Fresnel Equations

Introduction

A plane electromagnetic wave propagating through free space is incident at an angle upon an infinite dielectric medium. This model computes the reflection and transmission coefficients and compares the results to the Fresnel equations.

Model Definition

A plane wave propagating through free space ($n = 1$) as shown in Figure 1 is incident upon an infinite dielectric medium ($n = 1.5$) and is partially reflected and partially transmitted. If the electric field is *p-polarized*—that is, if the electric field vector is in the same plane as the Poynting vector and the surface normal—then there will be no reflections at an incident angle of roughly 56°, known as the *Brewster angle*.

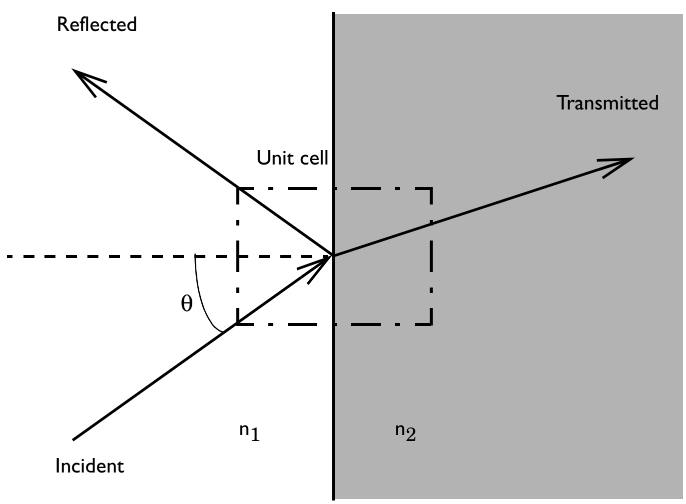


Figure 1: A plane wave propagating through free space incident upon an infinite dielectric medium.

Although, by assumption, space extends to infinity in all directions, it is sufficient to model a small unit cell, as shown in Figure 1; a Floquet-periodic boundary condition applies on the top and bottom unit-cell boundaries because the solution is periodic along the interface. This model uses a 3D unit cell, and applies perfect electric conductor and perfect magnetic conductor boundary conditions as appropriate to model out-of-plane symmetry. The angle of incidence ranges between 0–90° for both polarizations.

For comparison, Ref. 1 and Ref. 2 provide analytic expressions for the reflectance and transmittance. Reflection and transmission coefficients for s-polarization and p-polarization are defined respectively as

$$r_s = \frac{n_1\cos\theta_{\text{incident}} - n_2\cos\theta_{\text{transmitted}}}{n_1\cos\theta_{\text{incident}} + n_2\cos\theta_{\text{transmitted}}} \tag{1}$$

$$t_s = \frac{2n_1\cos\theta_{\text{incident}}}{n_1\cos\theta_{\text{incident}} + n_2\cos\theta_{\text{transmitted}}} \tag{2}$$

$$r_p = \frac{n_2\cos\theta_{\text{incident}} - n_1\cos\theta_{\text{transmitted}}}{n_1\cos\theta_{\text{transmitted}} + n_2\cos\theta_{\text{incident}}} \tag{3}$$

$$t_p = \frac{2n_1\cos\theta_{\text{incident}}}{n_1\cos\theta_{\text{transmitted}} + n_2\cos\theta_{\text{incident}}} \tag{4}$$

Reflectance and transmittance are defined as

$$R = |r|^2 \tag{5}$$

$$T = \frac{n_2\cos\theta_{\text{transmitted}}}{n_1\cos\theta_{\text{incident}}}|t|^2 \tag{6}$$

The Brewster angle at which $r_\text{p} = 0$ is defined as

$$\theta_B = \operatorname{atan}\frac{n_2}{n_1} \tag{7}$$

Results and Discussion

Figure 2 is a combined plot of the y component of the electric-field distribution and the power flow visualized as an arrow plot for the TE case.

