Microwave Cancer Therapy

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 model 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

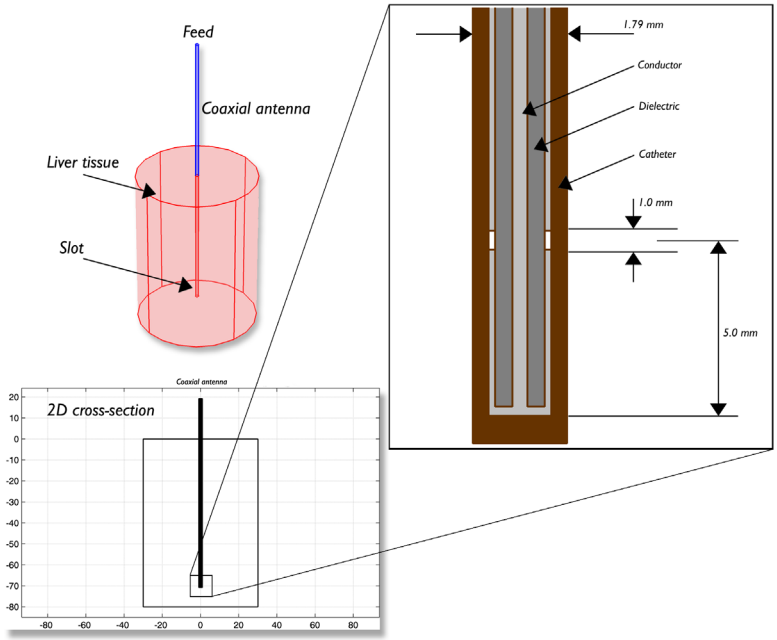


Figure 1: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.

©2013 COMSOL

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.

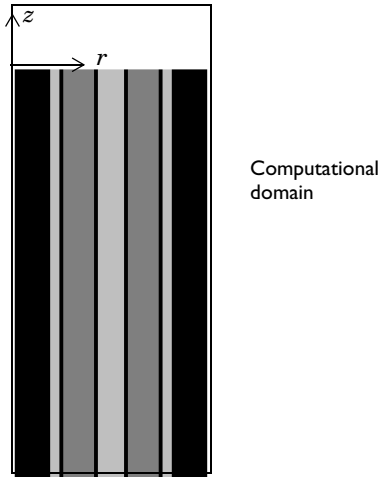


Figure 2: The computational domain appears as a rectangle in the rz-plane.

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, φ, and z are cylindrical coordinates centered on the axis of the coaxial cable. $\mathbf{P}_{\text{av}}$ is the time-averaged power flow in the cable, Z is the wave impedance in the dielectric of the cable, while r_{inner} and r_{outer} are the dielectric's inner and outer radii, respectively. Further, ω denotes the angular frequency. The propagation constant, k, relates to the wavelength in the medium, λ, 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_φ:

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

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

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

for an input power of $\mathbf{P}_{\text{av}}$ 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 stationary heat transfer problem as

 ©2013 COMSOL

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

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

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 ω_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

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

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.

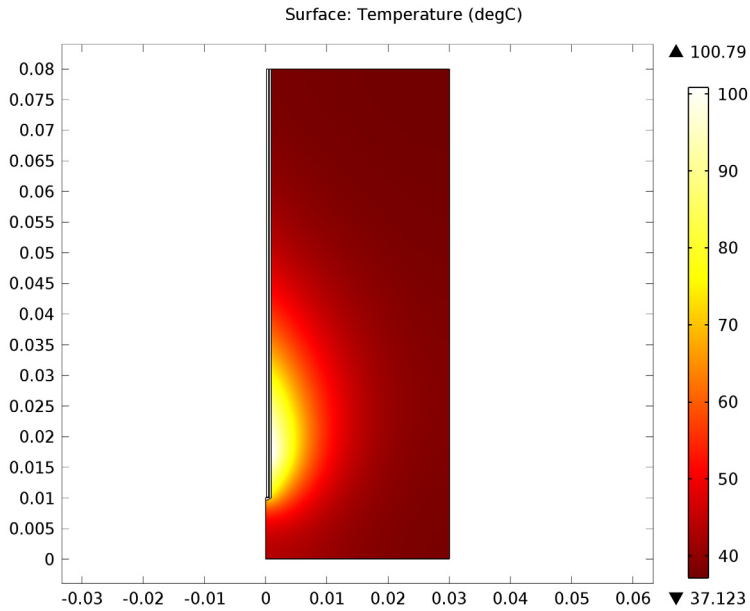


Figure 3: Temperature in the liver tissue.

