Created in COMSOL Multiphysics 6.2

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.

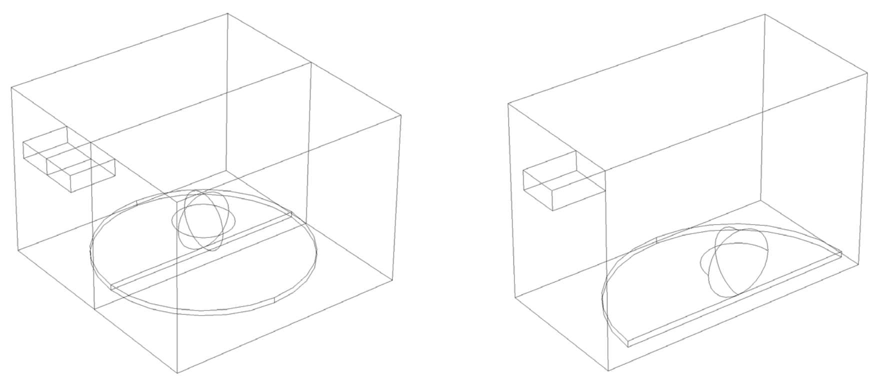


Figure 1: Geometry of the microwave oven, potato, and waveguide feed. Full size (left) and half size (right).

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 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 **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.

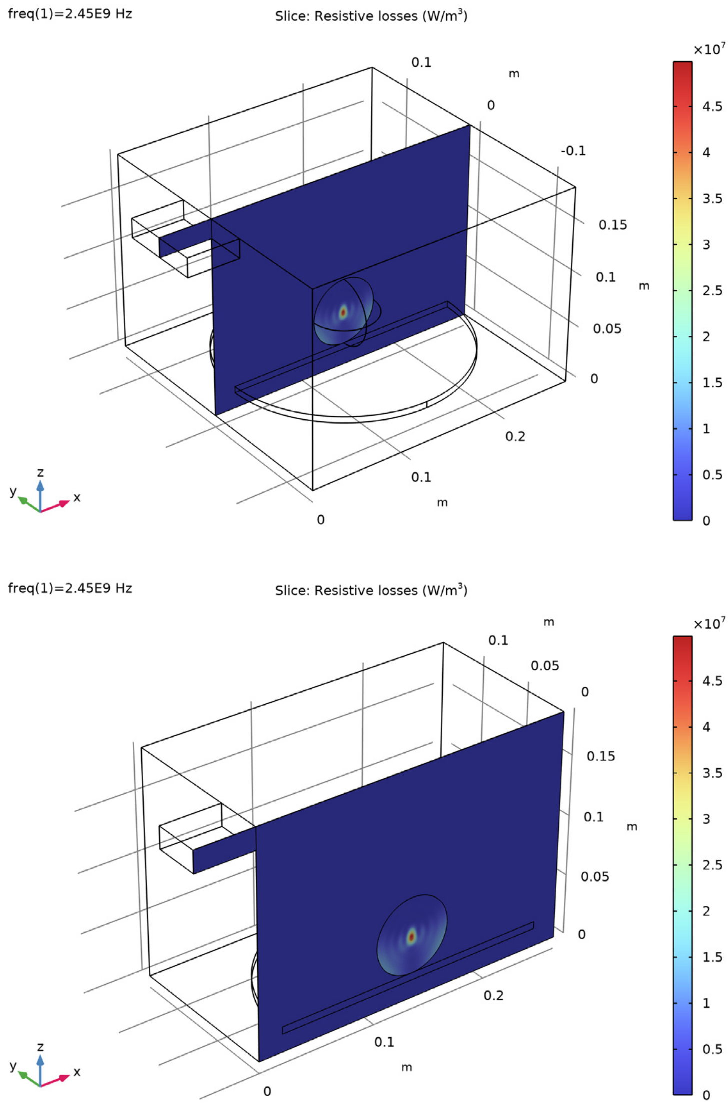


Figure 2: Dissipated microwave power distribution (W/m^3). Full size (top) and half size (bottom).