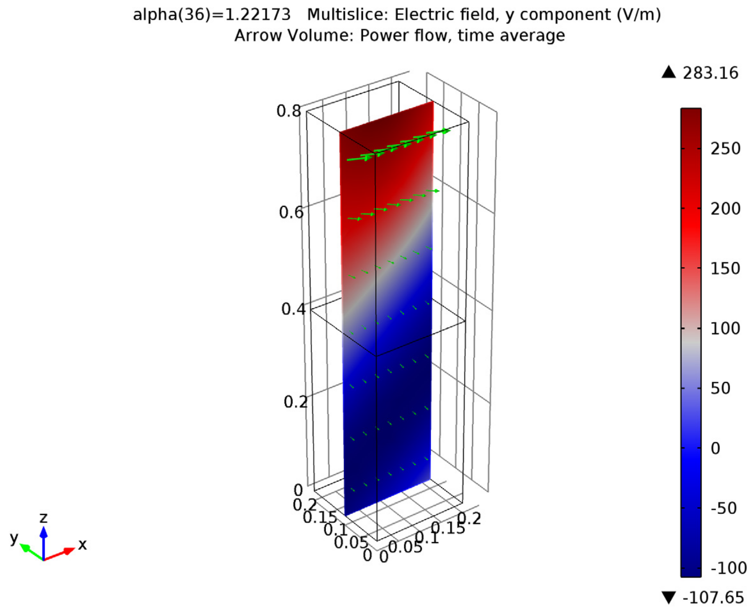


Figure 2: Electric field, E_y (slice) and power flow (arrows) for TE incidence at 70° inside the unit cell.

For the TM case, Figure 3 visualizes the *y* component of the magnetic-field distribution instead, again in combination with the power flow.

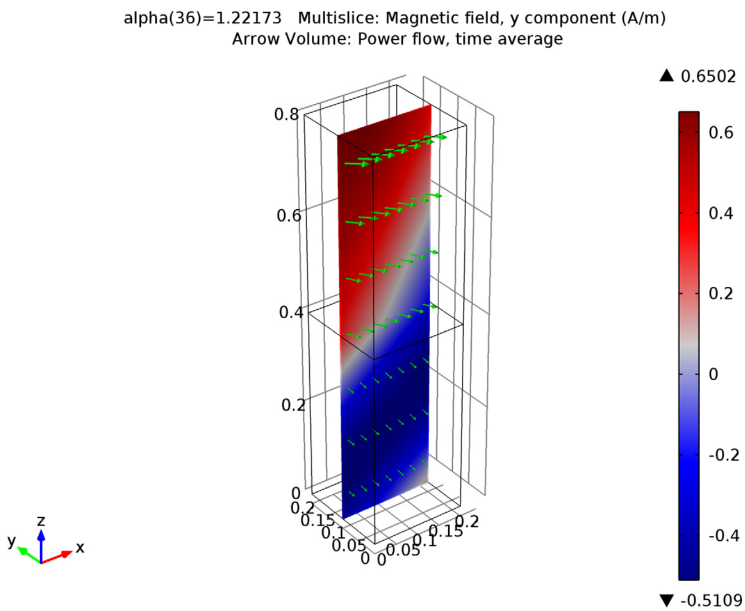


Figure 3: Magnetic field, H_y (slice) and power flow (arrows) for TM incidence at 70° inside the unit cell.

Note that the sum of reflectance and transmittance in Figure 4 and Figure 5 equals 1, showing conservation of power. Figure 5 also shows that the reflectance around 56°—the Brewster angle in the TM case—is close to zero.

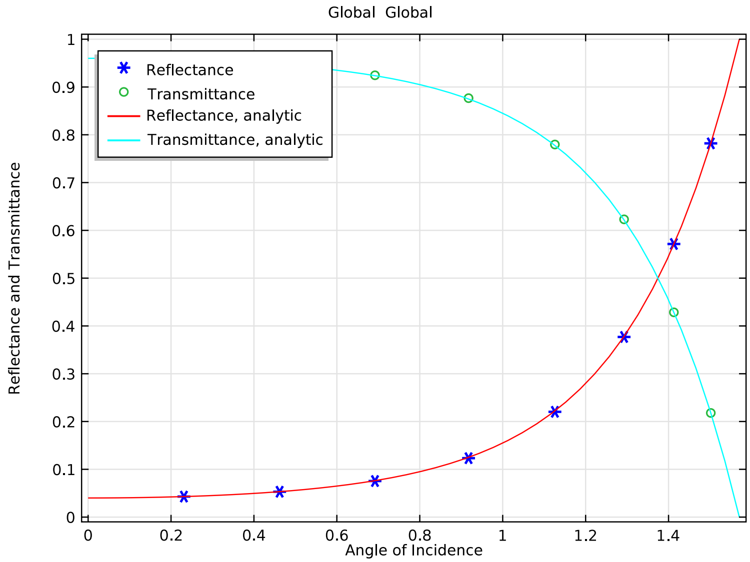


Figure 4: The reflectance and transmittance for TE incidence agree well with the analytic solutions.

