Created in COMSOL Multiphysics 6.2

Hepatic Tumor Ablation

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 Electric Currents interface, and this example couples them to 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. Tungjitkusolmun 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 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.

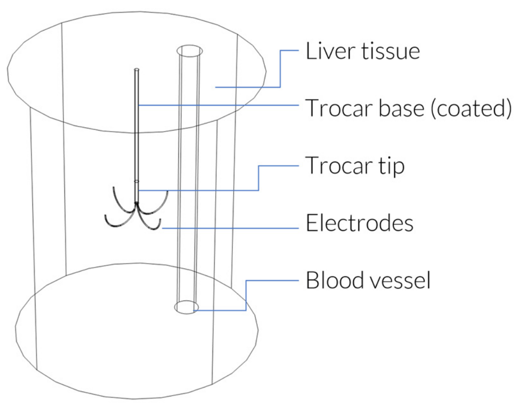

Figure 1: Cylindrical modeling domain with the four-armed electric probe in the middle, which is located next to a large blood vessel.

Model Definition

This tutorial uses the Bioheat Transfer interface, the Electric Currents interface and a multiphysics feature, Electromagnetic Heat Source, to implement a transient analysis.

The standard temperature unit in COMSOL Multiphysics is kelvin (K). This tutorial 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 centerline such that its electrodes span the region where the tumor is located. The geometry also includes a large blood vessel.

HEAT TRANSFER

The bioheat 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 (J/(kg·K)); and k is its thermal conductivity (W/(m·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 (J/(kg·K)); ω_b is its perfusion rate (1/s); T_b is the arterial blood temperature (K); while Q_{met} and Q_{ext} are the heat sources from metabolism and spatial heating, respectively (W/m^3).

In this example, 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.

In addition to the heat transfer equation this model provides a calculation of the tissue damage integral. This gives an idea about the degree of tissue injury α during the process, based on the Arrhenius equation:

$$\frac{d\alpha}{dt} = A\exp\left(-\frac{\Delta E}{RT}\right)$$

where A is the frequency factor (s^{-1}) and ΔE is the activation energy for irreversible damage reaction (J/mol). These two parameters are dependent on the type of tissue. The fraction of necrotic tissue, θ_d, is then expressed by:

$$\theta_d = 1 - \exp(-\alpha)$$

ELECTRIC CURRENT

The governing equation for the Electric Currents interface is

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

where V is the potential (V), σ the electrical conductivity (S/m), $\mathbf{J}^e$ an externally generated current density (A/m^2), Q_j the current source (A/m^3).

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 22 V. Assume continuity for all other boundaries.

The boundary conditions for the Electric Currents interface are:

$V = 0$	on the cylinder wall
$V = V_0$	on the electrode surfaces
$\mathbf{n} \cdot (J_1 - J_2) = 0$	on all other boundaries

The boundary conditions for the bioheat equation are:

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

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

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

The slice plot in Figure 2 illustrates the temperature field at the end of the procedure.

