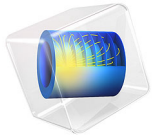


Optimization of a Photonic Crystal for Demultiplexing

Introduction

Photonic crystal devices are periodic structures of alternating layers of materials with different refractive indices. This model demonstrates how to apply shape optimization to a photonic crystal. The objective function is to maximize the output power ratio between for two narrow frequency bands, while constraining the loss from below. This is achieved by letting GaAs pillars change position but not shape. The implementation makes use of the **Free Shape Domain** and **Transformation** features, so that gradient-based optimization can be applied.

Model Definition

The objective function, Φ is defined in terms of the average magnitude of the output powers for a given wavelength, λ:

$$\Phi = \max(\varphi(\lambda_1), \varphi(\lambda_2), ..., \varphi(\lambda_i))$$

$$\left[\varphi(\lambda) = \begin{cases} \Phi_1(\lambda)/\Phi_2(\lambda) \text{ , for } f < (f_1 + f_2)/2 \\ \Phi_2(\lambda)/\Phi_1(\lambda) \text{ , for } (f_1 + f_2)/2 \le f \end{cases} \right.$$

$$\left. \varphi_1(\lambda) = \int_{\text{port 1}} P_{\text{out}} ds \quad \varphi_2(\lambda) = \int_{\text{port 2}} P_{\text{out}} ds \right]$$

The objective thus it to minimize the maximum of a list of objective functions. The MMA optimization solver is well suited to such problems. The constraint, ψ, is formulated as

$$(\varphi_1(\lambda) + \varphi_2(\lambda))/\psi_0 \le \psi(\lambda)$$

The topology of the mesh is fixed to allow for gradient-based optimization. To simplify manufacturing, the shape of the cylinder is also fixed. Thus, the only thing that is allowed to change is the cylinder positions. If they are allowed to move far, they might collide and cause error messages about inverted elements or NaN/Inf values. To avoid this, the cylinders are constrained to move 50 nm in the x and y directions.

Figure 1 and Figure 2 show the z component of the electric field in the optimized geometry for the two of the wavelengths.

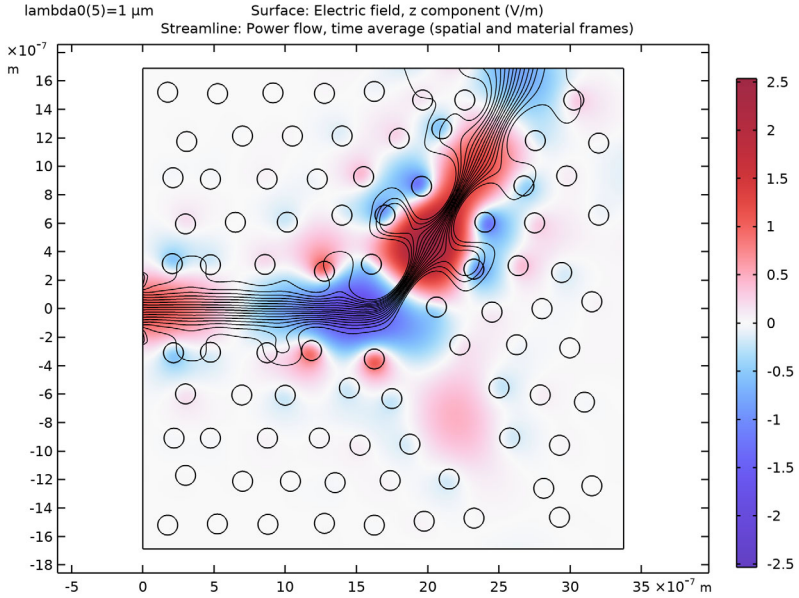


Figure 1: The z component of the electric field for the lower frequency band. The wave propagates to the lower output.

