Created in COMSOL Multiphysics 6.2

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.

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 strains are independent of the out-of -plane coordinate z, that is

$$\varepsilon_{ij} = \varepsilon_{ij}(x, y)$$

Under the small strain assumption, the above equations have the following 3D solution:

$$u = u_0(x, y) - \frac{e_1}{2} z^2$$

$$v = v_0(x, y) - \frac{e_2}{2} z^2$$

$$w = (e_0 + e_1 x + e_2 y) z$$

where e_0, e_1, and e_2 are constants. Thus, at the cross section $z = 0$, one has

$$u = u_0(x, y)$$

$$v = v_0(x, y)$$

$$w = 0$$

and

$$\varepsilon_z = e_0 + e_1 x + e_2 y$$
$$\varepsilon_{yz} = \varepsilon_{yz} = 0$$

The above conditions differ from the plane strain state only by the fact that the normal out-of -plane strain component can vary linearly throughout the cross section. This approximation is expected to be good when the structure is free to expand in the out-of-plane direction, and possible bending curvature is small with respect to the extents of the structure in the *xy*-plane. It corresponds to a small rotation that is representative of each cross section of the structure along the *z*-axis.

In COMSOL Multiphysics, the Generalized Plane Strain option is available under Structural Mechanics interface in 2D. Coefficients e_0, e_1, and e_2 in the expression for the ε_z strain are modeled as extra degrees of freedom that are constant throughout the model (global variables). See the *Structural Mechanics Module User's Guide* for more details.

Results and Discussion

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

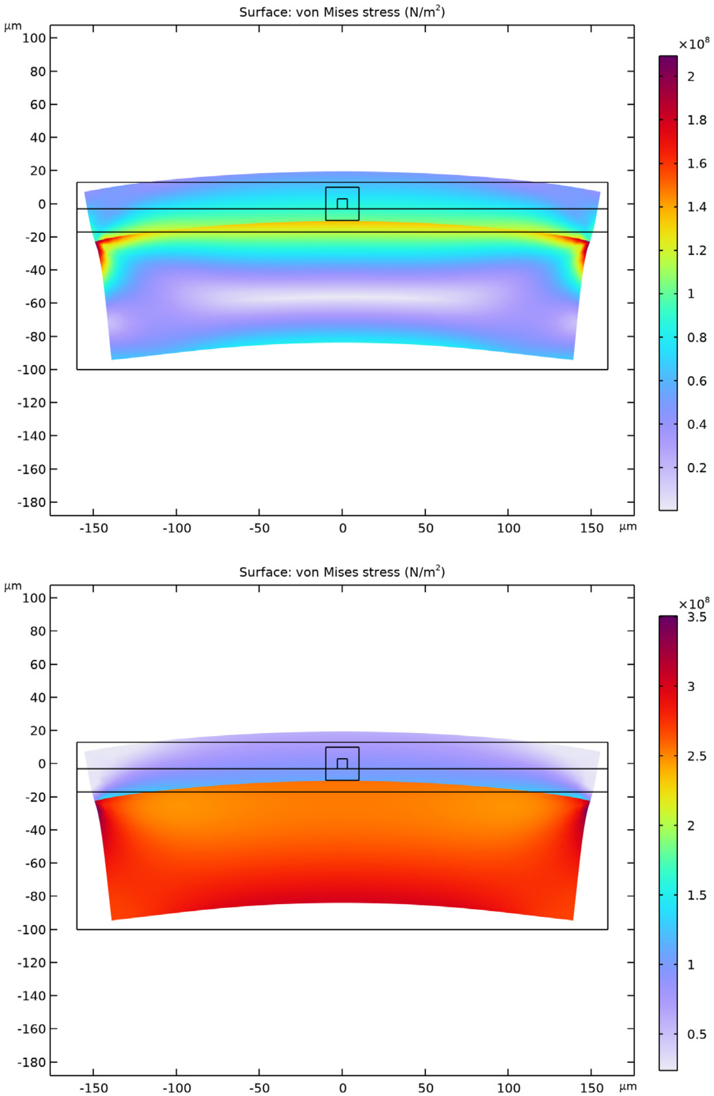
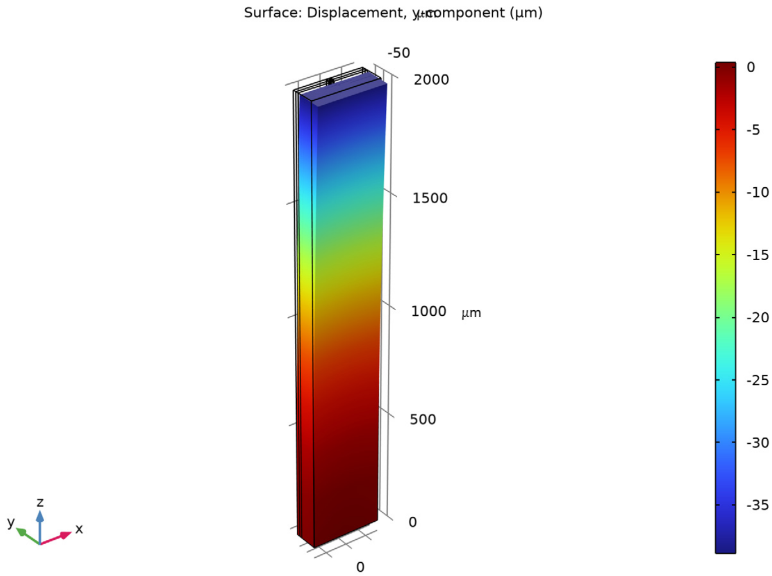


