



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 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. The microwave operates at 1 kW, but when we use symmetry to reduce the model size by one half, we only input 500 W in simulation. The symmetry cut is applied vertically through the oven, waveguide, potato, and plate. [Figure 1](#) below shows both the full and reduced size geometry.

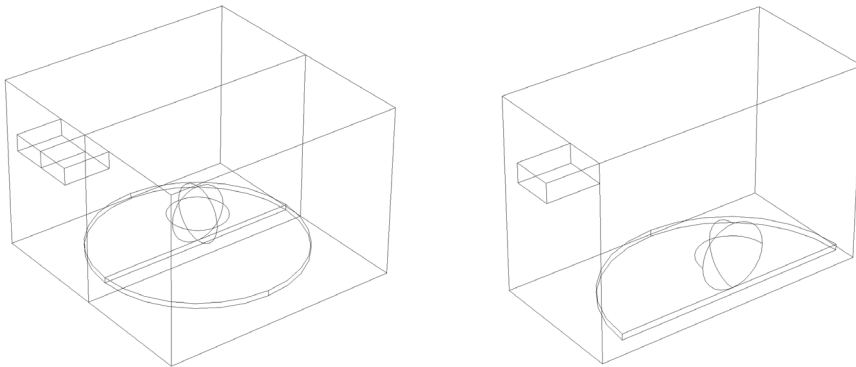


Figure 1: Geometry of the microwave oven, potato, and waveguide feed. Full size (left) and half size (right).

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 TE₁₀ mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

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

where m and n are the mode numbers and c denotes the speed of light. For the TE₁₀ mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 7.8$ cm and $b = 1.8$ cm), the TE₁₀ 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:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left(\epsilon_r - \frac{j\sigma}{\omega\epsilon_0} \right) \mathbf{E} = 0$$

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 $\mu_r = \epsilon_r = 1$. In the potato the same parameters are used except for the permittivity which is set to $\epsilon_r = 65 - 20j$ where the imaginary part accounts for dielectric losses. The glass plate has $\sigma = 0$, $\mu_r = 1$ and $\epsilon_r = 2.55$.

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.

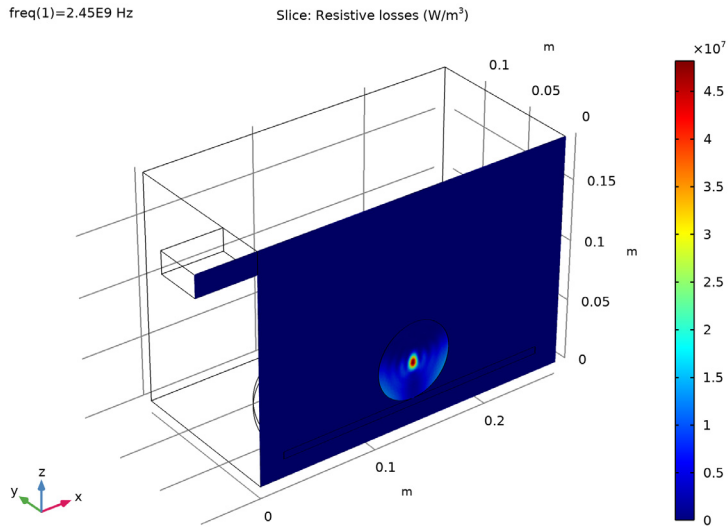
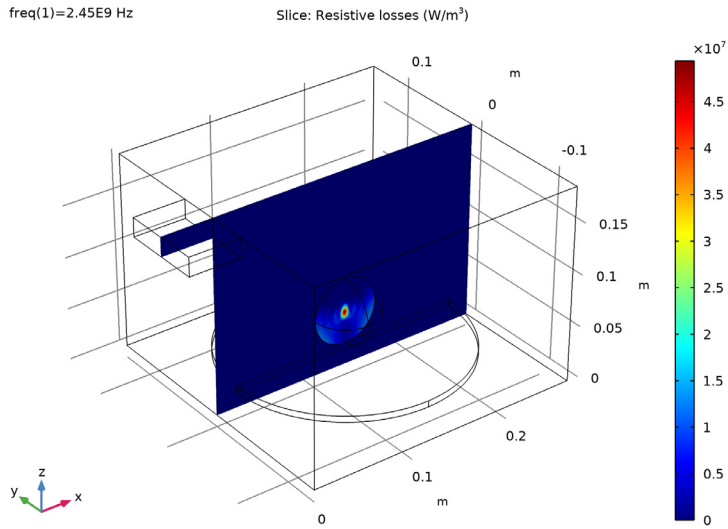


Figure 2: Dissipated microwave power distribution (W/m³). Full size (top) and half size (bottom).