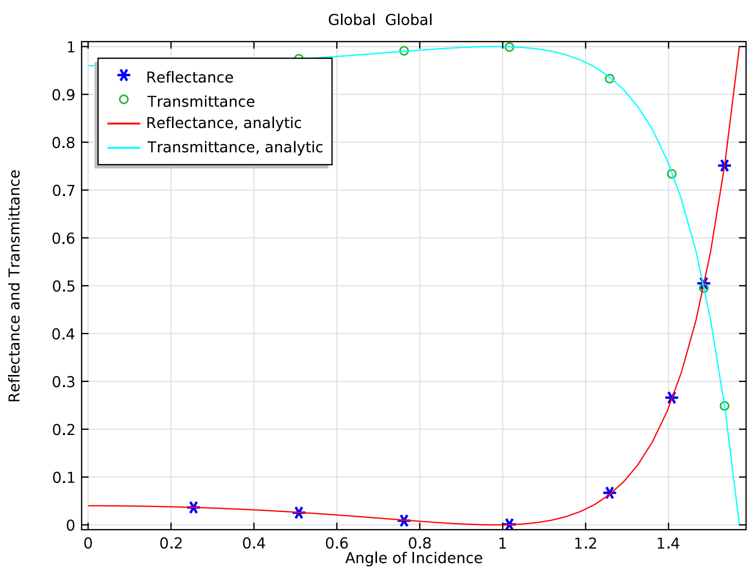


Figure 5: The reflectance and transmittance for TM incidence agree well with the analytic solutions. The Brewster angle is also observed at the expected location.

References

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, Wiley, 1989.

2. B.E.A. Saleh and M.C. Teich, *Fundamentals of Photonics*, Wiley, 1991.

Model Library path: `Wave_Optics_Module/Verification_Models/fresnel_equations`

Modeling Instructions

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

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **3D** button.

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

3 Click the **Add** button.

4 Click the **Study** button.

5 In the tree, select **Preset Studies>Frequency Domain**.

6 Click the **Done** button.

GLOBAL DEFINITIONS

Define some parameters that are useful when setting up the mesh and the study.

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
n_air	1	1.000	Refractive index, air
n_slab	1.5	1.500	Refractive index, slab
lda0	1[m]	1.000 m	Wavelength
f0	c_const/lda0	2.998E8 1/s	Frequency
alpha	70[deg]	1.222 rad	Angle of incidence
beta	asin(n_air*sin(alpha)/ n_slab)	0.6770 rad	Refraction angle
h_max	lda0/6	0.1667 m	Maximum element size, air
alpha_brewster	atan(n_slab/n_air)	0.9828 rad	Brewster angle, TM only
r_s	(n_air*cos(alpha)-n_slab* cos(beta))/ (n_air*cos(alpha)+n_slab* cos(beta))	-0.5474	Reflection coefficient, TE
r_p	(n_slab*cos(alpha)-n_air* cos(beta))/ (n_air*cos(beta)+n_slab*cos(alpha))	-0.2061	Reflection coefficient, TM
t_s	(2*n_air*cos(alpha))/ (n_air*cos(alpha)+n_slab* cos(beta))	0.4526	Transmission coefficient, TE
t_p	(2*n_air*cos(alpha))/ (n_air*cos(beta)+n_slab*cos(alpha))	0.5292	Transmission coefficient, TM

The angle of incidence is updated while running the parametric sweep. The refraction (transmitted) angle is defined by Snell's law with the updated angle of incidence. The Brewster angle exists only for TM incidence, p-polarization, and parallel polarization.

DEFINITIONS

Variables 1

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

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
ka	ewfd.k0	rad/m	Propagation constant, air
kax	ka*sin(alpha)	rad/m	kx for incident wave
kay	0		ky for incident wave
kaz	ka*cos(alpha)	rad/m	kz for incident wave
kb	n_slab*ewfd.k0	rad/m	Propagation constant, slab
kbx	kb*sin(beta)	rad/m	kx for refracted wave
kby	0		ky for refracted wave
kbz	kb*cos(beta)	rad/m	kz for refracted wave

GEOMETRY 1

First, create a block composed of two domains. Use layers to split the block.

