



Band-Gap Analysis of a Photonic Crystal

This application performs a band-gap analysis of a photonic crystal similar to the one used in the [Photonic Crystal](#) model.

Introduction

The model investigates the wave propagation in a photonic crystal that consists of GaAs pillars placed equidistant from each other. The distance between the pillars determines a relationship between the wave number and the frequency of the light that prevents light of certain wavelengths to propagate inside the crystal structure. This frequency range is called the photonic band gap ([Ref. 2](#)). There are several band gaps for a certain structure, and this application extracts the band gaps for the lowest bands of the crystal.

Model Definition

This application is similar to the [Photonic Crystal](#) waveguide model. The difference is that in this application the crystal itself is analyzed instead of a waveguide. Because it has a repeated pattern it is possible to use periodic boundary conditions. As a result, only one pillar is needed for this simulation.

There are two main complications with this band-gap analysis. Firstly, the refractive index of GaAs is frequency dependent. Secondly, the wave vector must be ramped for the band diagram. Although you can solve each of these complications with the eigenvalue solver separately, the two complications combined make it difficult without reformulating the problem. Thus, formulate a nonlinear eigenvalue problem, using a stationary solver with the eigenvalue as an unknown. The equation for the eigenvalue is a normalization of the electric field, so the average field is unity over the domain. The nonlinear solver finds the correct eigenvalue with an updated refractive index to the found eigenvalue. Furthermore, the parametric solver can sweep the wave vector, k .

The wave vector for the propagating wave, k , enters the simulation as Floquet periodicity boundary conditions ([Ref. 1](#)),

$$E_z(2) = E_z(1)e^{-i\beta}$$

where β is a phase factor determined by the wave vector and the distance, d , between the periodic boundaries:

$$\beta = kd$$

The range for the swept k is determined by the reciprocal lattice vectors of the photonic crystal, and these are determined from the primitive lattice vectors. For a 2D crystal there are two lattice vectors, \mathbf{a}_1 and \mathbf{a}_2 , defined in [Figure 1](#).

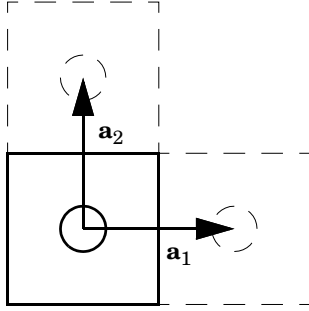


Figure 1: Definition of the square primitive cells and the lattice vectors \mathbf{a}_1 and \mathbf{a}_2 .

The reciprocal lattice vectors are calculated from \mathbf{a}_1 and \mathbf{a}_2 using the relations

$$\mathbf{b}_1 = 2\pi \frac{\mathbf{a}_2 \times \mathbf{a}_3}{\mathbf{a}_1 \cdot (\mathbf{a}_2 \times \mathbf{a}_3)}$$

$$\mathbf{b}_2 = 2\pi \frac{\mathbf{a}_3 \times \mathbf{a}_1}{\mathbf{a}_1 \cdot (\mathbf{a}_2 \times \mathbf{a}_3)}$$

where \mathbf{a}_3 is assumed to be the unit vector \mathbf{e}_z . When \mathbf{a}_1 and \mathbf{a}_2 are perpendicular to each other and to \mathbf{a}_3 , \mathbf{b}_1 and \mathbf{b}_2 become

$$\mathbf{b}_1 = 2\pi \frac{\mathbf{a}_1}{|\mathbf{a}_1||\mathbf{a}_1|}$$

$$\mathbf{b}_2 = 2\pi \frac{\mathbf{a}_2}{|\mathbf{a}_2||\mathbf{a}_2|}$$

Results and Discussion

Figure 2 shows the z -component of the electric field, as determined by the eigenfrequency solver for $k = 0$.