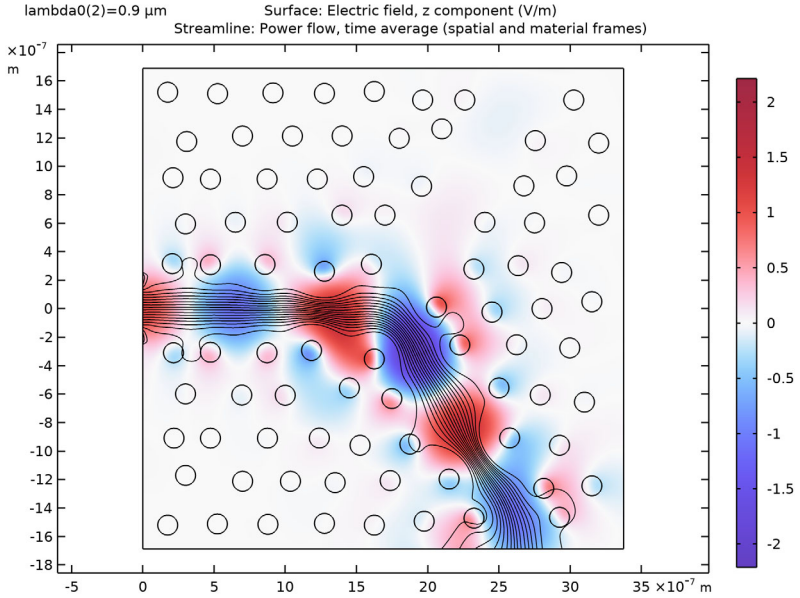


Figure 2: The z component of the electric field for the higher frequency band. The wave propagates to the upper output.

The optimization is able to achieve an output power ratio of less than 1 %. The constraints is not satisfied initially, so this takes priority in the beginning of the optimization. Figure 3 shows that it is satisfied in the end ($\psi = 0.25$ nW/m). The line in the graph is based on an analysis, where the mesh has been regenerated in the deformed configuration. The purpose of this is to ensure that the optimization result does not rely on unphysical numerical effects.

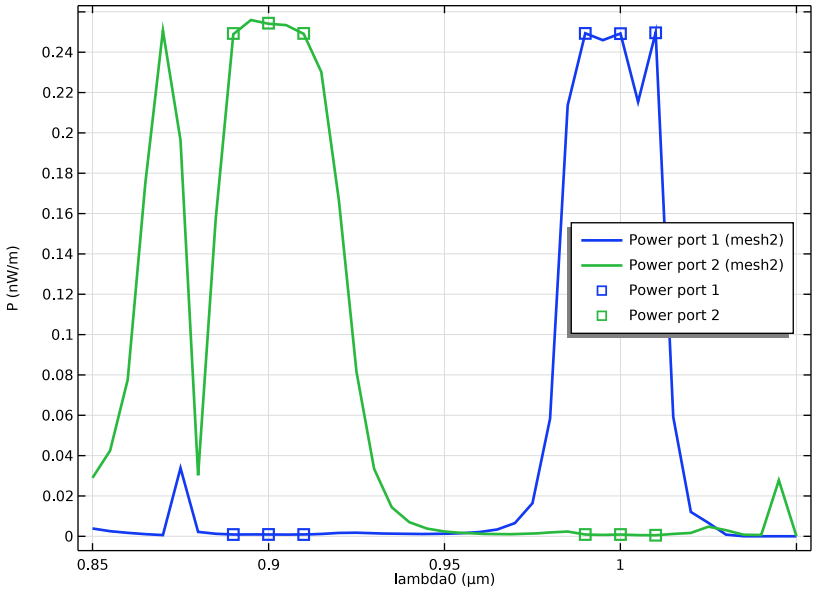


Figure 3: The two auxiliary objective functions are plotted as a function of the frequency. The points indicate the frequencies used to approximate the two frequency bands.

Reference

1. J.D. Joannopoulus, R.D. Meade, and J.N. Winn, *Photonic Crystals (Modeling the Flow of Light)*, Princeton University Press, 1995.

Application Library path: `Wave_Optics_Module/Waveguides_and_Couplers/photonic_crystal_demultiplexer_optimization`

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 **Preset Studies for Selected Physics Interfaces> Wavelength Domain**.

