Created in COMSOL Multiphysics 6.1

Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis

Introduction

The response of a millimeter wave with frequencies of 35 GHz and 95 GHz is known to be very sensitive to water content. This model utilizes a low-power 35 GHz Ka-band millimeter wave and its reflectivity to moisture for non-invasive cancer diagnosis. Since skin tumors contain more moisture than healthy skin, it leads to stronger reflections on this frequency band. Hence the probe detects abnormalities in terms of S-parameters at the tumor locations. A circular waveguide at the dominant mode and a conically tapered dielectric probe are quickly analyzed, along with the probe's radiation characteristics, using a 2D axisymmetric model. Temperature variation of the skin and the fraction of necrotic tissue analysis are also performed as well.

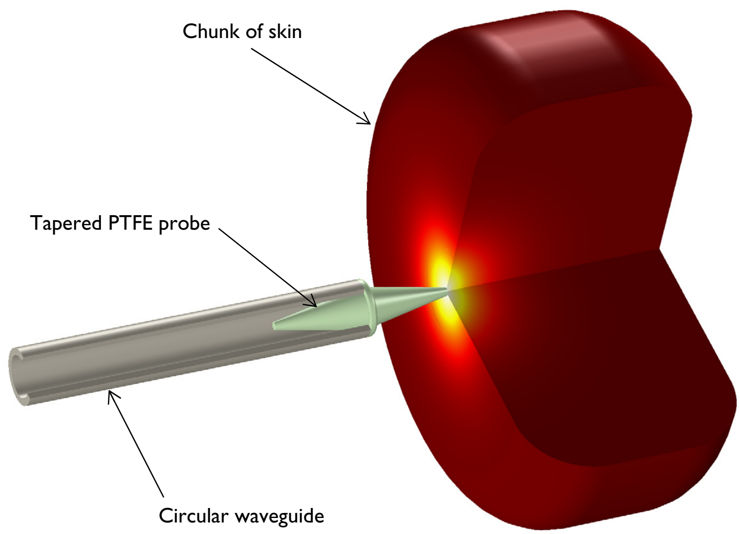


Figure 1: 3D visualization of the 2D axisymmetric model. The probe consists of a circular waveguide and a tapered dielectric rod.

Model Definition

The model consists of a metallic circular waveguide, a tapered PTFE dielectric rod, and a phantom of skin chunk shown in Figure 1. The entire model is enclosed by an air domain which is truncated at its outermost shell with perfectly matched layers (PML) to absorb any radiation directly from the rod or reflected from the skin phantom. One end of the waveguide is terminated with a circular port and excited using the dominant TE_{1m} mode, where m is the azimuthal mode number of this 2D axisymmetric model defined as 1 in the Electromagnetic Waves, Frequency Domain physics interface settings. The other end is connected to a tapered conical PTFE dielectric ($\varepsilon_r = 2.1$) rod. The shape of the rod is symmetrically tapered so the radius is increasing from the inside to the outside of the waveguide, then it is decreasing gradually for the impedance matching between the waveguide and the air domain. There is a ring structure in the middle to support the rod on the rim of the waveguide. The tip of the rod is touching the skin phantom.

The conductivity of the metallic waveguide is assumed to be high enough to neglect any loss and is modeled as perfect electric conductor (PEC). With the given radius of the waveguide and excited TE mode, the cutoff frequency is around 29.3 GHz, which is calculated by

$$f_{c_{ml}} = \frac{c_0 p'_{nm}}{2\pi a}$$

where c_0 is the speed of light, p'_{nm} are the roots of the derivative of the Bessel functions $J_n(x)$, m and n are the mode indices, and a is the radius of the waveguide. The value of p'_{11} is approximately 1.841. The operating frequency of the probe, 35 GHz, is higher than the waveguide cutoff frequency. The excited wave is propagating along the waveguide.

The circular port boundary condition is placed on the interior boundary where the reflection and transmission characteristics are computed automatically in terms of S-parameters. The interior port boundary with PEC backing for one-way excitation requires the slit condition. The port orientation is specified to define the inward direction for the S-parameter calculation.

First, the electromagnetic properties of the model are analyzed without a phantom to check the design validity of the waveguide and dielectric rod. Then, complexity is added, first with a healthy phantom, then a phantom with a skin tumor. See Table 1.

TABLE 1: MATERIAL PROPERTY VARIATION.

