Plasmonic Wire Grating

Introduction

A plane electromagnetic wave is incident on a wire grating on a dielectric substrate. The model computes transmission and reflection coefficients for the refraction, specular reflection, and first order diffraction.

Model Definition

Figure 1 shows the considered grating, with a gold wire on a dielectric material with refractive index n_β. The grating constant, or the distance between the wires, is d. A plane-polarized wave traveling through a medium with refractive index n_α is incident on the grating, at an angle α in a plane perpendicular to the grating.

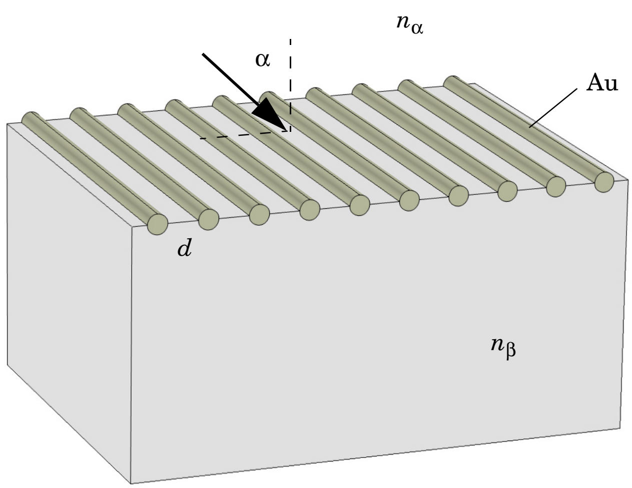


Figure 1: The modeled grating. The model considers a unit cell of a slice through this geometry. The grating is assumed to consist of an infinite number of infinitely long wires.

If the wavelengths involved in the model are sufficiently short compared to the grating constant, one or several diffraction orders can be present. The diagram in Figure 2 shows two transmissive paths taken by light incident on adjacent cells of the grating, exactly one grating constant apart.

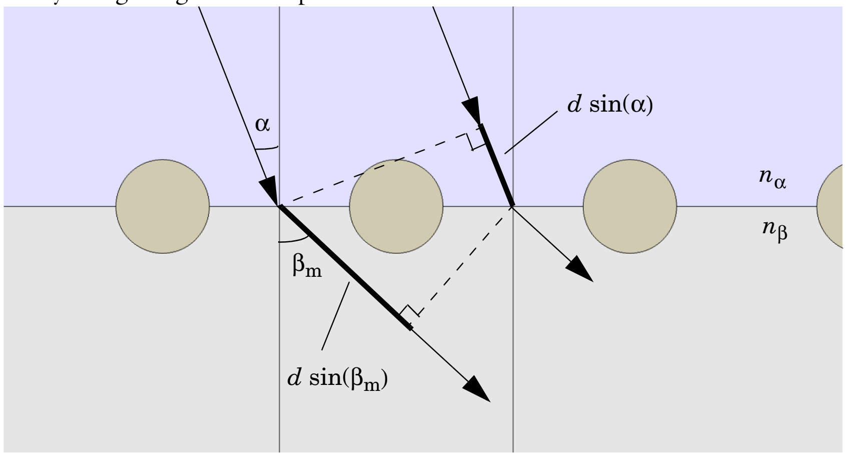


Figure 2: The geometric path lengths of two transmitted parallel beams. The optical path length is the geometric path length multiplied by the local refractive index.

The criterion for positive interference is that the difference in optical path length along the two paths equals an integer number of vacuum wavelengths, or:

$$m\lambda_0 = d(n_\beta \sin\beta_m - n_\alpha \sin\alpha) \tag{1}$$

with $m = 0, \pm 1, \pm 2, \ldots$, λ_0 the vacuum wavelength, and β_m the transmitted diffracted beam of order m. For $m = 0$, this reduces to refraction, as described by Snell's law:

$$\sin\beta_0 = \frac{n_\alpha}{n_\beta}\sin\alpha$$

Because the sine functions can only vary between -1 and 1, the existence of higher diffraction order requires that

$$-(n_\alpha + n_\beta) < \frac{m\lambda_0}{d} < (n_\alpha + n_\beta)$$

The model instructions cover only first order diffraction, and are hence only valid for under the condition

$$2\lambda_0 > d(n_\alpha|\sin\alpha| + n_\beta) \tag{2}$$

Note that for the special cases of perpendicular and grazing incidence, the right-hand side of the inequality evaluates to dn_β and $d(n_\alpha + n_\beta)$ respectively.

