Created in COMSOL Multiphysics 6.1

# Microwave Heating of a Cancer Tumor

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

## *Introduction*

Electromagnetic heating appears in a wide range of engineering problems and is ideally suited for modeling in COMSOL Multiphysics because of its multiphysics capabilities. This example comes from the area of hyperthermic oncology and it models the electromagnetic field coupled to the bioheat equation. The modeling issues and techniques are generally applicable to any problem involving electromagnetic heating.

In hyperthermic oncology, cancer is treated by applying localized heating to the tumor tissue, often in combination with chemotherapy or radiotherapy. Some of the challenges associated with the selective heating of deep-seated tumors without damaging surrounding tissue are:

- Control of heating power and spatial distribution
- Design and placement of temperature sensors

Among possible heating techniques, RF and microwave heating have attracted much attention from clinical researchers. Microwave coagulation therapy is one such technique where a thin microwave antenna is inserted into the tumor. The microwaves heat up the tumor, producing a coagulated region where the cancer cells are killed.

This model computes the temperature field, the radiation field, and the specific absorption rate (SAR) — defined as the ratio of absorbed heat power and tissue density — in liver tissue when using a thin coaxial slot antenna for microwave coagulation therapy. It closely follows the analysis found in Ref. 1. It computes the temperature distribution in the tissue using the bioheat equation.

---

**Note:** This application requires the RF Module and the Heat Transfer Module.

---

## *Model Definition*

Figure 1 shows the antenna geometry. It consists of a thin coaxial cable with a ring-shaped slot measuring 1 mm cut on the outer conductor 5 mm from the short-circuited tip. For hygienic purposes, the antenna is enclosed in a sleeve (catheter) made of PTFE (polytetrafluoroethylene). The following tables give the relevant geometrical dimensions and material data. The antenna operates at 2.45 GHz, a frequency widely used in microwave coagulation therapy.

TABLE 1: DIMENSIONS OF THE COAXIAL SLOT ANTENNA.

| PROPERTY | VALUE |
|---|---|
| Diameter of the central conductor | 0.29 mm |
| Inner diameter of the outer conductor | 0.94 mm |
| Outer diameter of the outer conductor | 1.19 mm |
| Diameter of catheter | 1.79 mm |

TABLE 2: MATERIAL PROPERTIES.

| PROPERTY | INNER DIELECTRIC OF COAXIAL CABLE | CATHETER | LIVER TISSUE |
|---|---|---|---|
| Relative permittivity | 2.03 | 2.60 | 43.03 |
| Conductivity | | | 1.69 S/m |

