Created in COMSOL Multiphysics 6.2

Time to Frequency FFT Analysis of a Distributed Bragg Reflector

Introduction

A distributed Bragg reflector (DBR), or dielectric mirror, is a structure used in bulk optics and waveguides to reflect light. It has extremely low losses at optical and infrared frequencies compared to ordinary metallic mirrors. The structure is formed from alternating thin layers of materials with high and low refractive indices, where each layer is one quarter of a material wavelength thick.

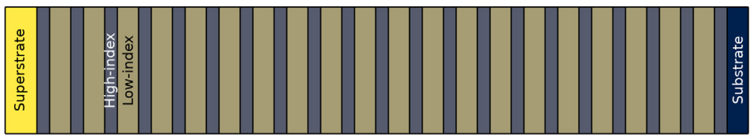


Figure 1: The distributed Bragg reflector structure. The first layer represents the infinite superstrate. In this case air is used. The high-index layers are made of titanium dioxide and the low-index layers use silica. Also the substrate is assumed to use silica.

Each layer boundary causes a partial reflection of an optical wave. When the wavelength is close to four times the optical thickness of the layers, the many reflected waves tend to interfere constructively, causing the layers to act as a high-quality reflector. The range of wavelengths in which most of the incident intensity is reflected is called the photonic stopband. In the limit in which the reflector contains a very large number of layers, radiation in this range of wavelengths cannot propagate into the structure.

Distributed Bragg reflectors are critical components in for instance vertical cavity surface emitting lasers (see Threshold Gain Calculations for Vertical-Cavity Surface-Emitting Lasers (VCSELs)).

This model demonstrates how to setup a Time to Frequency FFT study for a distributed Bragg reflector (DBR) structure. It also compares the results with results from a regular Frequency domain study.

Model Definition

The model consists of a structure of twenty periods of alternating high and low refractive index layers. The refractive indexes are tabulated in Table 1 below.

TABLE 1: REFRACTIVE INDICES.

LAYER TYPE	MATERIAL	REFRACTIVE INDEX
High-index	Titanium dioxide	2.5
Low-index	Silica	1.5

Each layer is a quarter of a material wavelength thick.

In the first study, a modulated Gaussian pulse is used for exciting the structure. The input electric field for the Scattering boundary condition is defined by

$$\mathbf{E}_{\mathrm{in}} = E_0 \exp\left(-\frac{(t-T_c)^2}{T_d^2}\right) \sin(\omega_0(t-T_c))\mathbf{z},$$

where E_0 is the amplitude, t is time, $T_\mathrm{d} = 1/(2f_0)$ is the pulse duration, f_0 is the center frequency, $T_\mathrm{c} = 3T_\mathrm{d}$ is a delay time, $\omega_0 = 2\pi f_0$ is the center angular frequency, and $\mathbf{z}$ is the unit vector in the z direction.

In a Time to Frequency FFT, the time dependent solution, from an initial Time Dependent study step, is transformed to a frequency dependent solution. To make it possible to also perform an FFT on the input electric field above, a dependent variable is assigned the value of the input electric field function. Then the FFT is performed also on that dependent variable.

To generate the frequency-domain data without significant distortion in the frequency range between 0 and $2f_0$, the time step, satisfying the Nyquist criterion, is set to $1/4f_0$ = $1/2\mathrm{B}$, where B is the bandwidth $2f_0$.

Results and Discussion

Figure 1 shows the electric field distribution in the DBR structure. The plot is calculated using an FFT of the time dependent solution. At the center frequency for the stop band, the field decays quickly in the structure. Almost no field is transmitted.

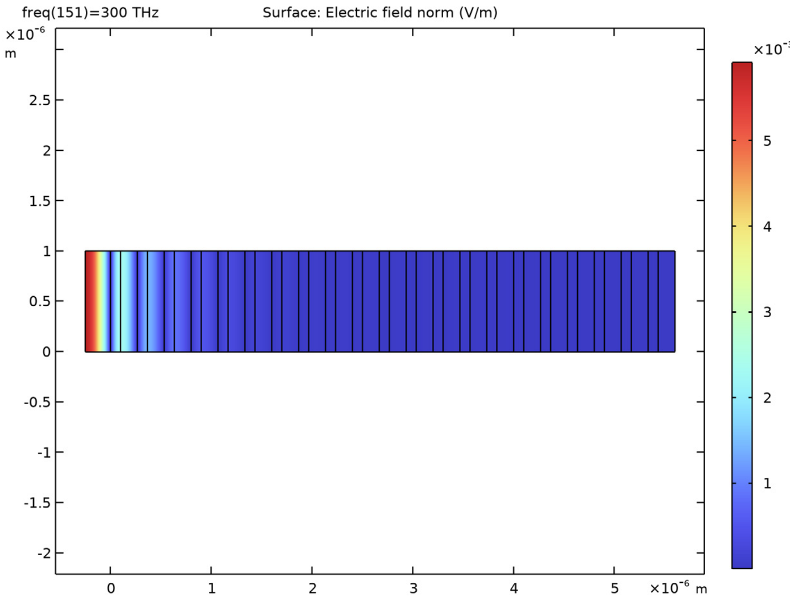


Figure 2: The electric field distribution at the center of the stopband (300 THz). The plot is calculated by performing an FFT of the time dependent solution.

As a comparison, Figure 3 shows a similar plot as in Figure 2, but calculated by a regular Frequency domain study. The two field distributions look the same, except for the different amplitudes.

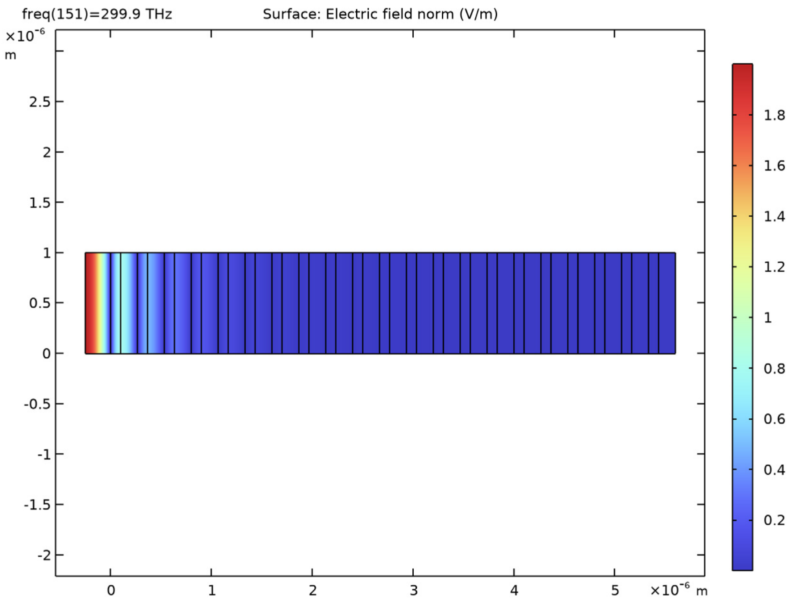


Figure 3: The electric field distribution at the center of the stopband (299.9 THz), as computed by a Frequency domain study.

Figure 4 shows that the transmittance spectra for the two types of simulations agree very well.