Figure 3 shows the corresponding paths of the reflected light.

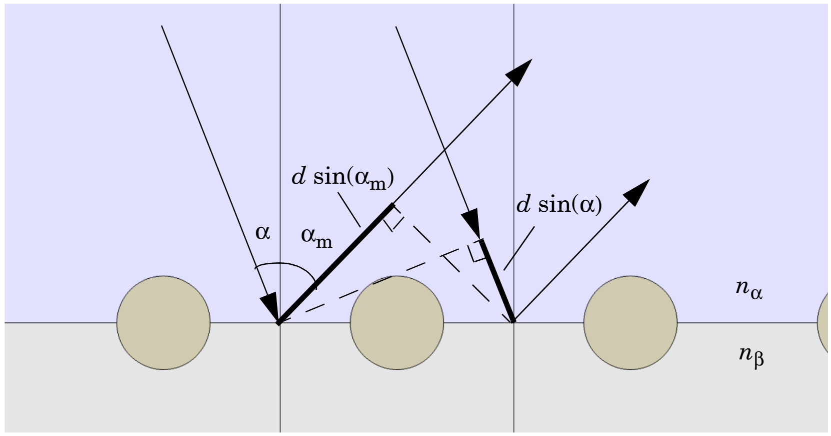


Figure 3: The geometric path lengths of two parallel reflected beams.

For positive interference we get

$$m\lambda_0 = dn_\alpha(\sin\alpha_m - \sin\alpha), \tag{3}$$

where α_m is the reflected beam of diffraction order m. Setting $m = 0$ in this equation renders

$$\sin\alpha_0 = \sin\alpha,$$

or specular reflection. The condition for no reflected diffracted beams of order 2 or greater being present is

$$2\lambda_0 > dn_a(1 + |\sin\alpha|). \tag{4}$$

The model uses $n_\alpha = 1$ for air and $n_\beta = 1.2$ for the dielectric substrate. Allowing for arbitrary angles of incidence and with a grating constant d = 400 nm, Equation 2 sets the validity limit to vacuum wavelengths greater than 440 nm. The model uses λ_0 = 441 nm. For the wire, a complex-valued permittivity of $-1.75 - 5.4i$ approximates that of gold at the corresponding frequency.

The performance of the grating depends on the polarization of the incident wave. Therefore both a transverse electric (TE) and a transverse magnetic (TM) case are considered. The TE wave has the electric field component in the *z* direction, out of the modeling *xy*-plane. For the TM wave, the incident electric field is specified in the *xy*-plane and perpendicular to the direction of propagation. The angle of incidence is for both cases swept from 0 to $\pi/2$, with a pitch of $\pi/40$.

Results and Discussion

As an example of the output from the model, Figure 4 and Figure 5 show the electric field norm for an angle of incidence equal to $\pi/5$, for the TE and TM case respectively.

