Created in COMSOL Multiphysics 5.2a

Stress-Optical Effects with Generalized Plane Strain

Introduction

The assumptions made for plane strain in the previous analysis of the waveguide structure (see the Application Library model *Stress-Optical Effects in a Photonic Waveguide,* the model name is `stress_optical`) 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 application 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,

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

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.

Results and Discussion

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

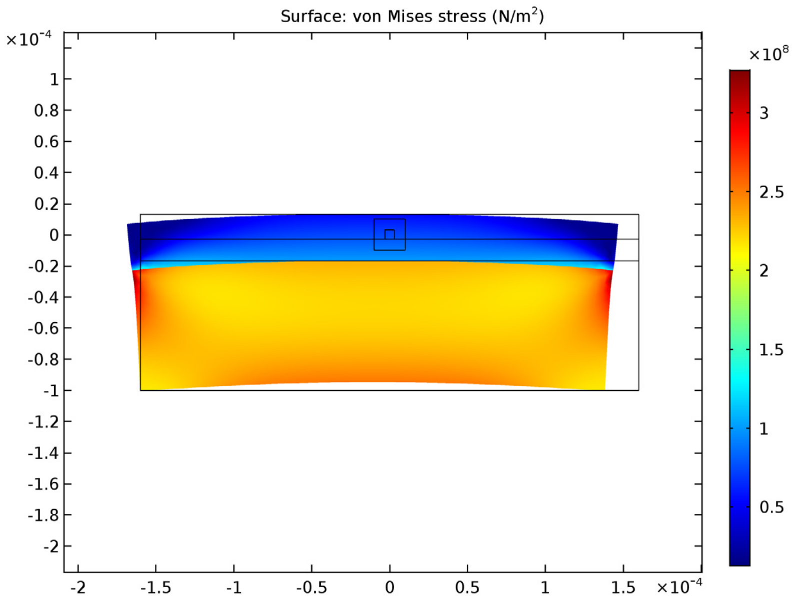

Figure 1: von Mises effective stress computed for the generalized plane strain.

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.

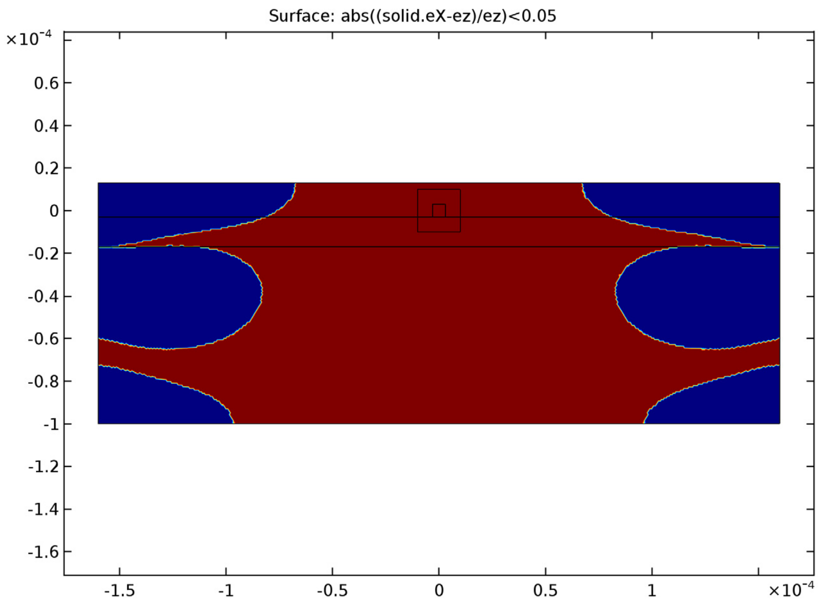

Figure 2: In the red-colored region, the relative difference between the x and z strain components is within 5%.

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.