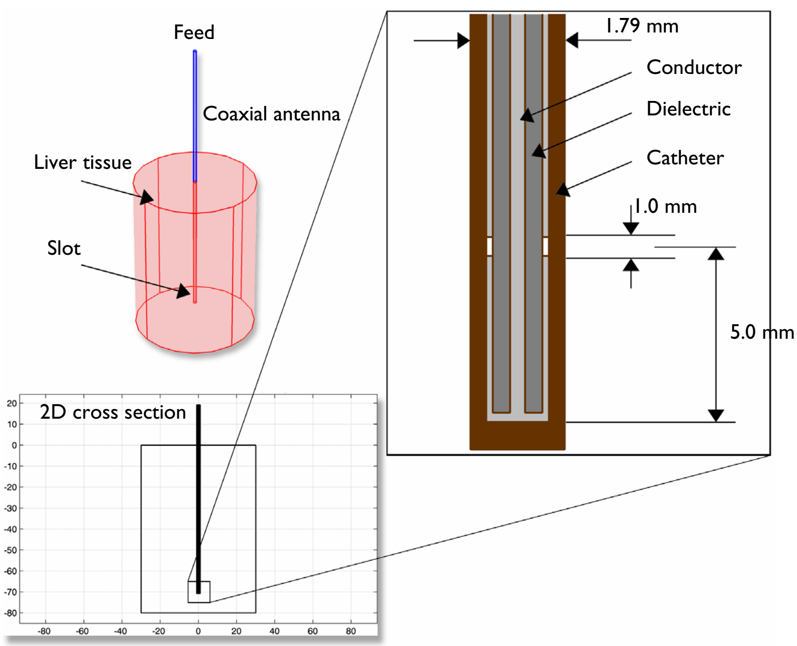



*Figure 1: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.*

The model takes advantage of the problem's rotational symmetry, which allows modeling in 2D using cylindrical coordinates as indicated in Figure 2. When modeling in 2D, you can select a fine mesh and achieve excellent accuracy. The model uses a frequency-domain problem formulation with the complex-valued azimuthal component of the magnetic field as the unknown.

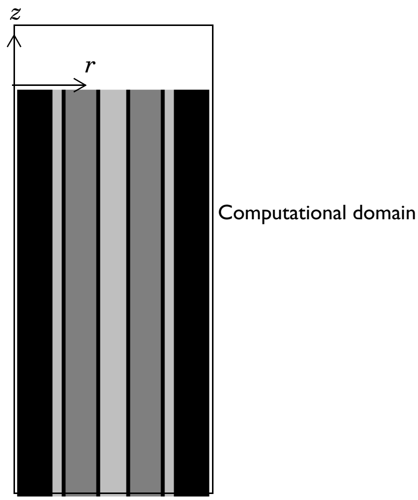



*Figure 2: The computational domain appears as a rectangle in the rz-plane.*

The radial and axial extent of the computational domain is in reality larger than indicated in Figure 2. This problem does not model the interior of the metallic conductors, and it models metallic parts using boundary conditions, setting the tangential component of the electric field to zero.

#### DOMAIN AND BOUNDARY EQUATIONS — ELECTROMAGNETICS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, the appropriate equations are

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\varphi \frac{C}{rZ} e^{j(\omega t - kz)}$$

$$\mathbf{P}_{\mathrm{av}} = \int_{r_{\mathrm{inner}}}^{r_{\mathrm{outer}}} \mathrm{Re}\left(\frac{1}{2}\mathbf{E} \times \mathbf{H}^*\right) 2\pi r dr = \mathbf{e}_z \pi \frac{C^2}{Z} \ln\left(\frac{r_{\mathrm{outer}}}{r_{\mathrm{inner}}}\right)$$

where $z$ is the direction of propagation, and $r$, $\varphi$, and $z$ are cylindrical coordinates centered on the axis of the coaxial cable. $\mathbf{P}_{\mathrm{av}}$ is the time-averaged power flow in the cable, $Z$ is the wave impedance in the dielectric of the cable, while $r_{\mathrm{inner}}$ and $r_{\mathrm{outer}}$ are the dielectric's inner and outer radii, respectively. Further, $\omega$ denotes the angular frequency. The propagation constant, $k$, relates to the wavelength in the medium, $\lambda$, as

$$k = \frac{2\pi}{\lambda}$$

In the tissue, the electric field also has a finite axial component whereas the magnetic field is purely in the azimuthal direction. Thus, you can model the antenna using an axisymmetric transverse magnetic (TM) formulation. The wave equation then becomes scalar in $H_\varphi$:

$$\nabla \times \left( \left( \varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0} \right)^{-1} \nabla \times H_\varphi \right) - \mu_r k_0^{\;2} H_\varphi = 0$$

The boundary conditions for the metallic surfaces are

$$\mathbf{n} \times \mathbf{E} = 0$$

The feed point is modeled using a port boundary condition with a power level set to 10 W. This is essentially a first-order low-reflecting boundary condition with an input field $H_{\varphi 0}$:

$$\mathbf{n} \times \sqrt{\varepsilon}\mathbf{E} - \sqrt{\mu} H_\varphi = -2\sqrt{\mu} H_{\varphi 0}$$

where

$$H_{\varphi 0} = \frac{1}{r} \sqrt{\frac{\mathbf{P}_{\mathrm{av}} Z}{\pi r \ln\left( \frac{r_{\mathrm{outer}}}{r_{\mathrm{inner}}} \right)}}$$

for an input power of $\mathbf{P}_{\mathrm{av}}$ deduced from the time-average power flow.