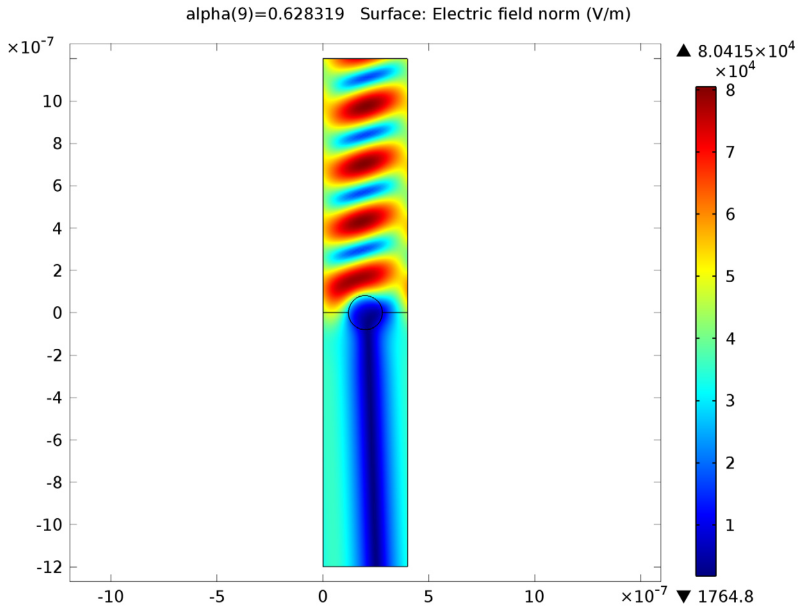


Figure 4: Electric field norm for TE incidence at $\pi/5$*.*

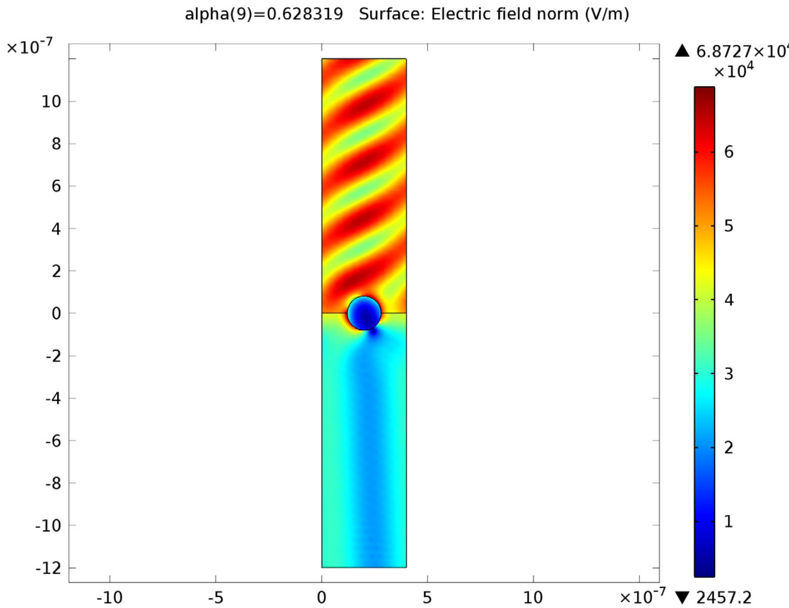


Figure 5: Electric field norm for TM incidence at π/5.

All the computed transmission and reflection coefficients for TE incidence are plotted in Figure 6. R_0, the coefficient for specular reflection, increases rather steadily with the angle of incidence. This is both because of reflection in the material interface and because the wave "sees" the wire as increasingly wider at greater angles—the same effect as achieved by a Venetian blind. T_0, the refracted but not diffracted transmission, decreases accordingly. For the considered wavelength to diffraction constant ratio, positive first order diffraction only exists as a transmitted beam (T_1) for nearly perpendicular incidence only. The reflected positive first order diffraction (R_1) would need a shorter wavelength or a wider grating to show up. Instead, the most prominent diffraction order is -1, both for reflection (R_{-1}) and transmission (T_{-1}).

Note first that the sum of all coefficients is consistently less than 1. This is because of the dielectric losses in the wire. This is even more apparent for TM incidence, as Figure 7 shows. Here, approximately half of the wave is absorbed in the wire. Another important feature of the TM case is that there is very little specular reflection (R_0) around 60 degrees.

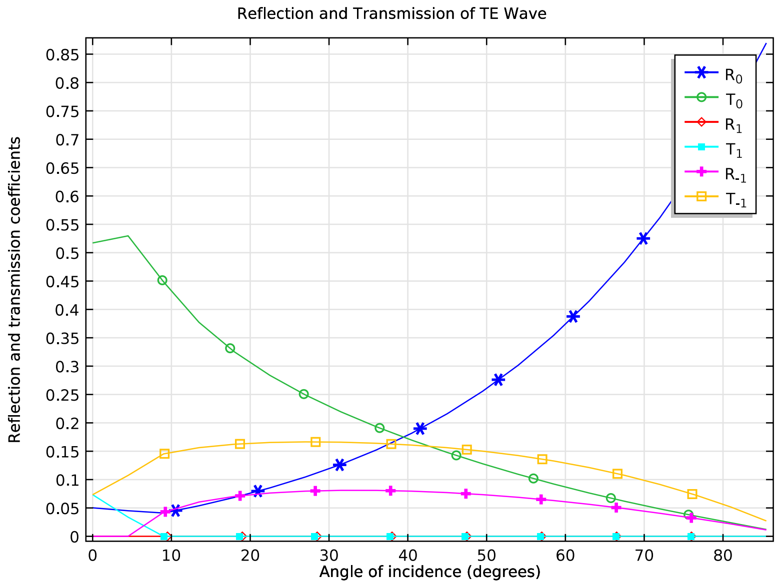


Figure 6: Transmission and reflection coefficients for TE incidence.