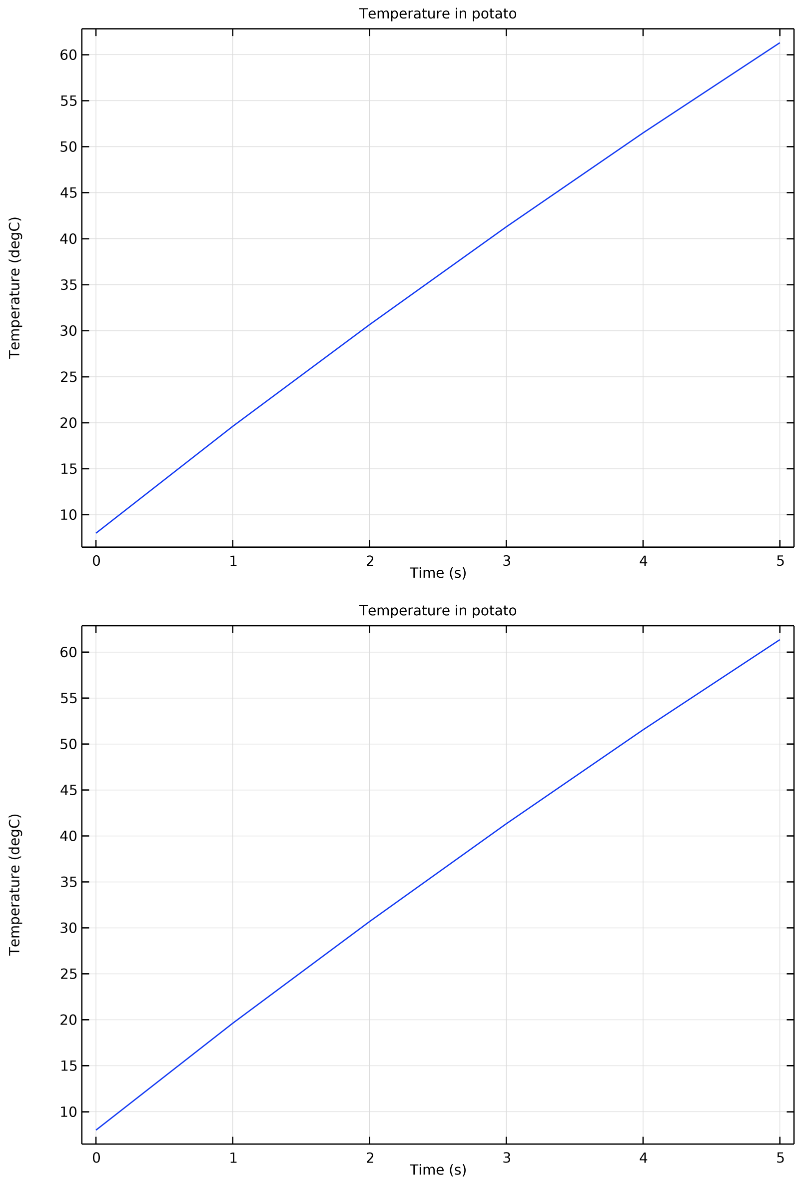


Figure 3: Temperature in the center of the potato during the first 5 seconds of heating. Full size (top) and half size (bottom).