Block 1

1 On the **Geometry** toolbar, click **Block**.

2 In the **Block** settings window, locate the **Size** section.

3 In the **Width** edit field, type 0.2.

4 In the **Depth** edit field, type 0.2.

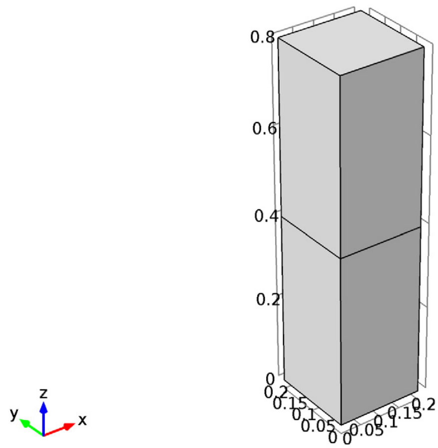
5 In the **Height** edit field, type 0.8.

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

Layer name	Thickness (m)
Layer 1	0.4

7 Click the **Build All Objects** button.

8 Click the **Zoom Extents** button on the Graphics toolbar.



Choose wireframe rendering to get a better view of each boundary.

9 Click the **Wireframe Rendering** button on the Graphics toolbar.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Set up the physics based on the direction of propagation and the E-field polarization. First, assume a TE-polarized wave which is equivalent to s-polarization and perpendicular polarization. E_x and E_z are zero while E_y is dominant.

The wave is excited from the port on the top.

Port 1

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

2 Select Boundary 7 only.

3 In the **Port** settings window, locate the **Port Properties** section.

4 From the **Wave excitation at this port** list, choose **On**.

5 Locate the **Port Mode Settings** section. Specify the $\mathbf{E}_0$ vector as

`0`	x
`exp(-i*kax*x)[V/m]`	y
`0`	z

6 In the β edit field, type `abs(kaz)`.

Port 2

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

2 Select Boundary 3 only.

3 In the **Port** settings window, locate the **Port Mode Settings** section.

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

`0`	x
`exp(-i*kbx*x)[V/m]`	y
`0`	z

5 In the β edit field, type `abs(kbz)`.

The bottom surface is an observation port. The S21-parameter from Port 1 and Port 2 provides the transmission characteristics.

The E-field polarization has E_y only and the boundaries are always either parallel or perpendicular to the E-field polarization. Apply periodic boundary conditions on the boundaries parallel to the E-field except those you already assigned to the ports.

Periodic Condition 1

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

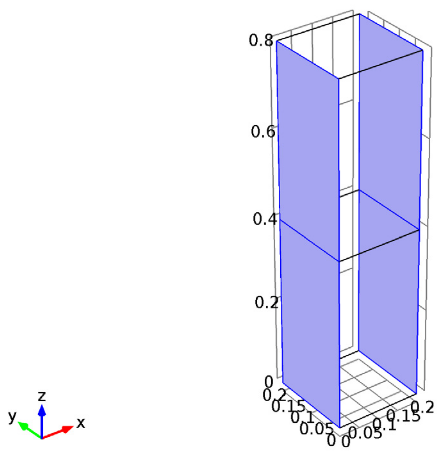
2 Select Boundaries 1, 4, 10, and 11 only.

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

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

5 Specify the $\mathbf{k}_\mathbf{F}$ vector as

kax	x
0	y
0	z

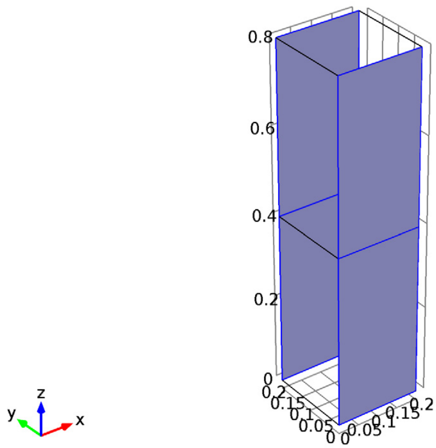


Apply a perfect electric conductor condition on the boundaries perpendicular to the E-field. This condition creates a virtually infinite modeling space.

Perfect Electric Conductor 2

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

2 Select Boundaries 2, 5, 8, and 9 only.



MATERIALS

Now set up the material properties based on refractive index. The top half is filled with air.