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.

©2013 COMSOL

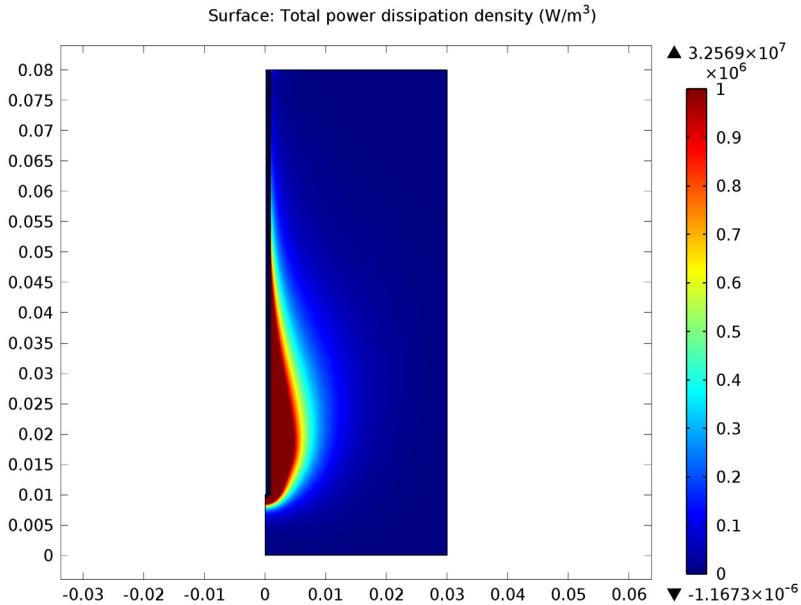


Figure 4: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at 1 W/cm^3.

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.

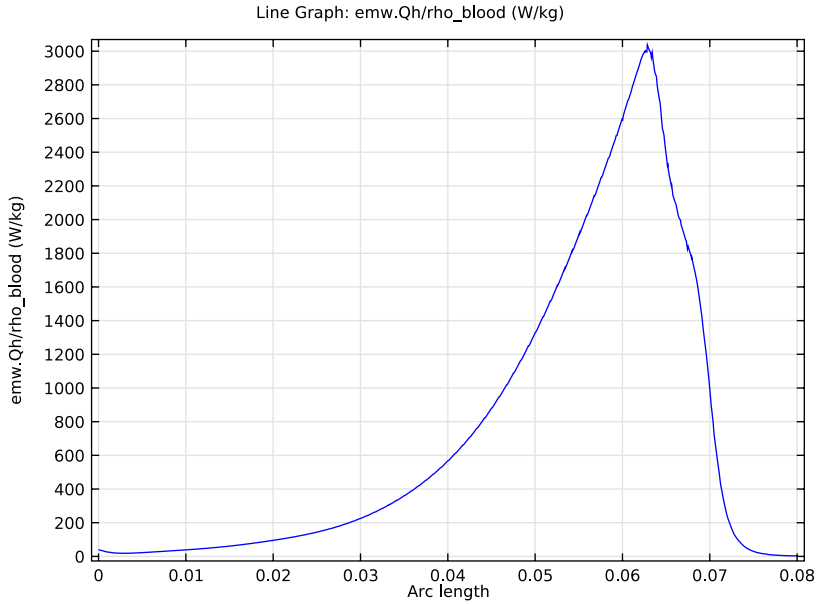


Figure 5: SAR in W/kg along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.

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.