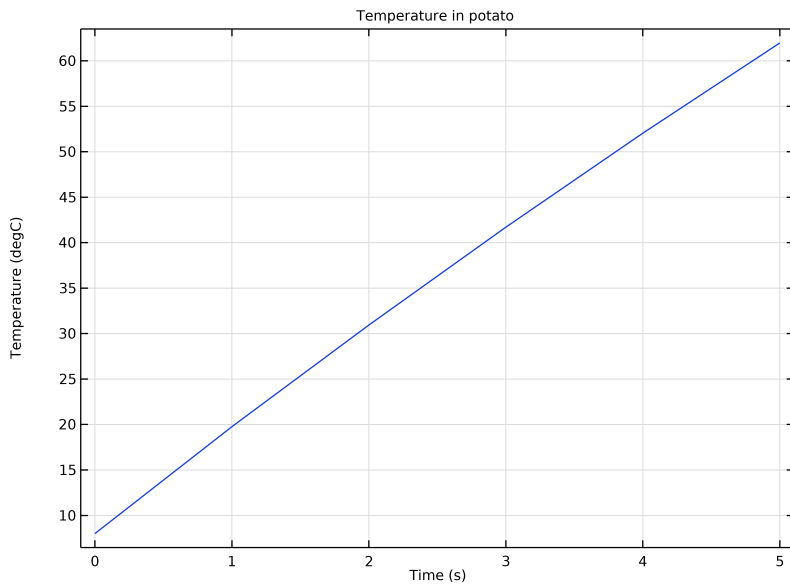
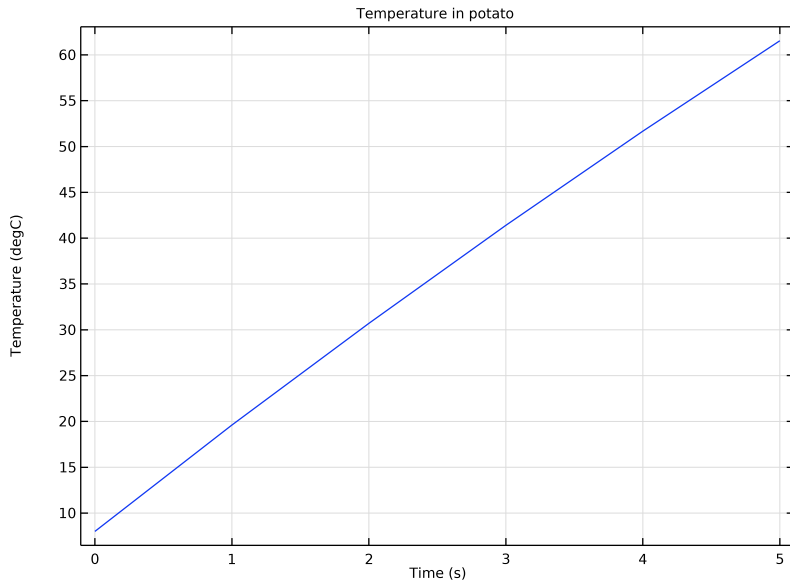
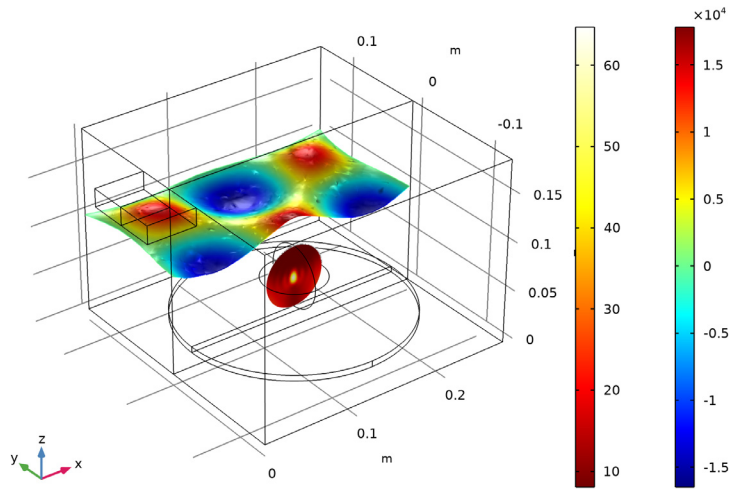


Figure 3: Temperature in the center of the potato during the first 5 seconds of heating. Full size (top) and half size (bottom).

full_geometry(1)=1 Time=5 s Slice: Temperature (degC) Slice: Electric field, z component (V/m)



full_geometry(1)=0 Time=5 s Surface: Temperature (degC) Slice: Electric field, z component (V/m)