PROPERTY	PROBE ONLY	WITH A HEALTHY PHANTOM	TUMOR ADDED
Relative permittivity (imaginary part)	0	10	15
Relative permittivity (real part)	1	5	8

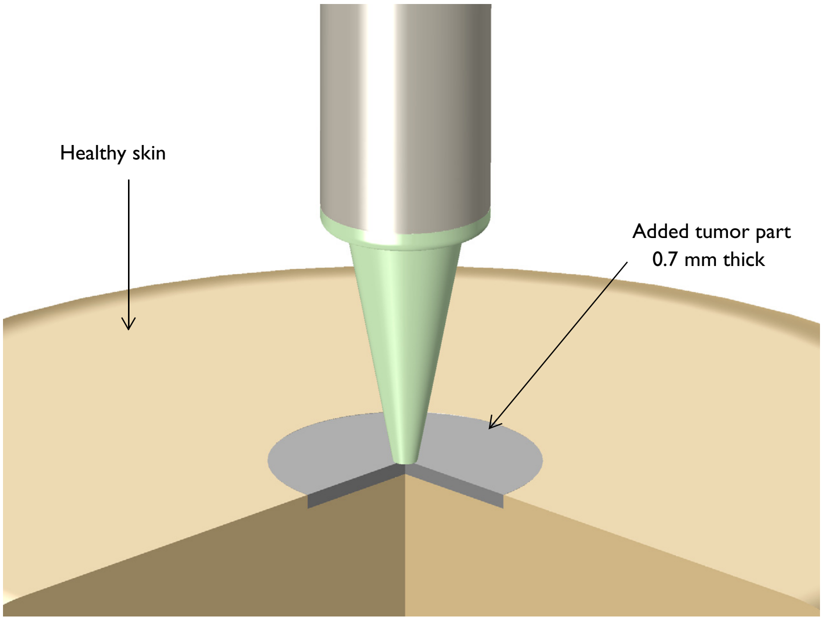


Figure 2: Zoomed 3D visualization of the skin tumor area. The entire probe model is simulated in a 2D axisymmetric space dimension. The measured S-parameters vary due to the different moisture content in each skin phantom.

Though the waveguide excited by low power is expected to be harmless, its effect on necrotic tissue is reviewed by studying Bioheat Transfer as well as temperature, over a 10 minute period.

Figure 3 shows the real part of the electric field $\mathbf{E}_\mathrm{r}$ excited from one end of the waveguide without a phantom. Its radiation pattern is visualized in the Modeling Instructions section.

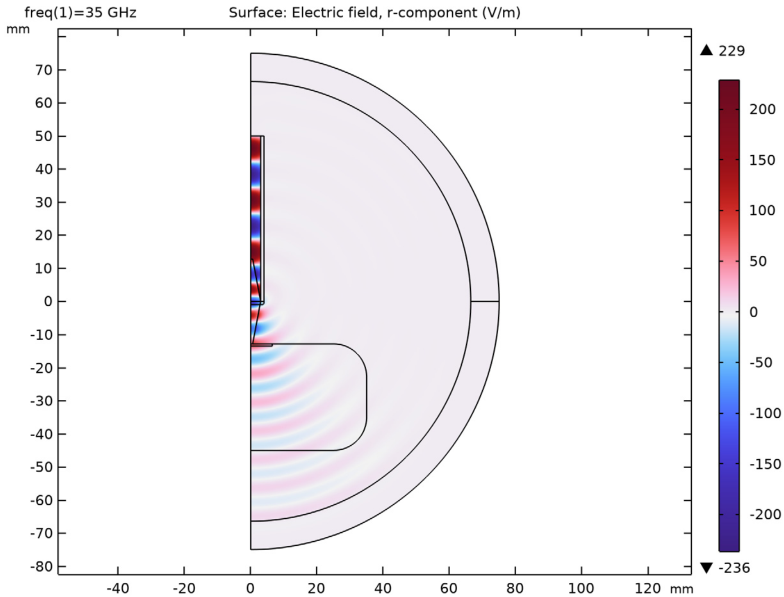


Figure 3: Wave propagation from the dielectric rod plotted with the E_r component of the E-field (probe-only case without a phantom).

Temperature change on the surface of the phantom with the tumor is plotted in Figure 4. Since the input power from the waveguide port is low, 1 mW, the temperature change is within 0.06 K even after 10 minutes of millimeter wave exposure. The color difference shows the relatively hotter spot where the temperature is still very close to the initial temperature, 34°C. Though the temperature analysis for the healthy phantom case is not included, it is easily expected that the temperature variation is less than the case with the tumor because the resistive loss should be lower due to the smaller imaginary part of the permittivity of the healthy skin. So the visualized temperature profile is the worst-case scenario of temperature increase among all three cases. The damaged tissue ratio is visualized in Figure 5. It shows that the effect of the low-power millimeter wave is negligible.

The computed S-parameters indicate more reflection when touching the skin with the tumor due to its higher moisture content, and they are approximately summarized below:

TABLE 2: S-PARAMETER RESPONSE OF THE PROBE.

