Microwave Oven

Introduction

This is a model of the heating process in a microwave oven. The distributed heat source is computed in a stationary, frequency-domain electromagnetic analysis followed by a transient heat transfer simulation showing how the heat redistributes in the food.

Model Definition

The microwave oven is a metallic box connected to a 500 W, 2.45 GHz microwave source via a rectangular waveguide operating in the TE_{10} mode. Near the bottom of the oven there is a cylindrical glass plate with a spherical potato placed on top of it. A part of the potato is cut away for mechanical stability, which also facilitates the creation of a finite element mesh in the region where it is in contact with the plate. Symmetry is utilized by simulating only half of the problem. The symmetry cut is applied vertically through the oven, waveguide, potato, and plate. Figure 1 below shows the reduced geometry.

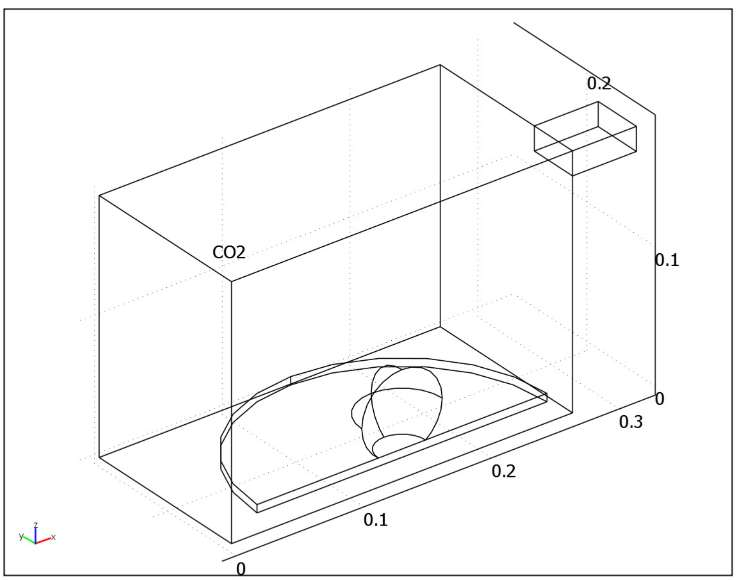

Figure 1: Geometry of microwave oven, potato, and waveguide feed.

The model uses copper for the walls of the oven and the waveguide. Although resistive metals losses are expected to be small, the *impedance boundary condition* on these walls ensures that they get accounted for. For more information on this boundary condition, see the section Impedance Boundary Condition in the *RF Module User's Guide*. The symmetry cut has mirror symmetry for the electric field and is represented by the boundary condition $\mathbf{n} \times \mathbf{H} = \mathbf{0}$.

The rectangular port is excited by a transverse electric (TE) wave, which is a wave that has no electric field component in the direction of propagation. At an excitation frequency of 2.45 GHz, the TE_{10} mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

$$(\nu_c)_{mn} = \frac{c}{2}\sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where m and n are the mode numbers and c denotes the speed of light. For the TE_{10} mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 7.8$ cm and $b = 1.8$ cm), the TE_{10} mode is the only propagating mode for frequencies between 1.92 GHz and 3.84 GHz.

The port condition requires a propagation constant β, which at the frequency ν is given by the expression

$$\beta = \frac{2\pi}{c}\sqrt{\nu^2 - \nu_c^2}$$

With the stipulated excitation at the rectangular port, the following equation is solved for the electric field vector $\mathbf{E}$ inside the waveguide and oven:

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

where μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ε_r the relative permittivity, and ε_0 the permittivity of free space. The model uses material parameters for air: $\sigma = 0$ and $\mu_r = \varepsilon_r = 1$. In the potato the same parameters are used except for the permittivity which is set to $\varepsilon_r = 65 - 20j$ where the imaginary part accounts for dielectric losses. The glass plate has $\sigma = 0$, $\mu_r = 1$ and $\varepsilon_r = 2.55$.