Material 1

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

2 Select Domain 2 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Refractive index	n	n_air	1	Refractive index

5 Right-click **Component 1>Materials>Material 1** and choose **Rename**.

6 Go to the **Rename Material** dialog box and type `Air` in the **New name** edit field.

7 Click **OK**.

The bottom half is glass.

Material 2

1 Right-click **Materials** and choose **New Material**.
2 Select Domain 1 only.
3 In the **Material** settings window, locate the **Material Contents** section.
4 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Refractive index	n	`n_slab`	1	Refractive index

5 Right-click **Component 1>Materials>Material 2** and choose **Rename**.
6 Go to the **Rename Material** dialog box and type `Glass` in the **New name** edit field.
7 Click **OK**.

MESH 1

The periodic boundary condition performs better if the mesh is identical on the periodicity boundaries. This is especially important when dealing with vector degrees of freedom, as will be the case in the TM version of this model. The maximum element size is smaller than 0.2 times the wavelength. The bottom half domain is scaled inversely by the refractive index of the material.

Size

1 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **Size**.
2 In the **Size** settings window, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** edit field, type `h_max`.

Size 1

1 In the **Model Builder** window, under **Component 1>Mesh 1** click **Size 1**.
2 In the **Size** settings window, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 1 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated edit field, type `h_max/n_slab`.

Free Triangular 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Triangular**.

2 Select Boundaries 1 and 4 only.

Copy Face 1

1 Right-click **Mesh 1** and choose **Copy Face**.

2 Select Boundaries 1 and 4 only.

3 In the **Copy Face** settings window, locate the **Destination Boundaries** section.

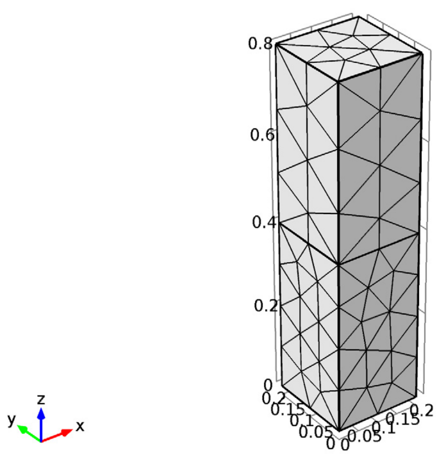
4 Select the **Destination group focus** toggle button.

5 Select Boundaries 10 and 11 only.

Free Tetrahedral 1

1 Right-click **Mesh 1** and choose **Free Tetrahedral**.

2 Right-click **Mesh 1** and choose **Build All**.



STUDY 1

Step 1: Frequency Domain

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

2 In the **Frequency Domain** settings window, locate the **Study Settings** section.

3 In the **Frequencies** edit field, type `f0`.

Parametric Sweep

1 On the **Study** toolbar, click **Parametric Sweep**.

2 In the **Parametric Sweep** settings window, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter names	Parameter value list
alpha	`range(0,2[deg],90[deg])`

Use a direct solver instead of an iterative one for faster convergence.

Solver 1

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

2 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** right-click **Direct** and choose **Enable**.

3 Right-click **Study 1>Solver Configurations>Solver 1>Stationary Solver 1>Direct** and choose **Compute**.

RESULTS

Electric Field (ewfd)

The default plot is the E-field norm for the last solution, which corresponds to tangential incidence. Replace the expression with E_y, add an arrow plot of the power flow (Poynting vector), and choose a more interesting angle of incidence for the plot.

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

2 In the **Multislice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field>Electric field, y component (ewfd.Ey)**.

3 Locate the **Multiplane Data** section. Find the **x-planes** settings and in the **Planes** edit field, type `0`.

4 Find the **z-planes** settings and in the **Planes** edit field, type `0`.

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

6 In the **Model Builder** window, right-click **Electric Field (ewfd)** and choose **Arrow Volume**.

7 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves,**

Frequency Domain>Energy and power>Power flow, time average (ewfd.Poavx,...,ewfd.Poavz).

8 Locate the **Arrow Positioning** section. Find the **y-grid points** and in the **Points** edit field, type 1.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

10 In the **Model Builder** window, click **Electric Field (ewfd)**.

11 In the **3D Plot Group** settings window, locate the **Data** section.

12 From the **Parameter value (alpha)** list, choose **1.22173**.

13 On the **3D plot group** toolbar, click **Plot**.

14 Click the **Zoom Extents** button on the Graphics toolbar. The plot should look like that in Figure 2.

Add a 1D plot to see the reflection and transmission versus the angle of incidence.