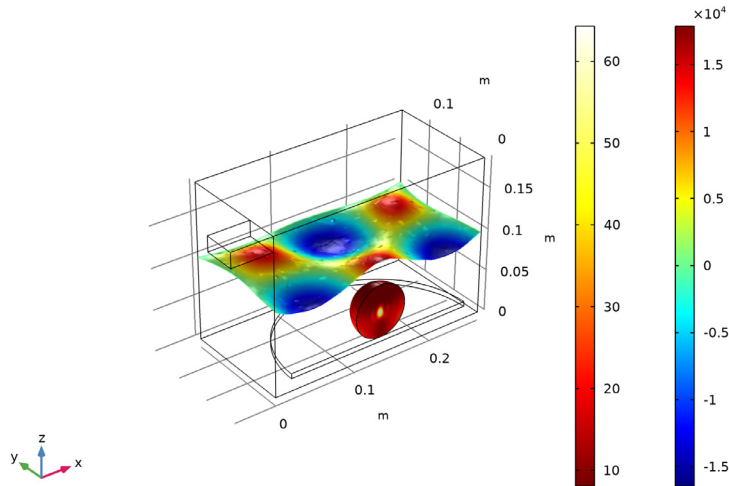


Figure 4: Deformed electric field and Temperature distribution after 5 seconds of heating. Full size (top) and half size (bottom).

Notes About the COMSOL Implementation


In this example model, the material properties of the potato are assumed to be constant as temperature rises, for a simpler and faster numerical modeling. It uses manually configured multiple study steps to perform one-way physics coupling from electromagnetics in the frequency domain to heat transfer in the time domain. Two-way bidirectional physics coupling between electromagnetics and heat transfer, using a predefined multiphysics study step, is addressed in another Application Libraries example, *RF Heating*.

Application Library path: RF_Module/Microwave_Heating/microwave_oven




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 **Heat Transfer>Electromagnetic Heating>Microwave Heating**.
- 3 Click **Add**.
- 4 In the **Added physics interfaces** tree, select **Electromagnetic Waves, Frequency Domain (emw)**.
- 5 Click  **Study**.
Add a **Frequency-Transient, One-Way Electromagnetic Heating** study sequence that add a **Frequency Domain** study type for the **Electromagnetic Waves, Frequency Domain** interface followed by a **Time Dependent** study type for the **Heat Transfer in Solids** interface.
- 6 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Frequency-Transient, One-Way Electromagnetic Heating**.
- 7 Click  **Done**.


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 2.45[GHz].
- 4 In the **Model Builder** window, click **Study 1**.
- 5 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 6 Select the **Store solution for all intermediate study steps** check box.


GLOBAL DEFINITIONS

Parameters 1


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

GEOMETRY 1

Block 1 (blk1)

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type `w0`.
- 4 In the **Depth** text field, type `d0`.
- 5 In the **Height** text field, type `h0`.
- 6 Locate the **Position** section. In the **y** text field, type `-d0/2`.


Block 2 (blk2)

- 1 In the **Geometry** toolbar, click  **Block**.
- 2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type `wg`.
- 4 In the **Depth** text field, type `dg`.
- 5 In the **Height** text field, type `hg`.
- 6 Locate the **Position** section. In the **x** text field, type `-wg`.

7 In the **y** text field, type $-dg/2$.

8 In the **z** text field, type $ho-hg$.

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 rp .

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

5 Locate the **Position** section. In the **x** text field, type $wo/2$.

6 In the **z** text field, type bp .

Sphere 1 (sph1)

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

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

3 In the **Radius** text field, type $rpot$.

4 Locate the **Position** section. In the **x** text field, type $wo/2$.


5 In the **z** text field, type $rpot+bp+hp$.

6 Click  **Build All Objects**.

Now, it is possible to exploit the mirror symmetry of the model by chopping the geometry and only simulating one half of the model. For this purpose, form a union of all geometric and build an intersection with a block that includes only half of the model.

Union 1 (uni1)

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

2 Click the  **Select All** button in the **Graphics** toolbar.

Block 3 (blk3)

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

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

3 In the **Width** text field, type 0.4 .




4 In the **Depth** text field, type 0.4 .

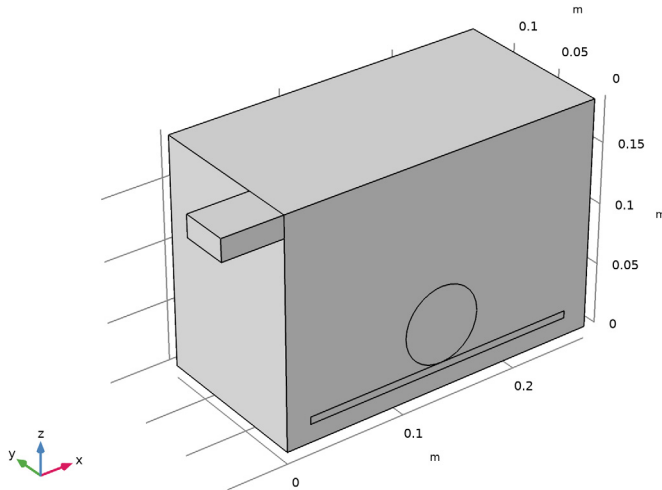
5 In the **Height** text field, type 0.4 .

