Stress-Optical Effects with Generalized Plane Strain

Introduction

The assumptions made for plane strain in the previous analysis of the waveguide structure, in the model Stress-Optical Effects in a Photonic WaveguideStress-Optical Effects in a Photonic Waveguide, do not hold in a situation where the silicon-silica laminate is free to expand in the z direction. Instead, it is necessary to use a *generalized plane strain* model that allows for free expansion in the z direction. The boundary conditions in the xy-plane already allow the structure to expand freely in all directions in the plane. When the different materials in a laminate expand with different expansion coefficients, the laminate bends. In this model, the silica-silicon laminate bends in both the x and z directions. The Solid Mechanics interface configured for 2D plane strain does not cover the bending in the z direction, so you need to make modifications to the plane strain equations at the equation-system level.

Note: This model requires the Wave Optics Module and the Structural Mechanics Module.

Model Definition

GENERALIZED PLANE STRAIN

One possible extension of the plane strain formulation is to assume that the normal out-of -plane strain component has the form:

$$\varepsilon_z = e_0 + e_1 x + e_2 y$$

That is, the strain is linearly varying throughout the cross section. This approximation is expected to be good when the bending curvature is small with respect to the extents of the structure in the xy-plane and corresponds to a small rotation that is representative of each cross section of the structure along the z-axis. (A more general model would include second-order terms in x and y.)

Extension of the Plane Strain Equations

In COMSOL Multiphysics, the coefficients e_0, e_1, and e_2 in the expression for the ε_z strain can be modeled as extra degrees of freedom that are constant throughout the model (global variables).

Start from the 3D stress-strain relation for linear isotropic material including thermal effects,

$$\begin{aligned} S_x &= S_{11} = D_{11}(\varepsilon_x - \varepsilon_{\text{th}}) + D_{12}(\varepsilon_y - \varepsilon_{\text{th}}) + D_{13}(\varepsilon_z - \varepsilon_{\text{th}}) \\ S_y &= S_{22} = D_{12}(\varepsilon_x - \varepsilon_{\text{th}}) + D_{22}(\varepsilon_y - \varepsilon_{\text{th}}) + D_{23}(\varepsilon_z - \varepsilon_{\text{th}}) \\ S_z &= S_{33} = D_{13}(\varepsilon_x - \varepsilon_{\text{th}}) + D_{23}(\varepsilon_y - \varepsilon_{\text{th}}) + D_{33}(\varepsilon_z - \varepsilon_{\text{th}}) \end{aligned}$$

where $\varepsilon_{\text{th}} = \alpha(T - T_{\text{ref}})$ and

$$\varepsilon_x = \varepsilon_{11} = \frac{\partial u}{\partial x}$$

$$\varepsilon_y = \varepsilon_{22} = \frac{\partial v}{\partial y}$$

$$D_{11} = D_{22} = D_{33} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)}$$

$$D_{12} = D_{23} = D_{31} = \frac{E\nu}{(1+\nu)(1-2\nu)}$$

where E is Young's modulus and ν is Poisson's ratio.

The Solid Mechanics equations are implemented via the virtual work principle, which leads to the following weak contribution (see the *Structural Mechanics Module User's Guide* for more details):

$$\sum_{i,j} S_{ij}\text{test}(\varepsilon_{ij})$$

Comparison of the above expression with and without the assumption of $\varepsilon_z = 0$ shows that the following weak terms need to be added to extend the plane strain equations:

$$D_{13}\varepsilon_z\text{test}(\varepsilon_x) + D_{23}\varepsilon_z\text{test}(\varepsilon_y) + S_z\text{test}(\varepsilon_z)$$

Note that

$$\text{test}(\varepsilon_z) = \text{test}(e_0) + \text{test}(e_1)x + \text{test}(e_2)y$$

which contributes to the equation for these extra three dependent variables.

 ©2013 COMSOL

Results and Discussion

Figure 1 shows the von Mises stress distribution together with the deformed shape of the waveguide.