	PROBE ONLY	WITH A HEALTHY PHANTOM	TUMOR ADDED
S_{11}	-29.4 dB	-9.83 dB	-8.97 dB

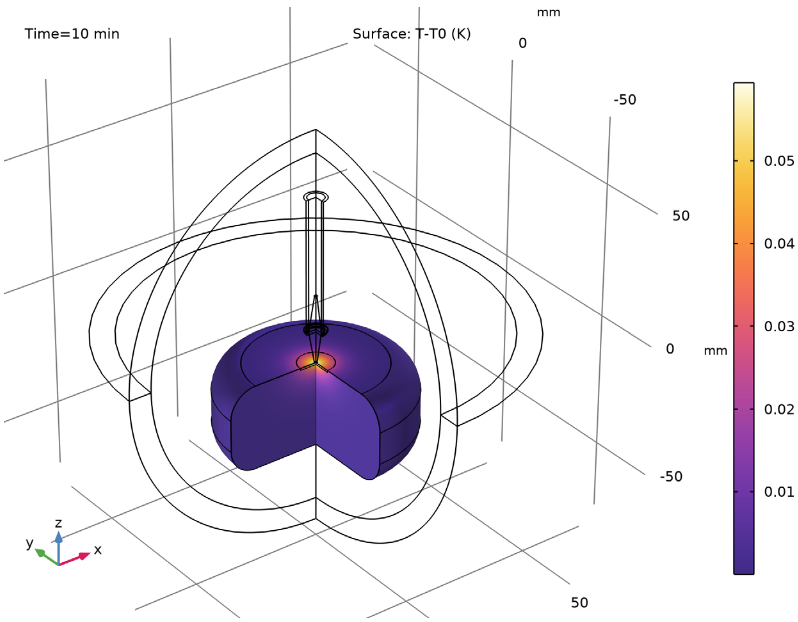


Figure 4: The temperature after 10 minutes. The variation compared to the initial temperature is negligible in the case where the tumor is added at the center of the center top of skin surface.

The modeling instructions show how to access the data plotted in Figure 6 which is not the dependent variable of the Electromagnetic Waves, Frequency Domain, by tweaking the solver settings.

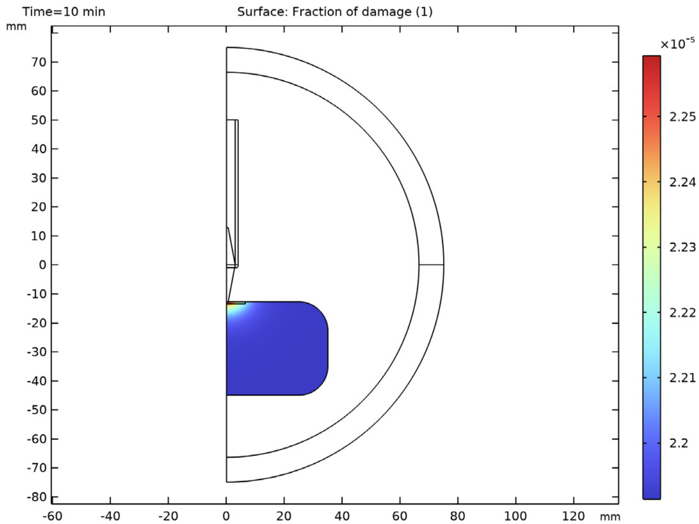


Figure 5: Fraction of necrotic (damaged) tissue is extremely low even after 10 minutes of millimeter wave exposure in the case where the tumor is added in contact with the probe, at the surface of the skin.

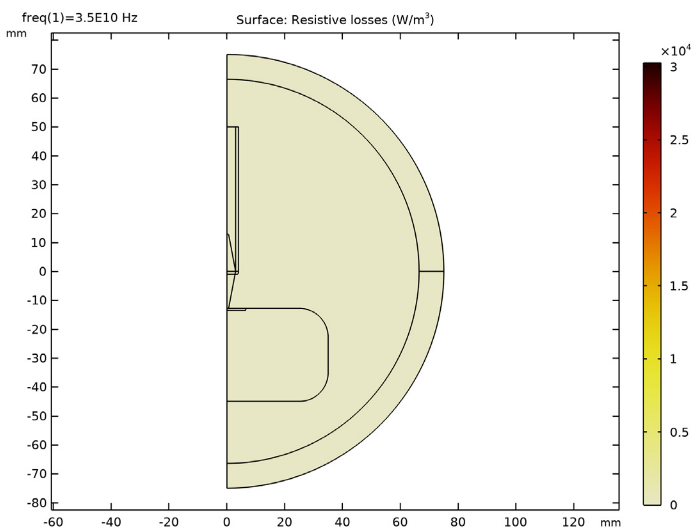


Figure 6: The resistive losses in the case where the tumor is added in contact with the probe, at the surface of the skin.

Notes About the COMSOL Implementation

The electromagnetic material properties of skin and tumor at 35 GHz are approximated to show the feasibility of the S-parameter method by detecting the areas with higher moisture content. For any further research, extracting accurate data in the given frequency range is recommended.

Application Library path: `RF_Module/Microwave_Heating/conical_dielectric_probe`

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 **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Frequency Domain**.