6 Click **Done**.

GEOMETRY 1

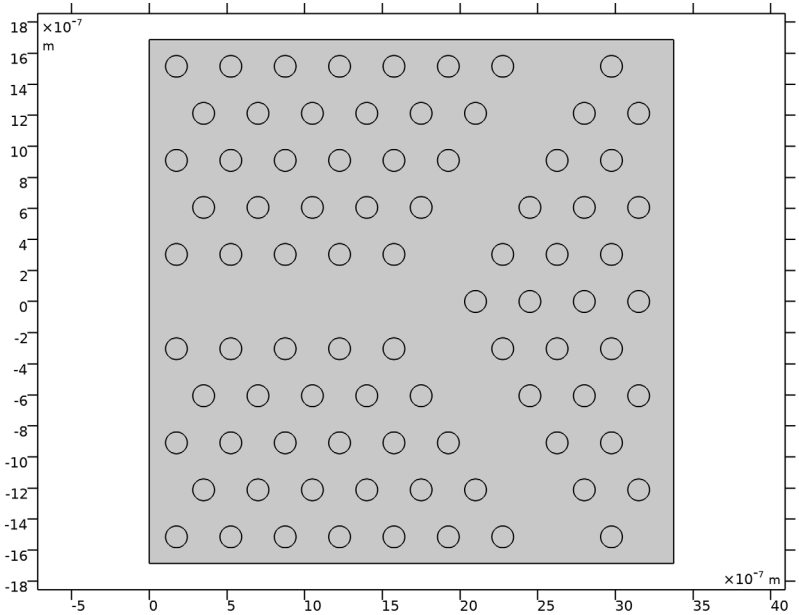
Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 In the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `photonic_crystal_demultiplexer_optimization_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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



5 In the **Model Builder** window, collapse the **Geometry 1** node.

MATERIALS

The refractive index of GaAs depends on the frequency. The material is added from the Optical Material Database.

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 **Optical>Inorganic Materials>As - Arsenides>Experimental data> GaAs (Gallium arsenide) (Skauli et al. 2003: n 0.97-17 um)**.

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

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

MATERIALS

GaAs (Gallium arsenide) (Skauli et al. 2003: n 0.97-17 um) (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Circle 1**.

Air

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

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

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	1	1	Refractive index

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

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**.

Scattering Boundary Condition 1

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

2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Scattering Boundary Condition 2

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

2 In the **Settings** window for **Scattering Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Input Port**.

4 Locate the **Scattering Boundary Condition** section. From the **Incident field** list, choose **Wave given by E field**.

5 From the **Incident field** list, choose **Wave given by E field**.

6 Specify the $\mathbf{E}_0$ vector as

0	x
0	y
1	z

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

DEFINITIONS

Power port 1

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `Power port 1` in the **Label** text field.

3 In the **Variable name** text field, type `obj1`.

4 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

5 Locate the **Source Selection** section. From the **Selection** list, choose **Output Port 1**.

6 Click **Section toolbar** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain> Energy and power>ewfd.nPoav - Power outflow, time average - W/m²**.

Power port 2

1 Right-click **Power port 1** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Probe**, type `Power port 2` in the **Label** text field.

3 In the **Variable name** text field, type `obj2`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Output Port 2**.

GLOBAL DEFINITIONS

Wave Parameter

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Wave Parameter` in the **Label** text field.

Use three parameters to define two frequency bands.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
`lambda1`	`0.9[um]`	9E-7 m	`First wavelength`
`lambda2`	`1[um]`	1E-6 m	`Second wavelength`
`dWave`	`0.02[um]`	2E-8 m	`Bandwidth`
`dWaveN`	3	3	`Frequencies per band`
`meshsz`	`lambda2/12`	8.3333E-8 m	`Mesh size`

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

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

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

3 In the **Maximum element size** text field, type `meshsz`.

4 In the **Minimum element size** text field, type `meshsz/2`.

5 In the **Curvature factor** text field, type `Inf`.

Size 1

1 In the **Model Builder** window, right-click **Size 1** and choose **Enable**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Circle 1**.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz/2`.

5 In the **Minimum element size** text field, type `meshsz/4`.

6 Clear the **Maximum element growth rate** check box.

7 Clear the **Curvature factor** check box.

8 Clear the **Resolution of narrow regions** check box.

9 Click **Build All**.

STUDY 1

Step 1: Wavelength Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

3 From the **Wavelength unit** list, choose **m**.

4 In the **Wavelengths** text field, type `range(lambda1-50[nm],5[nm],lambda2+50[nm])`.

5 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

6 In the **Model Builder** window, click **Study 1**.

7 In the **Settings** window for **Study**, type `Initial Design` in the **Label** text field.

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

DEFINITIONS

In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Free Shape Domain**.