Figure 1: von Mises equivalent stress computed with (upper) and without (lower) the generalized plane strain assumption.

A 3D representation of the generalized plane strain solution is shown in this figure:



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.

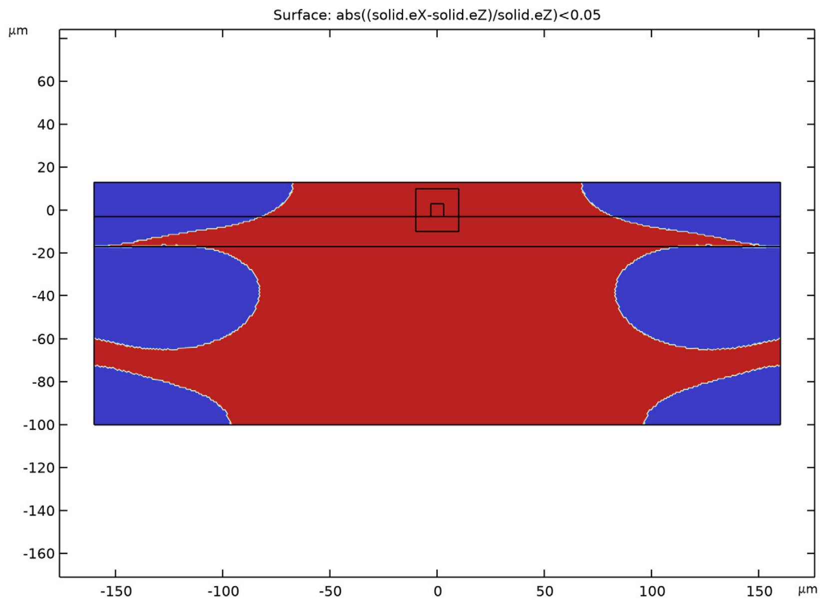


Figure 2: In the red-colored region, the relative difference between the x and z strain components is within 5%.

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.

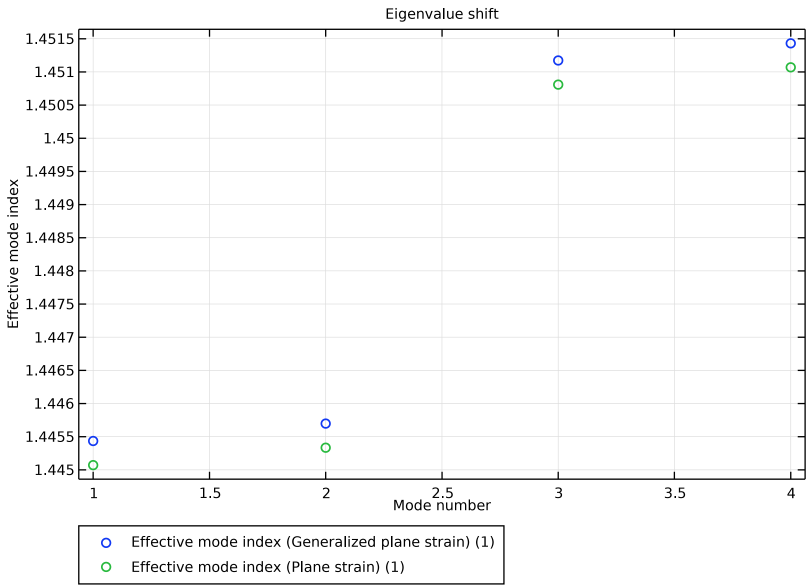


Figure 3: Effective mode indices assuming plane strain and generalized plane strain.

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.