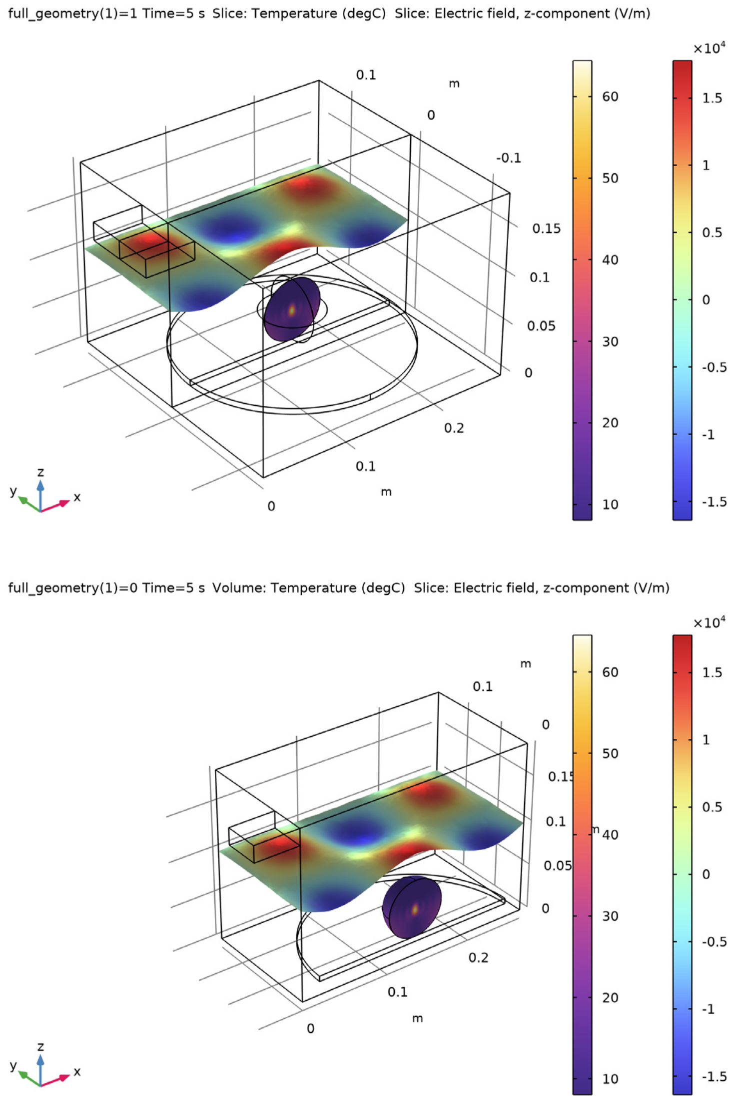


Figure 4: Deformed electric field and Temperature distribution after 5 seconds of heating. Full size (top) and half size (bottom).

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 RF Module Application Library 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

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the specified frequency value.

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 `wo`.

4 In the **Depth** text field, type `do`.

5 In the **Height** text field, type `ho`.

6 Locate the **Position** section. In the **y** text field, type `-do/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 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`.

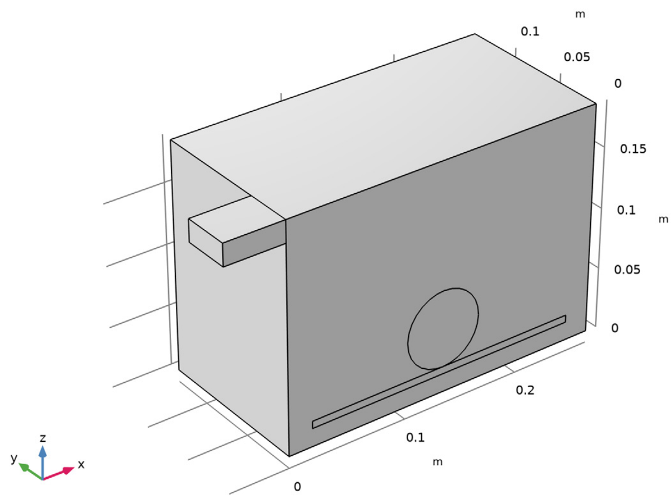
7 Click Build Selected.

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.

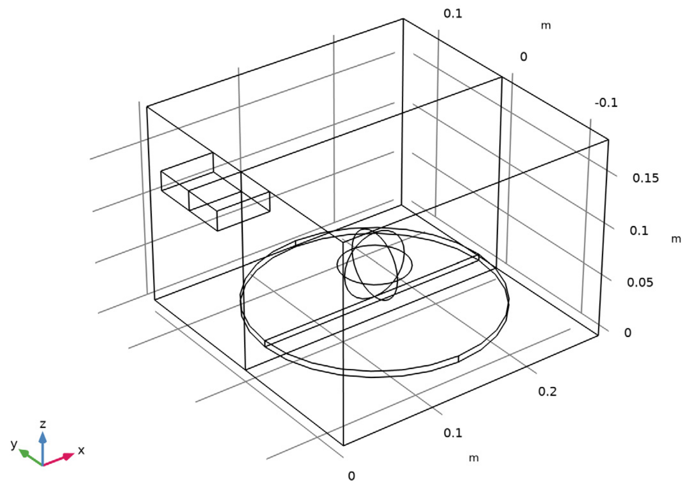
4 Select the **Keep input objects** check box.