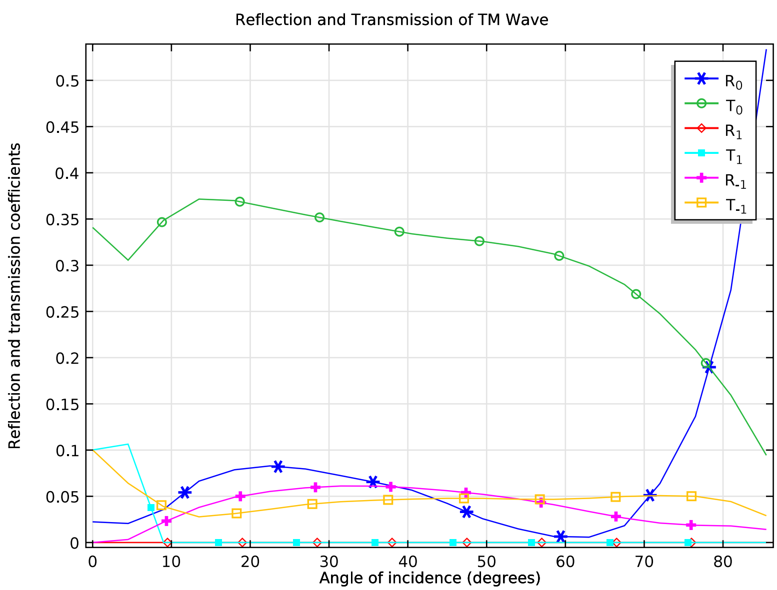


Figure 7: Transmission and reflection coefficients for TM incidence.

Notes About the COMSOL Implementation

The model is set up for one unit cell of the grating, flanked by Floquet boundary conditions describing the periodicity. As applied, this condition states that the solution on one side of the unit equals the solution on the other side multiplied by a complex-valued phase factor. The phase shift between the boundaries is evaluated from the perpendicular component of the wave vector. Because the periodicity boundaries are parallel with the y-axis, only the x-component is required. Note that due to the continuity of the field, the phase factor will be the same for the refracted and reflected beams as for the incident wave.

Port conditions are used both for specifying the incident wave and for letting the resulting solution leave the model without any non-physical reflections. In order to achieve perfect transmission through the port boundaries, one port for each mode $(m = 0, m = -1, m = 1)$ in each direction must be present. This gives a total of 6 ports.

The input to each port condition is an electric field vector and a propagation constant. The electric field should describe the specular reflection of the mode that you want the port to transmit. This convention makes it possible to have the input port automatically transmit the specular reflection of the incident wave, α_0. The propagation constant asked for is the component along the outwards-facing normal. It should be positive for all transmitted waves, so it safe to consistently refer to its absolute value.

The below table lists the variable names used in the model.

TABLE 4-1: VARIABLE NAMES

MODEL DESCRIPTION	MODEL	DESCRIPTION
n_α	`na`	Refractive index, air
n_β	`nb`	Refractive index, dielectric
α	`alpha`	Angle of incidence
α_1	`alpha_p`	Reflected diffraction angle, order 1
α_{-1}	`alpha_m`	Reflected diffraction angle, order -1
β_0	`beta`	Refraction angle
β_1	`beta_p`	Refracted diffraction angle, order 1
β_{-1}	`beta_m`	Refracted diffraction angle, order -1

Model Library path: `RF_Module/Optics_and_Photonics/plasmonic_wire_grating`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.
2 Click the **2D** button.
3 Click **Next**.
4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
5 Click **Next**.
6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.
2 In the **Parameters** settings window, locate the **Parameters** section.
3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
na	1	Refractive index, air
nb	1.2	Refractive index, dielectric
d	400[nm]	Grating constant
lam0	441[nm]	Vacuum wavelength
f0	c_const/lam0	Frequency
alpha	0	Angle of incidence

Although the angle of incidence will not remain constant at 0, it needs to be specified as a parameter to be accessible to the parametric solver.