6 Locate the **Position** section. In the **x** text field, type -0.1 .


7 Click  **Build Selected**.

Intersection 1 (int1)



- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Intersection**.
- 2 Click the  **Select All** button in the **Graphics** toolbar.
- 3 In the **Settings** window for **Intersection**, click  **Build All Objects**.



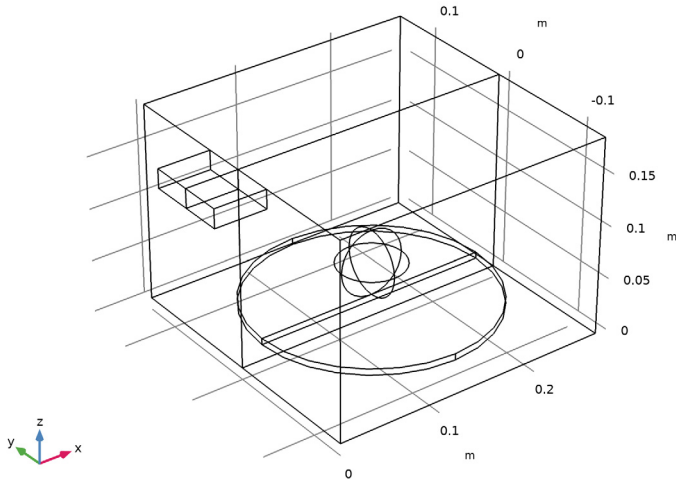
If Full Geometry

- 1 In the **Geometry** toolbar, click  **Programming** and choose **If + End If**.
- 2 In the **Settings** window for **If**, type If Full Geometry in the **Label** text field.
- 3 Locate the **If** section. In the **Condition** text field, type full_geometry.

Mirror 1 (mir1)

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Mirror**.
- 2 Select the object **int1** only.
- 3 In the **Settings** window for **Mirror**, locate the **Input** section.
- 4 Select the **Keep input objects** check box.
- 5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type 1.
- 6 In the **z** text field, type 0.
- 7 Click  **Build All Objects**.

8 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.



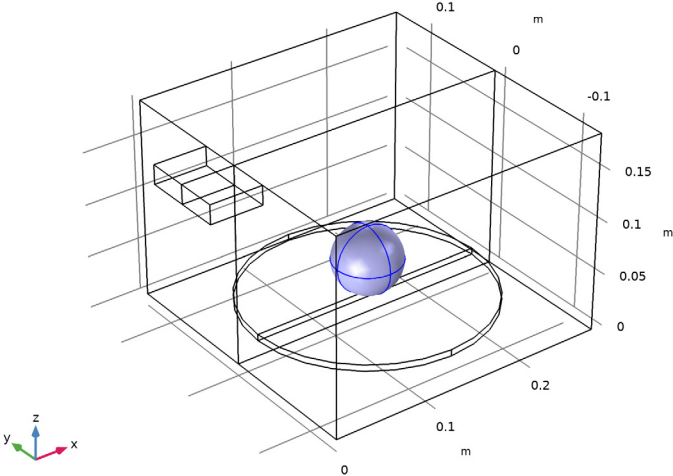
Create the following selections definitions in order to make Domain and Boundary selections easier as you walk through these model instructions. Note that if you have problems finding certain numbers, you can always choose View > Selection List.

DEFINITIONS


Potato

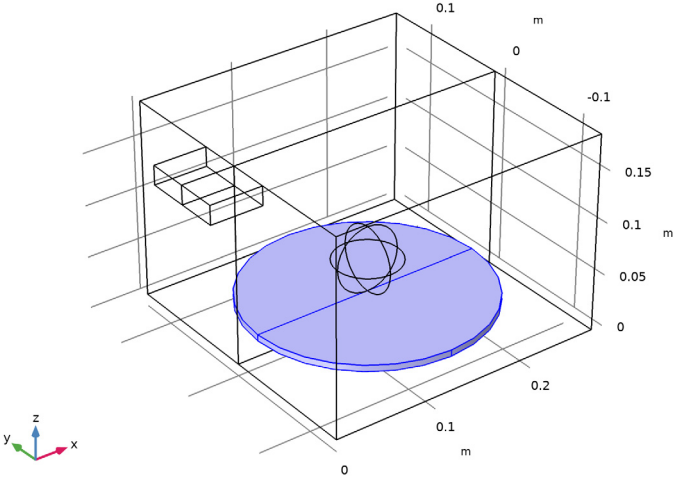
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Potato in the **Label** text field.

