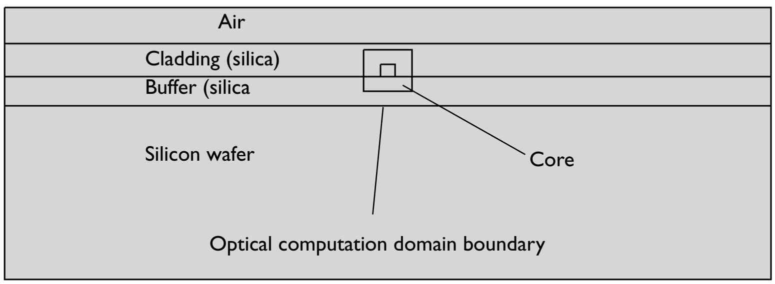
Stress-Optical Effects in a Photonic Waveguide

Introduction

Planar photonic waveguides in silica (SiO_2) have great potential for use in wavelength routing applications. The major problem with these kinds of waveguides is birefringence. Anisotropic refractive indices result in fundamental mode splitting and pulse broadening. The goal is to minimize birefringence effects by adapting materials and manufacturing processes. One source of birefringence is the use of a silicon (Si) wafer on which the waveguide structure is deposited. After annealing at high temperature (approximately 1000 °C), mismatch in thermal expansivity between the silica and silicon layers results in thermally induced stresses in the structure at the operating temperature (typically room temperature around 20 °C).



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

Model Definition

THE STRESS-OPTICAL EFFECT

The general linear stress-optical relation can be written, using tensor notation, as

$$\Delta n_{ij} = -B_{ijkl}S_{kl}$$

where $\Delta n_{ij} = n_{ij} - n_0 I_{ij}$, n_{ij} is the refractive index tensor, n_0 is the refractive index for a stress-free material, I_{ij} is the identity tensor, B_{ijkl} is the stress-optical tensor, and S_{kl} is the stress tensor. The number of independent parameters in the stress-optical tensor that characterizes this constitutive relation is reduced by symmetry. Because n_{ij} and S_{kl} are both symmetric, $B_{ijkl} = B_{jikl}$ and $B_{ijkl} = B_{ijlk}$. In many cases it is possible to further reduce the number of independent parameters. The model at hand considers only two independent parameters, B_1 and B_2. The stress-optical relation then simplifies to

$$\begin{bmatrix}\Delta n_x \\ \Delta n_y \\ \Delta n_z\end{bmatrix} = -\begin{bmatrix}B_1 & B_2 & B_2 \\ B_2 & B_1 & B_2 \\ B_2 & B_2 & B_1\end{bmatrix}\begin{bmatrix}S_x \\ S_y \\ S_z\end{bmatrix}$$

where $n_x = n_{11}$, $n_y = n_{22}$, $n_z = n_{33}$, $S_x = S_{11}$, $S_y = S_{22}$, and $S_z = S_{33}$.

This translates to

$$\begin{aligned} n_x &= n_0 - B_1 S_x - B_2(S_y + S_z) \\ n_y &= n_0 - B_1 S_y - B_2(S_z + S_x) \\ n_z &= n_0 - B_1 S_z - B_2(S_x + S_y) \end{aligned}$$

Using the two parameters B_1 and B_2, the model assumes that the nondiagonal parts of n_{ij} are negligible.

The first part of this model utilizes the plane strain approximation available in the Solid Mechanics interface. The resulting birefringent refractive index is computed using expression variables and can be considered a postprocessing step of the plane strain model. The refractive index tensor is used as material data for the second part of the model, the mode analysis.

The Application Library model *Stress-Optical Effects with Generalized Plane Strain* (the model name is `stress_optical_generalized`) demonstrates a computation, for which the structure is free to bend in the z direction, using a formulation called Generalized Plane Strain.