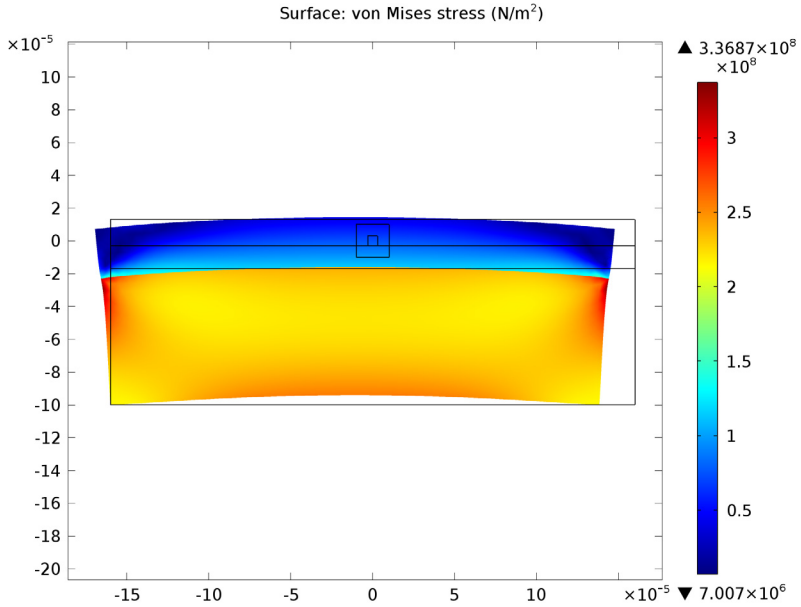

Figure 1: von Mises effective stress computed for the generalized plane strain.

For symmetry reasons, the strain components ε_x and ε_z should be equal. The plot in Figure 2 visualizes the area where the relative difference between ε_x and ε_z is within 5%. The model is most accurate in the regions close to the core, far from the boundaries on the far left and right.

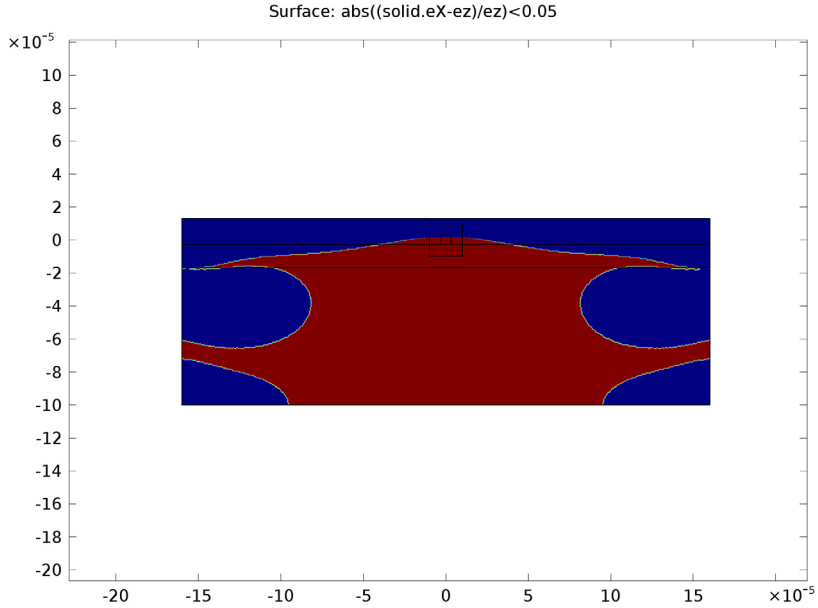

Figure 2: In the red-colored region, the relative difference between the x and z strain components is within 5%.

Figure 3 compares the effective mode indices for the first four propagating modes using the generalized plane strain equations with those obtained from the analysis in the previous model. As the plot shows, there is a systematic shift in the propagation constants when the strain in the z direction is taken into account.

