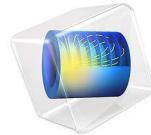


Created in COMSOL Multiphysics 5.6



# Hepatic Tumor Ablation

This model is licensed under the [COMSOL Software License Agreement 5.6](#).  
All trademarks are the property of their respective owners. See [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Material used under license agreement COMSOL Software License Agreement 5.6. Parts of the original instructions and result figures are omitted for educational purposes.

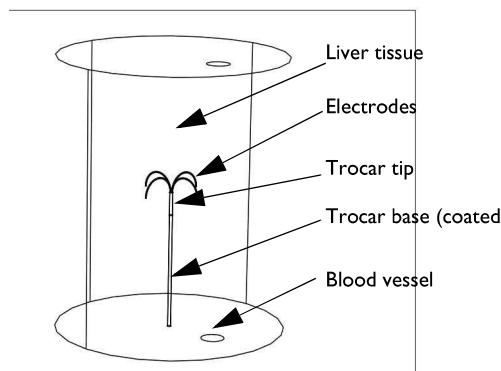
## *Introduction*

---

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. Tungjitskusolmun 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  $\delta_{ts}$  is a time-scaling coefficient;  $\rho$  is the tissue density ( $\text{kg}/\text{m}^3$ );  $C$  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,  $\rho_b$  gives the blood's density ( $\text{kg}/\text{m}^3$ );  $C_b$  is the blood's specific heat ( $\text{J}/(\text{kg}\cdot\text{K})$ );  $\omega_b$  is its perfusion rate ( $1/\text{s}$ );  $T_b$  is the arterial blood temperature (K); while  $Q_{met}$  and  $Q_{ext}$  are the heat sources from metabolism and spatial heating, respectively ( $\text{W}/\text{m}^3$ ).

In this example, the bioheat equation also models heat transfer in various parts of the probe with the appropriate values for the specific heat,  $C$  ( $\text{J}/(\text{kg}\cdot\text{K})$ ), and thermal conductivity,  $k$  ( $\text{W}/(\text{m}\cdot\text{K})$ ). 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  $\alpha$  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 ( $\text{s}^{-1}$ ) and  $\Delta E$  is the activation energy for irreversible damage reaction ( $\text{J/mol}$ ). These two parameters are dependent on the type of tissue. The fraction of necrotic tissue,  $\theta_d$ , 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),  $\sigma$  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:

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

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 of time.

### Reference

---

I.S. Tungjitusolmun, 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 I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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/m <sup>3</sup>	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 I**

- 1 In the **Model Builder** window, under **Component I (compI)** click **Geometry I**.
- 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.

#### *Trocars*

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

#### *Trocars*

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

#### *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 **Electrodes**, **Trocars**, and **Trocars**.
- 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 ; sigmaji = sigma_iso, sigmaji = 0	0.333[S/m]	S/m	Basic
Relative permittivity	epsilon_nr_iso ; epsilon_nrii = epsilon_nr_iso, epsilon_nrij = 0	1	I	Basic

#### Blood

- In the **Materials** toolbar, click  **Blank Material**.
- In the **Settings** window for **Material**, type **Blood** in the **Label** text field.
- Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Blood Vessel**.
- Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaji = sigma_iso, sigmaji = 0	0.667[S/m]	S/m	Basic
Relative permittivity	epsilon_nr_is_o ; epsilon_nrii = epsilon_nr_is_o, epsilon_nrij = 0	1	I	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.543[W/(m·K)]	W/(m·K)	Basic
Density	rho	rho_b	kg/m <sup>3</sup>	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	<code>sigma_iso ; sigmaji = sigma_iso, sigmaji = 0</code>	<code>1e8 [S/m]</code>	S/m	Basic
Relative permittivity	<code>epsilon_nr_is o ; epsilon_nrii = epsilon_nr_is o, epsilon_nr_ij = 0</code>	1	I	Basic
Thermal conductivity	<code>k_iso ; kii = k_iso, kij = 0</code>	<code>18 [W / (m·K)]</code>	W/(m·K)	Basic
Density	<code>rho</code>	<code>6450 [kg / m^3]</code>	kg/m <sup>3</sup>	Basic
Heat capacity at constant pressure	<code>Cp</code>	<code>840 [J / (kg·K)]</code>	J/(kg·K)	Basic

### *Trocars Tip*

- 1 In the **Materials** toolbar, click  **Blank Material**.
- 2 In the **Settings** window for **Material**, type **Trocars Tip** in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocars Tip**.