ELECTROMAGNETIC WAVES

For a given frequency ν, or equivalently, free-space wavelength $\lambda_0 = c_0/\nu$, the Wave Optics Module's Electromagnetic Waves, Frequency Domain interface can be used for the mode analysis. In this model the free-space wavelength is 1.55 µm. The simulation is set up with the electric field components $\mathbf{E} = (E_x, E_y, E_z)$ as dependent variables. The wave is assumed to have the form

$$\mathbf{E} = \mathbf{E}(x, y)e^{j(\omega t - \beta z)} = (E_x(x, y), E_y(x, y), E_z(x, y))e^{j(\omega t - \beta z)}$$

and the effective mode index, $n_{\mathrm{eff}} = \beta/k_0$, is obtained from the eigenvalues.

For propagating modes it must hold that

$$n_{eff} < n_{core} = 1.456$$

The optical core and planar waveguide layers are made of Silica (SiO2) which is deposited unto a silicon (Si) wafer. The material properties are shown in the following table:

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nSiO2	1.445	Refractive index, silica (SiO_2)
deltan	0.0075	Relative index difference: $\Delta = \frac{(n_{core}^2 - n_{cladding}^2)}{2n_{core}^2}$
nCore	nClad/sqrt(1-2*deltan)	Refractive index, core
alphaSi	2.5e-6[1/K]	Coefficient of thermal expansion Si
alphaSiO2	0.35e-6[1/K]	Coefficient of thermal expansion, SiO_2
ESi	110[GPa]	Young's modulus, Si
ESiO2	78[GPa]	Young's modulus, SiO_2
nuSi	0.19	Poisson's ratio, Si
nuSiO2	0.17	Poisson's ratio, SiO_2
B1	0.65e-12[m^2/N]	First stress optical coefficient
B2	4.2e-12[m^2/N]	Second stress optical coefficient
T1	20[degC]	Operating temperature
T0	1000[degC]	Reference temperature

The computational domain can be reduced significantly for the optical mode analysis, because the energy of the fundamental modes is concentrated in the core region and the energy density decays rapidly in the cladding and buffer regions. Thus, you do not need to model the air domain.

Results and Discussion

Figure 1 shows the von Mises stress distribution together with the deformed shape of the waveguide. Notice that the stress varies slowly in the horizontal direction. This means that the significant influence on the stress-induced changes in the refractive index comes from the stress variations in the vertical direction. This is expected because the extension of the domains in the x direction is chosen to minimize effects of the edges.

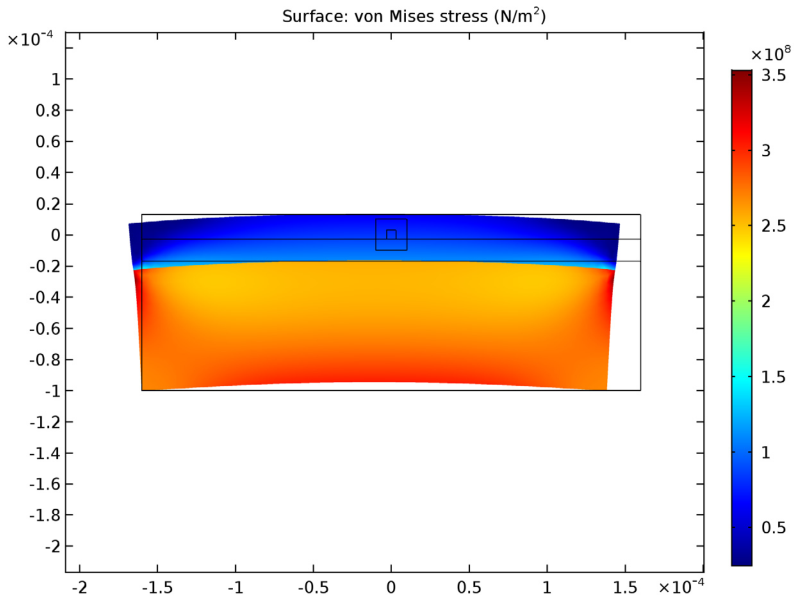


Figure 1: von Mises effective stress.