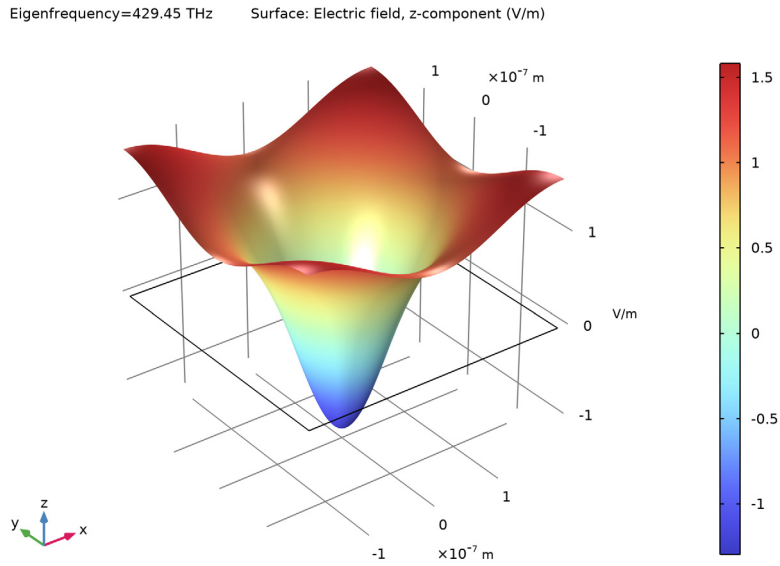


Figure 2: z -component of the electric field for $k = 0$.

Figure 3 shows the z -component of the electric field for $k = 0.5$ for the fifth band, as determined by the nonlinear solver.

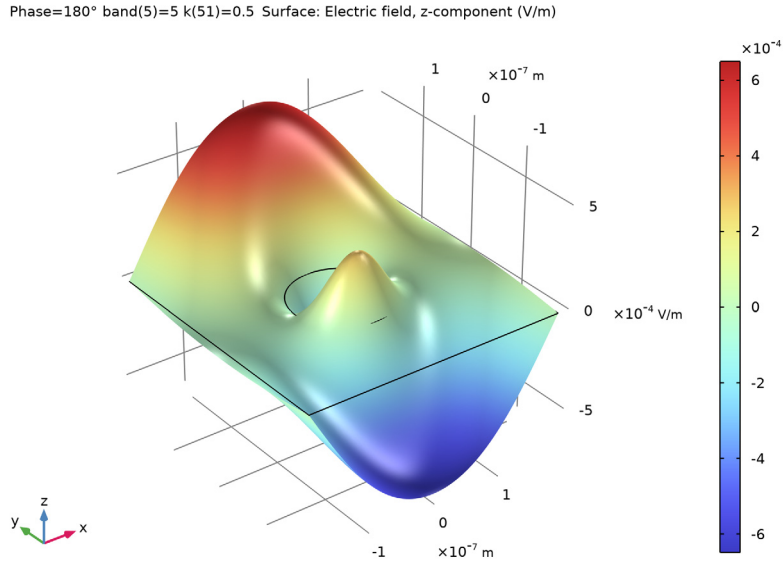


Figure 3: z -component of the electric field for the fifth band and $k = 0.5$.

Finally, Figure 4 shows the band diagram for k swept from 0 to 0.5 in the (1,1) direction. Notice that band two and three are degenerate for $k = 0$, and that band one and two and four and five are degenerate at $k = 0.5$.

Between band three and four there is a frequency range for which there are no states. This frequency range corresponds to a band gap in the structure, as there can be no propagating waves in the (1,1)-propagation direction for that frequency range.

Notice that there is actually a band with a lower frequency than for the lowest band in Figure 4. However, this band has so small frequencies that the approximation for the

frequency-dependent refractive index of GaAs is no longer valid. Thus, this band has not been included in the calculations.

