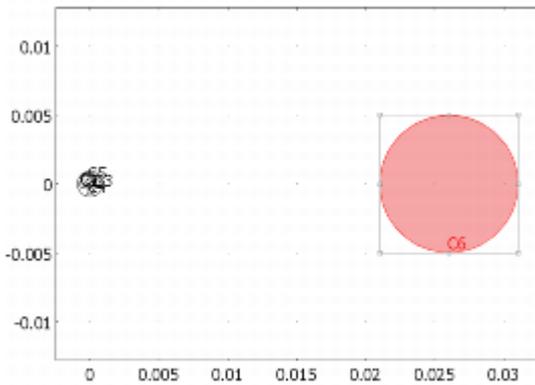


Figure 4-5: 2D working plane geometry.

The following steps describe how to create the 3D geometry by extruding and revolving the 2D geometry:

- 1 To create the conducting part of the trocar, go to the **Draw** menu and select **Extrude**.
- 2 In the dialog box that appears, select **C1** in the **Objects to extrude** list and type 10e-3 in the **Distance** edit field.
- 3 Click **OK**.
- 4 To create the insulated part of the trocar, return to the Geom2 work plane, then go to the **Draw** menu and select **Extrude**.
- 5 Select **C1** in the **Objects to extrude** list and type 50e-3 in the **Distance** edit field.
- 6 Click **OK**.
- 7 Select the EXT2 geometry object.
- 8 Click the **Move** button on the Draw toolbar.
- 9 Enter the following displacement values; when finished, click **OK**.

PROPERTY	VALUE
x	0
y	0
z	10e-3

To create the electrode arms, proceed as follows:

- 10 Return to the Geom2 work plane.
- 11 From the **Draw** menu select **Revolve**.
- 12 From the **Objects to revolve** list select **C2**.
- 13 Enter the following values in the dialog box; when finished, click **OK**.

PROPERTY	VALUE

α_1	0
α_2	180
x (point on axis)	$-8e-3$
y (point on axis)	0
x (second point)	$-8e-3$
y (second point)	1

14 Revolve the circles C3, C4, and C5 in the same manner using the following values.

Revolve parameters for the circle C3:

PROPERTY	VALUE
α_1	0
α_2	-180
x (point on axis)	$8e-3$
y (point on axis)	0
x (second point)	$8e-3$
y (second point)	1

Revolve parameters for the circle C4:

PROPERTY	VALUE
α_1	-180
α_2	0
x (point on axis)	0
y (point on axis)	$-8e-3$
x (second point)	1
y (second point)	$-8e-3$

Revolve parameters for the circle C5:

PROPERTY	VALUE
α_1	180
α_2	0
x (point on axis)	0
y (point on axis)	$8e-3$
x (second point)	1
y (second point)	$8e-3$

15 To create the blood vessel, return to the Geom2 work plane and select C6.

16 Go to the **Draw** menu and select **Extrude**.

17 In the **Distance** field enter $120e-3$.

18 Click **OK**.

19 Select the EXT3 geometry object, go to the **Draw** menu and select **Move** under **Modify**.

20 Enter the following displacement values; when finished, click **OK**.

PROPERTY	VALUE
x	0
y	0
z	-60e-3

21 To create the large cylinder, press the **Shift** key and click the **Cylinder** button.

22 Enter the following cylinder properties; when finished, click **OK**.

PROPERTY	VALUE
Radius	0.05
Height	0.12

23 Click the **Zoom Extents** button on the Main toolbar.

This concludes the drawing stage. To get a better view of the geometry you have created, do as follows:

- 1 To hide the coordinate axes, double-click the **AXIS** button on the status bar at the bottom of the user interface.
- 2 From the **Options** menu select **Visualization/Selection Settings**.
- 3 Clear the **Geometry labels** check box, then click **OK**.
- 4 Choose **Options>Suppress>Suppress Boundaries**.
- 5 Select Boundaries 34, 47, 58, 59, 62, and 63, then click **OK**.
- 6 Click the **Perspective Projection** button on the Camera toolbar.
- 7 Finally, after rotating the geometry in the drawing area upside down you should have a view similar to that in [Figure 4-1 on page 246](#).

PHYSICS SETTINGS

Subdomain Settings—Bioheat Equation

- 1 In the **Multiphysics** menu select **1 Bioheat Equation (htbh)**.
- 2 In the **Physics** menu select **Subdomain Settings**.
- 3 Select Subdomain 8 and clear the **Active in this domain** check box.
- 4 Enter the properties for the remaining subdomains as follows:

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2, 5-7	SUBDOMAIN 3	SUBDOMAIN 4
k	k_l	k_e	k_t	k_c
ρ	rho_l	rho_e	rho_t	rho_c
C	c_l	c_e	c_t	c_c
ρ_b	rho_b			
C _b	c_b			
ω_b	omega_b			
T _b	T_b			

Qmet	0			
Qext	Q_dc			

- On the **Init** page, select all active subdomains and in the **T(t₀)** edit field type T0.
- To reduce the size of the computation problem, select a lower element order by choosing the **Element** tab and then select **Lagrange - Linear** from the list of **Predefined elements** for all active subdomains.
- Click **OK**.

Boundary Conditions—Bioheat Equation

- Go to the **Physics** menu and select **Boundary Settings**.
- Enter boundary coefficients as follows:

SETTINGS	BOUNDARIES 1-4, 34 47, 58, 59, 62, 63	BOUNDARY 20
Boundary condition	Temperature	Thermal insulation
T0	T_b	

- Click **OK**.