6 Click **Done**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
r1	0.003[m]	0.003 m	Waveguide radius
fc	1.841*c_const/2/pi/r1	2.928E10 1/s	Cutoff frequency

Name	Expression	Value	Description
f0	35[GHz]	3.5E10 Hz	Frequency
lda0	c_const/f0	0.0085655 m	Wavelength, free space
l_probe	12.8[mm]	0.0128 m	Tapered probe length
w1_probe	3[mm]	0.003 m	Tapered probe width1
w2_probe	0.58[mm]	5.8E-4 m	Tapered probe width2
T0	34[degC]	307.15 K	Initial skin temperature

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 f0.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Primitives** and choose **Circle**.

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

3 In the **Radius** text field, type 75.

4 In the **Sector angle** text field, type 180.

5 Locate the **Rotation Angle** section. In the **Rotation** text field, type 270.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	lda0

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type r1.

4 In the **Height** text field, type 50.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Height** text field, type `50`.
4 Locate the **Position** section. In the **r** text field, type `3`.

Bézier Polygon 1 (b1)

1 In the **Geometry** toolbar, click **Primitives** and choose **Bézier Polygon**.
2 In the **Settings** window for **Bézier Polygon**, locate the **Polygon Segments** section.
3 Find the **Added segments** subsection. Click **Add Linear**.
4 Find the **Control points** subsection. In row **1**, set **z** to `-l_probe`.
5 In row **2**, set **r** to `w2_probe` and **z** to `-l_probe`.
6 Find the **Added segments** subsection. Click **Add Linear**.
7 Find the **Control points** subsection. In row **2**, set **r** to `w1_probe` and **z** to `0`.
8 Find the **Added segments** subsection. Click **Add Linear**.
9 Find the **Control points** subsection. In row **2**, set **r** to `0`.
10 Find the **Added segments** subsection. Click **Add Linear**.
11 Find the **Control points** subsection. In row **2**, set **z** to `-l_probe`.
12 Click **Build Selected**.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the object **b1** 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 Line of Reflection** section. In the **r** text field, type `0`.
6 In the **z** text field, type `1`.
7 Click **Build Selected**.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `4`.
4 Locate the **Position** section. In the **z** text field, type `-1`.

Fillet 1 (fil1)

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

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

3 On the object **r3**, select Point 2 only.

4 In the **Settings** window for **Fillet**, locate the **Radius** section.

5 In the **Radius** text field, type `0.5`.

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `35`.

4 In the **Height** text field, type `32.2`.

5 Locate the **Position** section. In the **z** text field, type `-45`.

Fillet 2 (fil2)

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

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

3 On the object **r4**, select Points 2 and 3 only.

4 In the **Settings** window for **Fillet**, locate the **Radius** section.

5 In the **Radius** text field, type `10`.

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

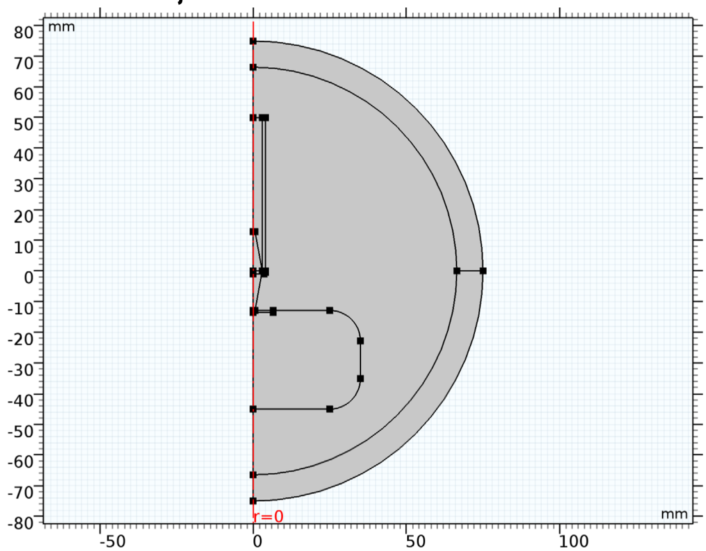
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `6.5`.