Figure 2 shows the birefringence, $n_x - n_y$, in the optical computation domain. The birefringence is nearly constant along any horizontal line, so the influence of the side edges is indeed reduced to a minimum.

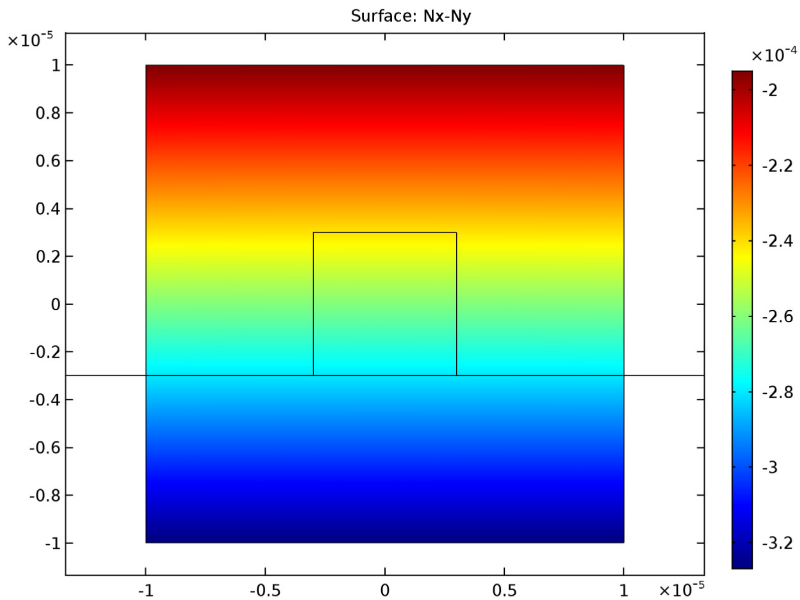


Figure 2: Birefringence.

Figure 3 shows the effective mode indices for the first four propagating modes. The value para = 0 corresponds to the case without stress-optical effect, while this effect is included for para = 1.

The difference is significant, which shows that the shift in the effective mode indices due to the stress-optical effect is indeed resolved.

The computations show a shift in effective mode index due to the stress-induced change in refractive index. The birefringence causes the otherwise two-fold degenerate fundamental mode to split.

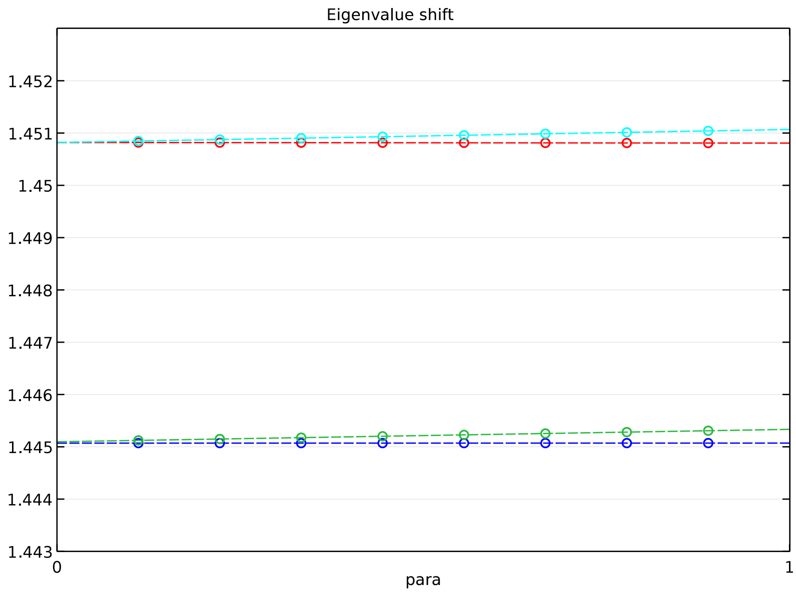


Figure 3: Effective mode indices for the first four propagating modes without (para = 0) and with (para = 1) the stress-optical effect.

Figure 4 shows the out-of-plane component (z component) of the Poynting vector for four eigenmodes computed without the stress-optical coupling.