Subdomain Settings—Conductive Media DC

- Go to the **Multiphysics** menu and select **2 Conductive Media DC (dc)**.
- Go to the **Physics** menu and select **Subdomain Settings**.
- Enter the subdomain properties as in the following table:

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2, 5-7	SUBDOMAIN 3	SUBDOMAIN 4	SUBDOMAIN 8
σ (isotropic)	sigma_l	sigma_e	sigma_t	sigma_c	sigma_b

- Reduce the element order also for this application mode: On the **Element** page, select **Lagrange - Linear** from the list of **Predefined elements** for all subdomains.
- Click **OK**.

Boundary Conditions—Conductive Media DC

- From the **Physics** menu open the **Boundary Settings** dialog box.
- Select the **Interior boundaries** check box and enter boundary coefficients as in the table below. You only need to set the boundary condition on the boundaries that use Electric potential. The remaining boundaries already have the correct conditions set by default, that is, Ground for outer boundaries and Continuity for interior boundaries.

SETTINGS	BOUNDARIES 5-16, 21-33, 35- 39, 41-43, 45, 46, 48-57	BOUNDARIES 1-4, 20, 34, 47, 60, 61	BOUNDARIES 17-19, 40, 44, 58, 59, 62, 63
Boundary condition	Electric potential	Ground	Continuity
V	v0		

- 3 Click **OK**.

MESH GENERATION

- 1 From the **Mesh** menu select **Free Mesh Parameters**.
- 2 From the **Predefined mesh sizes** list select **Fine**.
- 3 Click the **Custom mesh size** button, then modify the **Element growth rate** from the default value to 1.7.
- 4 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 From the **Solver** list select **Time dependent**.
- 3 On the **General** page, type 0 600 in the **Times** field.
- 4 From the **Linear system solver** list select **Direct (UMFPACK)**.
- 5 From the **Matrix symmetry** list select **Nonsymmetric**.
- 6 Go to the **Time Stepping** tab and select **Time steps from solver** in the **Times to store in output** list.
- 7 Select the **Manual tuning of step size** check box and type 0.01 in the **Initial time step** edit field and 50 in the **Maximum time step** edit field.
- 8 Click **OK**.
- 9 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot is a slice plot of the temperature at time = 600 seconds. In order to generate [Figure 4-2](#), follow these steps:

- 1 Click the **Plot Parameters** button on the Main toolbar.
- 2 On the **General** page select **Interpolated** from the **Solution at time** list.
- 3 In the **Time** edit field type 60.
- 4 Click the **Slice** tab and locate the **Slice data** area. From the **Predefined quantities** list select **Temperature** and from the **Unit** list select **°C**.
- 5 In the **Slice positioning** area set the number of **x levels**, **y levels**, and **z levels** to 1, 1, and 0, respectively.
- 6 Click **Apply**.

The following steps describe how to generate the plot in [Figure 4-4](#), which shows the isosurface for the temperature 50 °C, after 8 minutes:

- 1 Return to the **General** page, then clear the **Slice** check box and select the **Isosurface** check box.
- 2 In the **Time** edit field type 480.
- 3 Click the **Isosurface** tab. On the **Isosurface Data** page, select **Temperature** from the list of **Predefined quantities** and select **°C** from the **Unit** list.
- 4 In the **Isosurface levels** area, type 50 in the edit field for **Vector with isolevels**.
- 5 Click **OK**.

- To finish the plot, click the **Scene Light** button on the Plot toolbar, then click the **Increase Transparency** button 5–6 times.

Clicking the **Decrease Transparency** button repeatedly returns the transparency settings to the original state.

To create [Figure 4-3](#), which shows temperature versus time, do the following steps:

- From the **Postprocessing** menu select **Domain Plot Parameters**.
- On the **General** page, click the **Point plot** option button under **Plot type**.
- On the **Point** page, select Point 43. From the **Unit** list select **°C**, then click **OK**.

The postprocessing image that is shown when you open the tumor ablation model from the model library is created with the following steps:

- Click the **Plot Parameters** button on the Main toolbar.
- On the **General** page, clear the **Isosurface** check box and select the **Streamline** and **Slice** check boxes.
- On the **Streamline** page, select **Heat flux** from the list of **Predefined quantities**.
- On the **Start Points** page, select **Specify start point coordinates**, then specify the following settings in the **x**, **y**, and **z** edit fields:

COORD.	VALUE
x	linspace(0.02,0.02,30)
y	linspace(-0.04,0.04,10) linspace(-0.04,0.04,10) linspace(-0.04,0.04,10)
z	linspace(0.05,0.05,10) linspace(0.08,0.08,10) linspace(0.02,0.02,10)

- On the **Line Color** page, select the **Use expression** option button, then click the **Color Expression** button. In the dialog box that appears, clear the **Color scale** check box (leave the default settings in the **Streamline color data** area). Click **OK**.
- Set the **Line type** to **Tube**, then click the **Tube Radius** button. In the dialog box that appears, clear the **Auto** check box for the **Radius scale factor**, then type 0.1 in the edit field. Click **OK**.
- Click the **Slice** tab. In the **Slice positioning** area set the number of **x levels**, **y levels**, and **z levels** to 1, 0, and 0, respectively.
- Click **OK**.