 ©2013 COMSOL

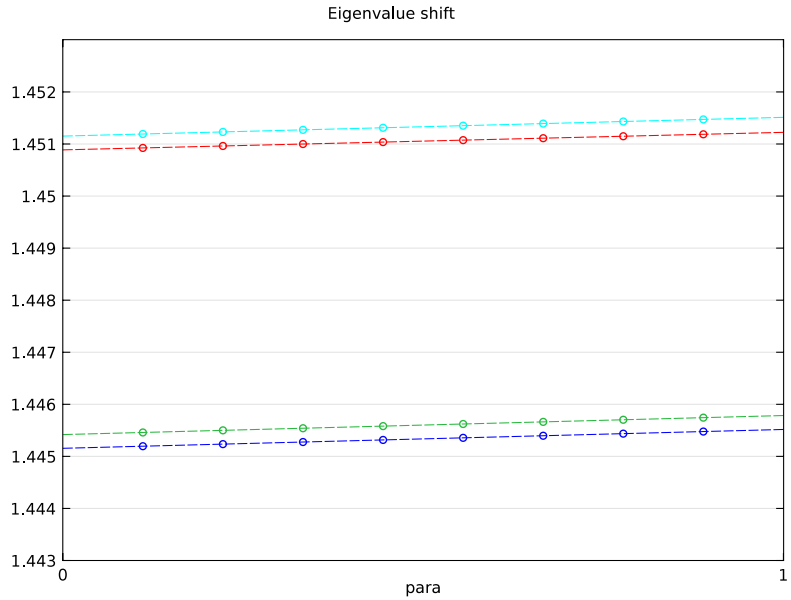

Figure 3: Effective mode indices assuming plane strain (para = 0) and generalized plane strain (para = 1).

Figure 4 shows the stress-induced birefringence along the symmetry line within the waveguide. Nonzero out-of-plane strain leads to an increase in the birefringence effect.

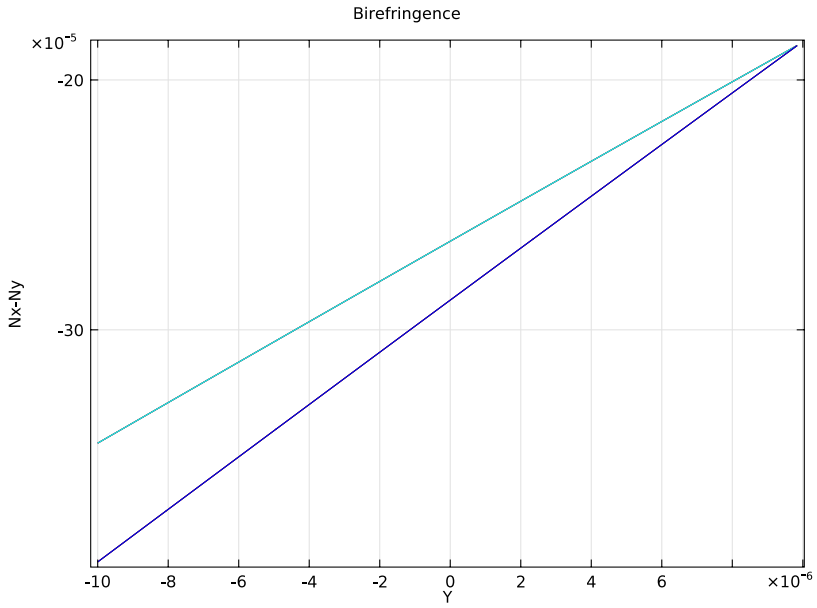

Figure 4: Birefringence along the vertical symmetry line within the waveguide for plane strain (upper curve) and generalized plane strain (lower curve).

Figure 5 gives the details of the eigenmode with the lowest effective mode index. It presents the visualization of the power flow, also called the optical intensity or the Poynting vector, in the out-of-plane direction.