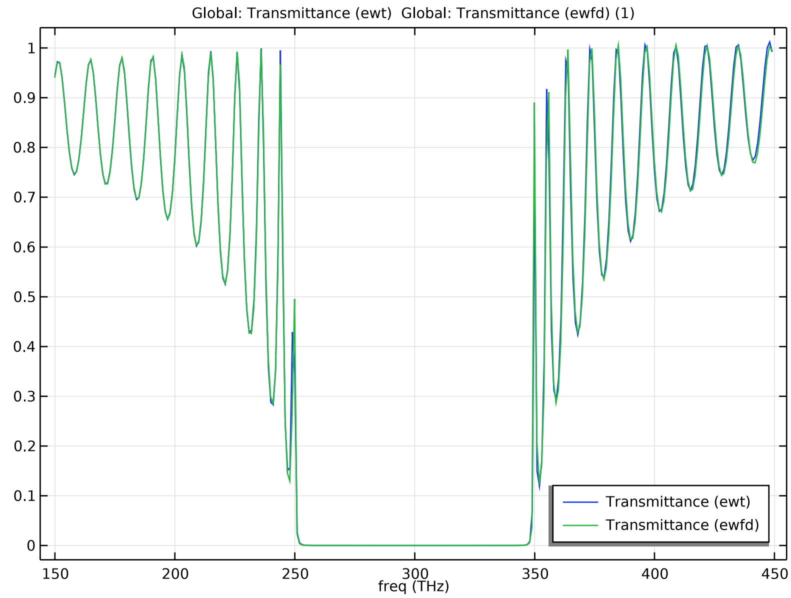


Figure 4: The transmittance for the two type of simulations. The blue curve is performed by performing an FFT on the time domain solution, and the green curve is the result of a regular frequency sweep.

Figure 5 shows the field distribution after 40 fs. At that point in time, the main pulse has almost passed the mirror structure. As shown in Figure 6, the electromagnetic energy decays slowly, as the radiation is being reflected back and forth in the layer structure before it escapes through the left and right boundaries. However, at the end of the simulation, there is not much energy left in the structure. Thus, as shown in Figure 4, the result from the FFT of the time dependent signal is very accurate, as almost the whole time dependent phenomena is included in the simulation.

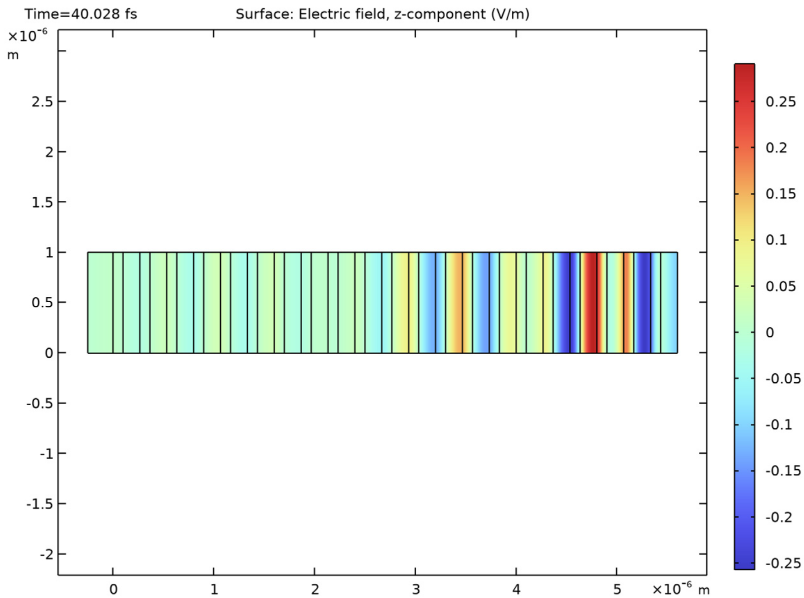


Figure 5: The electric field distribution after 40 fs, when the main pulse has almost propagated through the whole mirror structure.

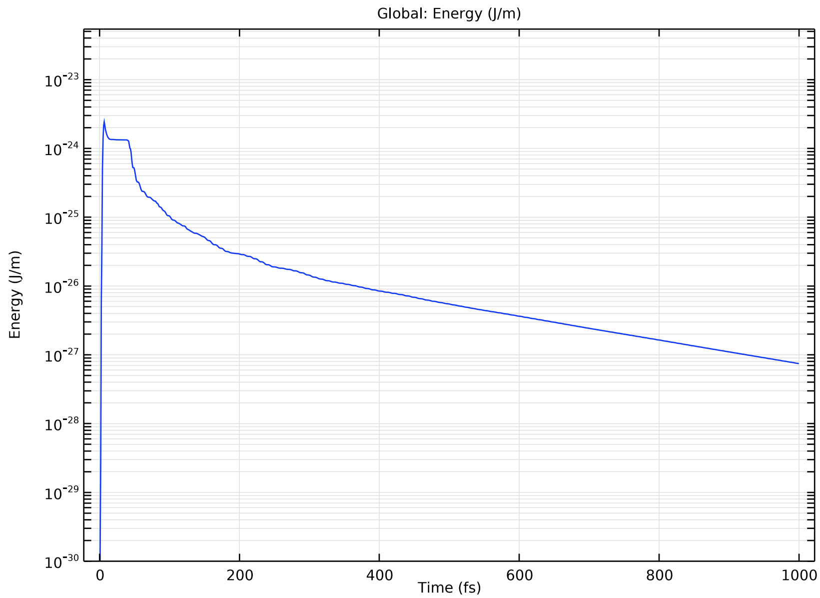


Figure 6: The electromagnetic energy in the mirror structure versus time.

Application Library path: `Wave_Optics_Module/Couplers_Filters_and_Mirrors/time_to_frequency_fft_distributed_bragg_reflector`

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, Transient (ewt)**.
3. Click **Add**.

4 In the **Select Physics** tree, select **Optics>Wave Optics>Electromagnetic Waves, Frequency Domain (ewfd)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Electromagnetic Waves, Transient>Time Dependent with FFT**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Add some parameters that are used for configuring the geometry, the materials, the mesh, and the solvers.

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `time_to_frequency_fft_distributed_bragg_reflector_parameters.txt`.

GEOMETRY 1

Now, add the geometry. Define cumulative selections to make it easier to add the materials to the right layers.

Superstrate

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

2 In the **Settings** window for **Rectangle**, type `Superstrate` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `d0`.

4 In the **Height** text field, type `lda0`.

5 Locate the **Position** section. In the **x** text field, type `-d0`.

6 Right-click **Superstrate** and choose **Duplicate**.

High-Index Layer

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Superstrate 1 (r2)**.

2 In the **Settings** window for **Rectangle**, type `High-Index Layer` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `dH`.

4 Locate the **Position** section. In the **x** text field, type `0`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `High-Index Material` in the **Name** text field.

7 Click **OK**.

8 Right-click **High-Index Layer** and choose **Duplicate**.

Low-Index Layer

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **High-Index Layer 1 (r3)**.

2 In the **Settings** window for **Rectangle**, type `Low-Index Layer` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `dL`.

4 Locate the **Position** section. In the **x** text field, type `dH`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Low-Index Material` in the **Name** text field.