3 Select Domains 7 and 8 only.




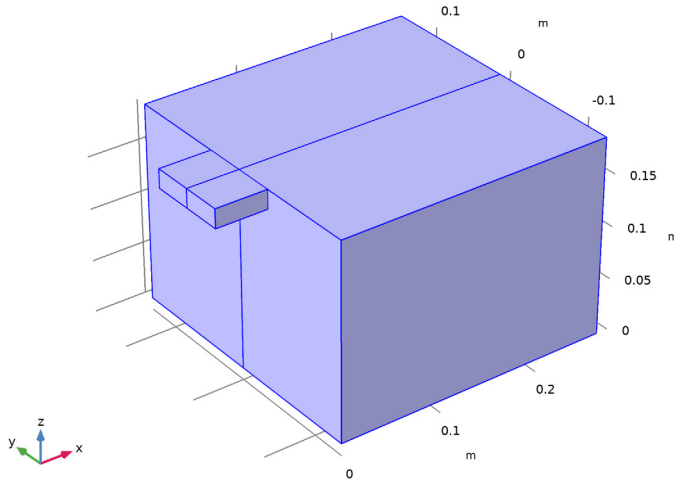
Plate

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type **Plate** in the **Label** text field.
- 3 Select Domains 5 and 6 only.




Air

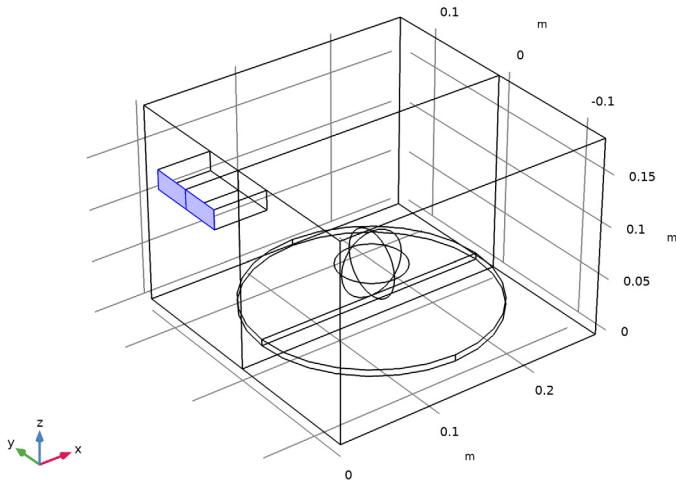
- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Air in the **Label** text field.
- 3 Select Domains 1–4 only.




Port Boundary

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Port Boundary in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

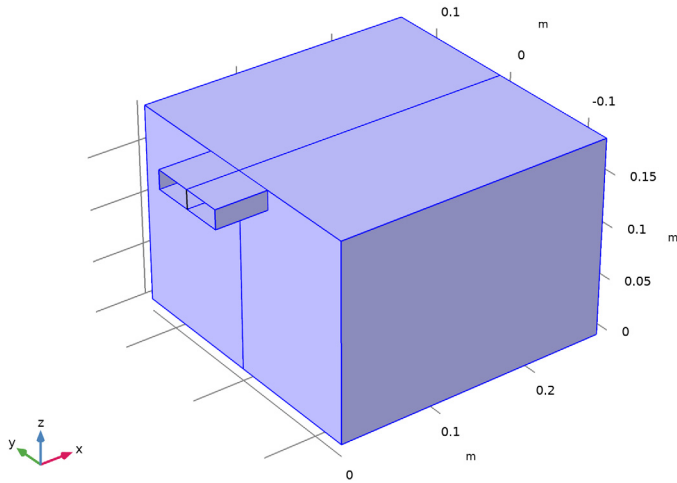
4 Select Boundaries 1 and 5 only.




Metal Boundaries

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Metal Boundaries in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

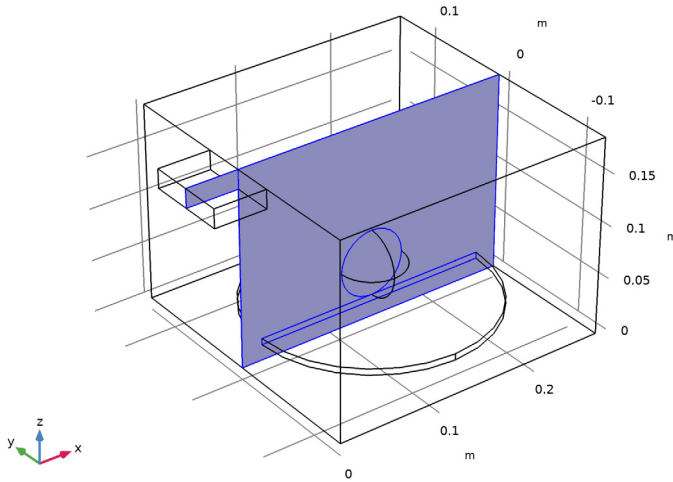
4 Select Boundaries 2–4, 7–13, 15, 17, 19, 20, 39, and 40 only.