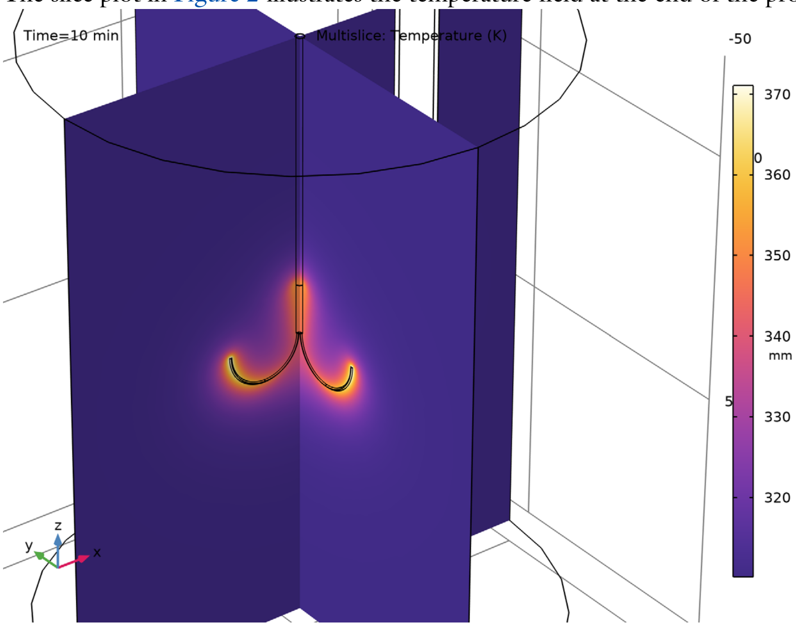

Figure 2: Temperature field at time 10 minutes.

Figure 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 97°C.

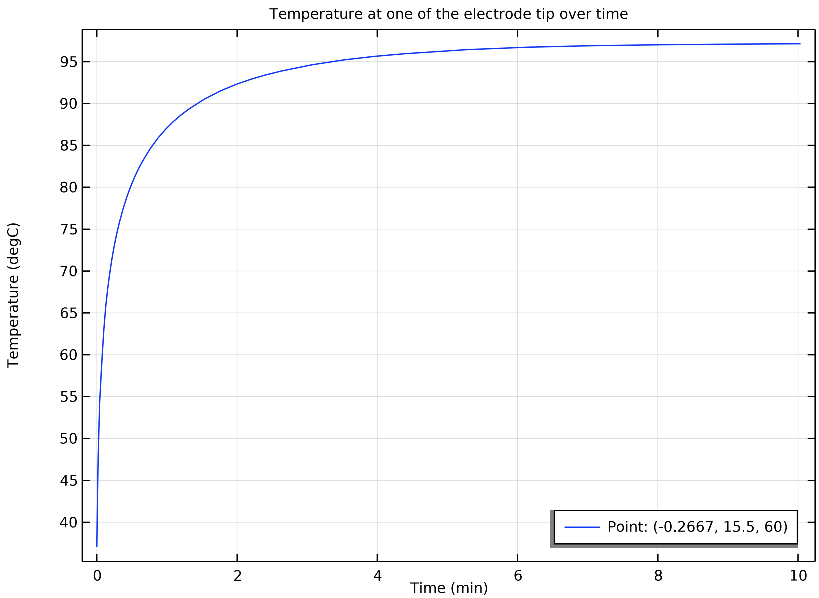

Figure 3: Temperature versus time at the tip of one of the electrode arms.

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 shows one after 10 minutes.

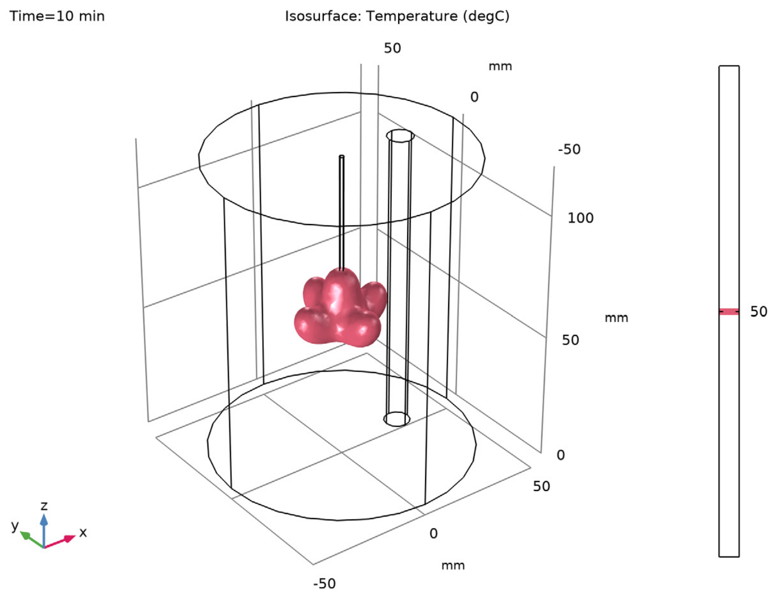

Figure 4: Visualization of the region that has reached 50°C after 10 minutes.