The antenna radiates into the tissue where a damped wave propagates. Because you can discretize only a finite region, you must truncate the geometry some distance from the antenna using a similar absorbing boundary condition without excitation. Apply this boundary condition to all exterior boundaries.

### DOMAIN AND BOUNDARY EQUATIONS — HEAT TRANSFER

The bioheat equation describes the time-dependent heat transfer problem as

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

where $k$ is the liver's thermal conductivity (W/(m·K)), $\rho_b$ represents the blood density (kg/m$^3$), $C_b$ is the blood's specific heat capacity (J/(kg·K)), $\omega_b$ denotes the blood perfusion rate (1/s), and $T_b$ is the arterial blood temperature (K). Further, $Q_{met}$ is the heat source from metabolism, and $Q_{ext}$ is an external heat source, both measured in W/m$^3$.

The initial temperature equals $T_b$ in all domains.

This model neglects the heat source from metabolism. The external heat source is equal to the resistive heat generated by the electromagnetic field:

$$Q_{ext} = \frac{1}{2} \mathrm{Re}[(\sigma - j\omega\varepsilon)\mathbf{E} \cdot \mathbf{E}^*]$$

The model assumes that the blood perfusion rate is $\omega_b = 0.0036$ s$^{-1}$, and that the blood enters the liver at the body temperature $T_b = 37$°C and is heated to a temperature, $T$. The blood's specific heat capacity is $C_b = 3639$ J/(kg·K).

For a more realistic model, you might consider letting $\omega_b$ be a function of the temperature. At least for external body parts such as hands and feet, it is evident that a temperature increase results in an increased blood flow.

This example models the heat transfer problem only in the liver domain. Where this domain is truncated, it uses insulation, that is

$$\mathbf{n} \cdot \nabla T = 0$$

In addition to the heat transfer equation, this model computes 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 (s$^{-1}$) and $\Delta E$ 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, $\theta_d$, is then expressed by:

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

## *Results and Discussion*

Figure 3 shows the resulting steady-state temperature distribution in the liver tissue for an input microwave power of 10 W. The temperature is highest near the antenna. It then decreases with distance from the antenna and reaches 37°C closer to the outer boundaries of the computational domain. The perfusion of relatively cold blood seems to limit the extent of the area that is heated.

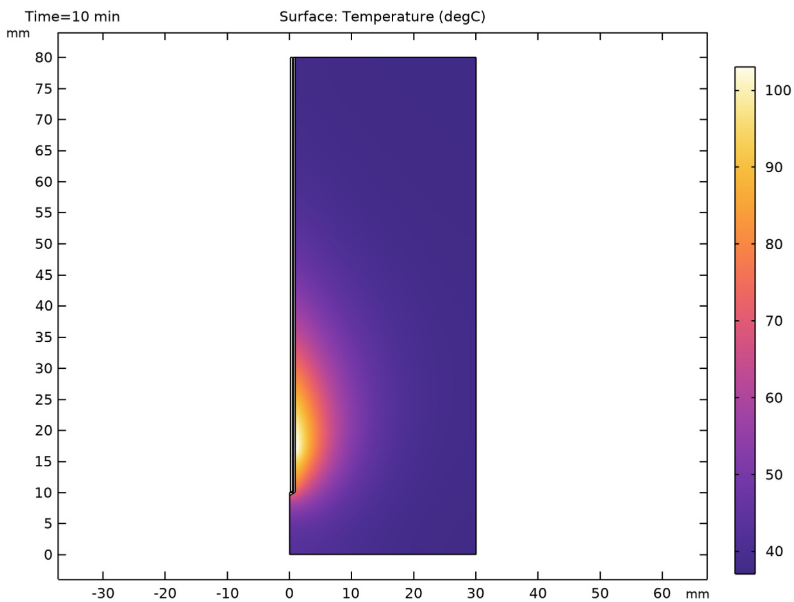



*Figure 3: Temperature in the liver tissue.*

Figure 4 shows the distribution of the microwave heat source. Clearly the temperature field follows the heat source distribution quite well. That is, near the antenna the heat source is strong, which leads to high temperatures, while far from the antenna, the heat source is weaker and the blood manages to keep the tissue at normal body temperature.