Symmetry


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Symmetry in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 6, 16, 23, and 30 only.



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

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 **Built-in>Air**.
- 4 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

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

Potato

- 1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type **Potato** in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Potato**.

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

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilon _{r_iso} ; epsilon _{r_ii} = epsilon _{r_iso} , epsilon _{r_ij} = 0	65 - 20*j	l	Basic
Relative permeability	mu _{r_iso} ; mu _{r_ii} = mu _{r_iso} , mu _{r_ij} = 0	1	l	Basic
Electrical conductivity	sigma _{iso} ; sigma _{ii} = sigma _{iso} , sigma _{ij} = 0	0	S/m	Basic
Thermal conductivity	k _{iso} ; k _{ii} = k _{iso} , k _{ij} = 0	0.55	W/(m·K)	Basic
Density	rho	1050	kg/m ³	Basic
Heat capacity at constant pressure	C _p	3.64e3	J/(kg·K)	Basic

Glass


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

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

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilon _{r_} iso ; epsilon _{r_} ii = epsilon _{r_} iso, epsilon _{r_} ij = 0	2.55	l	Basic
Relative permeability	mu _{r_} iso ; mu _{r_} ii = mu _{r_} iso, mu _{r_} ij = 0	1	l	Basic
Electrical conductivity	sigma _{iso} ; sigma _{ii} = sigma _{iso} , sigma _{ij} = 0	0	S/m	Basic
Thermal conductivity	k _{iso} ; k _{ii} = k _{iso} , k _{ij} = 0		W/(m·K)	Basic
Density	rho		kg/m ³	Basic
Heat capacity at constant pressure	C _p		J/(kg·K)	Basic

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 Click **Add to Component** in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat4)

- 1 In the **Settings** window for **Material**, 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 (EMW)


For the electromagnetic part of the problem, begin by defining the input port. In the full model, you can exploit the predefined settings of the rectangular port.

Port 1, Full Model

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.
- 2 In the **Settings** window for **Port**, type **Port 1**, **Full Model** in the **Label** text field.
- 3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Port Boundary**.
- 4 Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.
For the first port, wave excitation is **on** by default.
- 5 In the P_{in} text field, type **1 [kW]**.

Next, set up the remaining boundary conditions.

Impedance Boundary Condition 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Impedance Boundary Condition**.
- 2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Metal Boundaries**.


HEAT TRANSFER IN SOLIDS (HT)

In the **Physics** toolbar, click **Select Physics Interface** and choose **Heat Transfer in Solids**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

For a mirror symmetric rectangular port, there is not any predefined boundary condition available anymore. So we have to implement the port boundary condition manually.

Port 2, Half Model

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Port**.
Now select a User-defined port. Here you need to define the electric mode field as well as cut-off frequency manually. Also keep in mind that the excited power is only half of Port 1.
- 2 In the **Settings** window for **Port**, type **Port 2**, **Half Model** in the **Label** text field.
- 3 Locate the **Boundary Selection** section. 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} text field, type **1 [kW] / 2**.

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 text field, type $2*\pi/c_const*\sqrt{freq^2-c_const^2/(4*dg^2)}$.

Exploit the mirror symmetry of the model by adding a Perfect Magnetic Conductor.

Perfect Magnetic Conductor / Symmetry

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

2 In the **Settings** window for **Perfect Magnetic Conductor**, type Perfect Magnetic Conductor / Symmetry in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Symmetry**.

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 Heating coupling feature.

In order to solve for the temperature in the potato only, use the predefined potato selection.

HEAT TRANSFER IN SOLIDS (HT)

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

2 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

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

Initial Values I


Set the initial value for the temperature.

1 In the **Model Builder** window, under **Component 1 (comp1)**>**Heat Transfer in Solids (ht)** 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 .

MESH I

In the **Home** toolbar, click  **Build Mesh**.

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 **Physics and Variables Selection** section.
- 3 Select the **Modify model configuration for study step** check box.
- 4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 2, Half Model**.
- 5 Click  **Disable**.
- 6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Perfect Magnetic Conductor / Symmetry**.
- 7 Click  **Disable**.

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 In the **Output times** text field, type range (0, 1, 5).
This will give you output at every second from $t = 0$ s to $t = 5$ s.

Parametric Sweep

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click  **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
full_geometry (Symmetry flag)	1	

- 5 In the **Study** toolbar, click  **Compute**.