In addition to the previous figure, you can visualize the fraction of necrotic tissue in the slice plot of Figure 5.

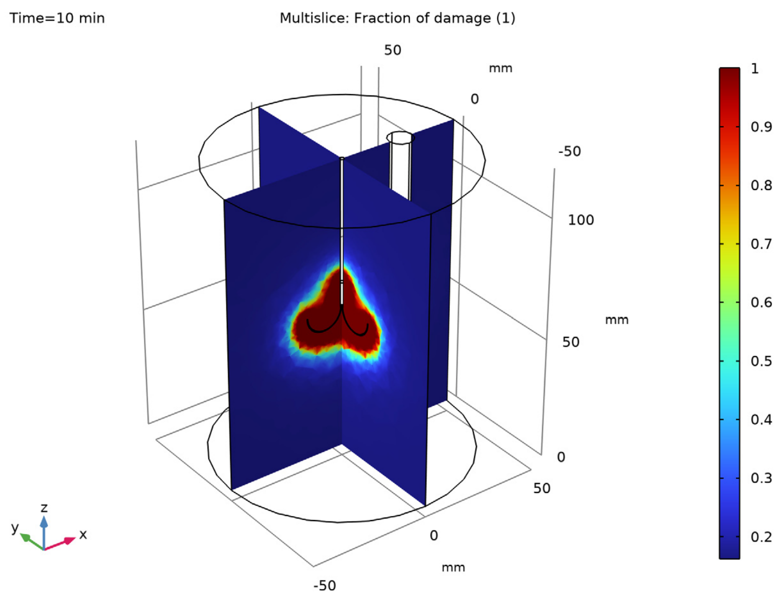

Figure 5: Fraction of necrotic tissue.

Finally, Figure 6 shows the fraction of necrotic tissue at three different points above the electrode arm. Observe that necrosis happens faster next to the electrode and the trocar tip.

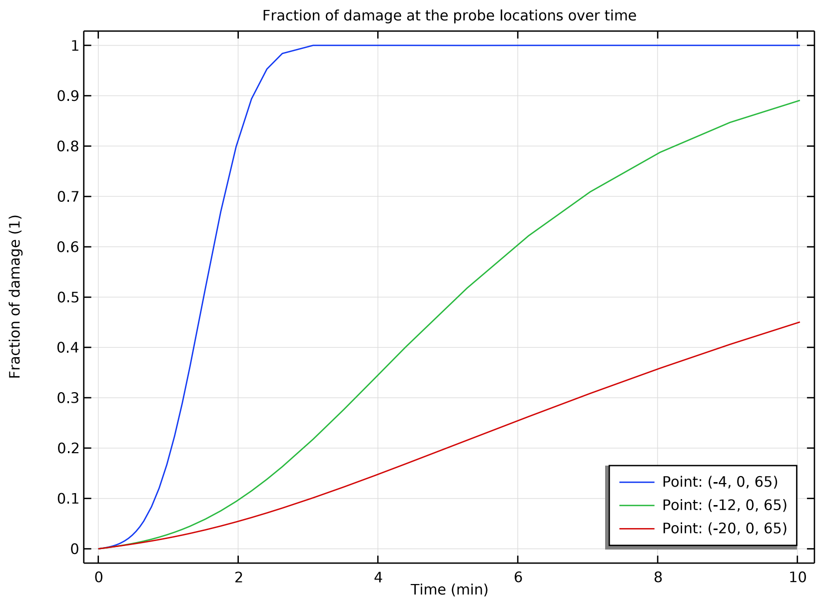

Figure 6: Fraction of necrotic tissue at three points above the electrode arm.