Results and Discussion

Figure 2 below shows the distributed microwave heat source as a slice plot through the center of the potato. The rather complicated oscillating pattern, which has a strong peak in the center, shows that the potato acts as a resonant cavity for the microwave field. The power absorbed in the potato is evaluated and amounts to about 60% of the input microwave power. Most of the remaining power is reflected back through the port.

Figure 3 shows the temperature in the center of the potato as a function of time for the first 5 seconds. Due to the low thermal conductivity of the potato, the heat distributes rather slowly, and the temperature profile after 5 seconds has a strong peak in the center (see Figure 4). When heating the potato further, the temperature in the center eventually reaches 100 °C and the water contents start boiling, drying out the center and transporting heat as steam to outer layers. This also affects the electromagnetic properties of the potato. The simple microwave absorption and heat conduction model used here does not capture these nonlinear effects. However, the model can serve as a starting point for a more advanced analysis.

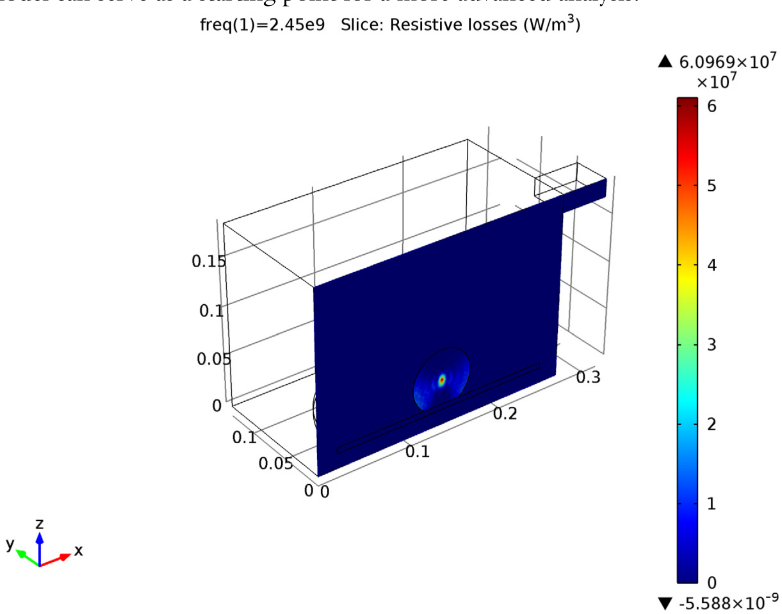

Figure 2: Dissipated microwave power distribution (W/m^3).

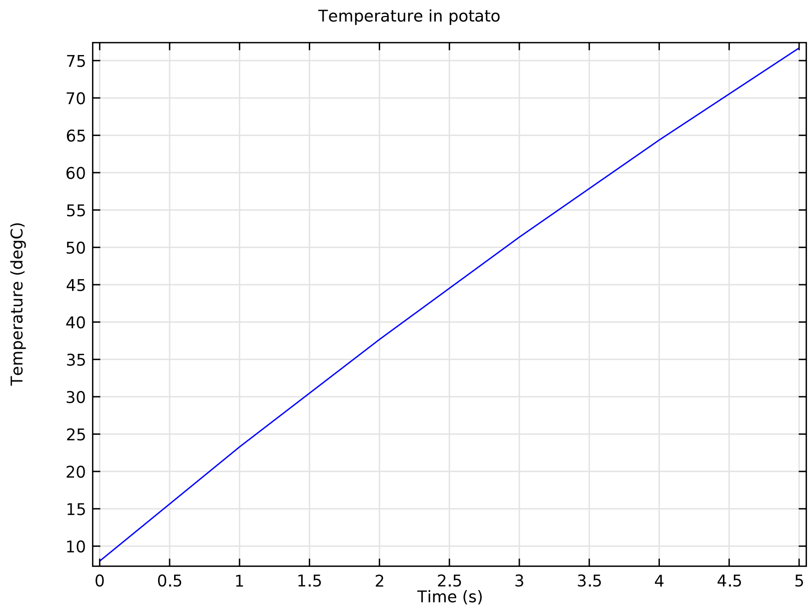

Figure 3: Temperature in the center of the potato during the first 5 seconds of heating.