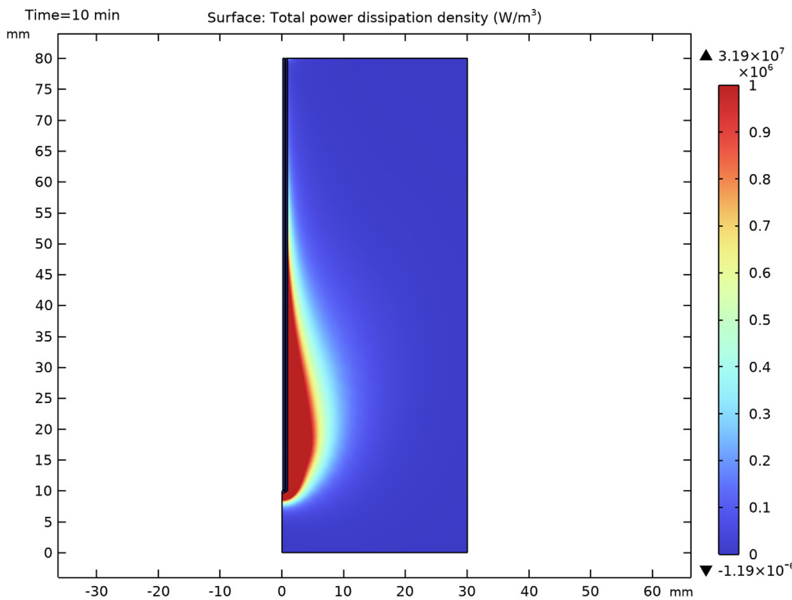



*Figure 4: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at 1 W/cm$^3$.*

Figure 5 plots the specific absorption rate (SAR) along a line parallel to the antenna and at a distance of 2.5 mm from the antenna axis. The results are in good agreement with those found in Ref. 1.

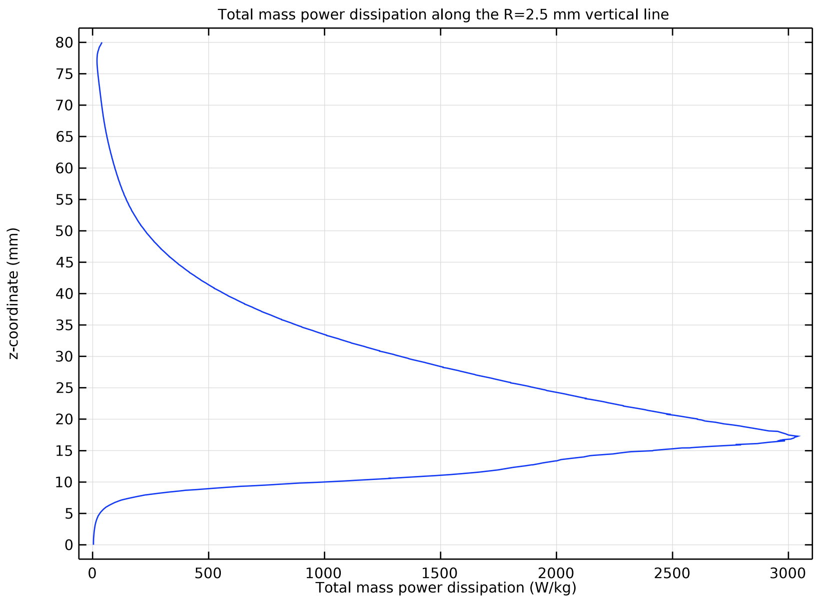



*Figure 5: SAR in W/kg along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.*

You can visualize the fraction of necrotic tissue in the surface plot of Figure 6.