4 In the **Height** text field, type `0.7`.

5 Locate the **Position** section. In the **z** text field, type `-13.5`.

6 Click **Build All Objects**.

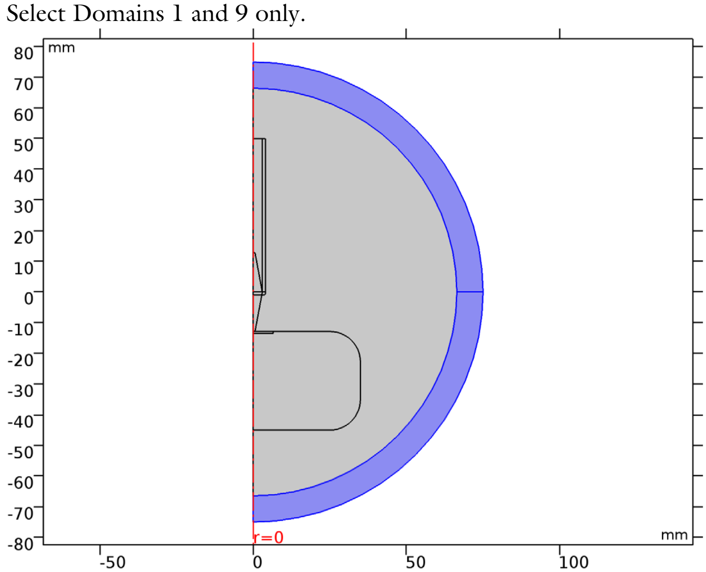


DEFINITIONS

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 9 only.



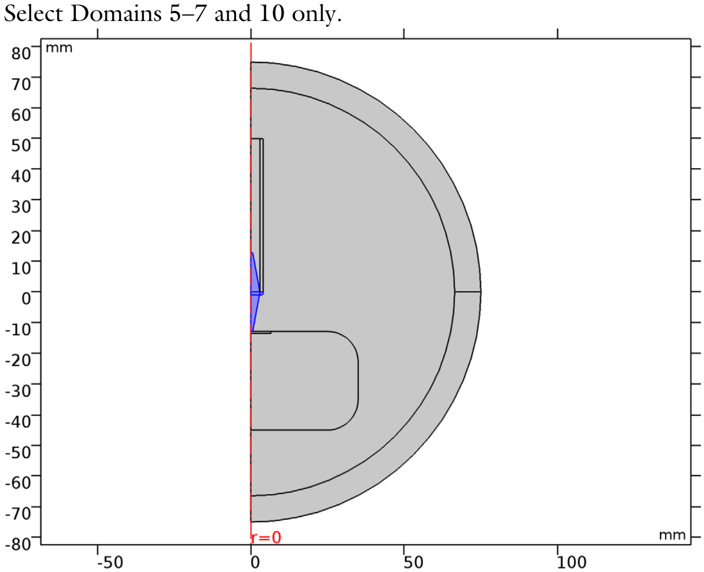
ADD MATERIAL

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.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Material 2 (mat2)

1. In the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for **Material**, type `PTFE` in the **Label** text field.
3. Select Domains 5–7 and 10 only.



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	2.1	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

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 **Out-of-Plane Wave Number** section.

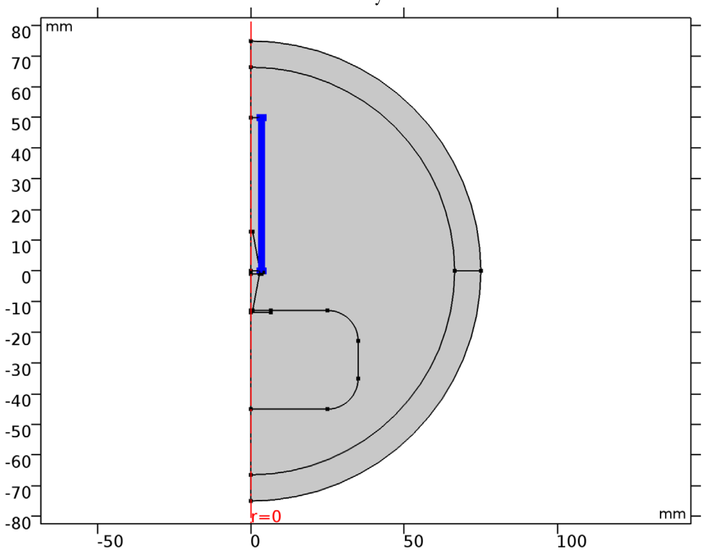
3 In the *m* text field, type 1.

Perfect Electric Conductor 2

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

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

3 Select Boundaries 23–25 and 27 only.



Port 1

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

2 Select Boundary 16 only.

3 In the **Settings** window for **Port**, locate the **Port Properties** section.

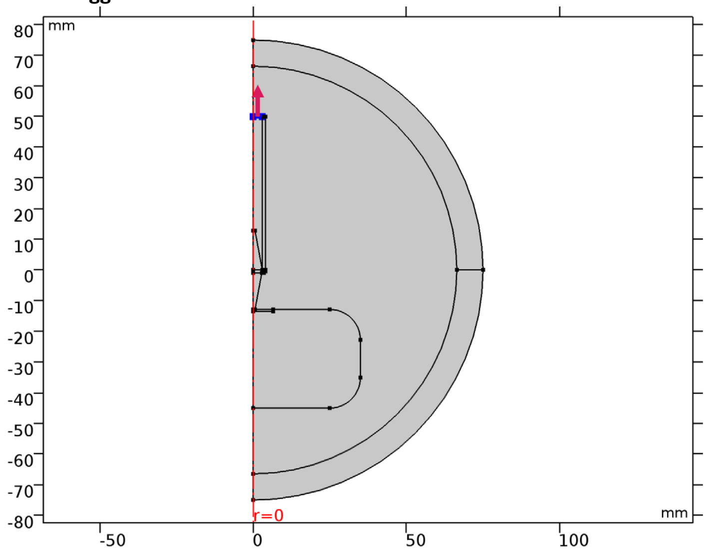
4 From the **Type of port** list, choose **Circular**.