 ©2013 COMSOL

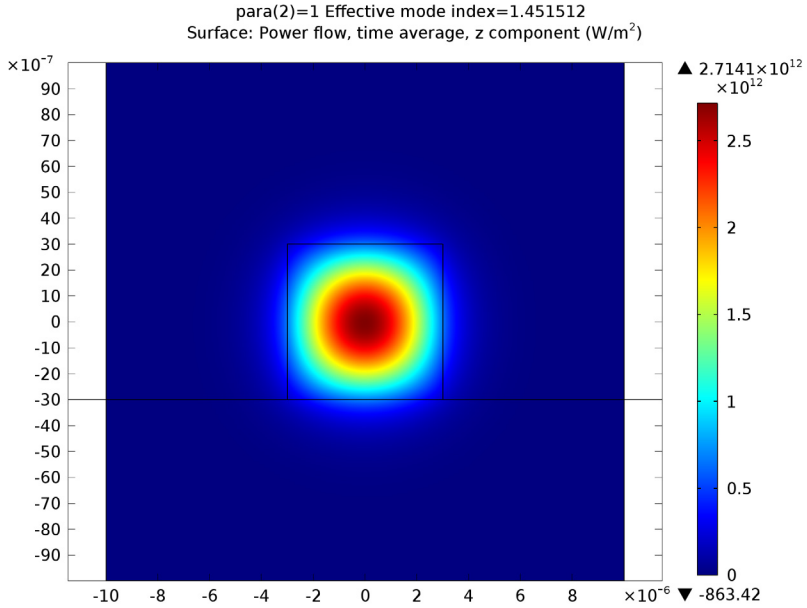

Figure 5: Eigenmode with lowest mode index, computed with the stress-optical ceffect under generalized plane strain assumption.

Notes About the COMSOL Implementation

The model includes an extension of the plane strain equation system. This implies that some of the results-processing variables available for the Solid Mechanics interface may no longer be valid or even give wrong results if used. Therefore, use the stress variables available under Definitions for any additional results processing.

Model Library path: `RF_Module/Optics_and_PhotonicsStructural_Mechanics_Module/Stress_Optical_Effects/stress_optical_generalized`

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 **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add Selected**.

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

7 Click **Add Selected**.

8 Click **Next**.

9 Find the **Studies** subsection. In the tree, select **Custom Studies>Preset Studies for Some Physics>Stationary**.

10 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 Click **Load from File**.

4 Browse to the model's Model Library folder and double-click the file `stress_optical_parameters.txt`.

Add a parameter to switch on and off the stress effects.

5 In the table, enter the following settings:

Name	Expression	Description
para	1	1: use generalized plane strain, 0: use plane strain

GEOMETRY 1

Rectangle 1

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

 ©2013 COMSOL

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

3 In the **Width** edit field, type `3.2E-4`.

4 In the **Height** edit field, type `8.3E-5`.

5 Locate the **Position** section. In the **x** edit field, type `-1.6E-4`.

6 In the **y** edit field, type `-1.0E-4`.

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 `3.2E-4`.

4 In the **Height** edit field, type `1.4E-5`.

5 Locate the **Position** section. In the **x** edit field, type `-1.6E-4`.

6 In the **y** edit field, type `-1.7E-5`.

Rectangle 3

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 `3.2E-4`.

4 In the **Height** edit field, type `1.6E-5`.

5 Locate the **Position** section. In the **x** edit field, type `-1.6E-4`.

6 In the **y** edit field, type `-3.0E-6`.

Rectangle 4

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 `6.0E-6`.

4 In the **Height** edit field, type `6.0E-6`.

5 Locate the **Position** section. In the **x** edit field, type `-3.0E-6`.

6 In the **y** edit field, type `-3.0E-6`.

Rectangle 5

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 `2.0E-5`.

4 In the **Height** edit field, type `2.0E-5`.

5 Locate the **Position** section. In the **x** edit field, type `-1.0E-5`.

6 In the **y** edit field, type `-1.0E-5`.

7 Click the **Build Selected** button.

The last rectangular region encloses the optical computational domain. It can be enlarged if needed for validating the results. The region should be chosen large enough so that the computed propagation constants do not change significantly if the region is enlarged.

SOLID MECHANICS

Thermal Expansion 1

1 In the **Model Builder** window, under **Model 1>Solid Mechanics** right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

2 In the **Thermal Expansion** settings window, locate the **Model Inputs** section.

3 In the T edit field, type `T1`.

4 Locate the **Thermal Expansion** section. In the T_{ref} edit field, type `T0`.

You add the coefficients in the out-of-plane strain expression as dependent variables in a global equation.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

This will allow you to add a global equation and other advanced modeling features to the Solid Mechanics interface.

Global Equations 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Global>Global Equations**.

2 In the **Global Equations** settings window, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t)
e0	e0*(1-para)
e1	e1*(1-para)
e2	e2*(1-para)

Setting `para` = 0 will constrain `e0`, `e1`, and `e2` to zero. For `para` = 1, an extra equation for these components will be set up via the weak contribution that you will add to all domains.