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.

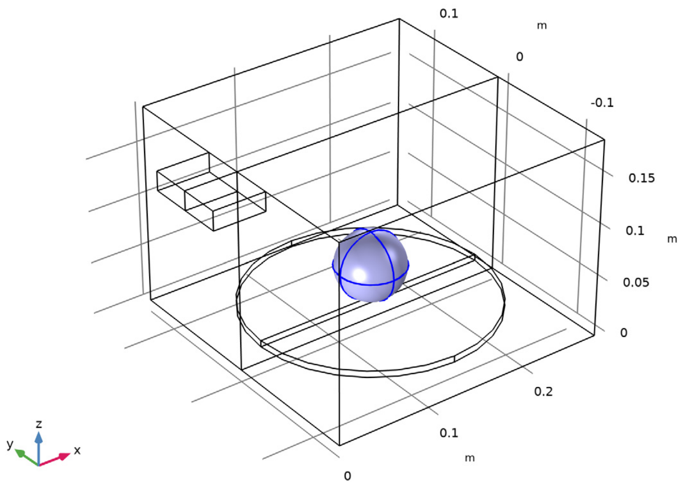
DEFINITIONS

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.

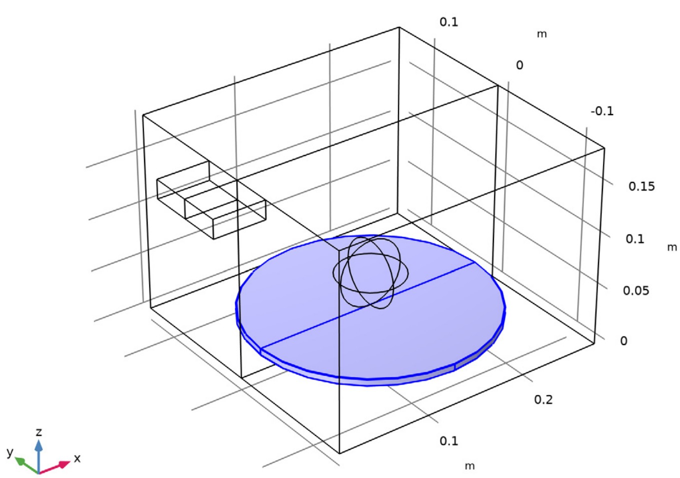


Plate

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.

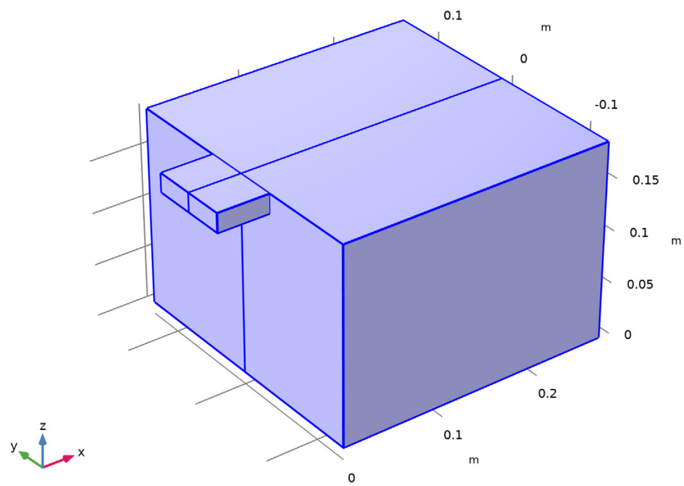


Air

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.