Reference

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

Application Library path: `Heat_Transfer_Module/Medical_Technology/tumor_ablation`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electric Currents (ec)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

GLOBAL DEFINITIONS

First, define the global parameters of the model and the geometry.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
rho_b	1000[kg/m^3]	1000 kg/m³	Density, blood
c_b	4180[J/(kg*K)]	4180 J/(kg·K)	Heat capacity, blood
omega_b	6.4e-3[1/s]	0.0064 1/s	Blood perfusion rate
T_b	37[degC]	310.15 K	Arterial blood temperature
T0	37[degC]	310.15 K	Initial and boundary temperature
V0	22[V]	22 V	Electric voltage
xc_v	26[mm]	0.026 m	Vessel cylinder center x-coordinate
a_time	10[min]	600 s	Ablation time

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type 0.9144.

4 In the **Height** text field, type 60.

5 Locate the **Position** section. In the **z** text field, type 60.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	10

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

9 Click **Build Selected**.

Torus 1 (tor1)

1 In the **Geometry** toolbar, click **Torus**.

2 In the **Settings** window for **Torus**, locate the **Size and Shape** section.

3 In the **Major radius** text field, type 7.5.

4 In the **Minor radius** text field, type 0.2667.

5 In the **Revolution angle** text field, type 180.

6 Locate the **Position** section. In the **x** text field, type 8.

7 In the **z** text field, type 60.

8 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

9 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.

10 Click **Build Selected**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **tor1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 Click **Range**.

5 In the **Range** dialog box, type 0 in the **Start** text field.

6 In the **Step** text field, type `90`.

7 In the **Stop** text field, type `270`.

8 Click **Replace**.

9 In the **Settings** window for **Rotate**, click **Build Selected**.

10 Click the **Zoom Extents** button in the **Graphics** toolbar.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `5`.

4 In the **Height** text field, type `120`.

5 Locate the **Position** section. In the **x** text field, type `xc_v`.

6 Click **Build Selected**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Cylinder 3 (cyl3)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `50`.

4 In the **Height** text field, type `120`.

5 In the **Geometry** toolbar, click **Build All**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

DEFINITIONS

Exterior Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** check box.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

Liver Tissue

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Liver Tissue` in the **Label** text field.

3 Select Domain 1 only.

Blood Vessel

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Blood Vessel` in the **Label** text field.
3 Select Domain 8 only.

Electrodes

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Electrodes` in the **Label** text field.
3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.
4 Select Domains 2 and 5–7 only.

Trocar Tip

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Trocar Tip` in the **Label** text field.
3 Select Domain 3 only.

Trocar Base

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Trocar Base` in the **Label** text field.
3 Select Domain 4 only.

Trocar

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Trocar` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
4 In the **Add** dialog box, in the **Selections to add** list, choose **Electrodes**, **Trocar Tip**, and **Trocar Base**.
5 Click **OK**.

Tissue and Trocar

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Tissue and Trocar` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
4 In the **Add** dialog box, in the **Selections to add** list, choose **Liver Tissue** and **Trocar**.
5 Click **OK**.

Tissue and Trocar, Exterior Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.
2 In the **Settings** window for **Adjacent**, type `Tissue and Trocar, Exterior Boundaries` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Input selections**, click + **Add**.
4 In the **Add** dialog box, select **Tissue and Trocar** in the **Input selections** list.
5 Click **OK**.