7 Click **OK**.

Period

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, type `Period` in the **Label** text field.

3 Select the objects **r2** and **r3** only.

Array 1 (arr1)

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

2 Select the object **uni1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `NPeriod`.

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

High-Index Layer (r2)

In the **Model Builder** window, right-click **High-Index Layer (r2)** and choose **Duplicate**.

Final High-Index Layer

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **High-Index Layer 1 (r4)**.

2 In the **Settings** window for **Rectangle**, type `Final High-Index Layer` in the **Label** text field.

3 Locate the **Position** section. In the **x** text field, type `L_tot-dH`.

Superstrate (r1)

In the **Model Builder** window, right-click **Superstrate (r1)** and choose **Duplicate**.

Substrate

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Superstrate 1 (r5)**.

2 In the **Settings** window for **Rectangle**, type `Substrate` in the **Label** text field.

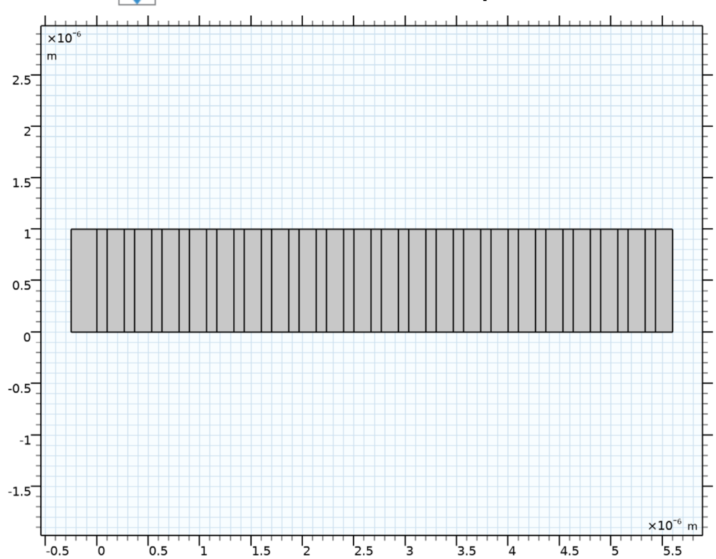
3 Locate the **Size and Shape** section. In the **Width** text field, type `ds`.

4 Locate the **Position** section. In the **x** text field, type `L_tot`.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Low-Index Material**.

6 Click **Build All Objects**.

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



MATERIALS

Add the materials now, using the cumulative selections defined with the geometry.

Air

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 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	n0	1	Refractive index
Refractive index, imaginary part	ki_iso ; kiii = ki_iso, kiij = 0	0	1	Refractive index

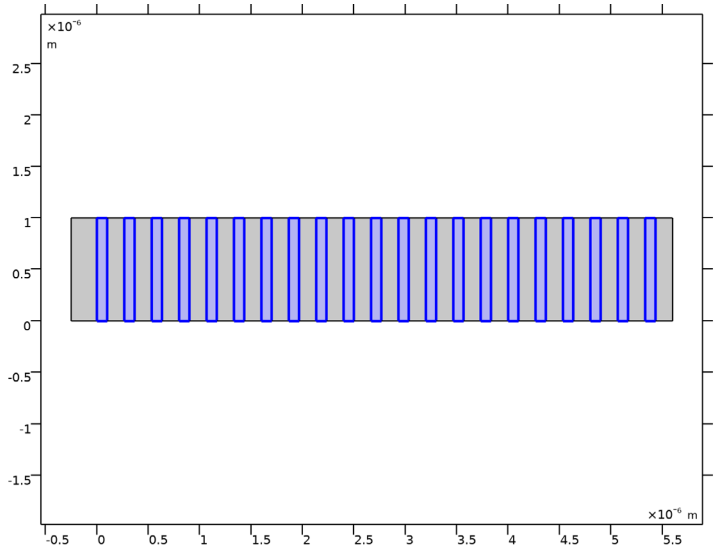
By default, the first added material will have All domains as the selection. The materials added later will override all selected domains, except for the leftmost superstrate domain.

Titanium Dioxide

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Titanium Dioxide` in the **Label** text field. This will be the high-index material.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **High-Index Material**.



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

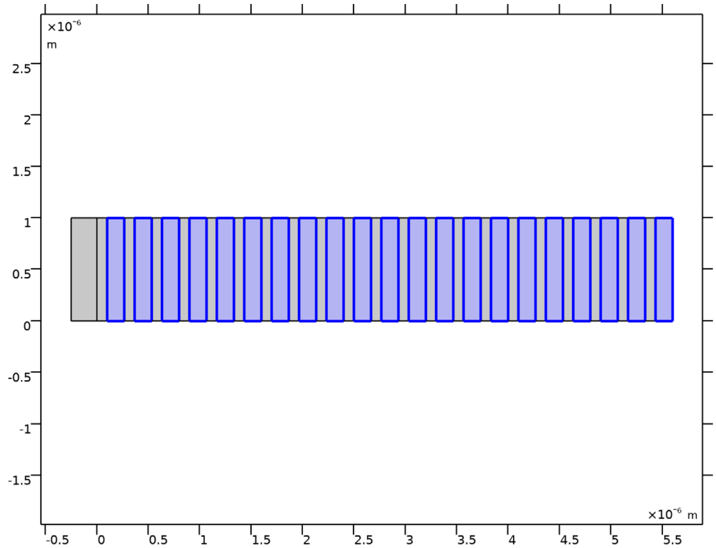
Property	**Variable**	**Value**	**Unit**	**Property group**
Refractive index, real part	n_iso ; nii = n_iso, nij = 0	nH	1	Refractive index
Refractive index, imaginary part	ki_iso ; kiii = ki_iso, kiij = 0	0	1	Refractive index

Silica

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Silica` in the **Label** text field. This will be the low-index material.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Low-Index Material**.



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	nL	1	Refractive index
Refractive index, imaginary part	ki_iso ; kiii = ki_iso, kiij = 0	0	1	Refractive index

DEFINITIONS

Next, add the modulated Gaussian input electric field.

Modulated Gaussian Input Field

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `Modulated Gaussian Input Field` in the **Label** text field.

3 In the **Function name** text field, type `Ein`.

4 Locate the **Definition** section. In the **Expression** text field, type `E0*exp(-(t-Tc)^2/Td^2)*sin(omega0*(t-Tc))`.

5 In the **Arguments** text field, type `t`.

6 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
`t`	`s`

7 In the **Function** text field, type `V/m`.

Top Exterior Boundaries

Before adding the physics, it is useful to add some selections that will be used by the physics features.

