**Beam Splitter**

**Introduction**

A beam splitter is used for splitting a beam of light in two. One way of making a splitter is to deposit a thin layer of metal between two glass prisms. The beam is slightly attenuated within the layer and then split into two paths. This example models the thin metal layer using a transition boundary condition, which reduces the memory requirements. Losses in the metal layer are also computed.

![Figure 1: A beam splitter composed of two prisms with a thin layer of metal between them.](image)

**Model Definition**

Model the beam splitter in the 2D plane, as shown in Figure 1, under the assumption that the electric field is polarized perpendicular to the plane. A Gaussian beam of wavelength 700 nm propagates in the $x$ direction through the glass prism of refractive index $n = 1.5$. A 13 nm thin layer of silver sandwiched between the two prisms splits the beams.

The model geometry is a square region around the region where the Gaussian beam crosses the silver layer. The focus of the beam is at the left boundary, so the expression for the beam intensity at the focal plane can be used as the excitation. The expression for the relative electric field intensity at the focal plane of a Gaussian is
where $w = 3500$ nm is the beam waist, and the $y = 0$ line is the centerline of the beam. Use this expression in a Port boundary condition on the left side to model the incident beam. Model all the other domain boundaries using Scattering Boundary Conditions. These conditions are appropriate when they are placed several wavelengths away from any scattering objects and the wave is known to be traveling at normal or almost normal incidence.

The thin silver layer is modeled using a Transition Boundary Condition. At a free-space wavelength of 700 nm, the dielectric of silver is about $\varepsilon_r = -16.5 - 1.06i$, where the imaginary part accounts for the losses. Thus, you can set the conductivity of the metal to zero. This boundary condition allows for a discontinuity in the fields across the interface by splitting the mesh at the boundary. It can introduce both losses and a phase shift across the interface. It does not require a mesh of the thickness of the domain, and thus saves significant memory. Mesh the two domains with triangular elements, with the maximum size set such that there are six elements per wavelength in the glass.

**Results and Discussion**

Figure 2 shows the electric field intensity in the modeling domain. The beam is split into two beams, one propagating in the $x$ direction and the other one in the $y$ direction. The splitting can be evaluated by computing the flux crossing the incoming boundary and the two outgoing boundaries. Figure 3 plots the power crossing these boundaries as well as the losses at the mirror.
Figure 2: The electric field intensity shows that the incoming beam is split into two beams of approximately equal intensity.

Figure 3: The power flux crossing the input boundary and the two output boundaries as well as the losses at the silver surface.
Model Library path: Wave_Optics_Module/Optical_Scattering/beam_splitter

Modeling Instructions

From the File menu, choose New.

NEW
1 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>Frequency Domain.
6 Click Done.

DEFINITIONS

Parameters
1 On the Model toolbar, click Parameters.
2 In the Settings window for Parameters, locate the Parameters section.
3 In the table, enter the following settings:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>lda0</td>
<td>700[nm]</td>
<td>7.000E-7 m</td>
<td>Wavelength</td>
</tr>
<tr>
<td>lda</td>
<td>lda0/1.5</td>
<td>4.667E-7 m</td>
<td>Material wavelength</td>
</tr>
<tr>
<td>f0</td>
<td>c_const/lda0</td>
<td>4.283E14 1/s</td>
<td>Frequency</td>
</tr>
<tr>
<td>h_max</td>
<td>0.2*lda</td>
<td>9.333E-8 m</td>
<td>Maximum mesh element size</td>
</tr>
<tr>
<td>eps_Ag</td>
<td>-16.5-1.06*i</td>
<td>-16.5 - 1.06i</td>
<td>Relative dielectric constant, Silver</td>
</tr>
</tbody>
</table>

Here, c_const is a predefined COMSOL constant for the speed of light in vacuum and lda is the material wavelength, defined as the vacuum wavelength divided by the refractive index. In this case, 1.5 is the refractive index of the glass material that will be used in the model.

GEOMETRY 1
1 In the Model Builder window, under Component 1 (comp1) click Geometry 1.
2 In the Settings window for Geometry, locate the Units section.
3 From the Length unit list, choose µm.

Create a triangle using Polygon for one prism.

Polygon 1 (pol1)
1 On the Geometry toolbar, click Primitives and choose Polygon.
2 In the Settings window for Polygon, locate the Coordinates section.
3 In the x text field, type -10 -10 10.
4 In the y text field, type -10 10 10.
5 Right-click Component 1 (comp1)>Geometry 1>Polygon 1 (pol1) and choose Build Selected.