RESULTS

Multislice

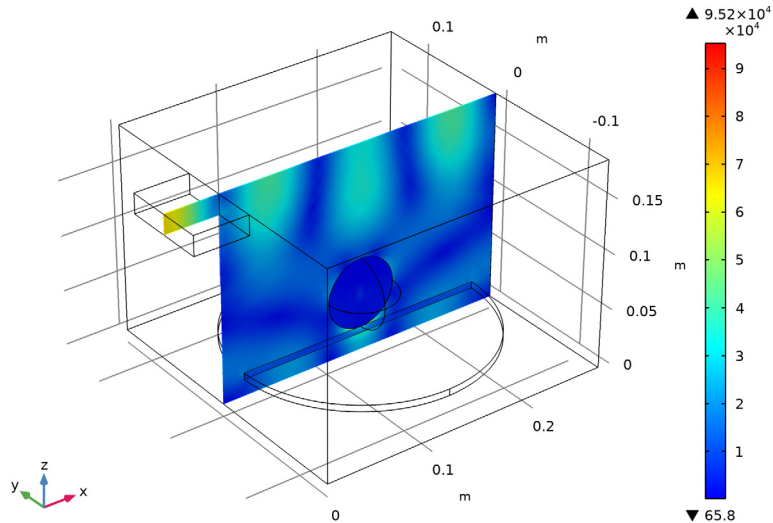
- 1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
- 2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
- 3 Find the **X-planes** subsection. In the **Planes** text field, type 0.

4 Find the **Z-planes** subsection. In the **Planes** text field, type 0.

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

full_geometry(1)=1 Time=5 s

Multislice: Electric field norm (V/m)



The results show the E-field norm distribution inside the microwave oven.

Surface


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, expand the **Results>Temperature (ht)** node, then click **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.
- 4 Right-click **Surface** and choose **Delete**.

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)>Heat Transfer in Solids>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**.
Next, add a nice visualization of the electromagnetic fields to the temperature plot.


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)>Electromagnetic Waves, Frequency Domain>Electric>Electric field - V/m>emw.Ez - Electric field, z component**.
- 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** text field, type 0.1.


Deformation 1

- 1 Right-click **Slice 2** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.
- 3 In the **Temperature (ht)** toolbar, click  **Plot**.

Add a filter to your plot to prevent the electric field plot from covering the potato.

Filter 1

- 1 In the **Model Builder** window, right-click **Slice 2** and choose **Filter**.
- 2 In the **Settings** window for **Filter**, locate the **Element Selection** section.
- 3 In the **Logical expression for inclusion** text field, type $y > 0$.

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

Compare the created plot to [Figure 4](#).

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


Resistive Heating

- 1 In the **Model Builder** window, under **Results** click **Isothermal Contours (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, type Resistive Heating in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

Isosurface

- 1 In the **Model Builder** window, expand the **Resistive Heating** node.
- 2 Right-click **Isosurface** and choose **Delete**.

Slice 1

- 1 In the **Model Builder** window, right-click **Resistive Heating** and choose **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)> Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qrh - Resistive losses - W/m³**.
- 3 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
- 4 From the **Entry method** list, choose **Coordinates**.
- 5 In the **Resistive Heating** toolbar, click  **Plot**.

The dissipated microwave power distribution inside the microwave oven. It is plotted in [Figure 2](#).

Volume Integration 1


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

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

- 2 In the **Settings** window for **Volume Integration**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

- 4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Heat sources>ht.Qtot - Total heat source - W/m³**.

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.


- 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  **Evaluate**.

The result is 631 W. Finally, to reproduce [Figure 3](#), create a plot of temperature in the center of the potato as a function of time.


TABLE

Go to the **Table** window.

Cut Point 3D 1


- 1 In the **Results** toolbar, click  **Cut Point 3D**.
- 2 In the **Settings** window for **Cut Point 3D**, locate the **Point Data** section.
- 3 In the **X** text field, type $w_0/2$.
- 4 In the **Y** text field, type 0.
- 5 In the **Z** text field, type $r_{pot}+b_p+h_p$.

ID Plot Group 4

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Data** section.
- 3 From the **Dataset** 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 text field, type Time (s).



Point Graph 1

- 1 Right-click **ID Plot Group 4** and choose **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 **T - Temperature - K**.
- 3 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

- 4 In the **ID Plot Group 4** toolbar, click  **Plot**.

The plot should now look like [Figure 3](#).



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.

STUDY 2

- 1 In the **Model Builder** window, click **Study 2**.
- 2 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 3 Select the **Store solution for all intermediate study steps** check box.



Step 1: Frequency Domain

- 1 In the **Model Builder** window, under **Study 2** 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 2.45[GHz].
- 4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.
- 5 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 1, Full Model**.
- 6 Click  **Disable**.
- 7 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Impedance Boundary Condition 1**.
- 8 Click  **Disable**.

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 In the **Output times** text field, type range(0, 1, 5).

Parametric Sweep

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click  **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
full_geometry (Symmetry flag)	0	

- 5 In the **Study** toolbar, click  **Compute**.