5 In the P_{in} text field, type `1[mW]`.

The input power is `0 dBm`.

6 Select the **Activate slit condition on interior port** check box.

7 Click **Toggle Power Flow Direction**.

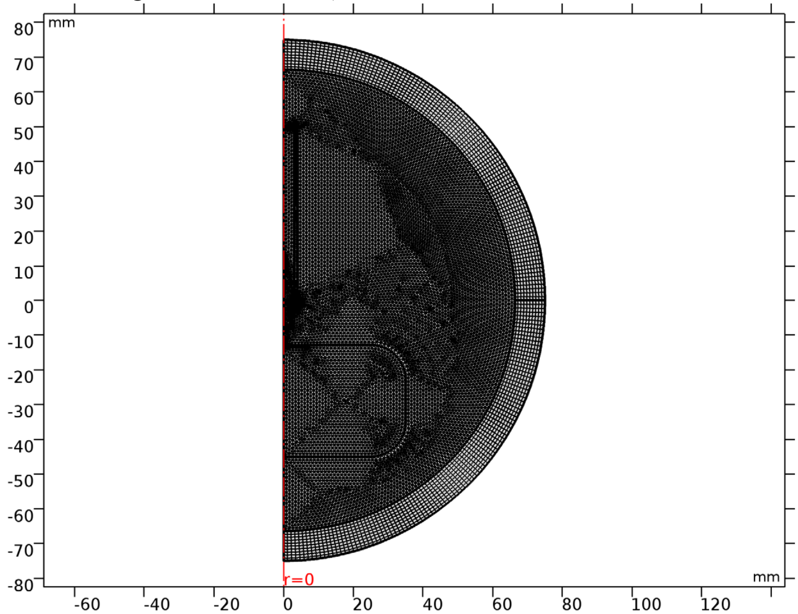


Far-Field Domain 1

In the **Physics** toolbar, click **Domains** and choose **Far-Field Domain**.

MESH 1

1 In the **Settings** window for **Mesh**, click **Build All**.



STUDY 1

Step 1: Frequency Domain

In the **Home** toolbar, click **Compute**.

RESULTS

Surface

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

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

3 In the **Expression** text field, type `emw.Er`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

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

See Figure 3 for the plot of the real part of E_r, showing wave propagation from the input port to the air domain via the tapered dielectric probe.

2D Far Field (emw)

In the **Settings** window for **Polar Plot Group**, type `Radiation Pattern, Polar` in the **Label** text field.

Radiation Pattern 1

1 In the **Model Builder** window, expand the **2D Far Field (emw)** node, then click **Results> Radiation Pattern, Polar>Radiation Pattern 1**.

2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.

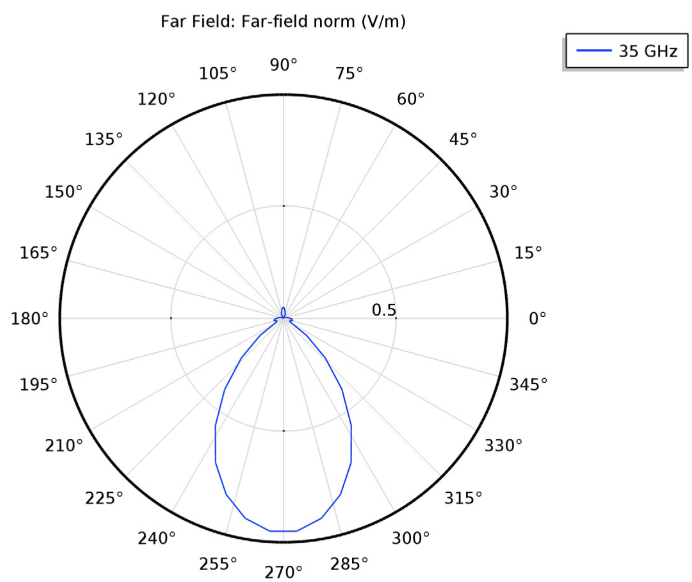
3 Find the **Reference direction** subsection. In the **y** text field, type `1`.

