

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

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

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

$$\beta = \frac{2\pi}{c} \sqrt{v^2 - v_c^2}$$

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) . 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)=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

- I In the Model Wizard window, click 间 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 \bigcirc 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 **M** Done.

STUDY I

Step 1: Frequency Domain

- I In the Model Builder window, under Study I click Step I: 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 I.
- 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

Block I (blk1)

- I 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)

- I 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 I (cyl1)

- I In the Geometry toolbar, click 问 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 I (sphI)

- I In the **Geometry** toolbar, click \bigoplus 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 I (unil)

- I In the Geometry toolbar, click 🛑 Booleans and Partitions and choose Union.
- 2 Click the **Select All** button in the **Graphics** toolbar.

Block 3 (blk3)

- I 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 틤 Build Selected.

Intersection 1 (int1)

- I In the Geometry toolbar, click pooleans 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

- I 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 I (mirl)

- I In the Geometry toolbar, click 📿 Transforms and choose Mirror.
- 2 Select the object intl 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

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

3 Select Domains 7 and 8 only.



Plate

- I 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

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



Port Boundary

- I In the **Definitions** toolbar, click **here 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

- I In the **Definitions** toolbar, click **here 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

- I 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

- I In the **Definitions** toolbar, click **herefore 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

- I 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)

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

Potato

- I 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.

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	65-20*j	I	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	I	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	Ср	3.64e3	J/(kg·K)	Basic

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

Glass

- I 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/m³	Basic
Heat capacity at constant pressure	Ср		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

- I 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)

- I 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 I, Full Model

- I In the Model Builder window, under Component I (compl) 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

- I 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)

- I In the Model Builder window, under Component I (compl) 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

I 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

- I 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)

- I In the Model Builder window, under Component I (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.

- I In the Model Builder window, under Component I (compl)>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 TO.

MESH I

In the Home toolbar, click 📗 Build Mesh.

STUDY I

Step 1: Frequency Domain

- I In the Model Builder window, under Study I click Step I: 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 I (compl)>Electromagnetic Waves, Frequency Domain (emw)>Port 2, Half Model.
- 5 Click 💋 Disable.
- 6 In the tree, select Component I (compl)>Electromagnetic Waves, Frequency Domain (emw)>Symmetry Plane I.
- 7 Click 🖉 Disable.

Step 2: Time Dependent

- I 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

- I 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

I 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 **9** Plot.

full_geometry(1)=1 freq(1)=2.45E9 Hz 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.

- I 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 Results>Temperature (ht)>Surface and choose Delete.

Temperature (ht)

- I 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 I/Parametric Solutions I (sol3).

Slice 1

- I In the Temperature (ht) toolbar, click 📜 Slice.
- In the Settings window for Slice, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 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

- I 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 I (compl)> 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

- I 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 I (compl)> 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

- I 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 **I** Plot.

Compare the created plot to Figure 4.

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

Resistive Heating

- I 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 I/Solution Store I (sol2).

Isosurface

- I In the Model Builder window, expand the Resistive Heating node.
- 2 Right-click Results>Resistive Heating>Isosurface and choose Delete.

Slice 1

- I In the Model Builder window, right-click Resistive Heating and choose Slice.
- In the Settings window for Slice, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 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 **O** Plot.

The dissipated microwave power distribution inside the microwave oven. It is plotted in Figure 2.

Volume Integration 1

I In the Results toolbar, click ^{8,85}_{e-12} 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 I/Parametric Solutions I (sol3).
- 4 Click Replace Expression in the upper-right corner of the Expressions section. From the menu, choose Component I (compl)>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

- I 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.

ID Plot Group 4

- I In the Results toolbar, click \sim 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 I.
- 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

- I 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 **ID Plot**.

The plot should now look like Figure 3.

ADD STUDY

- I In the Home toolbar, click 2 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

- I 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

- I In the Model Builder window, under Study 2 click Step I: 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 I (comp1)>Electromagnetic Waves, Frequency Domain (emw)>Port 1, Full Model.
- 6 Click 🕢 Disable.

Step 2: Time Dependent

- I 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

- I 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

- I In the Model Builder window, expand the Electric Field (emw) I 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 freq(1)=2.45E9 Hz 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.

Temperature (ht) 1

- I In the Model Builder window, under Results click Temperature (ht) I.
- 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).

Surface

- I In the Model Builder window, expand the 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

- I 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 I (compl)> 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 **V** Go to Default View button in the Graphics toolbar.

Deformation I

- I Right-click Slice I 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 I (compl)> Electromagnetic Waves, Frequency Domain>Electric>emw.Ex,emw.Ey,emw.Ez Electric field.
- 3 In the Temperature (ht) I toolbar, click 💿 Plot.

The plot is shown in Figure 4.

Resistive Heating Half Model

- I 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 (sol8).

Isosurface

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

Slice 1

- I In the Model Builder window, right-click Resistive Heating Half Model and choose Slice.
- In the Settings window for Slice, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
 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 **O** Plot.
- 6 Click the **V** Go to Default View button in the Graphics toolbar.

The created plot is shown in Figure 2.

Volume Integration 2

- I In the Results toolbar, click ^{8,85}_{e-12} 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 I (compl)>Heat Transfer in Solids>Heat sources>ht.Qtot Total heat source W/m³.
- 8 Click **= Evaluate**.

TABLE

I 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

I 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.

ID Plot Group 8

- I In the Results toolbar, click \sim 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.
- 7 Select the x-axis label check box. In the associated text field, type Time (s).

Point Graph 1

- I 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.