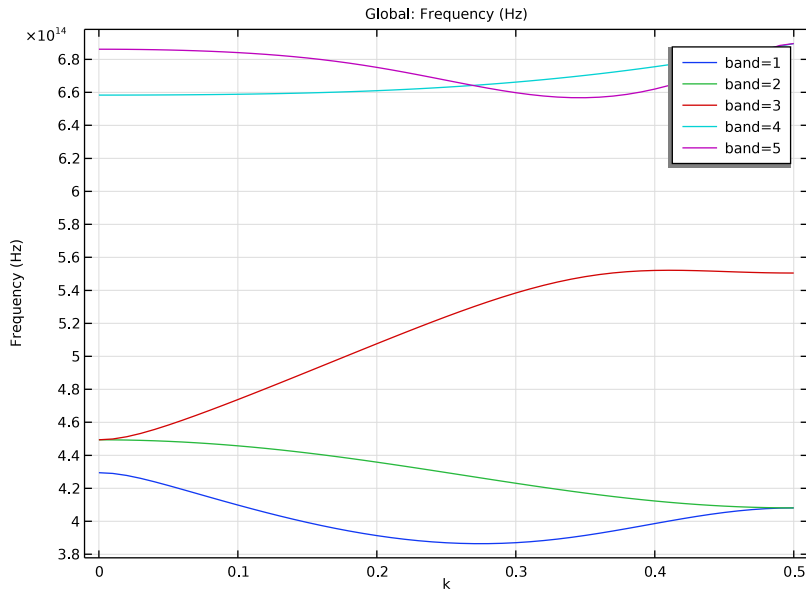


Figure 4: The dispersion relation (frequency versus wave number), when the wave vector is varied in the direction $(1,1)$, for the five lowest bands.

Notes About the COMSOL Implementation

One catch with the nonlinear formulation is that the mode normalization performed by the global equation involves setting the domain integral of $Ez \cdot \text{conj}(Ez)$ to unity. However, the conjugate function is nonanalytic in the complex analysis sense so a correct Jacobian cannot be obtained by the nonlinear solver.

To obtain a correct Jacobian, the complex splitting functionality of COMSOL Multiphysics is used, whereby complex entities are internally split into the constituent real and imaginary parts. Thus, the problem is converted from a complex-valued problem to a real-valued problem that is fully differentiable. After solution, COMSOL Multiphysics translates the solution back into complex form.

The complex splitting makes the problem solve faster and more robustly.

References


1. C. Kittel, *Introduction to Solid State Physics*, 7th ed., John Wiley & Sons, New York, 1996.
2. J.D. Joannopoulos, R.D. Meade, and J.N. Winn, *Photonic Crystals (Modeling the Flow of Light)*, Princeton University Press, 1995.

Application Library path: Wave_Optics_Module/Gratings_and_Metamaterials/
bandgap_photonic_crystal




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**.
- 2 In the **Select Physics** tree, select **Optics>Wave Optics>Electromagnetic Waves, Frequency Domain (ewfd)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Eigenfrequency**.
- 6 Click  **Done**.

GLOBAL DEFINITIONS

Parameters 1

First add parameters, characterizing the geometry of the periodic cell.

- 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
a	375[nm]	3.75E-7 m	Primitive cell side length
b	70[nm]	7E-8 m	GaAs pillar radius
k	0	0	Fraction of wave vector magnitude
k1	1	l	First component of wave direction vector
k2	1	l	Second component of wave direction vector
a1x	a	3.75E-7 m	First lattice vector, x-component
a1y	0[nm]	0 m	First lattice vector, y-component
a2x	0[nm]	0 m	Second lattice vector, x-component
a2y	a	3.75E-7 m	Second lattice vector, y-component
band	1	l	Band number

The last parameter, band, will be used for selecting what band to calculate the dispersion relation for.

DEFINITIONS

Variables l

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Add variables representing the reciprocal lattice vector and the Floquet wave vector (used later in the periodic boundary condition).

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

3 In the table, enter the following settings:


Name	Expression	Unit	Description
b1x	$2\pi a_2y / (a_1x a_2y - a_1y a_2x)$	l/m	First reciprocal lattice vector, x-component
b1y	$-2\pi a_2x / (a_1x a_2y - a_1y a_2x)$	l/m	First reciprocal lattice vector, y-component
b2x	$-2\pi a_1y / (a_1x a_2y - a_1y a_2x)$	l/m	Second reciprocal lattice vector, x-component
b2y	$2\pi a_1x / (a_1x a_2y - a_1y a_2x)$	l/m	Second reciprocal lattice vector, y-component

Name	Expression	Unit	Description
kx	$k*(k1*b1x+k2*b2x)$	l/m	Floquet vector, x-component
ky	$k*(k1*b1y+k2*b2y)$	l/m	Floquet vector, y-component