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

5 Find the **Normal** subsection. In the **x** text field, type `1`.

6 In the **y** text field, type `0`.

7 In the **Radiation Pattern, Polar** toolbar, click **Plot**.

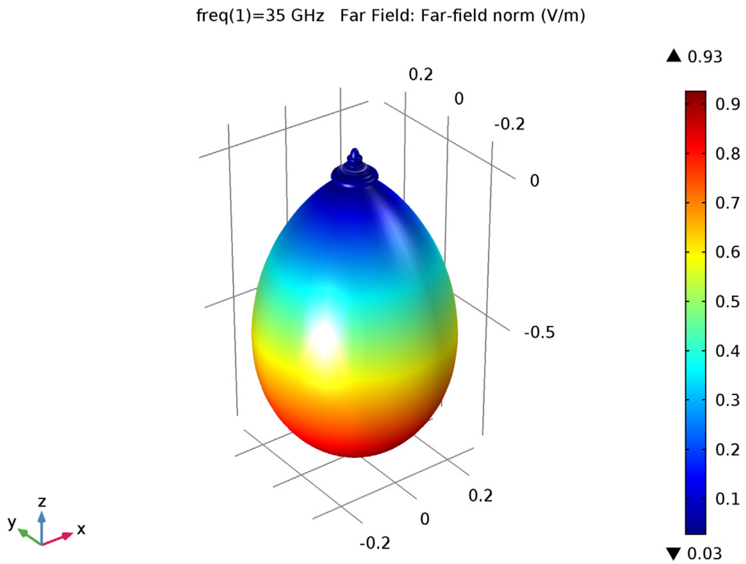


The Far-field pattern on the *yz*-plane shows the radiation from the tapered probe toward the bottom side.

3D Far Field (emw)

1 In the **Model Builder** window, expand the **Results>3D Far Field (emw)** node, then click **3D Far Field (emw)**.

2 In the **Settings** window for **3D Plot Group**, type `Radiation Pattern, 3D` in the **Label** text field.



The 3D far-field pattern is directed along the z-axis.

S-Parameter (emw)

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **S-Parameter (emw)**.

2 Click **Evaluate**.

The evaluated S-parameter is the input matching property of the circular waveguide without a human body phantom when the dominant mode is excited.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Add another Wave Equation which describe the human body phantom in term of dielectric loss using a complex permittivity.

Wave Equation, Electric 2

1 In the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

2 Select Domains 3 and 4 only.

3 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

4 From the **Electric displacement field model** list, choose **Dielectric loss**.

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 **Bioheat>Skin**.

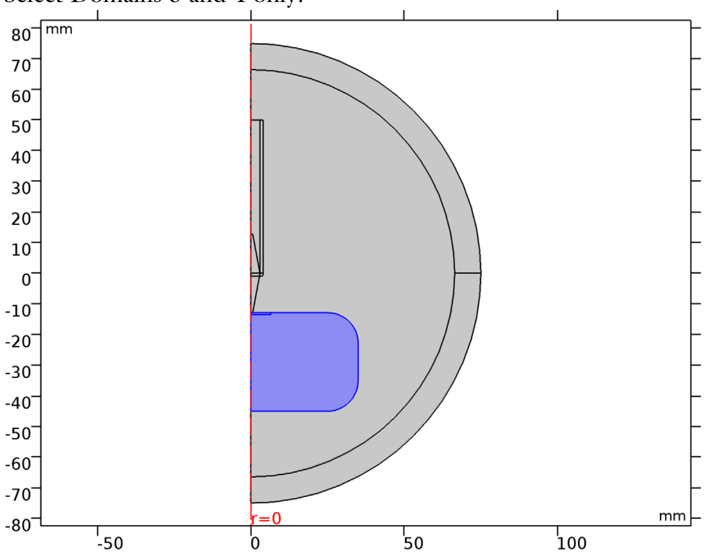
4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Skin (mat3)

1 Select Domains 3 and 4 only.



2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity (imaginary part)	epsilonBis_iso ; epsilonBisii = epsilonBis_iso, epsilonBisij = 0	10	1	Dielectric losses
Relative permittivity (real part)	epsilonPrim_iso ; epsilonPrimii = epsilonPrim_iso, epsilonPrimij = 0	5	1	Dielectric losses
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

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

RESULTS

S-Parameter (emw)