SHAPE OPTIMIZATION

Free Shape Domain 1

1 In the **Settings** window for **Free Shape Domain**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **All domains**.

Transformation 1

1 In the **Definitions** toolbar, click **Optimization** and choose **Shape Optimization> Transformation**.

2 In the **Settings** window for **Transformation**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Circle 1**.

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

	Lock	Lower bound (m)	Upper bound (m)
X		-5E-8	5E-8
Y		-5E-8	5E-8

5 Locate the **Scaling** section. From the **Scaling type** list, choose **No scaling**.

DEFINITIONS

Objective Function

1 In the **Definitions** toolbar, click  **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Objective Function` in the **Label** text field.

3 In the **Variable name** text field, type `obj`.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain> ewfd.lambda0 - Wavelength in free space - m**.

5 Locate the **Expression** section. In the **Expression** text field, type `if(ewfd.lambda0<(lambda1+lambda2)/2,obj1/obj2,obj2/obj1)`.

The if statement causes the ratio to be inverted for one frequency band relative to the other.

Constraint

1 In the **Definitions** toolbar, click  **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, type `Constraint` in the **Label** text field.

3 In the **Variable name** text field, type `constr`.

4 Locate the **Expression** section. In the **Expression** text field, type `(obj1+obj2)/0.25[nW/m]`.

`2.5e-10` provides a good compromise between demultiplexing and overall transmission.

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 Physics Interfaces>Wavelength Domain**.

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

5 In the **Select Study** tree, select **Empty Study**.

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

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

Disable the shape optimization for the first study.

INITIAL DESIGN

Step 1: Wavelength Domain

1 In the **Model Builder** window, under **Initial Design** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

STUDY 2

Step 1: Wavelength Domain

Approximate each frequency band by three frequencies.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Wavelength Domain**.

2 In the **Settings** window for **Wavelength Domain**, locate the **Study Settings** section.

3 In the **Wavelengths** text field, type `range(lambda1-dWave/2,dWave/(dWaveN-1), lambda1+dWave/2) range(lambda2-dWave/2,dWave/(dWaveN-1),lambda2+ dWave/2)`.

4 Locate the **Results While Solving** section. From the **Probes** list, choose **None**.

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `50`.

4 Clear the **Move limits** check box.

5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>comp1.obj - Objective Function**.

6 Locate the **Objective Function** section. From the **Solution** list, choose **Maximum of objectives**.

This causes the solver to identify the frequency associated with the maximum objective function and prioritize this over the other frequencies (while still taking all frequencies into account).

7 Click **Add Expression** in the upper-right corner of the **Constraints** section. From the menu, choose **Component 1 (comp1)>Definitions>comp1.constr - Constraint**.

8 Locate the **Constraints** section. In the table, enter the following settings:

Expression	Lower bound	Upper bound
comp1.constr	1	

This prevents the optimization solver from generating a design with poor transmission to the output ports.

9 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

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

11 In the **Settings** window for **Study**, type `Shape Optimization` in the **Label** text field.

12 In the **Study** toolbar, click **Get Initial Value**.

13 In the **Model Builder** window, click **Shape Optimization**.

14 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

15 Select the **Plot** check box.

16 From the **Plot group** list, choose **Shape Optimization**.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Shape Optimization>Solver Configurations** node.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Shape Optimization>Solver Configurations> Solution 2 (sol2)>Optimization Solver 1>Stationary 1** node.

4 Right-click **Stationary 1** and choose **Segregated**.

5 In the **Settings** window for **Segregated**, locate the **General** section.

6 From the **Termination technique** list, choose **Iterations**to reduce the computational time.

7 Right-click **Segregated 1** and choose **Segregated Step**.

8 In the **Settings** window for **Segregated Step**, locate the **General** section.

9 In the **Variables** list, select **Electric field (spatial and material frames) (comp1.E)**.

10 Under **Variables**, click **Delete**.

11 In the **Model Builder** window, click **Segregated Step 1**.

12 In the **Settings** window for **Segregated Step**, locate the **General** section.

13 Under **Variables**, click + **Add**.

14 In the **Add** dialog box, in the **Variables** list, choose **Electric field (spatial and material frames) (comp1.E)** and **Translation (geometry frame) (comp1.tsf1.move)**.