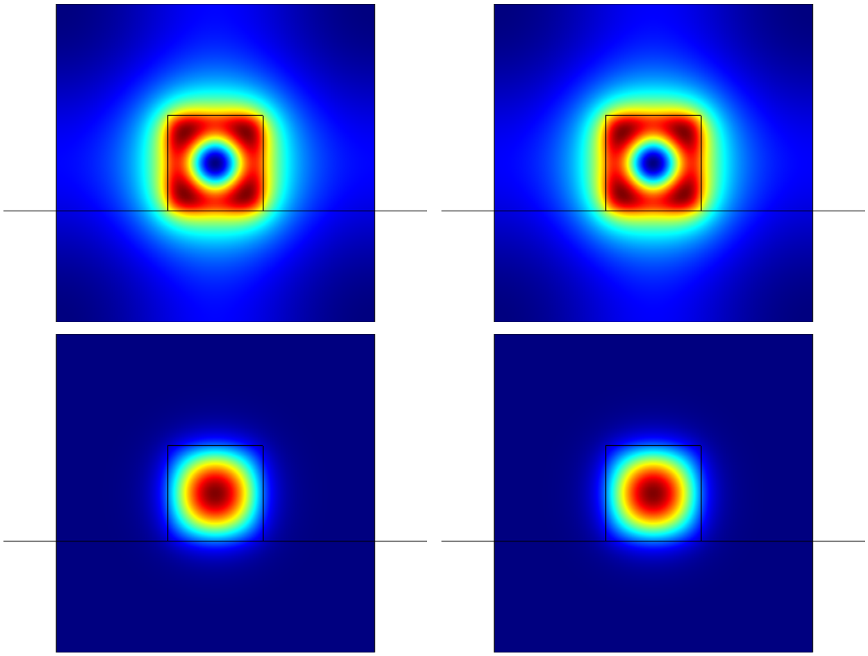


Figure 4: Eigenmodes computed without the stress-optical coupling.

Visual inspection of the higher eigenmodes indicates that they have a larger portion of energy leaking into the cladding and buffer, and are thus more affected than the fundamental modes of the distance to the air and silicon layers. Because of this leakage, the boundary condition affects the higher eigenmodes more than the fundamental mode.

The change in the eigenmodes shape due to the stress-optical effect is shown in Figure 5.

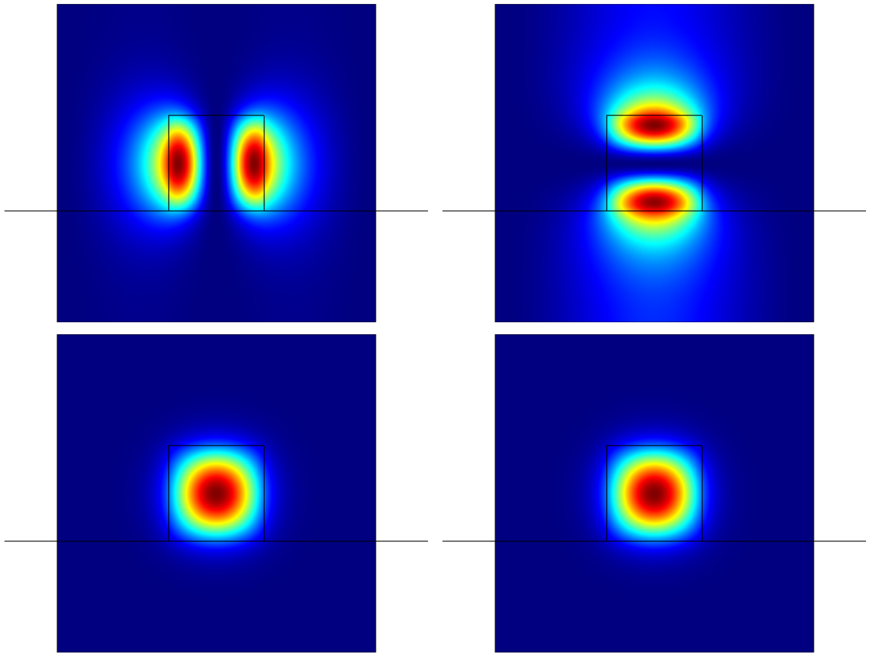


Figure 5: Eigenmodes computed with the stress-optical effect taken into account.