Model Library path: `Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

 ©2013 COMSOL

2 Click the **2D axisymmetric** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

5 Click **Add Selected**.

6 In the **Add physics** tree, select **Heat Transfer>Bioheat Transfer (ht)**.

7 Click **Add Selected**.

8 Click **Next**.

9 Find the **Studies** subsection. In the tree, select **Custom Studies>Preset Studies for Some Physics>Frequency Domain**.

The frequency domain study is for the electromagnetic part of the model. You add a stationary analysis for the heat transfer part before solving.

10 Click **Finish**.

GEOMETRY 1

Import 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Import**.

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `microwave_cancer_therapy.mphbin`.

5 Click the **Import** button.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

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

2 In the **Parameters** settings window, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `microwave_cancer_therapy_parameters.txt`.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Liver Tissue` in the **New name** edit field.

4 Click **OK**.

5 In the **Material** settings window, locate the **Geometric Entity Selection** section.

6 From the **Selection** list, choose **Manual**.

7 Click **Clear Selection**.

8 Select Domain 1 only.

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

Property	Name	Value
Relative permittivity	epsilonr	`eps_liver`
Relative permeability	mur	1
Electric conductivity	sigma	`sigma_liver`
Thermal conductivity	k	`k_liver`

The heat capacity and the density do not affect the stationary solution, only the time it takes to reach it. Because this example model does not include a transient analysis, it is safe to leave these properties undefined. The remaining materials take part only in the RF simulation, making any definitions of their thermal properties redundant.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 2** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Catheter` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Material** settings window, locate the **Material Contents** section.

 ©2013 COMSOL

7 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilonr	`eps_cat`
Relative permeability	mur	1
Electric conductivity	sigma	0

Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Right-click **Material 3** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Dielectric` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 3 only.

6 In the **Material** settings window, locate the **Material Contents** section.

7 In the table, enter the following settings:

Property	Name	Value
Relative permittivity	epsilonr	`eps_diel`
Relative permeability	mur	1
Electric conductivity	sigma	0

Material Browser

1 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domain 4 only.

The bioheat equation applies only in the liver tissue.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

1 In the **Model Builder** window, under **Model 1** click **Electromagnetic Waves, Frequency Domain**.

2 In the **Electromagnetic Waves, Frequency Domain** settings window, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Frequency domain**.

4 From the **Frequency** list, choose **User defined**. In the *f* edit field, type `f`.

Because you will use a study with a Frequency Domain step for a single frequency followed by a Stationary step, this is a convenient way to define the frequency in the second study step.

Port 1

1 Right-click **Model 1>Electromagnetic Waves, Frequency Domain** and choose **Port**.

2 Select Boundary 8 only.

3 In the **Port** settings window, locate the **Port Properties** section.

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

5 From the **Wave excitation at this port** list, choose **On**.

6 In the P_{in} edit field, type `P_in`.

Scattering Boundary Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose **Scattering Boundary Condition**.

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

BIOHEAT TRANSFER

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

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

3 Click **Clear Selection**.

4 Select Domain 1 only.

Heat Source 1

1 Right-click **Model 1>Bioheat Transfer** and choose **Heat Source**.

2 In the **Heat Source** settings window, locate the **Domain Selection** section.

3 From the **Selection** list, choose **All domains**.

4 Locate the **Heat Source** section. From the *Q* list, choose **Total power dissipation density (emw/wee1)**.

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

Bioheat 1

1 In the **Model Builder** window, expand the **Model 1>Bioheat Transfer>Biological Tissue 1** node, then click **Bioheat 1**.

 ©2013 COMSOL

2 In the **Bioheat** settings window, locate the **Bioheat** section.

3 In the ρ_b edit field, type `rho_blood`.

4 In the C_b edit field, type `Cp_blood`.

5 In the ω_b edit field, type `omega_blood`.

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

MESH 1

Size

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Free Triangular**.

2 In the **Size** settings window, locate the **Element Size Parameters** section.

3 In the **Maximum element size** edit field, type `3[mm]`.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Domain 3 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 edit field, type `0.15[mm]`.