Trocar Tip and Electrodes, Exterior Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.
2 In the **Settings** window for **Adjacent**, type `Trocar Tip and Electrodes, Exterior Boundaries` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Input selections**, click + **Add**.
4 In the **Add** dialog box, in the **Input selections** list, choose **Electrodes** and **Trocar Tip**.
5 Click **OK**.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Bioheat>Liver (human)**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Liver (human) (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2 From the **Selection** list, choose **Liver Tissue**.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`0.333[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Blood

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Blood` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Blood Vessel**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`0.667[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.543[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`rho_b`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`c_b`	J/(kg·K)	Basic

Electrodes

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Electrodes` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Electrodes**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1e8[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`1`	1	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`18[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`6450[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`840[J/(kg*K)]`	J/(kg·K)	Basic

Trocar Tip

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Trocar Tip` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocar Tip**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`4e6[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`71[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`21500[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`132[J/(kg*K)]`	J/(kg·K)	Basic

Trocar Base

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Trocar Base` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocar Base**.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1e-5[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.026[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`70[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`1045[J/(kg*K)]`	J/(kg·K)	Basic

ELECTRIC CURRENTS (EC)

At the time scale of the tumor ablation process, the electric field is stationary. Change the equation form accordingly.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

2 In the **Settings** window for **Electric Currents**, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Stationary**.

To reduce the size of the computation problem, select a lower element order.

4 Click to expand the **Discretization** section. From the **Electric potential** list, choose **Linear**.

Ground 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Ground**.

2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior Boundaries**.

Electric Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.

2 In the **Settings** window for **Electric Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Trocar Tip and Electrodes, Exterior Boundaries**.

4 Locate the **Electric Potential** section. In the V_0 text field, type `V0`.

BIOHEAT TRANSFER (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Bioheat Transfer (ht)**.

2 In the **Settings** window for **Bioheat Transfer**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Tissue and Trocar**.

Biological Tissue 1

In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Biological Tissue 1**.

Thermal Damage 1

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Damage**.

2 In the **Settings** window for **Thermal Damage**, locate the **Damaged Tissue** section.

3 From the **Transformation model** list, choose **Arrhenius kinetics**.

Bioheat 1

1 In the **Model Builder** window, click **Bioheat 1**.

2 In the **Settings** window for **Bioheat**, locate the **Bioheat** section.

3 In the T_b text field, type `T_b`.

4 In the $C_{p,b}$ text field, type `c_b`.

5 In the ω_b text field, type `omega_b`.

6 In the ρ_b text field, type `rho_b`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T0`.

Solid 1

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Trocar**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 In the **Settings** window for **Temperature**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Tissue and Trocar, Exterior Boundaries**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_b`.

MULTIPHYSICS

Electromagnetic Heating 1 (emh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Electromagnetic Heating**.

MESH 1

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 5–7 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.38`.

8 Select the **Minimum element size** check box. In the associated text field, type `0.35`.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 3 and 4 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `1.3`.

8 Select the **Minimum element size** check box. In the associated text field, type `1.1`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Resolution of narrow regions** text field, type `0`.

5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **min**.

4 In the **Output times** text field, type `range(0,a_time/4,a_time)`.

Add probes to save the fraction of necrotic tissue and the temperature over time at some specified points.

DEFINITIONS

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-4`.

4 In row **Coordinates**, set **z** to `65`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Bioheat Transfer>Irreversible transformation>ht.theta_d - Fraction of damage - 1**.

3 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Domain Point Probe 1

In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Duplicate**.

Domain Point Probe 2

1 In the **Model Builder** window, click **Domain Point Probe 2**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-12`.

Domain Point Probe 1

In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Duplicate**.

Domain Point Probe 3

1 In the **Model Builder** window, click **Domain Point Probe 3**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **x** to `-20`.

Domain Point Probe 4

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.
2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.
3 In row **Coordinates**, set **x** to `-0.2667`.
4 In row **Coordinates**, set **y** to `15.5`.
5 In row **Coordinates**, set **z** to `60`.

Point Probe Expression 4 (ppb4)

1 In the **Model Builder** window, expand the **Domain Point Probe 4** node, then click **Point Probe Expression 4 (ppb4)**.
2 In the **Settings** window for **Point Probe Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Bioheat Transfer>Temperature>T - Temperature - K**.
3 Locate the **Table and Window Settings** section. Click **Add Plot Window**.
4 Locate the **Expression** section. From the **Table and plot unit** list, choose **degC**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Damaged Tissue, 1D

The first two default plots show the change of fraction and damage in probe locations over time.