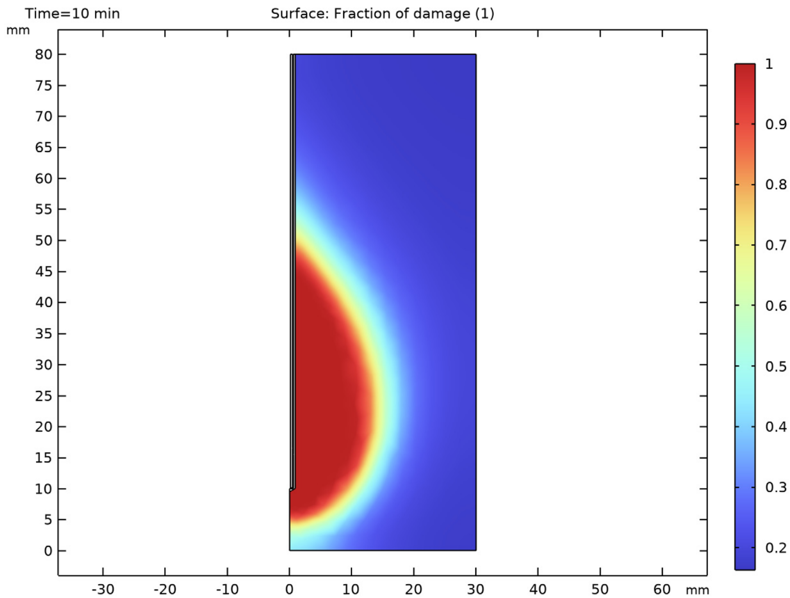



*Figure 6: Fraction of necrotic tissue.*

Figure 7 shows the fraction of necrotic tissue at four different point of the domain. Observe that necrosis happens faster near the antenna.

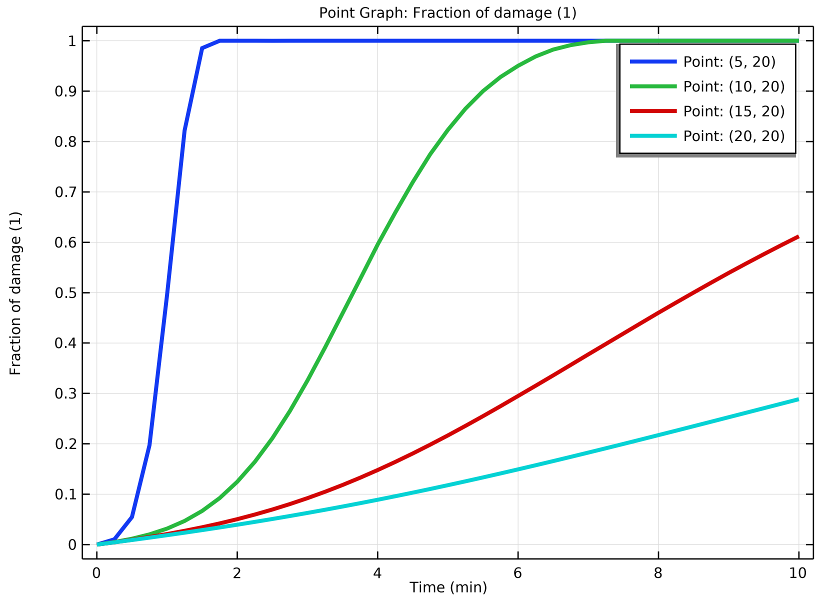



*Figure 7: Fraction of necrotic tissue at four points of the domain.*

## *Reference*


1. K. Saito, T. Taniguchi, H. Yoshimura, and K. Ito, "Estimation of SAR Distribution of a Tip-Split Array Applicator for Microwave Coagulation Therapy Using the Finite Element Method," *IEICE Trans. Electronics*, vol. E84-C, 7, pp. 948–954, July 2001.


**Application Library path:** `RF_Module/Microwave_Heating/microwave_cancer_therapy`

## *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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.
5 Click **Add**.

   Do not add the study right now, as it will be easier to define it once the multiphysics coupling has been added.
6 Click **Done**.