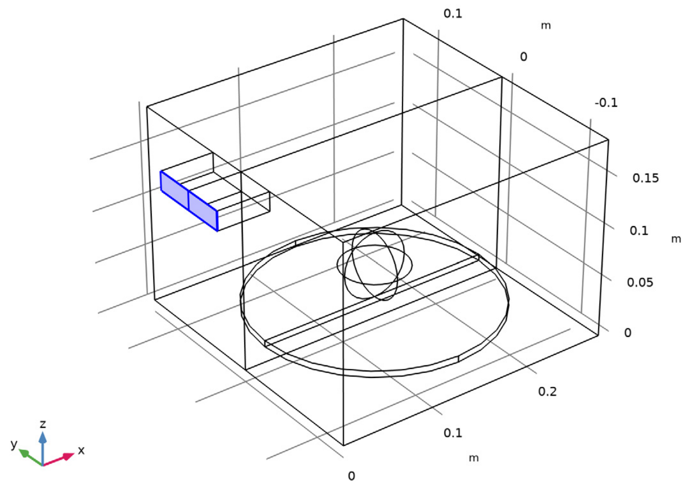
Port Boundary

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.



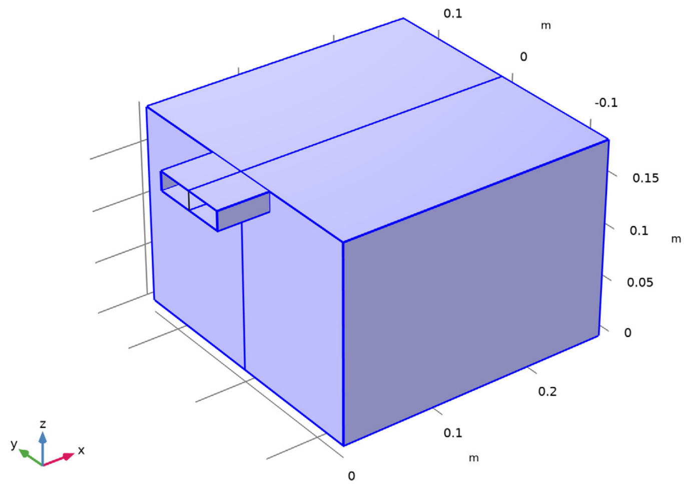
Metal Boundaries

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.



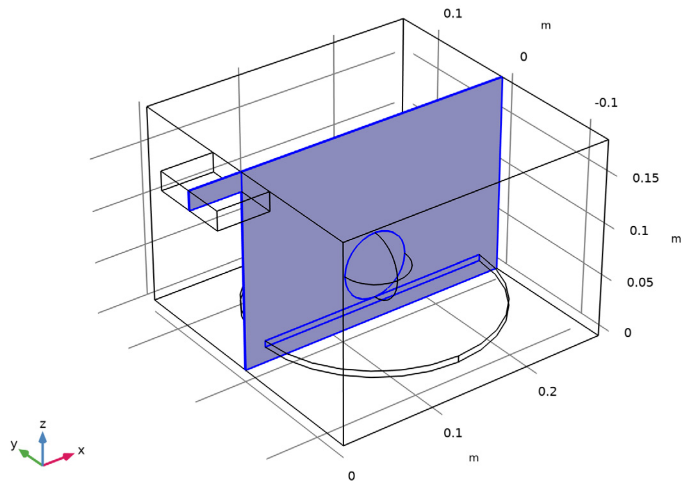
Symmetry

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.