GEOMETRY I

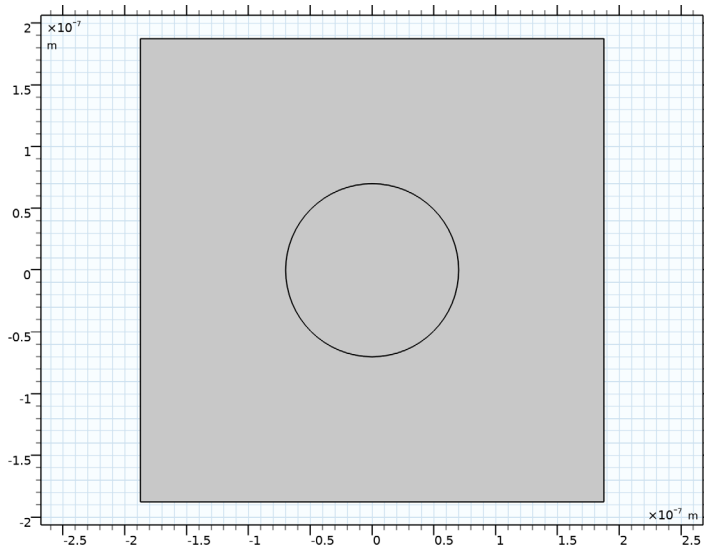
The geometry consists of a square air cell surrounding a circular GaAs pillar.

Square 1 (sq1)



- 1 In the **Geometry** toolbar, click  **Square**.
- 2 In the **Settings** window for **Square**, locate the **Size** section.
- 3 In the **Side length** text field, type a.
- 4 Locate the **Position** section. From the **Base** list, choose **Center**.

Circle 1 (c1)

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type b.
- 4 Click  **Build All Objects**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.




ADD MATERIAL

- 1 In the **Home** toolbar, click  **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
Define the air, that will surround the GaAs pillar.
- 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.

DEFINITIONS

Analytic 1 (an1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global>Analytic**.
The dispersion relation for the refractive index of GaAs will be used in more than one place, so it is best to define it as an analytical function.
- 2 In the **Settings** window for **Analytic**, type n_{GaAs} in the **Function name** text field.
- 3 Locate the **Definition** section. In the **Expression** text field, type $-3.3285e5[1/m] * c_{\text{const}}/f+3.8359$.
- 4 In the **Arguments** text field, type f .
- 5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
f	Hz


- 6 In the **Function** text field, type 1.


ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

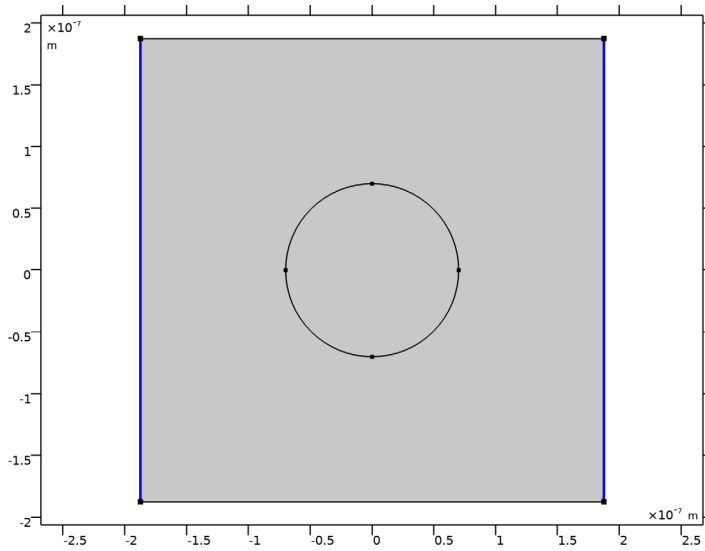
Compute the solution for out-of-plane polarization.

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (ewfd)**.
- 2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Components** section.
- 3 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Periodic Condition 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Condition**.
Define the periodic boundary conditions, using the Floquet wave vector.

- 2 Click the  **Zoom Extents** button in the **Graphics** toolbar.
- 3 Select Boundaries 1 and 4 only.