## MULTIPHYSICS

### *Electromagnetic Heating 1 (emh1)*

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

   This brings the heat created by the electromagnetic waves to the heat transfer simulation.

   Now add a **Frequency-Transient, One-Way Electromagnetic Heating** study sequence that first adds a **Frequency Domain** study for the electromagnetic part and then adds a **Time Dependent** study for the heat transfer part.

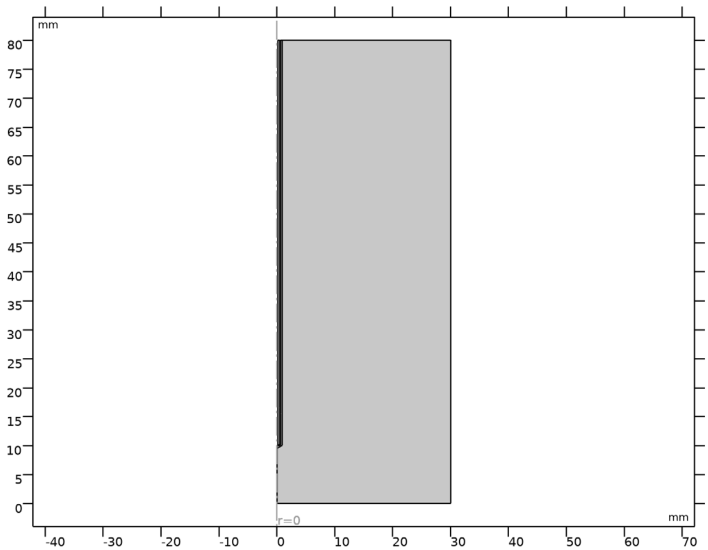
## ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.
3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Transient, One-Way Electromagnetic Heating**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

## GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch by yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1. In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `microwave_cancer_therapy_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.




You should now see the geometry shown above.

## GLOBAL DEFINITIONS

### *Parameters 1*

The relevant material properties and other model data are provided in a text file.

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `microwave_cancer_therapy_parameters.txt`.

### 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.

### MATERIALS

*Liver (human) (mat1)*

1 Select Domain 1 only.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | `eps_liver` | 1 | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | 1 | Basic |
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | `sigma_liver` | S/m | Basic |

The remaining materials take part only in the RF simulation, making any definitions of their thermal properties redundant.

*Catheter*

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

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

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

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | `eps_cat` | 1 | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | 1 | Basic |
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 0 | S/m | Basic |

*Dielectric*

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

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

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

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | `eps_diel` | 1 | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | 1 | Basic |
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 0 | S/m | Basic |

### ADD MATERIAL

**1** Go to the **Add Material** window.

**2** In the tree, select **Built-in>Air**.

**3** Click **Add to Component** in the window toolbar.

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

### *Air (mat4)*

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Air**.

## ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

### *Port 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Coaxial**.

5 In the $P_{\mathrm{in}}$ text field, type `P_in`.

### *Scattering Boundary Condition 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundaries 2, 17, 19, and 20 only.

## 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 Click **Clear Selection**.

The bioheat equation applies only in the liver tissue.

4 Select Domain 1 only.

### *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 $T_\mathrm{b}$ text field, type `T_blood`.

4 In the $\rho_\mathrm{b}$ text field, type `rho_blood`.

5 In the $C_{\mathrm{p,b}}$ text field, type `Cp_blood`.

6 In the $\omega_\mathrm{b}$ text field, type `omega_blood`.

You have now supplied all the parameters needed for the heat removal by the blood perfusion.

#### *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 `T_blood`.

### MESH 1

#### *Free Triangular 1*

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

#### *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 **Maximum element size** text field, type `3[mm]`.

#### *Size 1*

1 In the **Model Builder** window, right-click **Free Triangular 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 From the **Selection** list, choose **Dielectric**.

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 `0.15[mm]`.

**8** Click **Build All**.

The mesh is now reasonably fine everywhere, and especially fine in the coaxial cable, where the wave is created.

## STUDY 1

### *Step 1: Frequency Domain*

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f`.