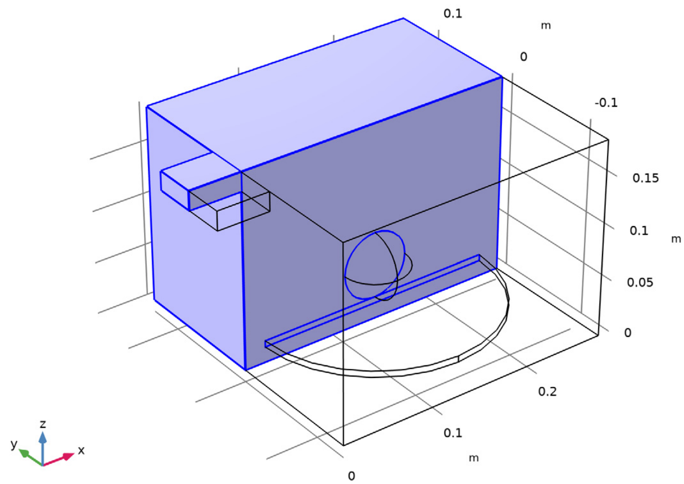


Half Model

1 In the **Definitions** toolbar, click  **Explicit**.

2 In the **Settings** window for **Explicit**, type `Half Model` in the **Label** text field.

3 Select Domains 2, 4, 6, and 8 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	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`65-20*j`	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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.55`	W/(m·K)	Basic
Density	rho	`1050`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`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	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	2.55	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
Thermal conductivity	k_iso ; kii = k_iso, kij = 0		W/(m·K)	Basic
Density	rho		kg/m^3	Basic
Heat capacity at constant pressure	Cp		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)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Half Model**.

Port 2, Half Model

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

   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 **Type of port** list, choose **Rectangular**.

5. From the **Wave excitation at this port** list, choose **On**.
6. In the P_{in} text field, type `1[kW]/2`.

   If you want to configure the port manually, the **Rectangular** type port can be replaced with **User defined** where the z-component of electric field is `cos(pi*y/dg)[V/m]` and the propagation constant is `2*pi/c_const*sqrt(freq^2-c_const^2/(4*dg^2))`.

Exploit the mirror symmetry of the model by adding a PMC type symmetry plane.

Symmetry Plane 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.
2. In the **Settings** window for **Symmetry Plane**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Symmetry**.
4. In the **Model Builder** window, click **Electromagnetic Waves, Frequency Domain (emw)**.
5. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.
6. From the **Selection** list, choose **All domains**.

   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 1

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 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. In the T text field, type `T0`.

MESH 1

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 tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 2, Half Model**.

5 Click **Disable**.

6 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Symmetry Plane 1**.

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

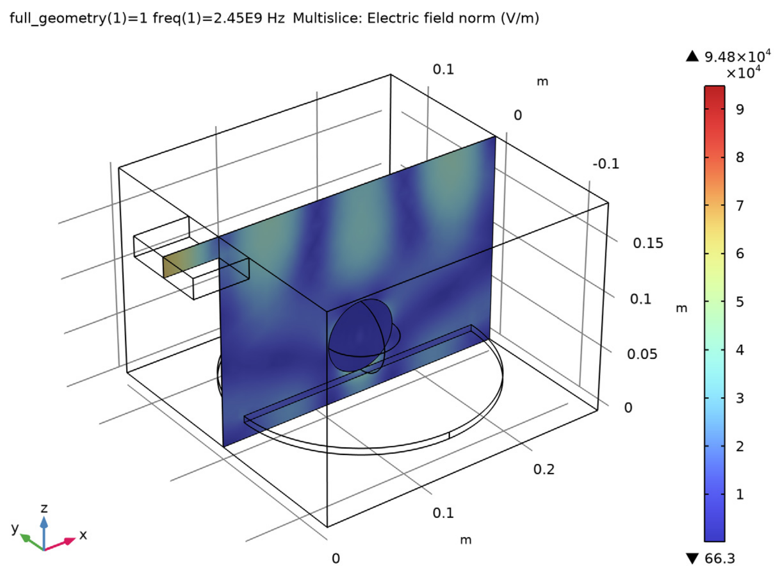
1 In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.

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**.



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

Volume 1

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 **Volume 1**.

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

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

4 Right-click **Volume 1** and choose **Delete**.

Temperature (ht)

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

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. Click **Change Color Table**.
7 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.
8 Click **OK**.

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**, locate the **Expression** section.
3 In the **X-component** text field, type `0`.
4 In the **Y-component** text field, type `0`.
5 In the **Z-component** text field, type `patcheval(emw.Ez,2)`. The `patcheval` operator ensures a smoother color distribution on the deformed plotting plane.
6 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.