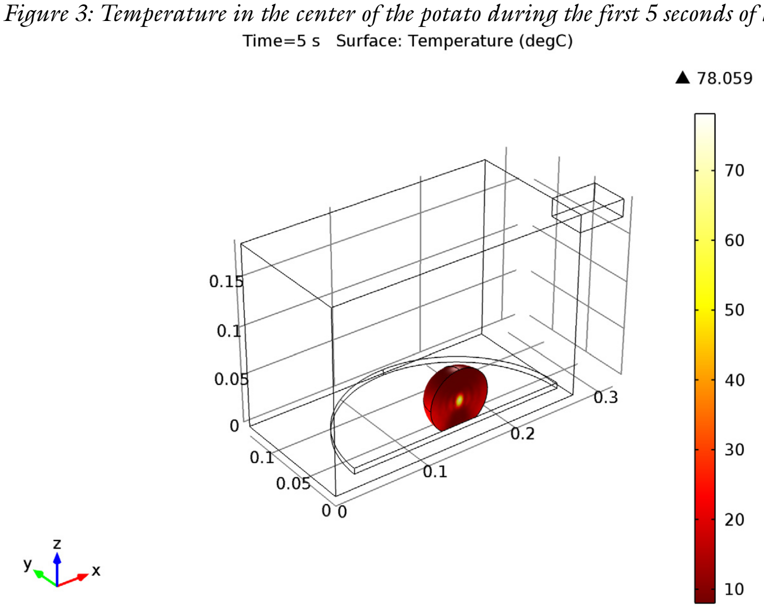

Figure 4: Temperature distribution after 5 seconds of heating.

Model Library path: `RF_Module/Microwave_Heating/microwave_oven`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

2 In the **Select physics** tree, select **Heat Transfer>Electromagnetic Heating>Microwave Heating**.

3 Click **Add**.

4 Click **Study**.

Add a Frequency Domain study type for the Electromagnetic Waves, Frequency Domain interface.

5 In the **Select study** tree, select **Custom Studies>Preset Studies for Some Physics>Frequency Domain**.

6 Click **Done**.

STUDY 1

Add a Time Dependent study for the Heat Transfer in Solids interface.

Step 2: Time Dependent

1 On the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

The Frequency Domain study is only used for the electromagnetics physics interface, whereas the time-dependent study is only applicable for the heat transfer physics interface. Notice that the electromagnetic heat source will be computed first, and then used in the time-dependent heat transfer study step.

2 In the **Time Dependent** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	**Solve for**	**Discretization**
Electromagnetic Waves, Frequency Domain (emw)	×	Physics settings

Step 1: Frequency Domain

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

2 In the **Frequency Domain** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	**Solve for**	**Discretization**
Heat Transfer in Solids (ht)	×	Physics settings

GLOBAL DEFINITIONS

First, define a set of parameters for creating the geometry.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
wo	267[mm]	0.26700 m	Oven width
do	270[mm]	0.27000 m	Oven depth
ho	188[mm]	0.18800 m	Oven height
wg	50[mm]	0.050000 m	Waveguide width
dg	78[mm]	0.078000 m	Waveguide depth
hg	18[mm]	0.018000 m	Waveguide height
rp	113.5[mm]	0.11350 m	Glass plate radius
hp	6[mm]	0.0060000 m	Glass plate height
bp	15[mm]	0.015000 m	Glass plate base
rpot	31.5[mm]	0.031500 m	Potato radius
T0	8[degC]	281.15 K	Initial potato temperature

GEOMETRY 1

Block 1

1. On the **Geometry** toolbar, click **Block**.
2. In the **Block** settings window, locate the **Size** section.
3. In the **Width** edit field, type `wo`.
4. In the **Depth** edit field, type `do/2`.
5. In the **Height** edit field, type `ho`.