Although the fundamental modes have converged to 5 decimal places, the known modeling errors makes the exactness of the numbers uncertain. One major modeling error is due to the fact that the model contains a plane strain assumption in a case where the real-world model does not necessarily conform to this deformation state. This modeling error is reduced in the refined model *Stress-Optical Effects with Generalized Plane Strain* (the model name is `stress_optical_generalized`).

References

1. H.P. Schriemer and M. Cada, “Modal birefringence and power density distribution in strained buried-core square waveguides,” *IEEE Journal of Quantum Electronics*, vol. 40, pp. 1131–1139, 2004.

2. J. Stone, “Stress-Optic Effects, Birefringence, and Reduction of Birefringence by Annealing in Fiber Fabry-Perot Interferometers,” *J. Lightwave Technol.*, vol. 6, pp. 1245–1248, 1988.

Application Library path: `Wave_Optics_Module//Waveguides_and_Couplers/stress_optical`

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 **Structural Mechanics>Solid Mechanics (solid)**.
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 **Custom Studies>Preset Studies for Some Physics Interfaces>Stationary**.
8 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.
2 In the **Settings** window for Parameters, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the application's Application Libraries 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	Value	Description
`para`	`1`	1	`1: stress-optical coupling, 0: no coupling`

GEOMETRY 1

Rectangle 1 (r1)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3. In the **Width** text field, type `3.2E-4`.
4. In the **Height** text field, type `8.3E-5`.
5. Locate the **Position** section. In the **x** text field, type `-1.6E-4`.
6. In the **y** text field, type `-1E-4`.

Rectangle 2 (r2)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3. In the **Width** text field, type `3.2E-4`.
4. In the **Height** text field, type `1.4E-5`.
5. Locate the **Position** section. In the **x** text field, type `-1.6E-4`.
6. In the **y** text field, type `-1.7E-5`.

Rectangle 3 (r3)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3. In the **Width** text field, type `3.2E-4`.
4. In the **Height** text field, type `1.6E-5`.
5. Locate the **Position** section. In the **x** text field, type `-1.6E-4`.
6. In the **y** text field, type `-3E-6`.

Rectangle 4 (r4)

1. On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.
2. In the **Settings** window for Rectangle, locate the **Size and Shape** section.
3. In the **Width** text field, type `6E-6`.

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

5 Locate the **Position** section. In the **x** text field, type `-3E-6`.

6 In the **y** text field, type `-3E-6`.

Rectangle 5 (r5)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

2 In the **Settings** window for Rectangle, locate the **Size and Shape** section.

3 In the **Width** text field, type `2E-5`.

4 In the **Height** text field, type `2E-5`.

5 Locate the **Position** section. In the **x** text field, type `-1E-5`.

6 In the **y** text field, type `-1E-5`.

7 Right-click **Rectangle 5 (r5)** and choose **Build Selected**.

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 (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Thermal Expansion 1

1 On the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.

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

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

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

DEFINITIONS

Variables 1

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	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 2

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
N	nSiO2		Refractive index for stress-free material

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

5 Select Domains 4 and 5 only.

Variables 3

1 On the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

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

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

5 Select Domains 4–6 only.

SOLID MECHANICS (SOLID)

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 On the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 1 only.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 In the **Settings** window for Prescribed Displacement, locate the **Prescribed Displacement** section.

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

4 Select Point 15 only.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EWFD)

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

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (ewfd)**.

2 Select Domains 4–6 only.

Wave Equation, Electric 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (ewfd)** click **Wave Equation, Electric 1**.