15 Click **OK**.

16 In the **Settings** window for **Segregated Step**, click = **Compute**.

RESULTS

Shape Optimization/Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Shape Optimization/Solution 2 (sol2)** and choose **Remesh Deformed Configuration**.

INITIAL DESIGN

Step 1: Wavelength Domain

In the **Model Builder** window, under **Initial Design** right-click **Step 1: Wavelength Domain** and choose **Copy**.

STUDY 3

In the **Model Builder** window, right-click **Study 3** and choose **Paste Wavelength Domain**.

Step 1: Wavelength Domain

1 In the **Settings** window for **Wavelength Domain**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

3 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Shape Optimization, Wavelength Domain**.

6 Find the **Store fields in output** subsection. From the **Settings** list, choose **For selections**.

7 Under **Selections**, click + **Add**.

8 In the **Add** dialog box, in the **Selections** list, choose **Output Port 1** and **Output Port 2**.

9 Click **OK**.

10 In the **Settings** window for **Wavelength Domain**, click to expand the **Mesh Selection** section.

11 In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 2

12 In the **Model Builder** window, click **Study 3**.

13 In the **Settings** window for **Study**, type `Verification` in the **Label** text field.

14 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

RESULTS

Electric Field (initial)

1 In the **Model Builder** window, under **Results** click **Electric Field (ewfd)**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field (initial)` in the **Label** text field.

Surface 1

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

2 In the **Settings** window for **Surface**, click **Section toolbar** 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>ewfd.Ez - Electric field, z component**.

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

Streamline 1

1 In the **Model Builder** window, right-click **Electric Field (ewfd) 1** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Energy and power>ewfd.Poavx,ewfd.Poavy - Power flow, time average (spatial and material frames)**.

3 Locate the **Selection** section. From the **Selection** list, choose **Input Port**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

5 Select the **Radius scale factor** check box.

6 In the **Tube radius expression** text field, type `3e-9`.

7 Find the **Point style** subsection. From the **Color** list, choose **Black**.

Surface 1

1 In the **Model Builder** window, click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Section toolbar** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Electric>Electric field (spatial and material frames) - V/m>ewfd.Ez - Electric field, z component**.

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

4 From the **Scale** list, choose **Linear symmetric**.

Electric Field (optimized)

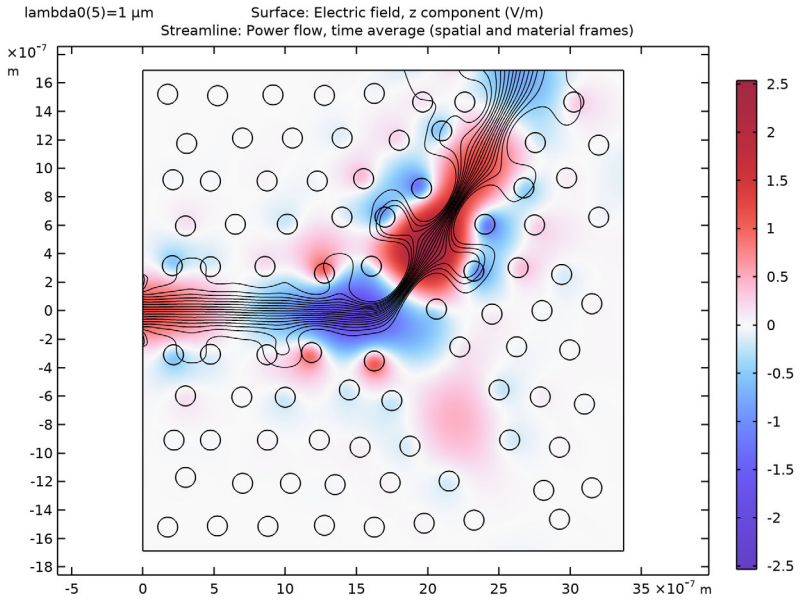
1 In the **Model Builder** window, under **Results** click **Electric Field (ewfd) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Electric Field (optimized)` in the **Label** text field.