1D Plot Group 2

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

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

3 Select the **x-axis label** check box.

4 In the associated edit field, type `Angle of Incidence`.

5 Select the **y-axis label** check box.

6 In the associated edit field, type `Reflectance and Transmittance`.

7 Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.

8 On the **1D plot group** toolbar, click **Global**.

9 In the **Global** settings window, locate the **y-Axis Data** section.

10 In the table, enter the following settings:

Expression	Unit	Description
abs(ewfd.S11)^2	1	Reflectance
abs(ewfd.S21)^2	1	Transmittance

11 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Line** list, choose **None**.

12 From the **Marker** list, choose **Cycle**.

13 From the **Line** list, choose **None**.

14 From the **Marker** list, choose **Cycle**.

15 On the **1D plot group** toolbar, click **Global**.

16 In the **Global** settings window, locate the **y-Axis Data** section.

17 In the table, enter the following settings:

Expression	Unit	Description
`abs(r_s)^2`		`Reflectance, analytic`
`n_slab*cos(beta)/(n_air*cos(alpha))*abs(t_s)^2`		`Transmittance, analytic`

18 On the **1D plot group** toolbar, click **Plot**.

19 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Rename**.

20 Go to the **Rename 1D Plot Group** dialog box and type `Reflection and Transmission` in the **New name** edit field.

21 Click **OK**. Compare the resulting plots with Figure 4.

The remaining instructions are for the case of TM-polarized wave, p-polarization, and parallel polarization. In this case, E_y is zero while E_x and E_z characterize the wave. In other words, H_y is dominant while H_x and H_z are effectless. Thus, the H-field is perpendicular to the plane of incidence and it is convenient to solve the model for the H-field.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Port 1

1 In the **Port** settings window, locate the **Port Mode Settings** section.

2 From the **Input quantity** list, choose **Magnetic field**.

3 Specify the $\mathbf{H}_0$ vector as

`0`	x
`exp(-i*kax*x)[A/m]`	y
`0`	z

Port 2

1 In the **Model Builder** window, under **Component 1>Electromagnetic Waves, Frequency Domain** click **Port 2**.

2 In the **Port** settings window, locate the **Port Mode Settings** section.

3 From the **Input quantity** list, choose **Magnetic field**.

4 Specify the $\mathbf{H}_0$ vector as

0	x
exp(-i*kbx*x)[A/m]	y
0	z

Perfect Electric Conductor 2

The model utilizes the H-field for the TM case and the remaining boundaries need to be perfect magnetic conductors.

1 In the **Model Builder** window, under **Component 1>Electromagnetic Waves, Frequency Domain** right-click **Perfect Electric Conductor 2** and choose **Disable**.

Perfect Magnetic Conductor 1

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

2 Select Boundaries 2, 5, 8, and 9 only.

To keep the solution and plots for the TE case, do as follows:

STUDY 1

Solver 1

1 In the **Model Builder** window, under **Study 1>Solver Configurations** right-click **Solver 1** and choose **Solution>Copy**.

RESULTS

Electric Field (ewfd)

1 In the **Model Builder** window, under **Results** Ctrl-click to select both **Results>Electric Field (ewfd)** and **Results>Reflection and Transmission,** then right-click and choose **Duplicate**.

Electric Field (ewfd)

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

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 2**.

Reflection and Transmission

1 In the **Model Builder** window, under **Results** click **Reflection and Transmission**.

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

3 From the **Data set** list, choose **Solution 2**.

STUDY 1

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

RESULTS

Electric Field (ewfd) 1

1 In the **Multislice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field>Magnetic field, y component (ewfd.Hy)**.

2 On the **3D plot group** toolbar, click **Plot**. This reproduces Figure 3.

Reflection and Transmission 1

1 In the **Model Builder** window, expand the **Results>Reflection and Transmission 1** node, then click **Global 2**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`abs(r_p)^2`		`Reflectance, analytic`
`n_slab*cos(beta)/(n_air*cos(alpha))*abs(t_p)^2`		`Transmittance, analytic`

4 On the **1D plot group** toolbar, click **Plot**. The plot should look like Figure 5. The Brewster angle is observed around 56 degrees, which is close to the analytic value.