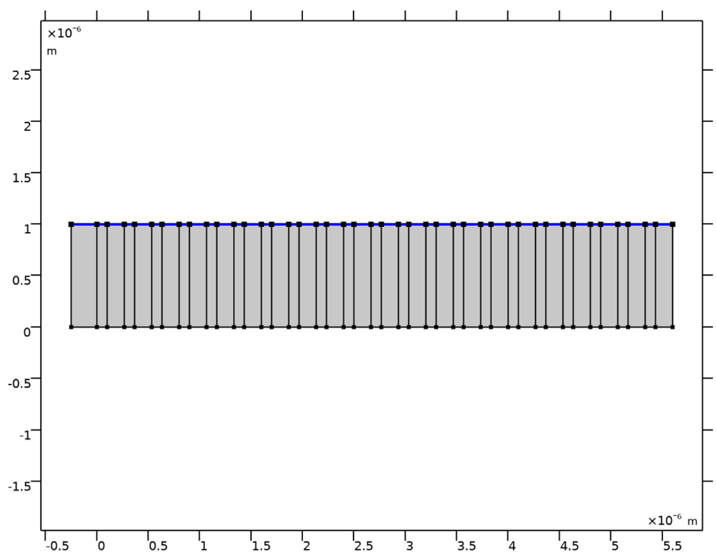
1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Top Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

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

5 Select Boundaries 3, 6, 9, 12, 15, 18, 21, 24, 27, 30, 33, 36, 39, 42, 45, 48, 51, 54, 57, 60, 63, 66, 69, 72, 75, 78, 81, 84, 87, 90, 93, 96, 99, 102, 105, 108, 111, 114, 117, 120, 123, 126, and 129 only. These are the entities representing the top horizontal boundary.



6 Right-click **Top Exterior Boundaries** and choose **Duplicate**.

Bottom Exterior Boundaries

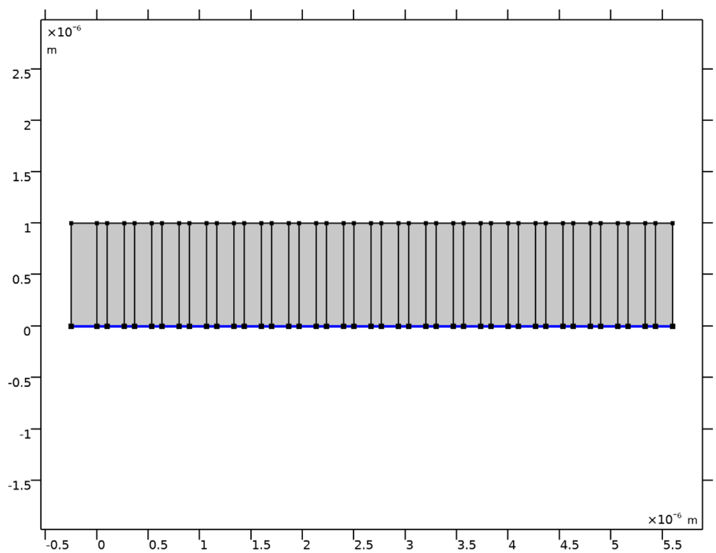
1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Top Exterior Boundaries 1**.

2 In the **Settings** window for **Explicit**, type `Bottom Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Clear Selection**.

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

5 Select Boundaries 2, 5, 8, 11, 14, 17, 20, 23, 26, 29, 32, 35, 38, 41, 44, 47, 50, 53, 56, 59, 62, 65, 68, 71, 74, 77, 80, 83, 86, 89, 92, 95, 98, 101, 104, 107, 110, 113, 116, 119, 122, 125, and 128 only. These are the entities representing the bottom horizontal boundary.



Top and Bottom Exterior Boundaries

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

2 In the **Settings** window for **Union**, type `Top and Bottom Exterior Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Top Exterior Boundaries** and **Bottom Exterior Boundaries**.

6 Click **OK**.

Top High-Index Material Boundaries

1 In the **Definitions** toolbar, click **Intersection**.
2 In the **Settings** window for **Intersection**, type `Top High-Index Material Boundaries` in the **Label** text field.
3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.
4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.
5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Top Exterior Boundaries** and **High-Index Material**.
6 Click **OK**.
7 Right-click **Top High-Index Material Boundaries** and choose **Duplicate**.

Top Low-Index Material Boundaries

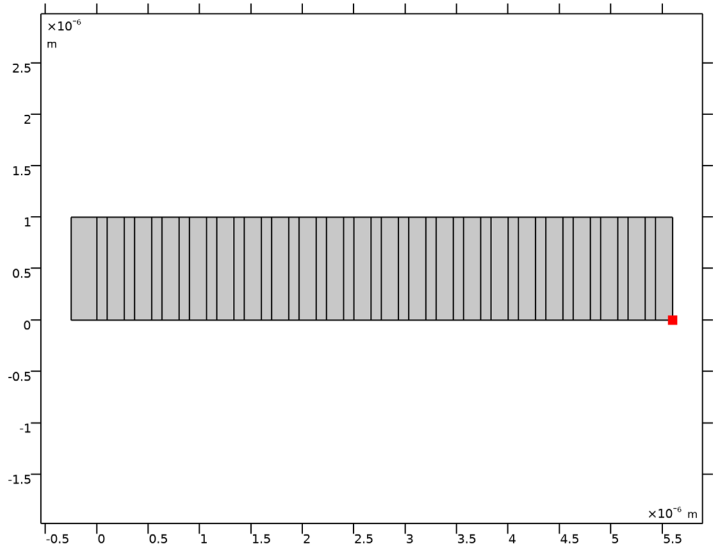
1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Top High-Index Material Boundaries 1**.
2 In the **Settings** window for **Intersection**, type `Top Low-Index Material Boundaries` in the **Label** text field.
3 Locate the **Input Entities** section. In the **Selections to intersect** list, select **High-Index Material**.
4 Under **Selections to intersect**, click **Delete**.
5 Under **Selections to intersect**, click **Add**.
6 In the **Add** dialog box, select **Low-Index Material** in the **Selections to intersect** list.
7 Click **OK**.

Domain Point Probe 1

A probe will be added to visualize the progress while solving.

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.
2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.
3 In row **Coordinates**, set **x** to `L_tot+ds`, to select the point in the lower right corner.

4 Select the **Snap to closest boundary** check box.



Point Probe Expression 1 (ppb1)

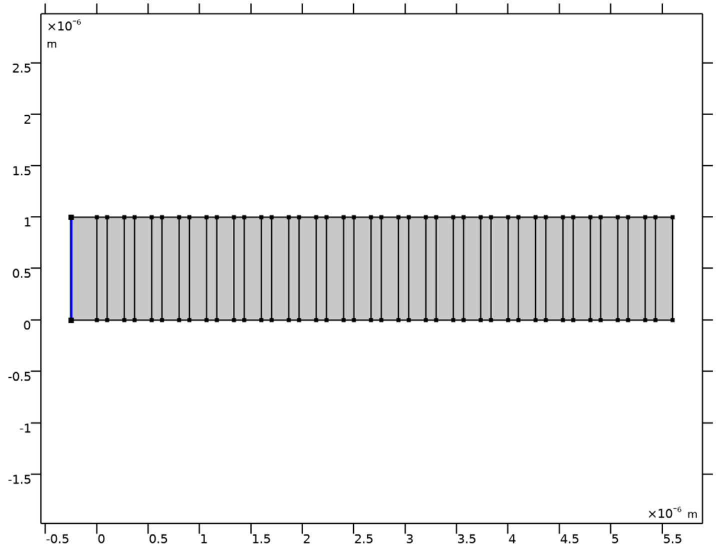
1. In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.
2. In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `ewt.Ez`.

Integration 1 (intop1)

Add a few integration operators that will be used for calculating the input and transmitted powers, and the energy stored in the mirror structure.