8 Click the **Build All** button.

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, expand the **Study 1** node, then click **Step 1: Frequency Domain**.

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

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

Add a stationary analysis for the heat transfer problem.

4 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Stationary**.

5 Right-click **Study 1** and choose **Compute**.

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.

RESULTS

Electric Field (emw)

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 **Results>Electric Field (emw)** node, then click **Surface**.

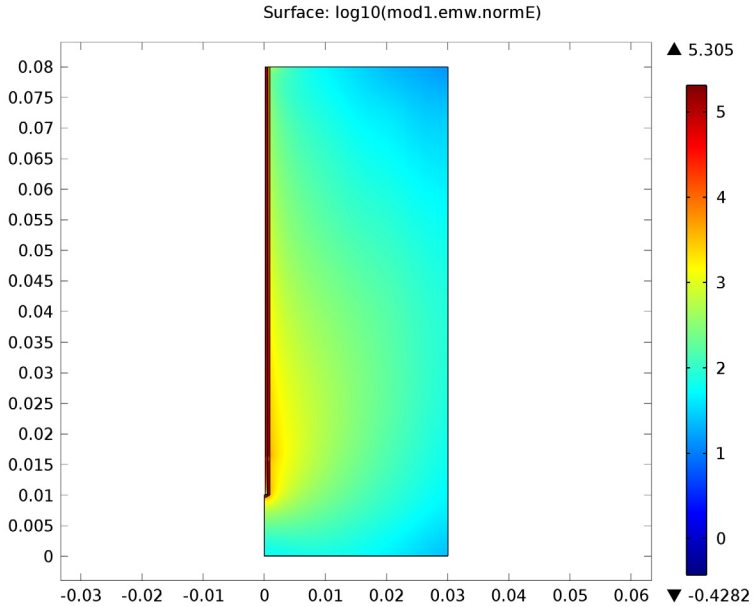
2 In the **Surface** settings window, locate the **Expression** section.

3 In the **Expression** edit field, type `log10(mod1.emw.normE)`.

4 Click the **Plot** button.

 ©2013 COMSOL

5 Click the **Zoom Extents** button on 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.

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

7 Click the **Plot** button.

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

9 In the **Maximum** edit field, type `1e6`.

10 Click the **Plot** button.

11 Click the **Zoom Extents** button on 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.

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Line 2D**.

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

3 In row **Point 1**, set **r** to `2.5e-3`.

4 In row **Point 1**, set **z** to `0.08`.

5 In row **Point 2**, set **r** to `2.5e-3`.

6 In row **Point 2**, set **z** to `0`.

1D Plot Group 4

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 In the **1D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Line Graph**.

5 In the **Line Graph** settings window, locate the **y-Axis Data** section.

6 In the **Expression** edit field, type `emw.Qh/rho_blood`.

7 Click the **Plot** button.

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

9 Right-click **1D Plot Group 4** and choose **Rename**.

10 Go to the **Rename 1D Plot Group** dialog box and type `Qh/rho vs arc length` in the **New name** edit field.

11 Click **OK**.

The plot you just created should look like Figure 5.

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

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration>Surface Integration**.

2 Select Domain 1 only.

3 In the **Surface Integration** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Heating and losses>Total power dissipation density (emw.Qh)**.

4 Click to expand the **Integration Settings** section. Select the **Compute volume integral** check box.

 ©2013 COMSOL

5 Click the **Evaluate** button.

The value of 9.35 W indicates that the tissue absorbs most of the 10 W input power.

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

2D Plot Group 5

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 5** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Bioheat Transfer>Temperature>Temperature (T)**.

4 Locate the **Expression** section. From the **Unit** list, choose **degC**.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

6 Click the **Plot** button.

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

8 In the **Model Builder** window, right-click **2D Plot Group 5** and choose **Rename**.

9 Go to the **Rename 2D Plot Group** dialog box and type `Temperature, 2D` in the **New name** edit field.

10 Click **OK**.

©2013 COMSOL