3 Click ← **Plot Previous**.

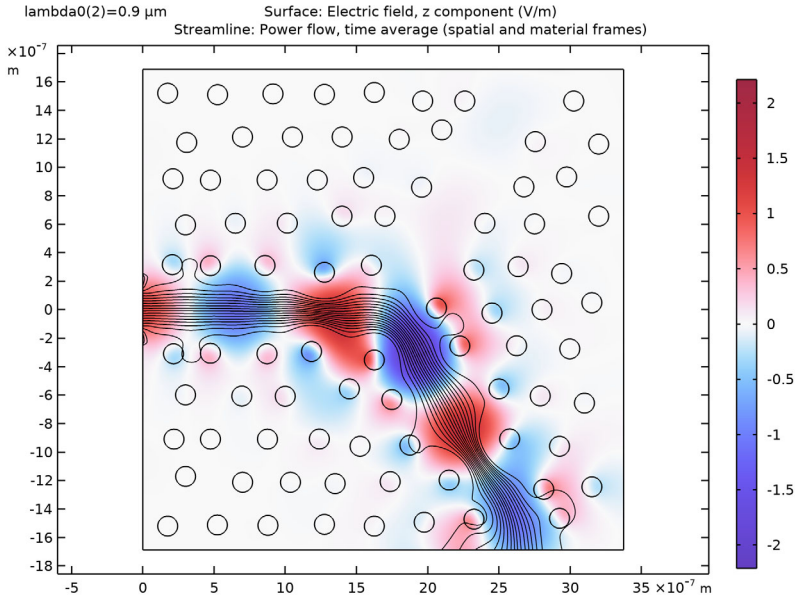
4 In the **Electric Field (optimized)** toolbar, click **Plot**.

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



6 Click ← **Plot Previous** three times.

7 In the **Electric Field (optimized)** toolbar, click **Plot**.



Create a new plot **1D Plot Group** for the spectrum.

Spectrum

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

2 In the **Settings** window for **1D Plot Group**, type `Spectrum` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Verification/Solution 3 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section. Select the **y-axis label** check box.

6 In the associated text field, type `P (nW/m)`.

7 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Global 1

1 Right-click **Spectrum** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>obj1 - Power port 1 - W/m**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>obj2 - Power port 2 - W/m**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
obj1	nW/m	Power port 1 (mesh2)
obj2	nW/m	Power port 2 (mesh2)

5 Locate the **x-Axis Data** section. From the **Unit** list, choose **µm**.

6 Click to expand the **Coloring and Style** section. In the **Width** text field, type 2.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Shape Optimization/Solution 2 (sol2)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

6 From the **Positioning** list, choose **In data points**.

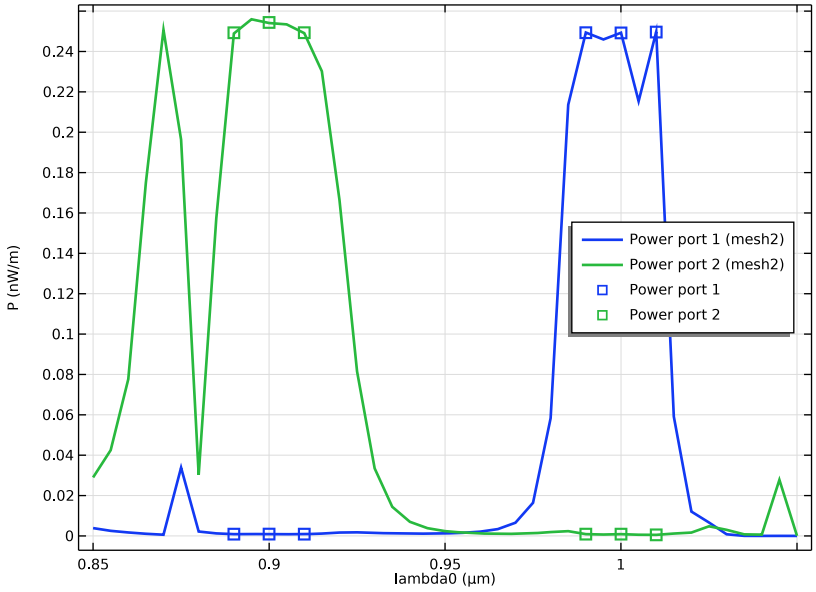
7 From the **Color** list, choose **Cycle (reset)**.