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2. In the **Settings** window for **Integration**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.



5 Right-click **Integration 1 (intop1)** and choose **Duplicate**.

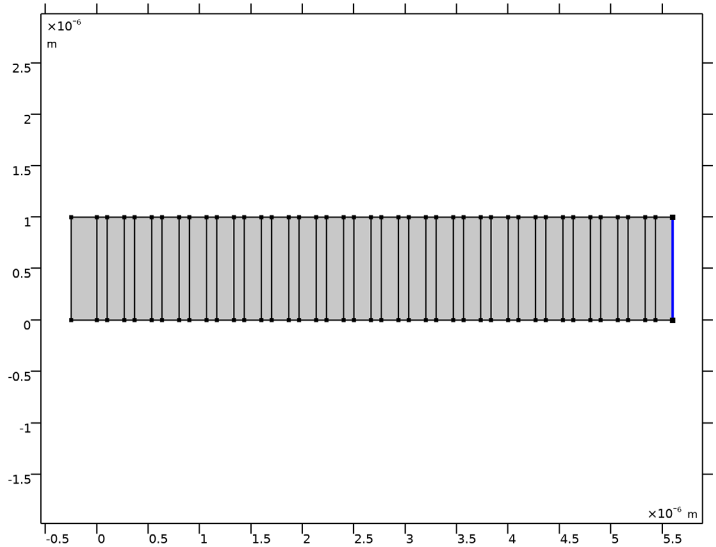
Integration 2 (intop2)

1 In the **Model Builder** window, click **Integration 2 (intop2)**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 Click  **Clear Selection**.

4 Select Boundary 130 only.

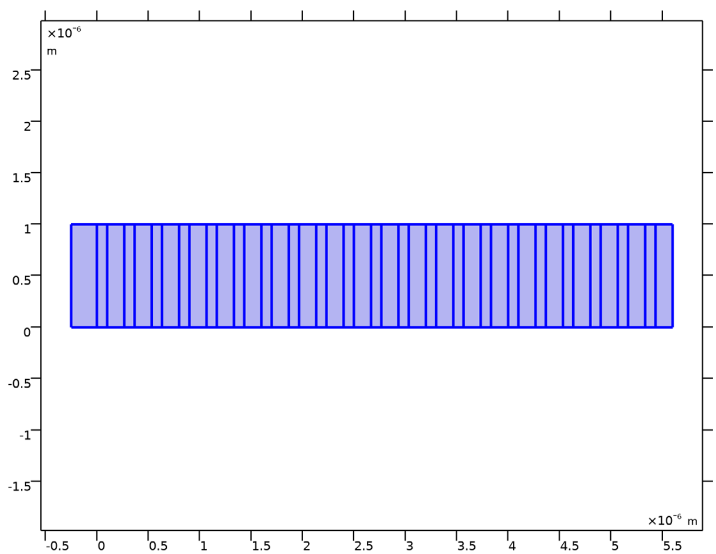


Integration 3 (intop3)

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

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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



Variables 1

Add variables for the input power and the transmitted power, using the integration operators defined above.

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`Pin`	`0.5*intop1(abs(EinODE)^2/Z0_const* ewt.nxx)`		`Input power`
`Pt`	`intop2(ewt.nPoav)`		`Transmitted power`

ELECTROMAGNETIC WAVES, TRANSIENT (EWT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Transient (ewt)**.

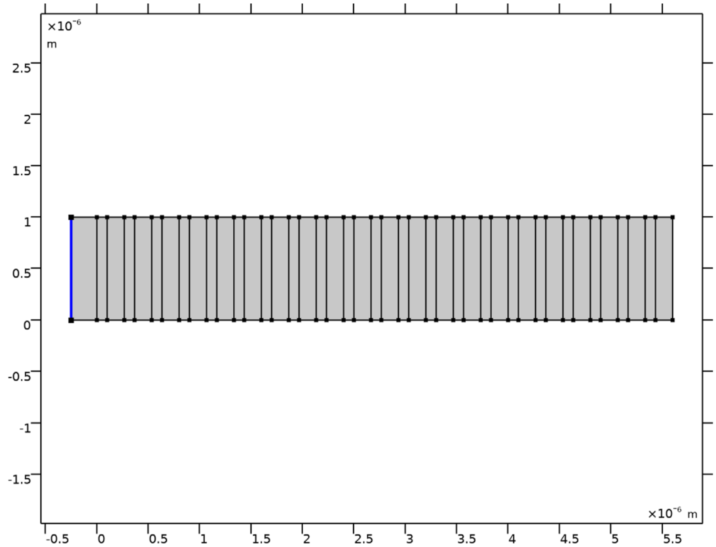
2 In the **Settings** window for **Electromagnetic Waves, Transient**, locate the **Components** section.

3 From the **Electric field components solved for** list, choose **Out-of-plane vector**, to solve for only the out-of-plane (z) polarization.

Scattering Boundary Condition 1

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

2 Select Boundary 1 only.



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

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

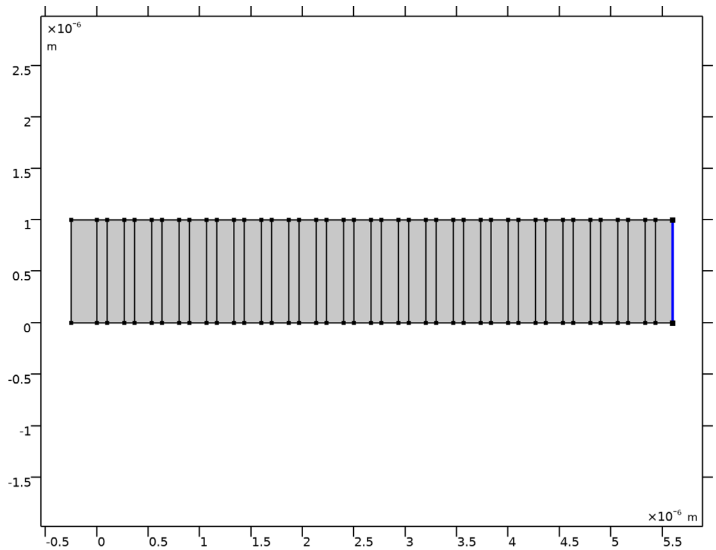
5 Specify the $\mathbf{E}_0$ vector as

0	x
0	y
Ein(t)	z

Scattering Boundary Condition 2

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

2 Select Boundary 130 only.

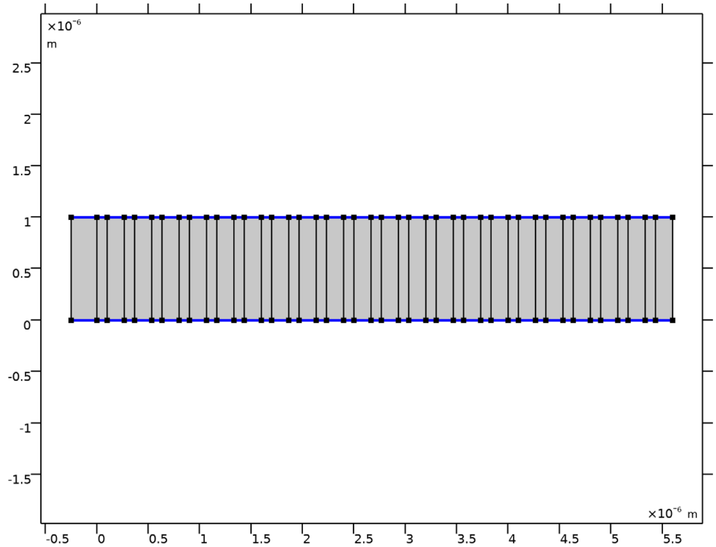


Perfect Magnetic Conductor 1

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

2 In the **Settings** window for **Perfect Magnetic Conductor**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Top and Bottom Exterior Boundaries**.