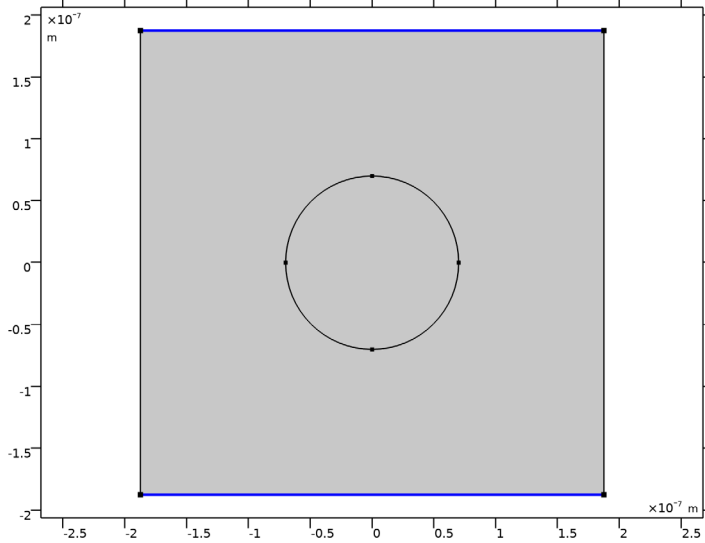
- 4 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.
- 5 From the **Type of periodicity** list, choose **Floquet periodicity**.
- 6 Specify the \mathbf{k}_F vector as

k_x	x
k_y	y

Periodic Condition 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 2 and 3 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

5 Specify the \mathbf{k}_F vector as

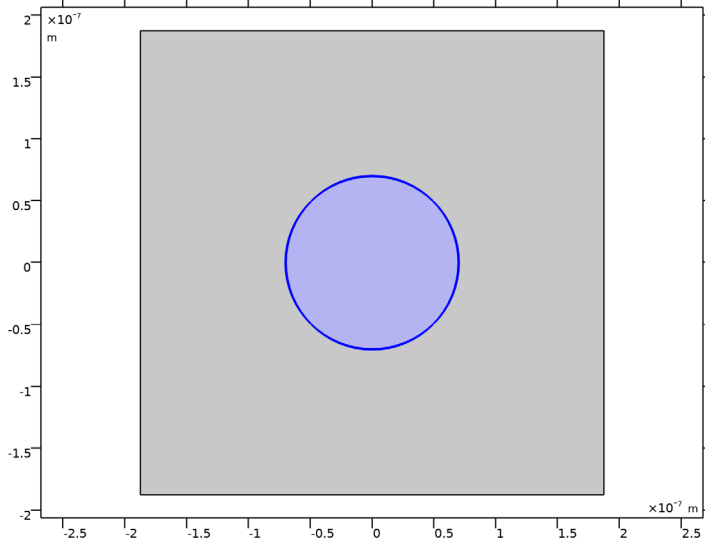
k_x	x
k_y	y

Add a wave equation feature, representing the GaAs pillar.

Wave Equation, Electric 2

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

2 Select Domain 2 only.



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

4 From the n list, choose **User defined**. In the associated text field, type `n_GaAs(freq)`.

5 From the k list, choose **User defined**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

Define the mesh so that the meshes on opposite edges are identical. This is important for the periodic boundary conditions, so that the source and destination meshes are the same.


Edge 1

1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.


2 Select Boundaries 1 and 2 only.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations> Copy Edge**.

- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.
- 4 Click to select the  **Activate Selection** toggle button.
- 5 Select Boundary 4 only.

Copy Edge 2

- 1 Right-click **Mesh 1** and choose **Copying Operations>Copy Edge**.
- 2 Select Boundary 2 only.
- 3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.
- 4 Click to select the  **Activate Selection** toggle button.
- 5 Select Boundary 3 only.

Free Triangular 1

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, click  **Build All**.



STUDY 1

Step 1: Eigenfrequency

Find the eigenfrequencies, for $k = 0$, around 550 THz.

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.
- 2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
- 3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type 5.
- 4 In the **Search for eigenfrequencies around shift** text field, type 550[THz].

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
Define the Transform point to the initial eigenfrequency guess, to make sure that a zero frequency will not be assigned to the denominator in any expression.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Eigenvalue Solver 1**.
- 3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.
- 4 Find the **Eigenvalue linearization point** subsection. In the **Value of eigenvalue linearization point** text field, type 550[THz].
- 5 In the **Study** toolbar, click  **Compute**.

RESULTS


Electric Field (ewfd)

Visualize the z component of the electric field, deforming the surface using a height expression.