### *Step 2: Time Dependent*

1 In the **Model Builder** window, click **Step 2: 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(0,15[s],10)`.

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

## RESULTS

### *Electric Field (emw)*

You have now solved the model first for the electromagnetic wave distribution, then for the temperature distribution resulting from the electromagnetic heating. Such a sequential solution is faster and consumes less memory than a fully coupled analysis, but works only if the material properties do not depend on the temperature.

### *Surface*

The default plot shows the distribution of the electric field norm. The range is dominated by the locally very high values in and in the near vicinity of the coaxial cable. One way to get a more useful picture is to plot the logarithm of the field.

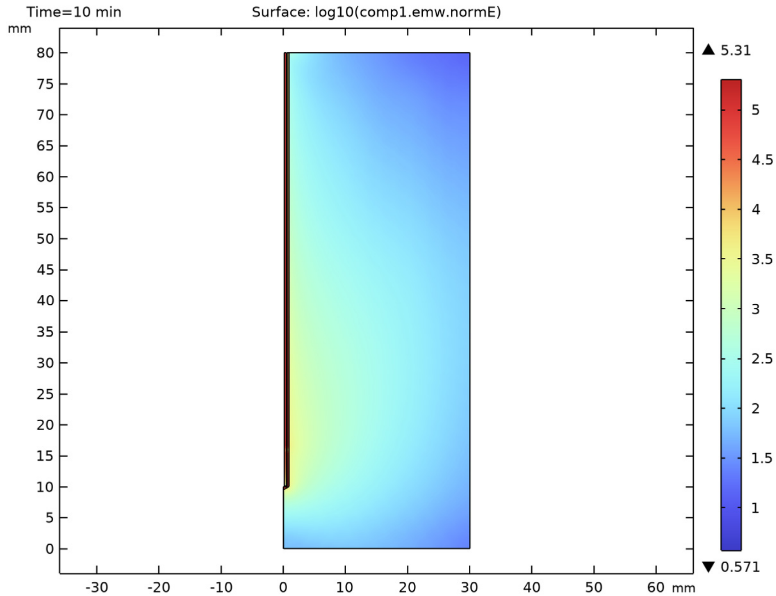
1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Surface**.

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

3 In the **Expression** text field, type `log10(comp1.emw.normE)`.

4 In the **Electric Field (emw)** toolbar, click **Plot**.

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




The local heating power density is an important result of this model. As it is proportional to the electric field squared, this entity is also going to have a very uneven distribution. Manually specifying the range is another option to keep the plot readable.

**1** Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qh - Total power dissipation density - W/m³**.

**2** In the **Electric Field (emw)** toolbar, click **Plot**.

**3** Click to expand the **Range** section. Select the **Manual color range** check box.

**4** In the **Maximum** text field, type `1e6`.

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

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

Any values greater than 1 $MW/m^3$ are now displayed as red.

If you divide the power loss density with the density of the liver tissue, you get the SAR. Try plotting this on a vertical line some distance away from the antenna. Take the liver density to be the same as that of blood.

*Total Power Dissipation Density (emw)*

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Settings** window for **2D Plot Group**, type `Total Power Dissipation Density (emw)` in the **Label** text field.

*Cut Line 2D 1*

1 In the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Line Data** section.

3 In row **Point 1**, set **R** to `2.5`, and **z** to `80`.

4 In row **Point 2**, set **R** to `2.5`.

*Qh/rho vs. Height*

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Qh/rho vs. Height` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.

4 From the **Time selection** list, choose **Last**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Total mass power dissipation along the R=2.5 mm vertical line`.

*Line Graph 1*

1 In the **Qh/rho vs. Height** toolbar, click **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `z`.

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `emw.Qh/rho_blood`.

6 Select the **Description** check box. In the associated text field, type `Total mass power dissipation`.

7 In the **Qh/rho vs. Height** toolbar, click **Plot**.

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

The plot you just created should look like Figure 5.

To evaluate the total deposited power, integrate the power loss in the liver domain.