Block 2

1. On the **Geometry** toolbar, click **Block**.
2. In the **Block** settings window, locate the **Size** section.
3. In the **Width** edit field, type `wg`.
4. In the **Depth** edit field, type `dg/2`.
5. In the **Height** edit field, type `hg`.
6. Locate the **Position** section. In the **x** edit field, type `wo`.
7. In the **z** edit field, type `ho-hg`.

Cylinder 1

1. On the **Geometry** toolbar, click **Cylinder**.
2. In the **Cylinder** settings window, locate the **Size and Shape** section.
3. In the **Radius** edit field, type `rp`.
4. In the **Height** edit field, type `hp`.
5. Locate the **Position** section. In the **x** edit field, type `wo/2`.
6. In the **z** edit field, type `bp`.

Sphere 1

1. On the **Geometry** toolbar, click **Sphere**.
2. In the **Sphere** settings window, locate the **Size** section.
3. In the **Radius** edit field, type `rpot`.
4. Locate the **Position** section. In the **x** edit field, type `wo/2`.
5. In the **z** edit field, type `rpot+bp`.
6. Click the **Build All Objects** button.

   The sphere you have created for the potato now overlaps the glass plate. This in itself is not a problem, but where the sphere touches the bottom of the glass plate, you risk getting very thin mesh elements. To avoid this problem, you will delete the part of the sphere that overlaps the cylinder. To retain the cylinder after this operation, begin by making a copy of it.

Copy 1

1. On the **Geometry** toolbar, click **Copy**.
2. Select the object **cyl1** only.

   The object **cyl1** is the cylinder.

Difference 1

1. On the **Geometry** toolbar, click **Difference**.
2. Select the object **sph1** only to add it to the **Objects to add** list.

   The object **sph1** is the sphere.
3. In the **Difference** settings window, locate the **Difference** section.
4. Click the **Active** button next to the **Objects to subtract** selection list.
5. Select the object **cyl1** only.
6. Click the **Build All Objects** button.

Finally, make a geometric operation to keep only the part of the potato and the plate that overlaps the half oven.

Compose 1

1. On the **Geometry** toolbar, click **Compose**.
2. Click in the **Graphics** window and then press Ctrl+A to select all objects.
3. In the **Compose** settings window, locate the **Compose** section.
4. In the **Set formula** edit field, type `(blk1+blk2)*(dif1+copy1)`.
5. Select the **Keep input objects** check box.
6. Click the **Build All Objects** button.

Delete Entities 1

1. In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
2. In the **Delete Entities** settings window, locate the **Entities or Objects to Delete** section.
3. From the **Geometric entity level** list, choose **Object**.
4. Select the objects **dif1** and **copy1** only.
5. Click the **Build All Objects** button.
6. Click the **Wireframe Rendering** button on the Graphics toolbar.

DEFINITIONS

Create the following selections definitions in order to make Domain and Boundary selections easier as you walk through these model instructions. Note that if you have problems finding certain numbers, you can always choose View > Selection List.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 1** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Potato` in the **New name** edit field.
4 Click **OK**.
5 Select Domain 3 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 2** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Plate` in the **New name** edit field.
4 Click **OK**.
5 Select Domain 2 only.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 3** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Air` in the **New name** edit field.
4 Click **OK**.
5 Select Domains 1 and 4 only.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 4** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `No Heat Transfer` in the **New name** edit field.
4 Click **OK**.

5 Select Domains 1, 2, and 4 only.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 5** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Port Boundary` in the **New name** edit field.
4 Click **OK**.
5 In the **Explicit** settings window, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Boundary**.
7 Select Boundary 23 only.

Explicit 6

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 6** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Symmetry Boundaries` in the **New name** edit field.
4 Click **OK**.
5 In the **Explicit** settings window, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Boundary**.
7 Select Boundaries 2, 7, 10, and 19 only.