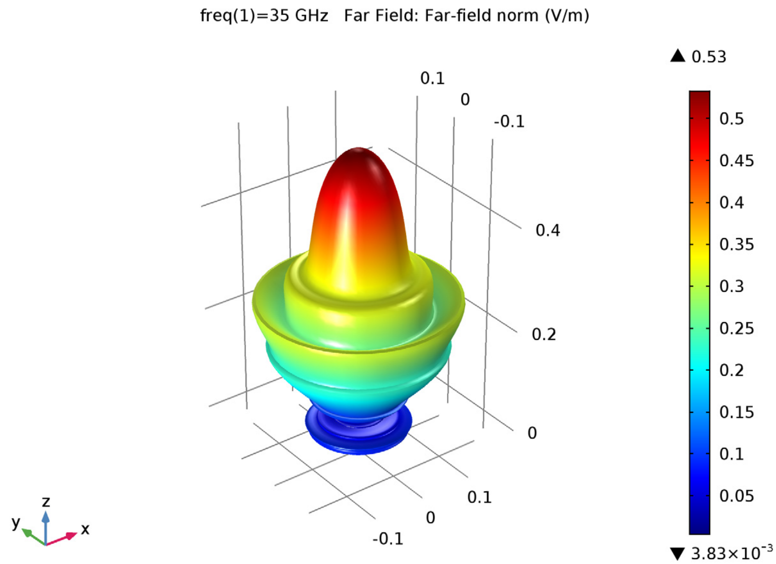
Evaluate the S-parameter assuming the probe is touching a phantom representing a healthy body.

1 In the **Model Builder** window, under **Results>Derived Values** click **S-Parameter (emw)**.

2 Click **Evaluate**.

Radiation Pattern, 3D

1 Click the **Zoom Extents** button in the **Graphics** toolbar.



Due to the body, the radiation is reflected back.

MATERIALS

Skin (mat3)

Now, add a tip of tumor skin.

Skin 1 (mat4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Skin (mat3)** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Skin Tumor` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. Click **Clear Selection**.

4 Select Domain 4 only.

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

Property	Variable	Value	Unit	Property group
Relative permittivity (imaginary part)	epsilonBis_iso ; epsilonBisii = epsilonBis_iso, epsilonBisij = 0	15	1	Dielectric losses
Relative permittivity (real part)	epsilonPrim_iso ; epsilonPrimii = epsilonPrim_iso, epsilonPrimij = 0	8	1	Dielectric losses

The effect of millimeter wave radiation on a human body will be investigated using the **Bioheat Transfer** physics interface.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Heat Transfer>Bioheat Transfer (ht)**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

BIOHEAT TRANSFER (HT)

1 In the **Settings** window for **Bioheat Transfer**, locate the **Domain Selection** section.

2 Click **Clear Selection**.

3 Select Domains 3 and 4 only.

Biological Tissue 1

In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (ht)** click **Biological Tissue 1**.

Thermal Damage 1

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Damage**.

2 In the **Settings** window for **Thermal Damage**, locate the **Damaged Tissue** section.

3 From the **Transformation model** list, choose **Arrhenius kinetics**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Bioheat Transfer (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`.

Open Boundary 1

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

2 Select Boundaries 4, 8, 19, 29, 30, 37, and 38 only.

MULTIPHYSICS

Electromagnetic Heating 1 (emh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global>Electromagnetic Heating**.

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>Sequential Frequency-Transient**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `f0`.

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 2: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **min**.

4 In the **Times** text field, type `range(0,15[s],10)`.

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 Select the **Store solution for all intermediate study steps** check box.

9 In the **Study** toolbar, click **Show Default Solver**.

RESULTS

Revolution 2D 2

1 In the **Results** toolbar, click **More Data Sets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type -90.

5 In the **Revolution angle** text field, type 270.

6 In the **Home** toolbar, click **Compute**.

S-Parameter (emw)

1 In the **Model Builder** window, under **Results>Derived Values** click **S-Parameter (emw)**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

4 Click **Evaluate**.

The computed S-parameter shows more reflection on the probe due to the skin tumor.

3D Plot Group 4

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

2 In the **Settings** window for **3D Plot Group**, type Temperature in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Revolution 2D 2**.

4 From the **Time (min)** list, choose **10**.

Surface 1

1 In the **Temperature** toolbar, click **Surface**.

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

3 In the **Expression** text field, type T-T0.

4 In the **Temperature** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

6 Click the **Zoom In** button in the **Graphics** toolbar.

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

The temperature variation in the skin is shown in Figure 4.

2D Plot Group 5

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Fraction of Necrotic Tissue` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 From the **Time (min)** list, choose **10**.

Surface 1

1 In the **Fraction of Necrotic Tissue** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Bioheat Transfer>Irreversible transformation>ht.theta_d - Fraction of damage**.

3 In the **Fraction of Necrotic Tissue** toolbar, click **Plot**.

4 Click the **Zoom In** button in the **Graphics** toolbar.

The reproduced plot addresses the fraction of necrotic tissue as shown in Figure 5.

2D Plot Group 6

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Resistive Losses` in the **Label** text field.

Resistive Losses

1 In the **Resistive Losses** toolbar, click **Surface**.

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

3 From the **Data set** list, choose **Study 2/Solution Store 1 (sol3)**.

Surface 1

1 In the **Model Builder** window, under **Results>Resistive Losses** click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qrh - Resistive losses - W/m³**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

4 Select the **Reverse color table** check box.

5 In the **Resistive Losses** toolbar, click **Plot**.

Finish the result analysis by regenerating Figure 6, the resistive losses plot.