RESULTS

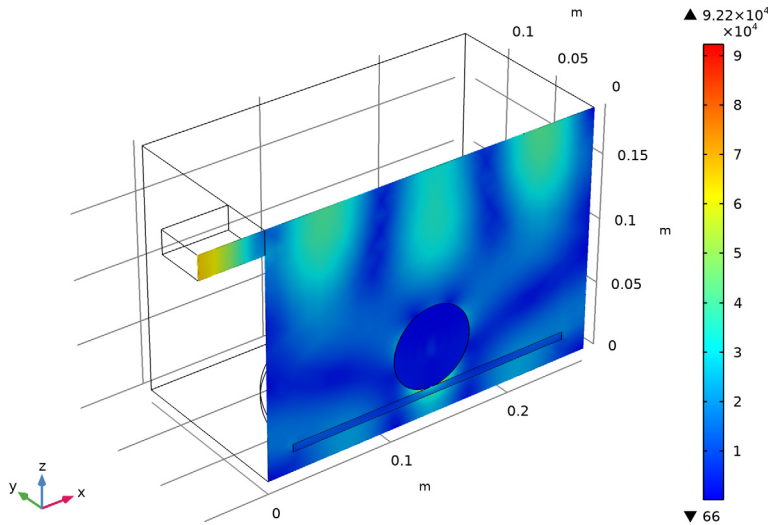
Multislice

- 1 In the **Model Builder** window, expand the **Electric Field (emw) 1** node, then click **Multislice**.
- 2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
- 3 Find the **X-planes** subsection. In the **Planes** text field, type 0.
- 4 Find the **Z-planes** subsection. In the **Planes** text field, type 0.
- 5 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type 0.

full_geometry(1)=0 Time=5 s

Multislice: Electric field norm (V/m)



Review the default plots of the half size model and modify them to compare your results with those of the full size model.

Surface


- 1 In the **Model Builder** window, expand the **Results>Temperature (ht) I** node, then click **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.

Slice 1

- 1 In the **Model Builder** window, right-click **Temperature (ht) I** and choose **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)> Electromagnetic Waves, Frequency Domain>Electric>Electric field - V/m>emw.Ez - Electric field, z component**.
- 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** text field, type 0.1.

6 Click the  **Go to Default View** button in the **Graphics** toolbar.

Deformation 1

- 1 Right-click **Slice 1** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.
- 3 In the **Temperature (ht) 1** toolbar, click  **Plot**.

The plot is shown in [Figure 4](#).



Resistive Heating Half Model

- 1 In the **Model Builder** window, expand the **Results>Isothermal Contours (ht)** node, then click **Isothermal Contours (ht)**.
- 2 In the **Settings** window for **3D Plot Group**, type Resistive Heating Half Model in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution Store 2 (sol6)**.

Isosurface


In the **Model Builder** window, right-click **Isosurface** and choose **Delete**.


Slice 1

- 1 In the **Model Builder** window, right-click **Resistive Heating Half Model** and choose **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)> Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qrh - Resistive losses - W/m³**.
- 3 Locate the **Plane Data** section. From the **Plane** list, choose **ZX-planes**.
- 4 From the **Entry method** list, choose **Coordinates**.
- 5 In the **Resistive Heating Half Model** toolbar, click  **Plot**.
- 6 Click the  **Go to Default View** button in the **Graphics** toolbar.

The created plot is shown in [Figure 2](#).

Volume Integration 2

- 1 In the **Results** toolbar, click  **More Derived Values** and choose **Integration> Volume Integration**.
- 2 In the **Settings** window for **Volume Integration**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Solution Store 2 (sol6)**.

- 4 From the **Parameter selection (freq)** list, choose **From list**.
- 5 Select Domain 3 only.
- 6 Locate the **Selection** section. From the **Selection** list, choose **Potato**.
- 7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Solids>Heat sources>ht.Qtot - Total heat source - W/m³**.
- 8 Click  **Evaluate**.


TABLE

- 1 Go to the **Table** window.


The result is 314 W. This is roughly half the power as for the full model.

RESULTS

Cut Point 3D 2

- 1 In the **Results** toolbar, click  **Cut Point 3D**.
- 2 In the **Settings** window for **Cut Point 3D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Solution 5 (sol5)**.
- 4 Locate the **Point Data** section. In the **X** text field, type $w_0/2$.
- 5 In the **Y** text field, type 0.
- 6 In the **Z** text field, type $r_{pot}+b_p+h_p$.

ID Plot Group 8

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Point 3D 2**.
- 4 Locate 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 text field, type Time (s).

Point Graph 1

- 1 Right-click **ID Plot Group 8** and choose **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 **T - Temperature - K**.
- 3 Locate the **y-Axis Data** section. From the **Unit** list, choose **degC**.

4 In the **ID Plot Group 8** toolbar, click  **Plot**.

The temperature plot is in good agreement with the temperature plot of Plot Group 4 of the full model. See [Figure 3](#).