Rotate the triangle to create the other prism.

Rotate 1 (rot1)
1 On the Geometry toolbar, click Transforms and choose Rotate.
2 Select the object pol1 only.
3 In the Settings window for Rotate, locate the Input section.
4 Select the Keep input objects check box.
5 Locate the Rotation Angle section. In the Rotation text field, type 180.
6 Click the **Build All Objects** button.

**ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)**

Now set up the physics.

*Scattering Boundary Condition 1*

1 On the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.
2 Select Boundaries 2, 4, and 5 only.

Port 1
1 On the Physics toolbar, click Boundaries and choose Port.
2 Select Boundary 1 only.

3 In the Settings window for Port, 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

\[
\begin{array}{c|c}
0 & x \\
0 & y \\
\exp(-(y/3500\text{[nm]})^2) & z \\
\end{array}
\]

6 In the $\beta$ text field, type ewfd.k.

Transition Boundary Condition 1
1 On the Physics toolbar, click Boundaries and choose Transition Boundary Condition.
2 Select Boundary 3 only.
3 In the Settings window for Transition Boundary Condition, locate the Transition Boundary Condition section.
4 From the Electric displacement field model list, choose Relative permittivity.
5 From the $\varepsilon_r$ list, choose User defined. In the associated text field, type $\text{eps}_\text{Ag}$.
6 From the $\mu_r$ list, choose User defined. Leave the default value of 1.
7 From the $\sigma$ list, choose User defined. Leave the default value of 0.
8 In the $d$ text field, type 13[nm].
MATERIALS
Next, assign material properties. Use Glass (quartz) for all domains.

ADD MATERIAL
1. On the Model toolbar, click Add Material to open the Add Material window.
2. Go to the Add Material window.
3. In the tree, select Built-In>Glass (quartz).
4. Click Add to Component in the window toolbar.

MESH 1
Choose the maximum mesh element size in the glass domain smaller than 0.2 material wavelengths using the parameter $h_{\text{max}}$ that you defined earlier.

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 Element Size section.
3. Click the Custom button.
4. Locate the Element Size Parameters section. Select the Maximum element size check box.
5. In the associated text field, type $h_{\text{max}}$.

Free Triangular 1
1. In the Model Builder window, right-click Mesh 1 and choose Free Triangular.
2. Right-click Free Triangular 1 and choose Build All.

STUDY 1
Step 1: Frequency Domain
1. In the Model Builder window, expand the Study 1 node, then click Step 1: Frequency Domain.
2. In the Settings window for Frequency Domain, locate the Study Settings section.
3. In the Frequencies text field, type $f_0$.
4. On the Model toolbar, click Compute.
RESULTS

Electric Field (ewfd)
The default plot shows the E-field norm. Compare the plot with Figure 2.

Follow the steps below to reproduce the plot in Figure 3.

1D Plot Group 2
1. On the Model toolbar, click Add Plot Group and choose 1D Plot Group.
2. On the 1D plot group toolbar, click Line Graph.
3. Select Boundary 1 only.
4. In the Settings window for Line Graph, locate the y-Axis Data section.
5. In the Expression text field, type \(-\text{ewfd}.\text{nPoav}\).
6. Locate the x-Axis Data section. From the Parameter list, choose Expression.
7. In the Expression text field, type \(y\).
8. On the 1D plot group toolbar, click Line Graph.
9. Select Boundary 5 only.
10. In the Settings window for Line Graph, locate the y-Axis Data section.
11. In the Expression text field, type \(\text{ewfd}.\text{nPoav}\).
12. Locate the x-Axis Data section. From the Parameter list, choose Expression.
13. In the Expression text field, type \(y\).
14. On the 1D plot group toolbar, click Line Graph.
15. Select Boundary 4 only.
16. In the Settings window for Line Graph, locate the y-Axis Data section.
17. In the Expression text field, type \(\text{ewfd}.\text{nPoav}\).
18. Locate the x-Axis Data section. From the Parameter list, choose Expression.
19. In the Expression text field, type \(x\).
20. On the 1D plot group toolbar, click Line Graph.
21. Select Boundary 3 only.
22. In the Settings window for Line Graph, locate the y-Axis Data section.
23. In the Expression text field, type \(\text{ewfd}.\text{Qsrh}\).
24. Locate the x-Axis Data section. From the Parameter list, choose Expression.
25. In the Expression text field, type \(x\).
26 On the **1D plot group** toolbar, click **Plot**. The plot describes the power flux crossing the input boundary and the two output boundaries together with the losses at the silver surface. Compare with Figure 3.