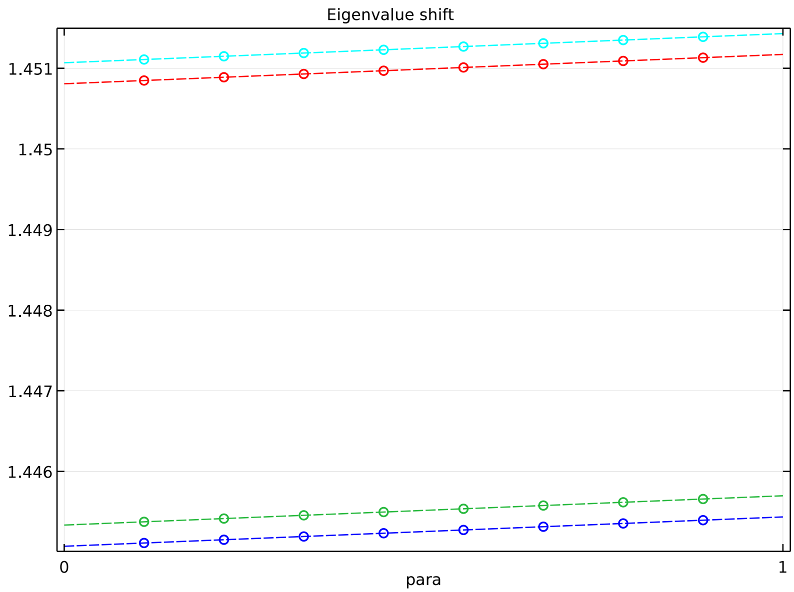

Figure 3: Effective mode indices assuming plane strain (para = 0) and generalized plane strain (para = 1).

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.

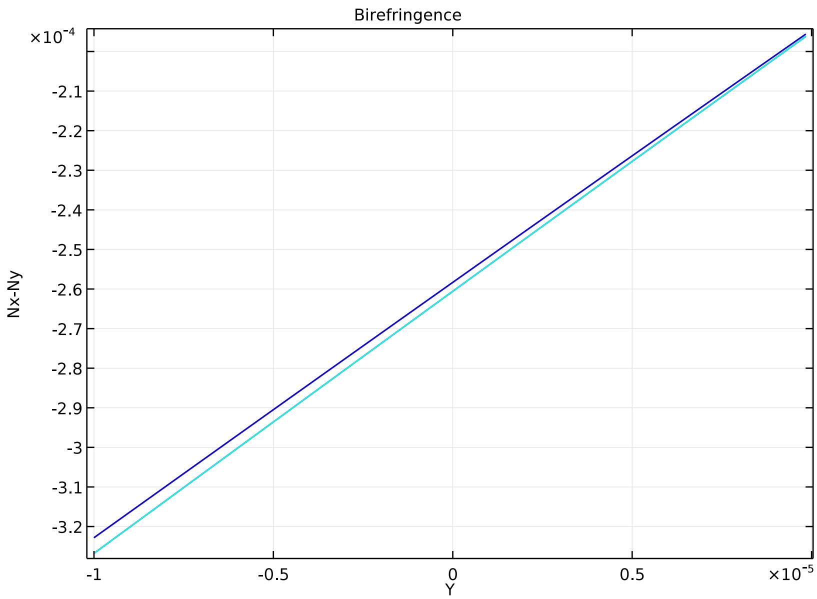

Figure 4: Birefringence along the vertical symmetry line within the waveguide for plane strain (lower curve) and generalized plane strain (upper curve).

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.