8 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
obj1	nW/m	Power port 1
obj2	nW/m	Power port 2

9 In the **Spectrum** toolbar, click **Plot**.

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



Shape Optimization

Finally, create a new **2D Plot Group** for the thumbnail.

Thumbnail

1 In the **Model Builder** window, right-click **Shape Optimization** and choose **Duplicate**.

2 In the **Model Builder** window, click **Shape Optimization 1**.

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Line 1

1 In the **Model Builder** window, expand the **Results>Thumbnail> Translation (Transformation 1)** node, then click **Results>Thumbnail>Line 1**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 From the **Line type** list, choose **Tube**.

4 Select the **Radius scale factor** check box.

5 In the **Tube radius expression** text field, type `5e-9`.

Line 2

1 In the **Model Builder** window, right-click **Line 1** and choose **Duplicate**.

2 In the **Settings** window for **Line**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **Gray**.

Deformation 1

1 Right-click **Line 2** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, click **Section toolbar** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions> Transformation 1>tsf1.dXg,tsf1.dYg - Boundary displacement (geometry frame)**.

3 Locate the **Expression** section. In the **X component** text field, type `-tsf1.dXg`.

4 In the **Y component** text field, type `-tsf1.dYg`.

5 Locate the **Scale** section. Select the **Scale factor** check box.

6 In the associated text field, type `1`.

Color Expression 1

1 In the **Model Builder** window, under **Results>Thumbnail>Translation (Transformation 1)** click **Color Expression 1**.

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

3 In the **Expression** text field, type `sqrt(material.dX^2+material.dY^2)`.

4 Click to expand the **Range** section. Clear the **Manual color range** check box.

Filter 1

1 In the **Model Builder** window, right-click **Line 1** 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 `(abs(X-xCenter)<1e-6)*(abs(Y)<1e-6)`.

4 Right-click **Filter 1** and choose **Copy**.

Filter 1

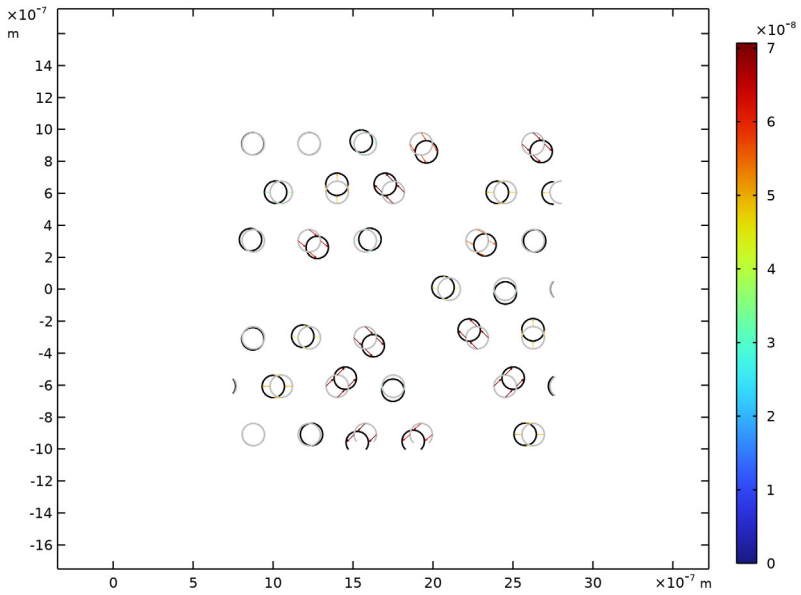
In the **Model Builder** window, right-click **Translation (Transformation 1)** and choose **Paste Filter**.

Filter 1

1 In the **Model Builder** window, right-click **Line 2** and choose **Paste Filter**.

2 In the **Thumbnail** toolbar, click  **Plot**.

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



Geometry Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Blank Model**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
W	3.375[um]	3.375E-6 m	Design Domain Width
H	W	3.375E-6 m	Design Domain Height
rHole	0.07[um]	7E-8 m	Hole Radius

Name	Expression	Value	Description
dPeriod	5*rHole	3.5E-7 m	Periodicity
xCenter	5*dPeriod	1.75E-6 m	Center Hole

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

Rectangle 1 (r1)

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `W`.
4. In the **Height** text field, type `H`.
5. Locate the **Position** section. In the **y** text field, type `-H/2`.