Variables 1

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

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

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
alpha_p	asin(sin(alpha)+lam0/(d*na))	Reflected diffraction angle, m=1
alpha_m	asin(sin(alpha)-lam0/(d*na))	Reflected diffraction angle, m=-1
beta	asin(na*sin(alpha)/nb)	Refraction angle
beta_p	asin(na/nb*sin(alpha)+lam0/(d*nb))	Refracted diffraction angle, m=1
beta_m	asin(na/nb*sin(alpha)-lam0/(d*nb))	Refracted diffraction angle, m=-1

The variables you just defined were extracted from Equation 1 and Equation 3. The corresponding wave vectors require input from the physics interface and are preferably created under the **Definitions** node.

DEFINITIONS

Variables 2

1 In the **Model Builder** window, under **Model 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	DESCRIPTION
ka	emw.k0	Propagation constant, air
kax	ka*sin(alpha)	kx for incident wave
kay	-ka*cos(alpha)	ky for incident wave
kapx	ka*sin(alpha_p)	kx for reflected diffracted wave, m=1
kapy	ka*cos(alpha_p)	ky for reflected diffracted wave, m=1
kamx	ka*sin(alpha_m)	kx for reflected diffracted wave, m=-1
kamy	ka*cos(alpha_m)	ky for reflected diffracted wave, m=-1

NAME	EXPRESSION	DESCRIPTION
kb	nb*emw.k0	Propagation constant, dielectric
kbx	kb*sin(beta)	kx for refracted wave
kby	-kb*cos(beta)	ky for refracted wave
kbpx	kb*sin(beta_p)	kx for refracted diffracted wave, m=1
kbpy	-kb*cos(beta_p)	ky for refracted diffracted wave, m=1
kbmx	kb*sin(beta_m)	kx for refracted diffracted wave, m=-1
kbmy	-kb*cos(beta_m)	ky for refracted diffracted wave, m=-1

GEOMETRY 1

Create the geometry entirely in terms of the grating constant, for easy scalability.

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

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

3 In the **Width** edit field, type `d`.

4 In the **Height** edit field, type `3*d`.

5 Click the **Build Selected** button.

Rectangle 2

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

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

3 In the **Width** edit field, type `d`.

4 In the **Height** edit field, type `3*d`.

5 Locate the **Position** section. In the **y** edit field, type `-3*d`.

6 Click the **Build Selected** button.

Circle 1

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

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

3 In the **Radius** edit field, type `d/5`.

4 Locate the **Position** section. In the **x** edit field, type `d/2`.

5 Click the **Build Selected** button.

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

The geometry now consists of two rectangular domains for the air and the dielectric, and a circle centered on their intersection. You can remove the line through the circle if you first create a union of the objects.

Union 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 From the **Edit** menu, choose **Select All**.

3 Click the **Build Selected** button.

Delete Entities 1

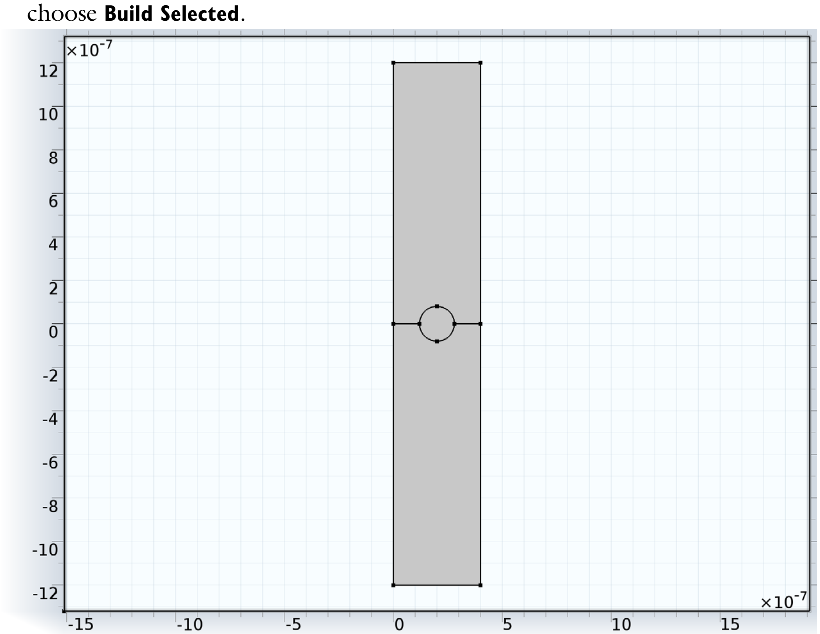
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundary 6 only. (This is the horizontal diameter of the circle in the center of the geometry.)