Surface 1

- 1 In the **Model Builder** window, expand the **Electric Field (ewfd)** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type $ewfd.Ez$.



Height Expression 1

- 1 Right-click **Surface 1** and choose **Height Expression**.
- 2 Click the  **Zoom Extents** button in the **Graphics** toolbar. Compare the results with [Figure 2](#).

COMPONENT 1 (COMP1)

To find the frequencies for the nonzero wave vectors, solve for the frequency using a nonlinear state equation, requiring the field to be normalized in the unit cell, and a second wave equation. The solution from the first wave equation is used as the initial value for the second wave equation.

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 **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.
- 4 Click **Add to Component 1** in the window toolbar.
- 5 In the tree, select **Recently Used>Electromagnetic Waves, Frequency Domain (ewfd)**.
- 6 Click **Add to Component 1** in the window toolbar.
- 7 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

GLOBAL ODES AND DAES (GE)




Global Equations 1 (ODE1)

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1 (ODE1)**.
- 2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	$f(u, ut, utt, t)$ (1)	Initial value (u_0) (1)	Initial value (u_{t0}) (1/s)	Description
freq1	$1[V^2/m^2] - nEz$	$-\text{imag}(\lambda) / (2\pi)$	0	Frequency

Here, the initial value for the frequency `freq1` is taken from the eigenvalue from the initial eigenfrequency analysis.

- 4 Locate the **Units** section. Click  **Select Dependent Variable Quantity**.
- 5 In the **Physical Quantity** dialog box, type `frequency` in the text field.
- 6 Click  **Filter**.
- 7 In the tree, select **General>Frequency (Hz)**.
- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click  **Define Source Term Unit**.
- 11 In the **Source term quantity** table, enter the following settings:

Source term quantity	Unit
Custom unit	V^2/m^2

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EWFD2)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain 2 (ewfd2)**.
- 2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, click to expand the **Equation** section.
The frequency used by this interface is the one solved for with the state equation, `freq1`.
- 3 From the **Equation form** list, choose **Frequency domain**.
- 4 From the **Frequency** list, choose **User defined**. In the f text field, type `freq1`.
- 5 Locate the **Components** section. From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Initial Values 1

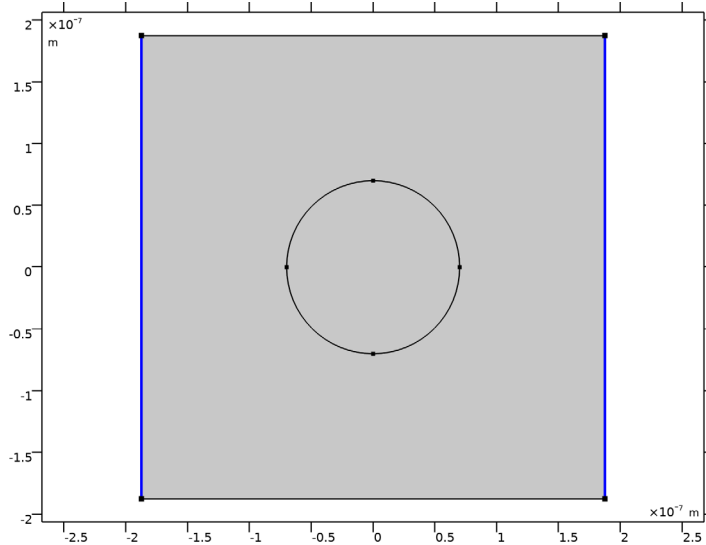
Assign the initial value to the solution from the initial eigenfrequency analysis. As more than one eigenfrequency solution was found, you will later specify, for the Stationary study step, which of the eigenfrequency solutions to select for the initial value.

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain 2 (ewfd2)** click **Initial Values 1**.
- 2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
- 3 Specify the $\mathbf{E2}$ vector as

0	x
0	y
ewfd.Ez	z

Periodic Condition 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Condition**.
- 2 Select Boundaries 1 and 4 only.