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 `rHole`.
4. Locate the **Position** section. In the **y** text field, type `-2*dPeriod*sin(pi/3)*round(H/dPeriod/sin(pi/3)/3)`.
5. Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Move 1 (mov1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Move**.
2. In the **Settings** window for **Move**, locate the **Input** section.
3. From the **Input objects** list, choose **Circle 1**.
4. Select the **Keep input objects** check box.
5. Locate the **Displacement** section. In the **y** text field, type `sin(pi/3)*dPeriod`.
6. In the **x** text field, type `cos(pi/3)*dPeriod`.

Array 1 (arr1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Array**.
2. In the **Settings** window for **Array**, locate the **Input** section.
3. From the **Input objects** list, choose **Circle 1**.

4 Locate the **Size** section. In the **x size** text field, type `round(W/dPeriod)+1`.

5 In the **y size** text field, type `round(H/dPeriod)`.

6 Locate the **Displacement** section. In the **x** text field, type `dPeriod`.

7 In the **y** text field, type `2*sin(pi/3)*dPeriod`.

Circles to Keep

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Circles to Keep` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `0`.

4 In the **x maximum** text field, type `W`.

5 In the **y minimum** text field, type `-H/2`.

6 In the **y maximum** text field, type `H/2`.

7 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

8 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

9 Click **Add**.

10 In the **Add** dialog box, select **Circle 1** in the **Selections** list.

11 Click **OK**.

Circles to Delete

1 In the **Geometry** toolbar, click **Selections** and choose **Difference Selection**.

2 In the **Settings** window for **Difference Selection**, type `Circles to Delete` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Circle 1** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click **Add**.

8 In the **Add** dialog box, select **Circles to Keep** in the **Selections to subtract** list.

9 Click **OK**.

Input Port

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, type `Input Port` in the **Label** text field.

3 Locate the **Starting Point** section. From the **Specify** list, choose **Coordinates**.

4 In the **y** text field, type `-dPeriod/2`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **y** text field, type `dPeriod/2`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Output Port 1

1 Right-click **Input Port** and choose **Duplicate**.

2 In the **Settings** window for **Line Segment**, type `Output Port 1` in the **Label** text field.

3 Locate the **Starting Point** section. In the **x** text field, type `xCenter+H/sqrt(3)/2-dPeriod/sqrt(3)`.

4 In the **y** text field, type `H/2`.

5 Locate the **Endpoint** section. In the **x** text field, type `xCenter+H/sqrt(3)/2+dPeriod/sqrt(3)`.

6 In the **y** text field, type `H/2`.

Output Port 2

1 Right-click **Output Port 1** and choose **Duplicate**.

2 In the **Settings** window for **Line Segment**, type `Output Port 2` in the **Label** text field.

3 Locate the **Starting Point** section. In the **y** text field, type `-H/2`.

4 Locate the **Endpoint** section. In the **y** text field, type `-H/2`.

Circles to Delete, row1

1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Circles to Delete, row1` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x maximum** text field, type `xCenter+2*rHole`.

4 In the **y minimum** text field, type `-rHole`.

5 In the **y maximum** text field, type `rHole`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Circle 1**.

4 Locate the **Rotation** section. In the **Angle** text field, type `120`.

5 Locate the **Center of Rotation** section. In the **x** text field, type `xCenter`.

Circles to Delete, row2

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Circles to Delete, row1 (boxsel2)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Circles to Delete, row2` in the **Label** text field.

Rotate 2 (rot2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rotate 1 (rot1)** and choose **Duplicate**.

Circles to Delete, row3

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Circles to Delete, row2 (boxsel3)** and choose **Duplicate**.

2 In the **Settings** window for **Box Selection**, type `Circles to Delete, row3` in the **Label** text field.

Rotate 3 (rot3)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rotate 2 (rot2)** and choose **Duplicate**.

Circles to Delete, rows

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Circles to Delete, rows` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Circles to Delete**, **Circles to Delete, row1**, **Circles to Delete, row2**, and **Circles to Delete, row3**.

5 Click **OK**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **Circles to Delete, rows**.

Circle Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Circle Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Circle 1** in the **Input selections** list.

5 Click **OK**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

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

The model geometry is now complete.