3 Click the **Build Selected** button.

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **Build Selected**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Before setting up the materials, define which constitutive relations you want to use in the Electromagnetic Waves interface.

Wave Equation, Electric 1

1 In the **Model Builder** window, expand the **Electromagnetic Waves, Frequency Domain** node, then click **Wave Equation, Electric 1**.

2 In the **Wave Equation, Electric** settings window, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Refractive index**.

Wave Equation, Electric 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the domain setting **Wave Equation, Electric**.

2 Select Domain 3 only.

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

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

MATERIALS

Material 1

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

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

3 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Refractive index	n	na

4 Right-click **Model 1>Materials>Material 1** and choose **Rename**.

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

6 Click **OK**.

Material 2

1 Right-click **Materials** and choose **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
Refractive index	n	nb

5 Right-click **Model 1>Materials>Material 2** and choose **Rename**.

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

7 Click **OK**.

Material 3

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

2 Select Domain 3 only.

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

4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Relative permittivity (imaginary part)	epsilonBis	5.4
Relative permittivity (real part)	epsilonPrim	-1.75
Relative permeability	mur	1
Electric conductivity	sigma	0

5 Right-click **Model 1>Materials>Material 3** and choose **Rename**.

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

7 Click **OK**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

In the first version of this model, you will assume a TE-polarized wave. This means that E_x and E_y will be zero throughout the geometry, and that you consequently only need to solve for E_z.

1 In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Components** section.

2 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Use the wave vector components you have defined in setting up the ports. Begin with the excitation port.

Port 1

1 Right-click **Model 1>Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 5 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
0	y
exp(-i*kax*x)[V/m]	z

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

The order in which you set up the ports will determine how the S-parameters are labeled. You have just created Port 1 for the excitation. If you set up the next port for the transmission of the purely refracted beam, the S21-parameter will contain information on the zero order transmission.

Port 2

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 2 only.

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

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

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

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

Continue with the ports for the positive and negative diffraction orders.

Port 3

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 5 only.

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

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

`0`	x
`0`	y
`exp(-i*kapx*x)`	z

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

Port 4

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 2 only.

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

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

`0`	x
`0`	y
`exp(-i*kbpx*x)`	z

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

Port 5

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 5 only.

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

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

`0`	x
`0`	y
`exp(-i*kamx*x)`	z

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

Port 6

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Port**.

2 Select Boundary 2 only.

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

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

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

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

Periodic Condition 1

1 In the **Model Builder** window, right-click **Electromagnetic Waves, Frequency Domain** and choose the boundary condition **Periodic Condition**.

2 Select Boundaries 1, 3, 7, and 8 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 $\mathbf{k}_F$ vector as

`kax`	x
`0`	y

MESH 1

The periodic boundary conditions perform 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.

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

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

Free Triangular 1

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

2 Click **Yes** to confirm.

Size

1 In the **Size** settings window, locate the **Element Size** section.

2 From the **Predefined** list, choose **Extra fine**.

Edge 1

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

2 Select Boundaries 1 and 3 only.

Copy Edge 1

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

2 Select Boundary 3 only.

3 In the **Copy Edge** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundary 8 only.

Copy Edge 2

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

2 Select Boundary 1 only.

3 In the **Copy Edge** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundary 7 only.

Free Triangular 1

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

2 In the **Free Triangular** settings window, click **Build All**.

STUDY 1

To set up the study to sweep for the angle of incidence, some modifications of the solver is required.

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then 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 In the **Model Builder** window, right-click **Study 1** and choose **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
alpha

5 Click **Range**.
6 Go to the **Range** dialog box.
7 In the **Start** edit field, type `0`.
8 In the **Stop** edit field, type `pi/2-pi/40`.
9 In the **Step** edit field, type `pi/40`.
10 Click the **Replace** button.
11 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Electric field

The default plot shows the electric field norm for the last solution, almost tangential incidence. Look at a more interesting angle of incidence.

1 In the **2D Plot Group** settings window, locate the **Data** section.
2 From the **Parameter value (alpha)** list, choose **0.628319**.
3 Click the **Plot** button.
4 Click the **Zoom Extents** button on the Graphics toolbar.

   The plot should now look like Figure 4.

   Rename the plot group to make it clear that it shows the TE solution.