 ©2013 COMSOL

DEFINITIONS

Variables 1

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
ez	para*(e0+(e1*X+e2*Y)/1[m])	Out-of-plane strain, z component
sx	solid.Sl11+solid.D13*ez	Stress, x component
sy	solid.Sl22+solid.D23*ez	Stress, y component
sz	solid.Sl33+solid.D33*ez	Stress, z component
wc1	-(solid.D13*ez*test(solid.el11)+solid.D23*ez*test(solid.el22)+sz*test(ez))*solid.d	Weak contribution due to out-of-plane strain

Variables 2

1 In the **Model Builder** window, 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
N	nCore	Refractive index for stress-free material

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 6 only.

Variables 3

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
N	nSiO2	Refractive index for stress-free material

4 In the **Variables** settings window, locate the **Geometric Entity Selection** section.

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

6 Select Domains 4 and 5 only.

Variables 4

1 Right-click **Definitions** and choose **Variables**.

2 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Description
Nx	N-(B1*sx+B2*(sy+sz))	Refractive index, x component
Ny	N-(B1*sy+B2*(sx+sz))	Refractive index, y component
Nz	N-(B1*sz+B2*(sx+sy))	Refractive index, z component

3 In the **Variables** settings window, locate the **Geometric Entity Selection** section.

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

5 Select Domains 4–6 only.

SOLID MECHANICS

Weak Contribution 1

1 In the **Model Builder** window, under **Model 1** right-click **Solid Mechanics** and choose the domain condition **More>Weak Contribution**.

2 In the **Weak Contribution** settings window, locate the **Domain Selection** section.

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

4 Locate the **Weak Contribution** section. In the **Weak expression** edit field, type `wc1`.

All regions have free boundaries, which also is the default boundary condition. However, these conditions will not be sufficient for creating a unique solution because the computational domain is allowed to move and rotate freely. The problem becomes well posed by adding constraints at two points to restrain such rigid body movements.

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose **Points>Fixed Constraint**.

2 Select Point 1 only.

 ©2013 COMSOL

Prescribed Displacement 1

1 Right-click **Solid Mechanics** and choose **Points>Prescribed Displacement**.

2 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

3 Select the **Prescribed in y direction** check box.

4 Select Point 15 only.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

The computational domain is reduced significantly for the optical mode analysis.

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

2 Select Domains 4–6 only.

Wave Equation, Electric 1

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

4 From the n list, choose **User defined**. From the list, choose **Diagonal**.

5 In the n table, enter the following settings:

Nx	0	0
0	Ny	0
0	0	Nz

6 From the k list, choose **User defined**.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 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
Young's modulus	E	ESi
Poisson's ratio	nu	nuSi
Density	rho	rhoSi
Coefficient of thermal expansion	alpha	alphaSi

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

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

7 Click **OK**.

Material 2

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

2 Select Domains 2–6 only.

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

4 In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	ESiO2
Poisson's ratio	nu	nuSiO2
Density	rho	rhoSiO2
Coefficient of thermal expansion	alpha	alphaSiO2

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

6 Go to the **Rename Material** dialog box and type `SiO2` in the **New name** edit field.

7 Click **OK**.

MESH 1

Free Triangular 1

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

Size 1

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

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

 ©2013 COMSOL

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

4 Select Domains 4–6 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 `2E-7`.

Size

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

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

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

4 Click the **Build All** button.

STUDY 1

Step 2: Mode Analysis

1 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Mode Analysis**.

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

3 In the **Search for modes around** edit field, type `1.46`.

4 In the **Desired number of modes** edit field, type `4`.

5 In the **Mode analysis frequency** edit field, type `c_const/lambda0_ewfd`.

These settings make the eigenmode solver search for the 4 eigenmodes with effective mode indices closest to the value 1.46. This value is an estimate of the effective mode index for the fundamental mode.