Explicit 7

1 On the **Definitions** toolbar, click **Explicit**.
2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 7** and choose **Rename**.
3 Go to the **Rename Explicit** dialog box and type `Metal Boundaries` in the **New name** edit field.
4 Click **OK**.
5 In the **Explicit** settings window, locate the **Input Entities** section.
6 From the **Geometric entity level** list, choose **Boundary**.
7 Select Boundaries 1, 3–5, 17, and 20–22 only.

MATERIALS

Next, define the materials. Air and Copper are already in the Material Library.

1 On the **Home** toolbar, click **More Windows** and choose **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.
2 In the tree, select **Built-In>Air**.
3 In the **Add material** window, click **Add to Component**.

Air

1 In the **Model Builder** window, under **Component 1>Materials** click **Air**.
2 In the **Material** settings window, locate the **Geometric Entity Selection** section.
3 From the **Selection** list, choose **Air**.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.
2 Right-click **Material 2** and choose **Rename**.
3 Go to the **Rename Material** dialog box and type `Potato` 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 **Potato**.
7 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit
Relative permittivity	epsilonr	65-20*j	1
Relative permeability	mur	1	1
Electrical conductivity	sigma	0	S/m
Thermal conductivity	k	0.55	W/(m·K)
Density	rho	1050	kg/m³
Heat capacity at constant pressure	Cp	3.64e3	J/(kg·K)

Material 3

1 In the **Model Builder** window, right-click **Materials** and choose **New Material**.
2 Right-click **Material 3** and choose **Rename**.
3 Go to the **Rename Material** dialog box and type `Glass` 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 **Plate**.

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

Property	Name	Value	Unit
Relative permittivity	epsilonr	2.55	1
Relative permeability	mur	1	1
Electrical conductivity	sigma	0	S/m

You do not need to define the listed thermal properties, as the glass plate will not be in the thermal part of the model.

ADD MATERIAL

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

2 In the tree, select **Built-In>Copper**.

3 In the **Add material** window, click **Add to Component**.

4 On the **Home** toolbar, click **Add Material**.

This closes the **Add Material** window.

Copper

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

2 From the **Geometric entity level** list, choose **Boundary**.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

For the electromagnetic part of the problem, begin by defining the input port.

Port 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Port**.

2 In the **Port** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Port Boundary**.

4 Locate the **Port Properties** section. From the **Wave excitation at this port** list, choose **On**.

5 In the P_{in} edit field, type 500.

6 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

`0`	x
`0`	y
`cos(pi*y/dg)[V/m]`	z

7 In the β edit field, type `2*pi/c_const*sqrt(freq^2-c_const^2/(4*dg^2))`.

This is the propagation constant for the first propagating mode.

Next, set up the remaining boundary conditions.

Impedance Boundary Condition 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

2 In the **Impedance Boundary Condition** settings window, locate the **Boundary Selection** section.

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

Perfect Magnetic Conductor 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Perfect Magnetic Conductor**.

2 In the **Perfect Magnetic Conductor** settings window, locate the **Boundary Selection** section.

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

This concludes the electromagnetic part of the physics.

The Heat Transfer physics will automatically use the electromagnetic heat source from the Electromagnetic Waves physics thanks to the Electromagnetic Heat Source coupling feature.

In order to solve for the temperature in the potato only, deselect all the other domains.

HEAT TRANSFER IN SOLIDS

1 In the **Model Builder** window, under **Component 1** click **Heat Transfer in Solids**.

2 Select Domain 3 only.

Set the initial value for the temperature.

Initial Values 1

1 In the **Model Builder** window, under **Component 1>Heat Transfer in Solids** click **Initial Values 1**.

2 In the **Initial Values** settings window, locate the **Initial Values** section.

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

MESH 1

To ensure convergence and get an accurate result, the mesh in this model is required to everywhere resolve the wavelength. This is most critical in the potato, where the high permittivity results in a wavelength of just above 15 mm.