Temperature (ht) and Ez

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

2 In the **Settings** window for **3D Plot Group**, type `Temperature (ht) and Ez` in the **Label** text field.

Create a plot showing the resistive heating on the symmetry plane.

Resistive Heating

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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)**.

Slice 1

In the **Model Builder** window, right-click **Resistive Heating** and choose **Slice**.

Slice 1

1 In the **Model Builder** window, expand the **Results>Resistive Heating** node, then click **Slice 1**.

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/Parametric Solutions 1 (sol3)**.

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 **Time selection** list, choose **First**.

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.

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 `wo/2`.

4 In the **Y** text field, type `0`.

5 In the **Z** text field, type `rpot+bp+hp`.

1D Plot Group 4

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

2 In the **Settings** window for **1D 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.

7 Select the **x-axis label** check box. In the associated text field, type `Time (s)`.

Point Graph 1

1. Right-click **1D 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 **1D 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 tree, select **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 1, Full Model**.
6. 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

Electric Field (emw), Half Model

In the **Settings** window for **3D Plot Group**, type `Electric Field (emw), Half Model` in the **Label** text field.

Multislice

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

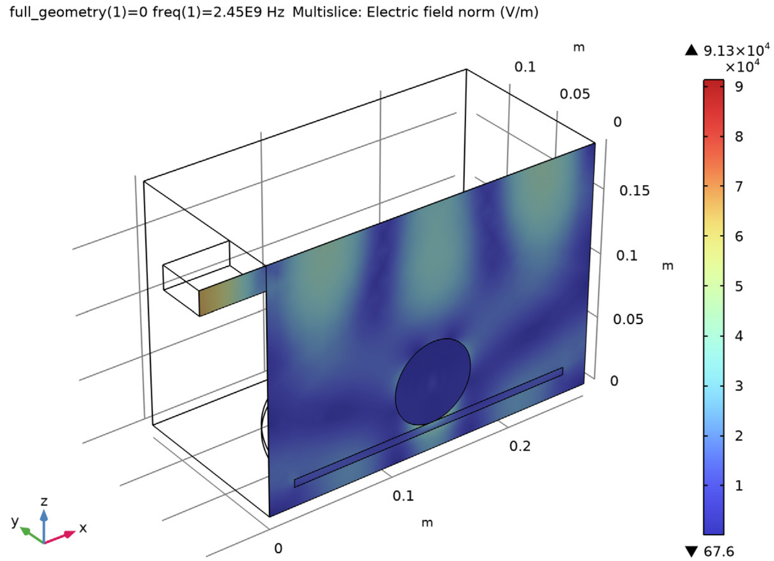
2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

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.



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

Temperature (ht)

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

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

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 3 (sol9)**.

Volume 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Volume 1**.

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

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

Slice 1

1 In the **Model Builder** window, right-click **Temperature (ht)** 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**, locate the **Expression** section.

3 In the **X-component** text field, type `0`.

4 In the **Y-component** text field, type `0`.

5 In the **Z-component** text field, type `patcheval(emw.Ez,2)`.

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

The plot is shown in Figure 4.

Temperature (ht) and Ez, Half Model

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

2 In the **Settings** window for **3D Plot Group**, type `Temperature (ht) and Ez, Half Model` in the **Label** text field.

Resistive Heating, Half Model

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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 (sol8)**.

Slice 1

In the **Model Builder** window, right-click **Resistive Heating, Half Model** and choose **Slice**.

Slice 1

1 In the **Model Builder** window, expand the **Results>Resistive Heating, Half Model** node, then click **Slice 1**.

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/Parametric Solutions 3 (sol9)**.

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

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 2

1 Go to the **Table 2** 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 7 (sol7)**.

4 Locate the **Point Data** section. In the **X** text field, type `wo/2`.

5 In the **Y** text field, type `0`.

6 In the **Z** text field, type `rpot+bp+hp`.

1D Plot Group 8

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

2 In the **Settings** window for **1D 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.
7. Select the **x-axis label** check box. In the associated text field, type `Time (s)`.

Point Graph 1

1. Right-click **1D 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 **1D 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.