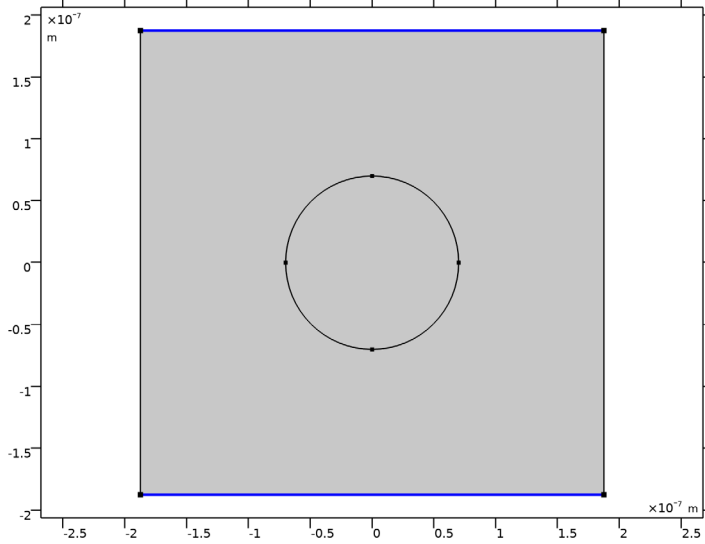
- 3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.
- 4 From the **Type of periodicity** list, choose **Floquet periodicity**.
- 5 Specify the \mathbf{k}_F vector as

kx	x
ky	y

Periodic Condition 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Condition**.

2 Select Boundaries 2 and 3 only.



3 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

4 From the **Type of periodicity** list, choose **Floquet periodicity**.

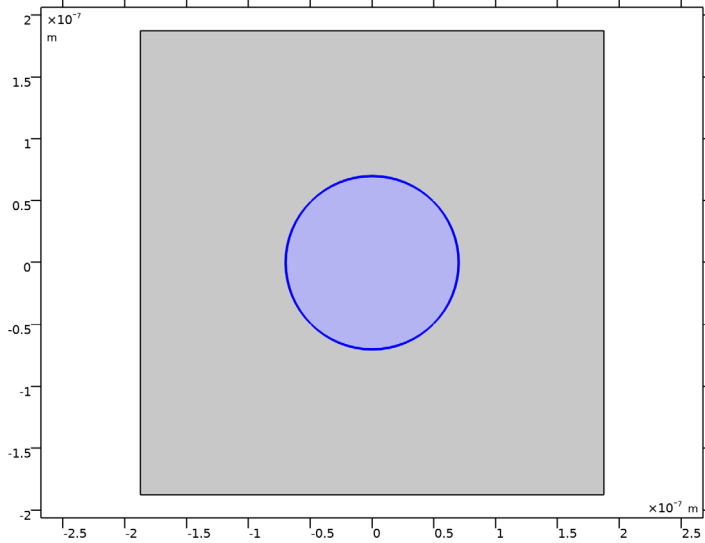
5 Specify the \mathbf{k}_F vector as

k_x	x
k_y	y

Wave Equation, Electric 2

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

2 Select Domain 2 only.



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

4 From the n list, choose **User defined**. In the associated text field, type $n_{\text{GaAs}}(\text{freq1})$. Notice that it must be the frequency freq1 that is used here.

5 From the k list, choose **User defined**.

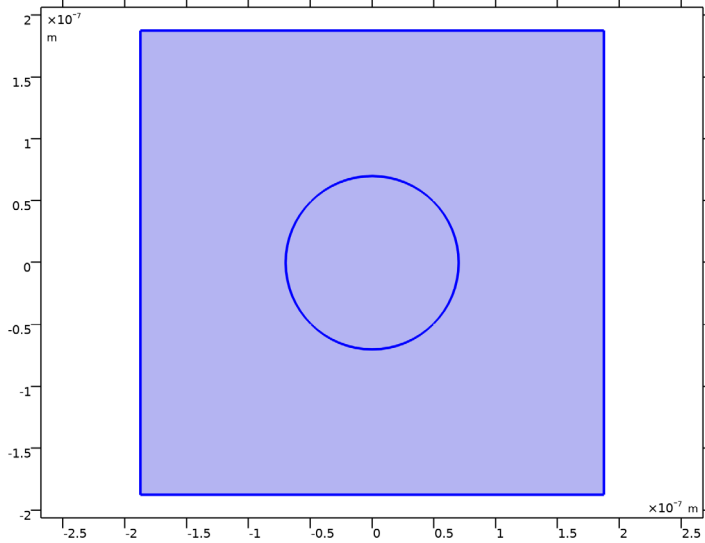
DEFINITIONS

Define the integration operator, used for the normalization of the field.