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

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaji = sigma_iso, sigmaji = 0	4e6[S/m]	S/m	Basic
Relative permittivity	epsilon_is_o ; epsilon_rii = epsilon_is_o, epsilon_rij = 0	1	I	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

- In the **Materials** toolbar, click  **Blank Material**.
- In the **Settings** window for **Material**, type **Trocar Base** in the **Label** text field.
- Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Trocar Base**.
- Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaji = sigma_iso, sigmaji = 0	1e-5[S/m]	S/m	Basic
Relative permittivity	epsilon_is_o ; epsilon_rii = epsilon_is_o, epsilon_rij = 0	1	I	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 **Trocars Tip and Electrodes, Exteriors Boundaries**.
- 4 Locate the **Electric Potential** section. In the  $V_0$  text field, type  $V_0$ .

### 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 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 In the **Model Builder** window, click **Bioheat I**.
- 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  $\omega_b$  text field, type  $\omega_b$ .
- 6 In the  $\rho_b$  text field, type  $\rho_b$ .

#### *Initial Values* /

- 1 In the **Model Builder** window, click **Initial Values I**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 In the  $T$  text field, type  $T_0$ .

#### *Solid* /

- 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 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* / (emhI)

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

## MESH I

### Free Tetrahedral I

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

#### Size 1

- 1 Right-click **Free Tetrahedral I** 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. Select the **Maximum element size** check box.
- 7 In the associated text field, type 0.38.
- 8 Select the **Minimum element size** check box.
- 9 In the associated text field, type 0.35.

#### Size 2

- 1 In the **Model Builder** window, right-click **Free Tetrahedral I** 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. Select the **Maximum element size** check box.
- 7 In the associated text field, type 1.3.
- 8 Select the **Minimum element size** check box.
- 9 In the associated text field, type 1.1.

#### Size

- 1 In the **Model Builder** window, 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**.
- 3 Click to expand the **Table and Window Settings** section. Click  **Add Plot Window**.

### Domain Point Probe 2

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Domain Point Probe 1** and choose **Duplicate**.
- 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 3

- 1 Right-click **Domain Point Probe 1** and choose **Duplicate**.
- 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 (compl)> 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**

##### *Electric Potential (ec)*

The first default plot shows the electric potential on slices.

##### *Temperature (ht)*

The second default plot shows the temperature at the final time.

To reproduce the two-slice plot of the temperature at 10 minutes shown in [Figure 2](#), proceed as follows.

Before adding slices, delete the default **Surface** node.

##### *Surface*

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node.
- 2 Right-click **Surface** and choose **Delete**. Click **Yes** to confirm.

##### *Temperature (ht)*

In the **Model Builder** window, click **Temperature (ht)**.

##### *Slice 1*

- 1 In the **Temperature (ht)** toolbar, click  **Slice**.

- 2 In the **Settings** window for **Slice**, 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 **Expression** section. From the **Unit** list, choose **degC**.
- 4 Locate the **Plane Data** section. In the **Planes** text field, type 1.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

*Temperature (ht)*

In the **Model Builder** window, click **Temperature (ht)**.

*Slice 2*

- 1 In the **Temperature (ht)** toolbar, click  **Slice**.
- 2 In the **Settings** window for **Slice**, 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 **Expression** section. From the **Unit** list, choose **degC**.
- 4 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
- 5 In the **Planes** text field, type 1.
- 6 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.
- 7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
- 8 In the **Temperature (ht)** toolbar, click  **Plot**.
- 9 Click the  **Zoom In** button in the **Graphics** toolbar.

*Isothermal Contours (ht)*

The third default plot shows isothermal contours for the final time.

To reproduce [Figure 4](#), modify this plot group as follows:

*Isosurface*

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

*Damaged Tissue, 1D*

- 1 In the **Model Builder** window, under **Results** click **Probe Plot Group 4**.
- 2 In the **Settings** window for **ID Plot Group**, type **Damaged Tissue, 1D** in the **Label** text field.
- 3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.
- 4 In the **Damaged Tissue, ID** 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 5**.
- 2 In the **Settings** window for **ID Plot Group**, type **Temperature at One Electrode Tip** in the **Label** text field.
- 3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.
- 4 In the **Temperature at One Electrode Tip** toolbar, click  **Plot**.

*Damaged Tissue, 3D*

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type **Damaged Tissue, 3D** in the **Label** text field.

*Slice 1*

- 1 In the **Damaged Tissue, 3D** toolbar, click  **Slice**.
- 2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (compl)>Bioheat Transfer>Irreversible transformation>ht.theta\_d - Fraction of damage**.
- 3 Locate the **Plane Data** section. In the **Planes** text field, type **1**.

*Slice 2*

- 1 Right-click **Slice 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
- 3 From the **Plane** list, choose **ZX-planes**.
- 4 In the **Planes** text field, type **1**.
- 5 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
- 6 In the **Damaged Tissue, 3D** toolbar, click  **Plot**.