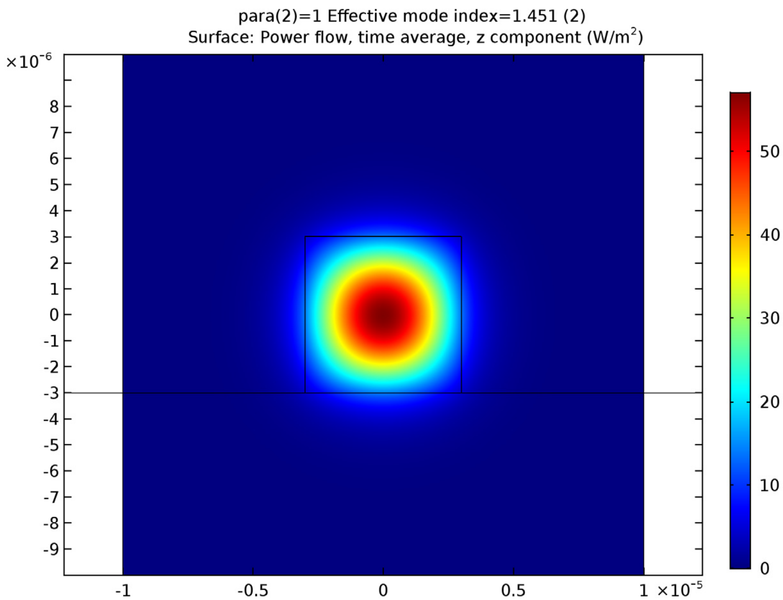

Figure 5: Eigenmode with lowest mode index, computed with the stress-optical effect under generalized plane strain assumption.

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.

Application Library path: `Wave_Optics_Module/Waveguides_and_Couplers/stress_optical_generalized`

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: use generalized plane strain, 0: use plane strain

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`.

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

5 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 On the **Physics** toolbar, click **Global** and choose **Global Equations**.
2 In the **Settings** window for Global Equations, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt, t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
e0	e0*(1-para)	0	0	
e1	e1*(1-para)	0	0	
e2	e2*(1-para)	0	0	

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

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
ez	para*(e0+(e1*X+e2*Y)/1[m])		Out-of-plane strain, z component
sx	solid.Sl11+solid.D13*ez	N/m²	Stress, x component
sy	solid.Sl22+solid.D23*ez	N/m²	Stress, y component
sz	solid.Sl33+solid.D33*ez	N/m²	Stress, z component
wc1	-(solid.D13*ez*test(solid.el11)+solid.D23*ez*test(solid.el22)+sz*test(ez))*solid.d	N/m	Weak contribution due to out-of-plane strain

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	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 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 4

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-(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

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

5 Select Domains 4–6 only.

SOLID MECHANICS (SOLID)

Weak Contribution 1

1 On the **Physics** toolbar, click **Domains** and choose **Weak Contribution**.
2 In the **Settings** window for Weak Contribution, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.
4 Locate the **Weak Contribution** section. In the **Weak expression** text field, type `wc1`.

Fixed Constraint 1

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.

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.

Electric Field (ewfd)

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

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

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

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

6 In the **Expression** text 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 On the **Electric Field (ewfd)** toolbar, click **Plot**.

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

Data Sets

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

Cut Line 2D 1

On the **Results** toolbar, click **Cut Line 2D**.

Data Sets

1 In the **Settings** window for Cut Line 2D, locate the **Data** section.

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

3 Locate the **Line Data** section. In row **Point 1**, set **Y** to `-1e-5`.

4 In row **Point 2**, set **X** to `0` and **Y** to `1e-5`.

5 Click **Plot**.

1D Plot Group 3

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

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

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

Line Graph 1

On the **1D Plot Group 3** toolbar, click **Line Graph**.

1D Plot Group 3

1 In the **Settings** window for Line Graph, locate the **y-Axis Data** section.

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

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

4 In the **Expression** text field, type `Y`.

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

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

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

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

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

10 In the associated text field, type `Y`.

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

12 In the associated text field, type `Nx-Ny`.

13 On the **1D Plot Group 3** toolbar, click **Plot**.

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.

14 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`.

2D Plot Group 4

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/Parametric Solutions 1 (sol3)**.

4 From the **Effective mode index** list, choose **1.451 (2)**.

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

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

7 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**.

8 On the **2D Plot Group 4** 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).

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 shift of the effective mode indices.

1D Plot Group 5

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

Table Graph 1

On the **1D Plot Group 5** toolbar, click **Table Graph**.

1D Plot Group 5

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 5**.

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 **Grid** section. Select the **Manual spacing** check box.

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

10 On the **1D Plot Group 5** toolbar, click **Plot**.

Electric Field (ewfd)

Note that the model includes an extension of the plane strain equation system. This implies that some of the results-processing variables available under the Solid Mechanics interface may no longer be valid or even give wrong results if used. Use the stress variables entered under the **Definition** for any additional postprocessing.