Add a dependent variable representing the input electric field. This will make it possible to perform an FFT on that variable. However, to be able to add the dependent variable, using a **Global Equation** node, the **Equation-Based Contributions** check box must be enabled in the **Show More Options** dialog.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

6 Click **OK**.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

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
`EinODE`	`EinODE-Ein(t)`	`Ein(0)`	`d(Ein(t),t)`	`Input electric field`

This will make the dependent variable `EinODE` follow the input electric field, defined by the function `Ein(t)`.

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `electric` in the text field.

6 Click **Filter**.

7 In the tree, select **Electromagnetics>Electric field (V/m)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, click **OK**, to select the Electric field.

MESH 1

Now, add a mesh that resolves also the higher frequencies properly.

Size 1

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

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

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

4 Select Boundary 3 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `lda0/2/n0/6`.

Here, we make sure that the mesh resolves the waves at the maximum frequency, which is twice the center frequency and where thereby the wavelength is half of the center wavelength.

8 Right-click **Size 1** and choose **Duplicate**.

Size 2

1 In the **Model Builder** window, click **Size 2**.

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

3 From the **Selection** list, choose **Top High-Index Material Boundaries**.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lda0/2/nH/6`.

5 Right-click **Size 2** and choose **Duplicate**.

Size 3

1 In the **Model Builder** window, click **Size 3**.

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

3 From the **Selection** list, choose **Top Low-Index Material Boundaries**.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `lda0/2/nL/6`.

Edge 1

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

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

3 From the **Selection** list, choose **Top Exterior Boundaries**.

Copy Edge 1

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

2 In the **Settings** window for **Copy Edge**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Top Exterior Boundaries**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Bottom Exterior Boundaries**.

Distribution 1

1 Right-click **Mesh 1** and choose **Distribution**.

2 Select Boundary 1 only.

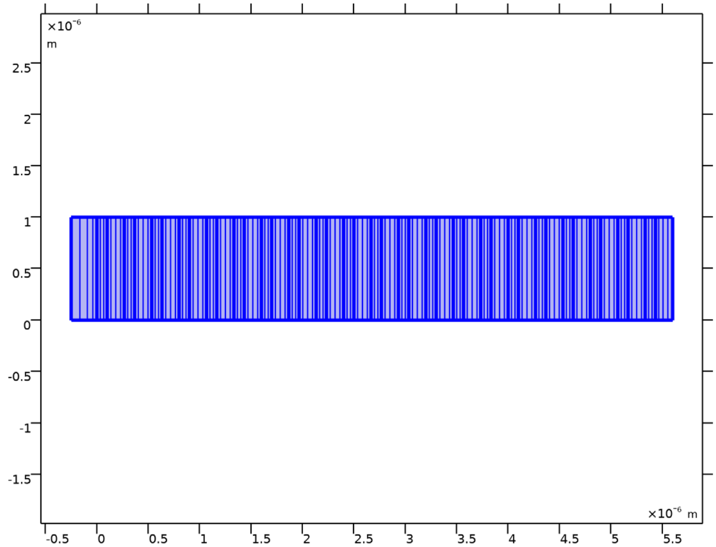
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `1`.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click **Build All**.



STUDY 1

Step 1: Time Dependent

1. In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3. From the **Time unit** list, choose **fs**.
4. In the **Output times** text field, type `range(0,0.25*T0,Tmax)`.
5. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electromagnetic Waves, Frequency Domain (ewfd)**.

Step 2: Time to Frequency FFT

1. In the **Model Builder** window, click **Step 2: Time to Frequency FFT**.
2. In the **Settings** window for **Time to Frequency FFT**, locate the **Study Settings** section.
3. From the **Time unit** list, choose **fs**.
4. In the **End time** text field, type `Tend`.
5. From the **Frequency unit** list, choose **THz**.
6. In the **Maximum output frequency** text field, type `2*f0`.
7. Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Electromagnetic Waves, Frequency Domain (ewfd)**.

Step 3: Combine Solutions

1 In the **Model Builder** window, click **Step 3: Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 In the **Excluded if** text field, type `freq<0.5*f0||freq>1.5*f0`, to remove too low and too high frequencies.

Solution 1 (sol1)

Create the default solver sequence and then edit the time stepping to use a constant step size.

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Manual**.

5 In the **Time step** text field, type `0.01*T0`.

6 In the **Study** toolbar, click **= Compute**.

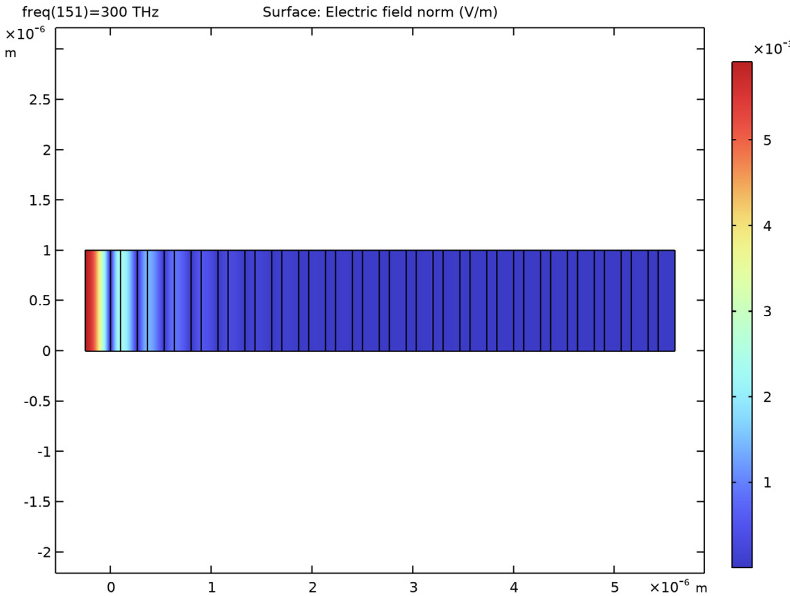
RESULTS

Electric Field (ewt, FD)

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

2 Locate the **Data** section. From the **Parameter value (freq (THz))** list, choose **300**, which is the center frequency (in the middle of the stopband).

3 In the **Electric Field (ewt, FD)** toolbar, click **Plot**.



Electric Field (ewt, TD)

Add an additional plot group to visualize the time dependent solution.