Integration I (intopI)

1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.

2 Click in the **Graphics** window and then press Ctrl+A to select both domains.





Variables 1

Add the variables that define the normalization of the field.

- 1 In the **Model Builder** window, click **Variables 1**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
A	intop1(1)	m ²	Area
nEz	intop1(real(dot(ewfd2.Ez, ewfd2.Ez)))/A	kg ² ·m ² / (s ⁶ ·A ²)	Normalization integral

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 Some Physics Interfaces>Stationary**.
- 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: Stationary

The stationary study will not include the first interface (**ewfd**).

Select what eigenfrequency solution for the first Electromagnetic Waves, Frequency Domain interface (**ewfd**) that will be used as the initial value to the second Electromagnetic Waves, Frequency Domain (**ewfd2**) interface.


- 1 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
- 2 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 3 From the **Study** list, choose **Study 1, Eigenfrequency**.
- 4 From the **Eigenfrequency** list, choose **Manual**.
- 5 In the **Index** text field, type **band**.
- 6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 7 Click **+ Add**.
Scan the fraction of the wave vector magnitude from 0 to 0.5 (half the Brillouin zone).
- 8 From the list in the **Parameter name** column, choose **k (Fraction of wave vector magnitude)**.
- 9 Click **Range**.
- 10 In the **Range** dialog box, type 0 in the **Start** text field.
- 11 In the **Step** text field, type 0.01.
- 12 In the **Stop** text field, type 0.5.
- 13 Click **Replace**.

Solution 2 (sol2)

- 1 In the **Study** toolbar, click **Show Default Solver**.
Solve the stationary problem using complex splitting, to split the complex expressions into their real and imaginary parts. Thereby the expressions become analytical and an exact Jacobian can be calculated.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Compile Equations: Stationary**.
- 3 In the **Settings** window for **Compile Equations**, locate the **Study and Step** section.
- 4 Select the **Split complex variables in real and imaginary parts** check box.

GLOBAL ODES AND DAES (GE)



Global Equations I (ODEI)

- 1 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 2 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
- 3 Click **OK**.
- 4 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations I (ODEI)**.
- 5 In the **Settings** window for **Global Equations**, click to expand the **Discretization** section.
- 6 From the **Value type when using splitting of complex variables** list, choose **Real**, as the frequency `freq1` has no imaginary part.

STUDY 2

Add a parametric sweep to calculate the dispersion relations for the five lowest bands.

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 From the list in the **Parameter name** column, choose **band (Band number)**.
- 5 Click  **Range**.
- 6 In the **Range** dialog box, type 1 in the **Start** text field.
- 7 In the **Step** text field, type 1.
- 8 In the **Stop** text field, type 5.
- 9 Click **Replace**.

Solution 2 (sol2)

- 1 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Parametric 1**.
- 2 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 3 Select the **Tuning of step size** check box, to make sure the solver takes small enough steps when starting the sweep.
- 4 In the **Initial step size** text field, type 0.0001.
- 5 In the **Minimum step size** text field, type 0.0001.
- 6 In the **Maximum step size** text field, type 0.01.

7 From the **Predictor** list, choose **Constant**. This makes the solver first try with the solution found for the previously calculated k value. This is preferred, as even though the states can be degenerate, the field solutions are orthogonal. Thus, the solver is forced to follow the right band.

8 In the **Study** toolbar, click  **Compute**.

RESULTS

Electric Field (ewfd2)

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

2 From the **Solution at angle (phase)** list, choose **Manual**.

3 In the **Phase** text field, type 180, to make the plot look better.

Surface 1


1 In the **Model Builder** window, expand the **Electric Field (ewfd2)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type $ewfd2.Ez$.

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

2 Click the  **Zoom Extents** button in the **Graphics** toolbar. Compare the results with [Figure 3](#).

ID Plot Group 3

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

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

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol3)**.

Global 1

1 Right-click **ID Plot Group 3** and choose **Global**.

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
band=1
band=2
band=3

Legends

band=4

band=5

5 In the **ID Plot Group 3** toolbar, click  **Plot**. Compare the results with [Figure 4](#).