5 Right-click **Results>Electric field** and choose **Rename**.
6 Go to the **Rename 2D Plot Group** dialog box and type `2D Plot Group TE` in the **New name** edit field.
7 Click **OK**.

Add a 1D plot to look at the various orders of reflection and transmission versus the angle of incidence.

1D Plot Group 2

1 Right-click **Results** and choose **1D Plot Group**.
2 In the **1D Plot Group** settings window, locate the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Reflection and Transmission of TE Wave`.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 In the associated edit field, type `Angle of incidence (degrees)`.
7 Select the **y-axis label** check box.

8 In the associated edit field, type `Reflection and transmission coefficients`.

9 Right-click **Results>1D Plot Group 2** and choose **Global**.

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

11 In the table, enter the following settings:

EXPRESSION
abs(emw.S11)^2
abs(emw.S21)^2
abs(emw.S31)^2
abs(emw.S41)^2
abs(emw.S51)^2
abs(emw.S61)^2

12 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

13 In the **Expression** edit field, type `alpha*180/pi`.

14 Click to expand the **Legends** section. Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

15 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

16 In the table, enter the following settings:

LEGENDS
R₀
T₀
R₁
T₁
R₋₁
T₋₁

17 Click the **Plot** button.

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

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

20 Click **OK**.

The plot should now look like Figure 6.

The remaining instructions show to alter the physics so that you solve for an incident TM wave.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

1 In the **Electromagnetic Waves, Frequency Domain** settings window, locate the **Components** section.

2 From the **Electric field components solved for** list, choose **In-plane vector**.

You will now solve for E_x and E_y instead of E_z; for a TM wave, E_z is zero.

Port 1

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

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

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

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

Port 2

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

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

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

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

Note that this is the specular reflection of the refracted beam.

Port 3

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 3**.

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

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

cos(alpha_p)*exp(-i*kapx*x)[V/m]	x
sin(alpha_p)*exp(-i*kapx*x)[V/m]	y
0	z

Port 4

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 4**.

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

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

-cos(beta_p)*exp(-i*kbpx*x)[V/m]	x
sin(beta_p)*exp(-i*kbpx*x)[V/m]	y
0	z

Port 5

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 5**.

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

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

cos(alpha_m)*exp(-i*kamx*x)[V/m]	x
sin(alpha_m)*exp(-i*kamx*x)[V/m]	y
0	z

Port 6

1 In the **Model Builder** window, under **Model 1>Electromagnetic Waves, Frequency Domain** click **Port 6**.

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

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

-cos(beta_m)*exp(-i*kbmx*x)[V/m]	x
sin(beta_m)*exp(-i*kbmx*x)[V/m]	y
0	z

Add a new study in order not to overwrite the TE solution.

MODEL WIZARD

1 In the **Model Builder** window, right-click the root node and choose **Add Study**.

2 Go to the **Model Wizard** window.

3 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

4 Click **Finish**.

STUDY 2

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 2** node, then 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 In the **Model Builder** window, right-click **Study 2** and choose **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
alpha

5 Click **Range**.

6 Go to the **Range** dialog box.

7 In the **Start** edit field, type `0`.

8 In the **Stop** edit field, type `pi/2-pi/40`.

9 In the **Step** edit field, type `pi/40`.

10 Click the **Replace** button.

11 In the **Model Builder** window, right-click **Study 2** and choose **Compute**.

RESULTS

Electric field

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

2 From the **Parameter value (alpha)** list, choose **0.628319**.

3 Click the **Plot** button.

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

5 Right-click **Results>Electric field** and choose **Rename**.

6 Go to the **Rename 2D Plot Group** dialog box and type `2D Plot Group TM` in the **New name** edit field.

7 Click **OK**.

You have now reproduced Figure 5. For the transmission and the reflection of the TM waves, copy and reuse the 1D plot for the TE waves.

1D Plot Group TE 1

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group TE** and choose **Duplicate**.
2 In the **1D Plot Group** settings window, locate the **Title** section.
3 In the **Title** text area, type `Reflection and Transmission of TM Wave`.
4 Locate the **Data** section. From the **Data set** list, choose **Solution 2**.
5 Click the **Plot** button.
6 Right-click **Results>1D Plot Group TE 1** and choose **Rename**.
7 Go to the **Rename 1D Plot Group** dialog box and type `1D Plot Group TM` in the **New name** edit field.
8 Click **OK**.

Compare the resulting plot with that in Figure 7.