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

Surface 1

1 Right-click **Electric Field (ewt, TD)** and choose **Surface**.

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

3 In the **Expression** text field, type `ewt.Ez`.

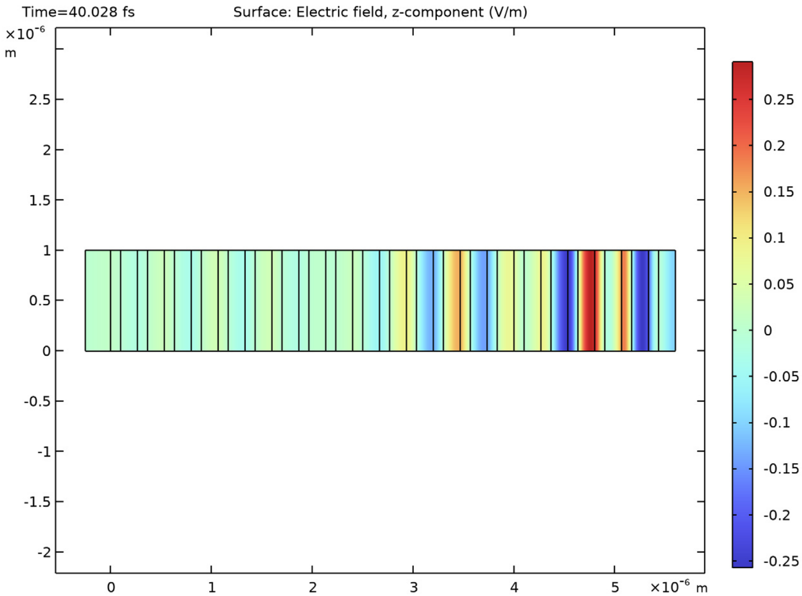
Electric Field (ewt, TD)

1 In the **Model Builder** window, click **Electric Field (ewt, TD)**.

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

3 From the **Time (fs)** list, choose **40.028**, showing the field distribution as the pulse has almost propagated through the mirror structure.

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



Animation 1

In the **Electric Field (ewt, TD)** toolbar, click **Animation** and choose **Player**, to display an animation of the temporal behavior of the field.

Transmittance

Add a plot group for displaying the transmittance versus frequency.

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

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

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`Pt/Pin`		`Transmittance (ewt)`

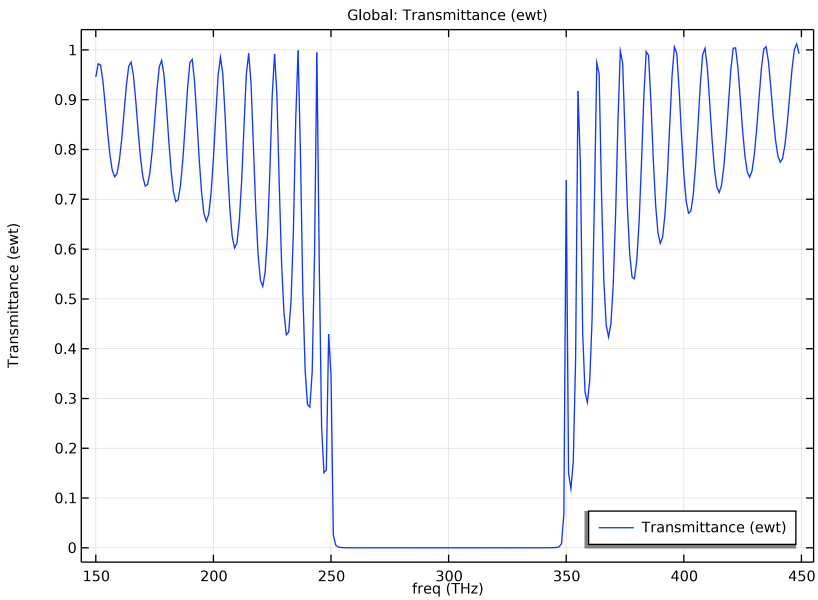
Transmittance

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

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

3 From the **Position** list, choose **Lower right**.

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



Energy

Verify that the energy has decayed from the mirror by the end of the simulation, by plotting the electromagnetic energy versus time.

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

4 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

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

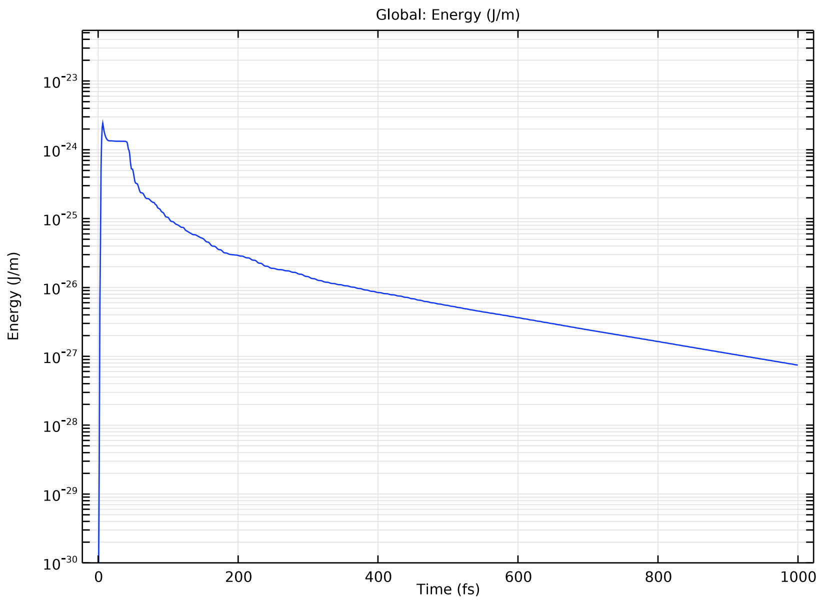
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	**Unit**	**Description**
`intop3(ewt.W)`	`J/m`	`Energy`

Energy

1 In the **Model Builder** window, click **Energy**.
2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.
3 Select the **y-axis log scale** check box.
4 Select the **Manual axis limits** check box.
5 In the **y minimum** text field, type `1e-30`.
6 In the **Energy** toolbar, click **Plot**.



Indeed, there is almost no energy left in the mirror at the end of the simulation.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

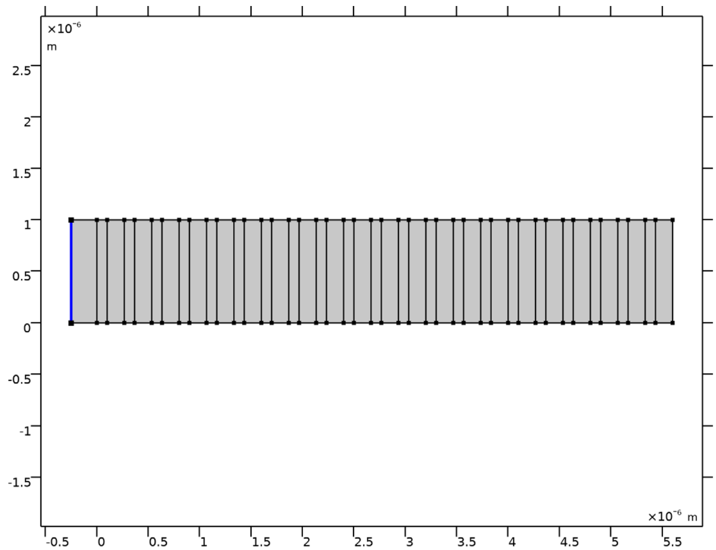
Now, make a frequency domain simulation, to compare with the results from the time-to-frequency FFT analysis.