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

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

4 In the n table, enter the following settings:

Nx	0	0
0	Ny	0
0	0	Nz

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

MATERIALS

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

Material 1 (mat1)

1. In the **Settings** window for Material, type `Si` in the **Label** text field.
2. Select Domain 1 only.
3. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`ESi`	Pa	Basic
Poisson's ratio	nu	`nuSi`	1	Basic
Density	rho	`rhoSi`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`alphaSi`	1/K	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, type `SiO2` in the **Label** text field.
3. Select Domains 2–6 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	`ESiO2`	Pa	Basic
Poisson's ratio	nu	`nuSiO2`	1	Basic
Density	rho	`rhoSiO2`	kg/m³	Basic
Coefficient of thermal expansion	alpha	`alphaSiO2`	1/K	Basic

MESH 1

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

Free Triangular 1

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

Size 1

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

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

3 Select Domains 4–6 only.

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

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type `2E-7`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for Size, locate the **Element Size** section.

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

4 Click **Build All**.

STUDY 1

Mode Analysis

On the **Study** toolbar, click **Study Steps** and choose **Other>Mode Analysis**.

Step 2: Mode Analysis

1 In the **Settings** window for Mode Analysis, locate the **Study Settings** section.

2 Select the **Search for modes around** check box.

3 In the associated text field, type `1.46`.

4 Select the **Desired number of modes** check box.

5 In the associated text field, type `4`.

6 In the **Mode analysis frequency** text 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 **Settings** window for Stationary, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for the **Electromagnetic Waves, Frequency Domain** interface.

Parametric Sweep

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para	0 1	

5 On the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

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

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

4 On the **Stress (solid)** toolbar, click **Plot**.

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

To visualize the details of the eigenmodes, you first set up a view that includes the optical computation domain only.

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

View 2D 2

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

2 In the **Settings** window for View 2D, locate the **View** section.

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

Axis

1 In the **Model Builder** window, expand the **View 2D 2** node, then click **Axis**.

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

3 In the **x minimum** text field, type -1E-5.

4 In the **x maximum** text field, type 1E-5.

5 In the **y minimum** text field, type -1E-5.

6 In the **y maximum** text field, type 1E-5.

Electric Field (ewfd)

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

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

3 From the **View** list, choose **View 2D 2**.

4 In the **Model Builder** window, expand the **Electric Field (ewfd)** node, then click **Surface 1**.

5 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Energy and power>Power flow, time average (Spatial)>ewfd.Poavz - Power flow, time average, z component**.

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

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

8 From the **Parameter value (para)** list, choose **0**.

9 On the **Electric Field (ewfd)** toolbar, click **Plot**. This creates a visualization of the power flow, also called optical intensity or the Poynting vector, in the z direction (out-of-plane direction). Continue to plot the power flow for the other computed eigenmodes, with and without stress effects.

2D Plot Group 3

Next, plot the birefringence Nx-Ny in the waveguide.

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

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

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

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

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Surface: Nx-Ny`.

7 Right-click **2D Plot Group 3** and choose **Surface**.

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

9 In the **Expression** text field, type `Nx-Ny`.

10 On the **2D Plot Group 3** toolbar, click **Plot**.

Derived Values

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

Global Evaluation 1

On the **Results** toolbar, click **Global Evaluation**.

Derived Values

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

2 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Electromagnetic Waves, Frequency Domain>Global>ewfd.neff - Effective mode index**.

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

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

1D Plot Group 4

On the **Results** toolbar, click **1D Plot Group**.

Table Graph 1

On the **1D Plot Group 4** toolbar, click **Table Graph**.

1D Plot Group 4

1 In the **Settings** window for Table Graph, locate the **Coloring and Style** section.

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

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

4 In the **Model Builder** window, click **1D Plot Group 4**.

5 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

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

7 In the **Title** text area, type `Eigenvalue shift`.

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

9 In the **x minimum** text field, type `0`.

10 In the **x maximum** text field, type `1`.

11 In the **y minimum** text field, type `1.443`.

12 In the **y maximum** text field, type `1.453`.

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

14 In the **y spacing** text field, type `1e-3`.

15 On the **1D Plot Group 4** toolbar, click **Plot**.