Free Tetrahedral 1

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

Size 1

1. In the **Model Builder** window, under **Component 1>Mesh 1** right-click **Free Tetrahedral 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. From the **Selection** list, choose **Potato**.
5. Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.
6. Click the **Custom** button.
7. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
8. In the associated edit field, type `6[mm]`.
9. Click the **Build All** button.

STUDY 1

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
2. In the **Frequency Domain** settings window, locate the **Study Settings** section.
3. In the **Frequencies** edit field, type `2.45[GHz]`.

Step 2: Time Dependent

1. In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.
2. In the **Time Dependent** settings window, locate the **Study Settings** section.
3. In the **Times** edit field, type `range(0,1,5)`.

   This will give you output at every second from t = 0 s to t = 5 s.

Solver 1

1 On the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solver 1** node.

3 Right-click **Study 1>Solver Configurations>Solver 1** and choose **Store Solution**.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The Graphics window shows the temperature distribution on the surface of the potato after 5 s. Change the unit to degC to reproduce Figure 4.

1 In the **Model Builder** window, under **Results>Temperature (ht)** click **Surface 1**.

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

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

4 On the **Temperature (ht)** toolbar, click **Plot**.

Modify an existing plot group to plot the resistive heating on the symmetry plane.

Isothermal Contours (ht)

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

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>Isothermal Contours (ht)** and choose **Rename**.

5 Go to the **Rename 3D Plot Group** dialog box and type `Resistive Heating` in the **New name** edit field.

6 Click **OK**.

Resistive Heating

1 In the **Model Builder** window, under **Results>Resistive Heating** right-click **Isosurface 1** and choose **Delete**.

2 Right-click **Results>Resistive Heating>Arrow Volume 1** and choose **Delete**.

3 Right-click **Resistive Heating** and choose **Slice**.

4 In the **Slice** 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>Resistive losses (emw.Qrh)**.

5 From the **Plane** list, choose **zx-planes**.

6 From the **Entry method** list, choose **Coordinates**.

7 On the **Resistive Heating** toolbar, click **Plot**.

The plot should now look like Figure 2. Next, add a nice visualization of the electromagnetic fields to the temperature plot.

Temperature (ht)

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Slice**.

2 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field>Electric field, z component (Ez)**.

3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **z-coordinates** edit field, type `0.1`.

6 Right-click **Results>Temperature (ht)>Slice 1** and choose **Deformation**.

7 In the **Deformation** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field (emw.Ex, emw.Ey, emw.Ez)**.

8 Click the **Plot** button.

Derived Values

Make a volume integral of the microwave heating to find out how much of the energy is absorbed in the potato.

1 On the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

Select one point in time for the output. Since the material parameters of the potato are independent of the temperature, it does not matter which time you choose.

2 In the **Volume Integration** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Heat Transfer in Solids>Heat sources>Total heat source (ht.Qtot)**.

5 Locate the **Data** section. From the **Parameter selection (freq)** list, choose **From list**.

6 Locate the **Selection** section. From the **Selection** list, choose **Potato**.

7 Click the **Evaluate** button.

TABLE

The result is 312 W. Finally, to reproduce Figure 3, create a plot of temperature in the center of the potato as a function of time.

RESULTS

Data Sets

1 On the **Results** toolbar, click **Cut Point 3D**.
2 In the **Cut Point 3D** settings window, locate the **Point Data** section.
3 In the **x** edit field, type `0.134`.
4 In the **y** edit field, type `0`.
5 In the **z** edit field, type `0.047`.

1D Plot Group 4

1 On the **Home** toolbar, click **Add Plot Group** 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 Point 3D 1**.
4 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
5 In the **Title** text area, type `Temperature in potato`.
6 Locate the **Plot Settings** section. Select the **x-axis label** check box.
7 In the associated edit field, type `Time (s)`.
8 On the **1D Plot Group 4** toolbar, click **Point Graph**.
9 In the **Point Graph** settings window, locate the **y-axis data** section.
10 Click **Temperature (T)** in the upper-right corner of the section. Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.
11 On the **1D Plot Group 4** toolbar, click **Plot**.