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 Select Boundary 1 only.



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

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

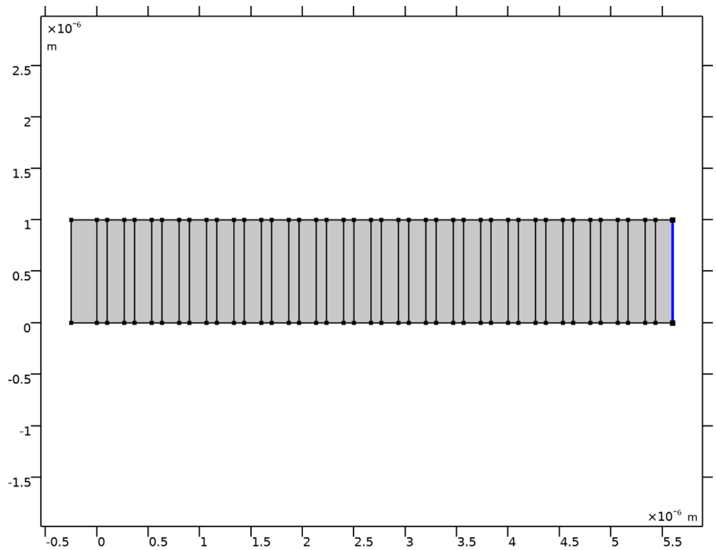
5 Specify the $\mathbf{E}_0$ vector as

0	x
0	y
E0	z

Scattering Boundary Condition 2

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

2 Select Boundary 130 only.

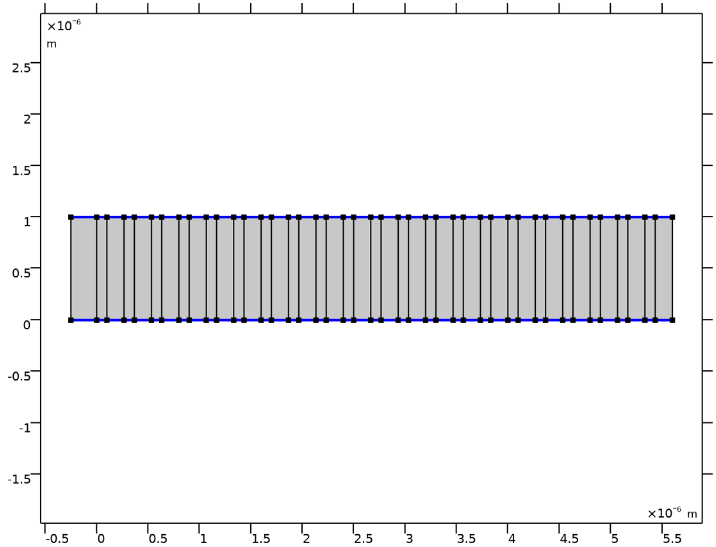


Perfect Magnetic Conductor 1

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

2 In the **Settings** window for **Perfect Magnetic Conductor**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Top and Bottom Exterior Boundaries**.



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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Transient (ewt)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

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

6 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 `range(0.5*f0,deltaf,1.5*f0)`.

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

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Electromagnetic Waves, Transient (ewt)**.

6 Right-click and choose **Disable in Model**.

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

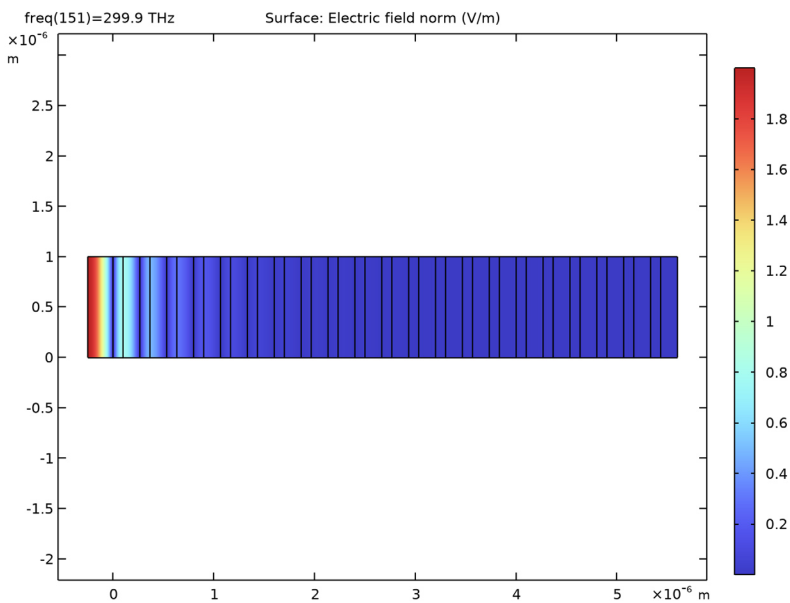
RESULTS

Electric Field (ewfd)

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

2 From the **Parameter value (freq (THz))** list, choose **299.9**, which is the center frequency in the sweep.

3 In the **Electric Field (ewfd)** toolbar, click **Plot**.



Notice that the field distribution is the same as for the time-to-frequency FFT result, except that the amplitude is different.

Global 2

1 In the **Model Builder** window, right-click **Transmittance** and choose **Global**.

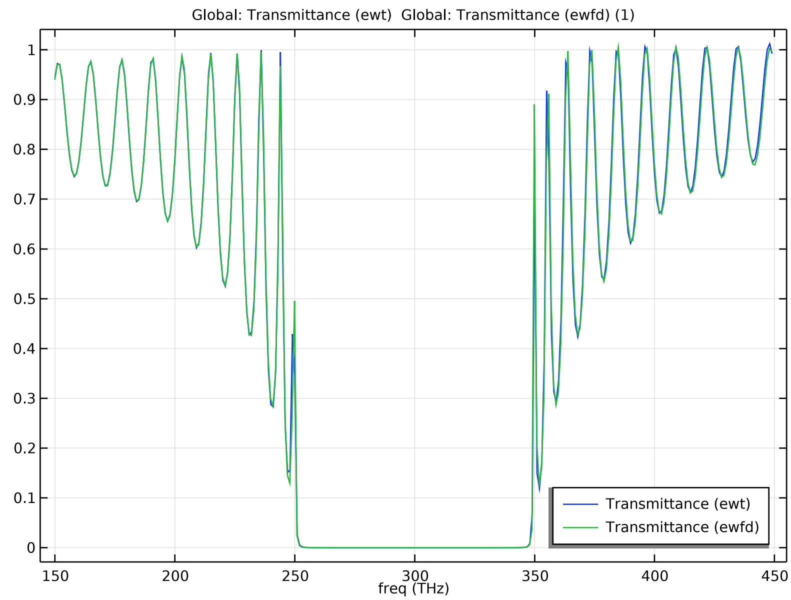
2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Probe Solution 4 (sol4)**.

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

Expression	Unit	Description
`intop2(ewfd.nPoav)/intop1(0.5*E0^2/Z0_const*n0)`	1	`Transmittance (ewfd)`

5 In the **Transmittance** toolbar, click **Plot**.



The curves from the two studies agree very well.