Step 1: Stationary

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

2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics	Solve for
Electromagnetic waves (ewfd)	×

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	Parameter value list
para	0 1

5 Right-click **Study 1** and choose **Compute**.

RESULTS

Stress (solid)

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

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

3 Click the **Plot** button.

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

To visualize the area, where the relative difference between the x and z strain components is within 5%, follow these steps:

Electric Field (ewfd)

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

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

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

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

5 In the **Surface** settings window, locate the **Expression** section.

6 In the **Expression** edit field, type `abs((solid.eX-ez)/ez)<0.05`.

7 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

8 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

9 Click the **Plot** button.

Next, plot the stress-induced birefringence along the symmetry line within the waveguide.

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Line 2D**.

2 In the **Cut Line 2D** settings window, locate the **Data** section.

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

 ©2013 COMSOL

4 Locate the **Line Data** section. In row **Point 2**, set **X** to `0`.

5 In row **Point 1**, set **Y** to `-1e-5`.

6 In row **Point 2**, set **Y** to `1e-5`.

7 Click the **Plot** button.

1D Plot Group 3

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

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

3 From the **Data set** list, choose **Cut Line 2D 1**.

4 Right-click **Results>1D Plot Group 3** and choose **Line Graph**.

5 In the **Line Graph** settings window, locate the **y-Axis Data** section.

6 In the **Expression** edit field, type `Nx-Ny`.

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

8 In the **Expression** edit field, type `Y`.

9 Click to expand the **Coloring and Style** section. In the **Model Builder** window, click **1D Plot Group 3**.

10 In the **1D Plot Group** settings window, locate the **Title** section.

11 From the **Title type** list, choose **Manual**.

12 In the **Title** text area, type `Birefringence`.

13 Locate the **Plot Settings** section. Select the **x-axis label** check box.

14 In the associated edit field, type `Y`.

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

16 In the associated edit field, type `Nx-Ny`.

17 Click the **Plot** button.

To visualize the details of the eigenmode with the lowest effective mode index, you first set up a view that includes the optical computation domain only.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Results Options** in the menu.

2 In the **Model Builder** window, under **Results** right-click **Views** and choose **View 2D**.

3 In the **View 2D** settings window, locate the **View** section.

4 Select the **Lock axis** check box.

5 In the **Model Builder** window, under **Results>Views>View 2D 2** click **Axis**.

6 In the **Axis** settings window, locate the **Axis** section.

7 In the **x minimum** edit field, type `-1e-5`.

8 In the **x maximum** edit field, type `1e-5`.

9 In the **y minimum** edit field, type `-1e-5`.

10 In the **y maximum** edit field, type `1e-5`.

2D Plot Group 4

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

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

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

4 From the **Effective mode index** list, choose **1.451512**.

5 Locate the **Plot Settings** section. From the **View** list, choose **View 2D 2**.

6 Right-click **Results>2D Plot Group 4** and choose **Surface**.

7 In the **Surface** 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 (Spatial)>Power flow, time average, z component (ewfd.Poavz)**.

8 Click the **Plot** button.

This creates a visualization of the power flow, also called optical intensity or the Poynting vector, in the z direction (out-of-plane direction).

To collect all computed effective mode indices in a table, follow these steps:

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.

2 In the **Global Evaluation** settings window, locate the **Data** section.

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

4 From the **Table columns** list, choose **Inner solutions**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electromagnetic Waves, Frequency Domain>Global>Effective mode index (ewfd.neff)**.

6 Click the **Evaluate** button.

If you see too few digits in the table, click the **Full Precision** toolbar button.

Finally, create a table plot to visualize the shift of the effective mode indices.

 ©2013 COMSOL

1D Plot Group 5

1 In the **Model Builder** window, 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 `Eigenvalue shift`.

5 Click to expand the **Axis** section. Select the **Manual axis limits** check box.

6 In the **x minimum** edit field, type `0`.

7 In the **x maximum** edit field, type `1`.

8 In the **y minimum** edit field, type `1.443`.

9 In the **y maximum** edit field, type `1.453`.

10 Click to expand the **Grid** section. Select the **Manual spacing** check box.

11 In the **y spacing** edit field, type `1e-3`.

12 Right-click **Results>1D Plot Group 5** and choose **Table Graph**.

13 In the **Table Graph** settings window, locate the **Coloring and Style** section.

14 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

15 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

 ©2013 COMSOL