### *Surface Integration 1*

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

2 In the **Settings** window for **Surface Integration**, locate the **Data** section.

3 From the **Time selection** list, choose **Last**.

4 Select Domain 1 only.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qh - Total power dissipation density - W/m³**.

6 Click **Evaluate**.

## TABLE

1 Go to the **Table** window.

As shown in the table, the tissue absorbs most of the 10 W input power.

## RESULTS

Create a new plot group for the surface plot of the temperature in the tissue (Figure 3).

### *Temperature, 2D*

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Temperature, 2D` in the **Label** text field.

### *Surface 1*

1 In the **Temperature, 2D** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, 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 **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

6 Click **OK**.

7 In the **Temperature, 2D** toolbar, click **Plot**.

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

*Damaged Tissue, 2D*

1. In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

   Generate a plot to show the fraction of necrotic tissue in 2D.

2. In the **Settings** window for **2D Plot Group**, type `Damaged Tissue, 2D` in the **Label** text field.

*Surface 1*

1. In the **Damaged Tissue, 2D** toolbar, click **Surface**.
2. In the **Settings** window for **Surface**, 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 **Quality** section. From the **Resolution** list, choose **No refinement**.
4. In the **Damaged Tissue, 2D** toolbar, click **Plot**.

*Cut Point 2D 1*

1. In the **Results** toolbar, click **Cut Point 2D**.
2. In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.
3. In the **R** text field, type `range(5,5,20)`.
4. In the **Z** text field, type `20`.
5. Click **Plot**.

*Temperature, 1D*

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Temperature, 1D` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.
4. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

*Point Graph 1*

1. In the **Temperature, 1D** toolbar, click **Point Graph**.
2. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Bioheat Transfer>Temperature>T - Temperature - K**.
3. Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.
4. Click to expand the **Legends** section. Select the **Show legends** check box.
5. Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Point: `.
6. In the **Temperature, 1D** toolbar, click **Plot**.

*Damaged Tissue, 1D*

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Damaged Tissue, 1D` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.

*Point Graph 1*

1 In the **Damaged Tissue, 1D** toolbar, click **Point Graph**.

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

3 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

4 Locate the **Legends** section. Select the **Show legends** check box.

5 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Point: `.

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

## *Geometry Modeling Instructions*

If you want to create the geometry by yourself, follow these steps.

### 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**.

*Rectangle 1 (r1)*

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

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

3 In the **Width** text field, type `30`.

4 In the **Height** text field, type `80`.

*Rectangle 2 (r2)*

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

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

3 In the **Width** text field, type `0.595`.

4 In the **Height** text field, type `70`.

**5** Locate the **Position** section. In the **z** text field, type `10`.

*Dielectric*

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

**2** In the **Settings** window for **Rectangle**, type `Dielectric` in the **Label** text field.

**3** Locate the **Size and Shape** section. In the **Width** text field, type `0.335`.

**4** In the **Height** text field, type `69.9`.

**5** Locate the **Position** section. In the **r** text field, type `0.135`.

**6** In the **z** text field, type `10.1`.

**7** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Rectangle 4 (r4)*

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

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

**3** In the **Width** text field, type `0.895`.

**4** In the **Height** text field, type `70`.

**5** Locate the **Position** section. In the **z** text field, type `10`.

*Air*

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

**2** In the **Settings** window for **Rectangle**, type `Air` in the **Label** text field.

**3** Locate the **Size and Shape** section. In the **Width** text field, type `0.125`.

**4** Locate the **Position** section. In the **r** text field, type `0.47`.

**5** In the **z** text field, type `15.5`.

**6** Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Polygon 1 (pol1)*

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

**2** In the **Settings** window for **Polygon**, locate the **Coordinates** section.

**3** From the **Data source** list, choose **Vectors**.

**4** In the **r** text field, type `0 0.895 0.895 0 0 0`.

**5** In the **z** text field, type `10 10 10 9.5 9.5 10`.

*Catheter*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Catheter` in the **Label** text field.

3 Select the objects **pol1** and **r4** only.

4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

*Difference 1 (dif1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **r1** and **uni1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **r2** only.