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.
2 In the **Settings** window for **1D Plot Group**, type `Damaged Tissue, 1D` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Fraction of damage at the probe locations over time`.
5 Locate the **Plot Settings** section.
6 Select the **y-axis label** check box. In the associated text field, type `Fraction of damage (1)`.
7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Damaged Tissue, 1D** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Point: (-4, 0, 65)`
`Point: (-12, 0, 65)`
`Point: (-20, 0, 65)`

5 In the **Damaged Tissue, 1D** toolbar, click **Plot**.

Generate plots to show the fraction of necrotic tissue.

Temperature at One Electrode Tip

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, type `Temperature at One Electrode Tip` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Temperature (degC)`.

5 Locate the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Temperature at one of the electrode tip over time`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Temperature at One Electrode Tip** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Point: (-0.2667, 15.5, 60)`

5 In the **Temperature at One Electrode Tip** toolbar, click **Plot**.

Electric Potential (ec)

The third default plot shows the electric potential in the volume. Modify it to display the results on the appropriate slices.

Volume 1

1 In the **Model Builder** window, expand the **Electric Potential (ec)** node.

2 Right-click **Volume 1** and choose **Delete**.

Electric Potential (ec)

In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

Multislice 1

1 In the **Electric Potential (ec)** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **z-planes** subsection. In the **Planes** text field, type 0.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Dipole** in the tree.

6 Click **OK**.

7 In the **Electric Potential (ec)** toolbar, click **Plot**.

Electric Field Norm (ec)

The fourth default plot shows the current density norm on slices. You can remove one of the slices for better visualization.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field Norm (ec)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **z-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type 0.

Streamline Multislice 1

1 In the **Model Builder** window, click **Streamline Multislice 1**.

2 In the **Settings** window for **Streamline Multislice**, locate the **Multiplane Data** section.

3 Find the **z-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type 0.

5 In the **Electric Field Norm (ec)** toolbar, click **Plot**.

Temperature (ht)

The last default plot shows the volume distribution of temperature at the final time.

To reproduce the two-slice plot of the temperature at 10 minutes shown in Figure 2, use one of the predefined plots.

Before adding the multislice plot, delete the default **Temperature** node.

1 In the **Model Builder** window, expand the **Temperature (ht)** node.

2 Right-click **Temperature (ht)** and choose **Delete**. Click **Yes** to confirm.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Bioheat Transfer>Temperature, Multislice (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Temperature, Multislice (ht)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type 1.

4 Find the **y-planes** subsection. In the **Planes** text field, type 1.

5 Find the **z-planes** subsection. In the **Planes** text field, type 0.

6 In the **Temperature, Multislice (ht)** toolbar, click **Plot**.

7 Click the **Zoom In** button in the **Graphics** toolbar.

Use another predefined plot to see isothermal contours for the final time.

To reproduce Figure 4, follow the next steps:

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Bioheat Transfer>Isothermal Contours (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Isosurface 1

1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Isosurface 1**.

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

3 From the **Unit** list, choose **degC**.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type 50.

Isothermal Contours (ht)

1 In the **Model Builder** window, click **Isothermal Contours (ht)**.

2 In the **Isothermal Contours (ht)** toolbar, click **Plot**.

Add another predefined plots to see two-slice distribution of damaged tissue.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Bioheat Transfer>Damaged Tissue, Multislice (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Damaged Tissue, Multislice (ht)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. In the **Planes** text field, type 1.

4 Find the **y-planes** subsection. In the **Planes** text field, type 1.

5 Find the **z-planes** subsection. In the **Planes** text field, type 0.

6 In the **Damaged Tissue, Multislice (ht)** toolbar, click **Plot**.