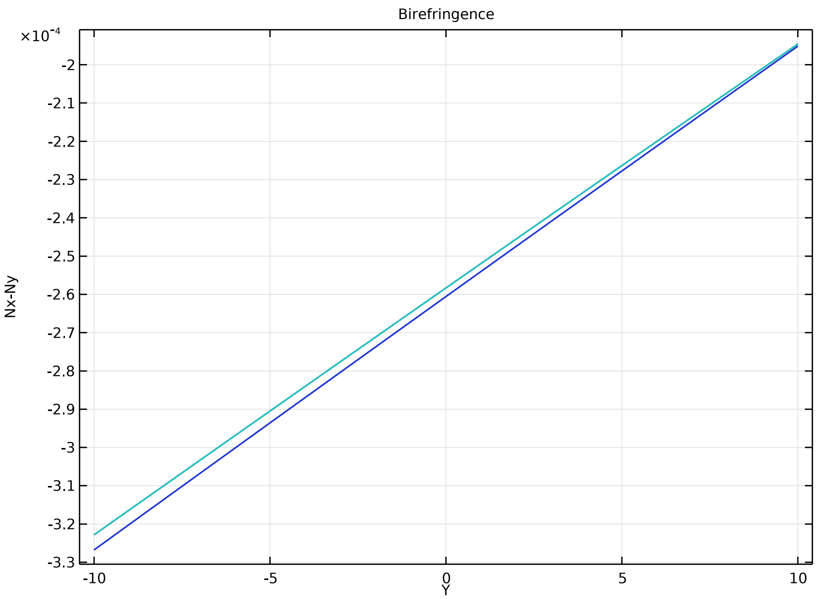


Figure 4: Birefringence along the vertical symmetry line within the waveguide for plane strain (lower curve) and generalized plane strain (upper curve).

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.

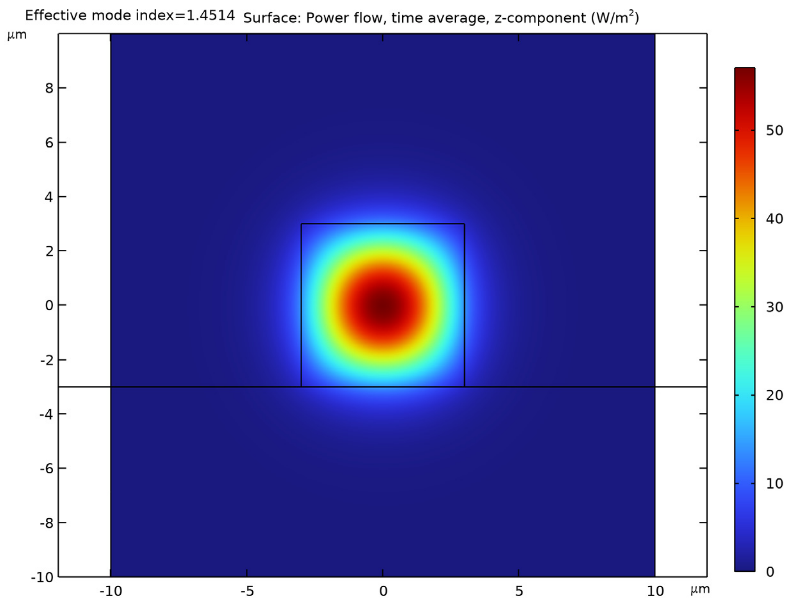


Figure 5: Eigenmode with lowest mode index, computed with the stress-optical effect under generalized plane strain assumption.

Application Library path: `Wave_Optics_Module/Waveguides/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 **Preset Studies for Some Physics Interfaces>Stationary**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

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**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `320`.

4 In the **Height** text field, type `83`.

5 Locate the **Position** section. In the **x** text field, type `-160`.

6 In the **y** text field, type `-100`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `320`.

4 In the **Height** text field, type `14`.

5 Locate the **Position** section. In the **x** text field, type `-160`.

6 In the **y** text field, type `-17`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type 320.

4 In the **Height** text field, type 16.

5 Locate the **Position** section. In the **x** text field, type -160.

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

Rectangle 4 (r4)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type 6.

4 In the **Height** text field, type 6.

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

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

Rectangle 5 (r5)

1 In the **Geometry** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type 20.

4 In the **Height** text field, type 20.

5 Locate the **Position** section. In the **x** text field, type -10.

6 In the **y** text field, type -10.

7 Click **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)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the d text field, type 2*d.

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 In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.
2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3 From the T list, choose **User defined**. In the associated text field, type `T1`.
4 Click **Go to Source** for **Volume reference temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.
3 Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `T0`.

DEFINITIONS

Variables 1

1 In 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 In 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 In the **Home** toolbar, click a= **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*solid.sx+B2*(solid.sy+solid.sz))		Refractive index, x-component
Ny	N-(B1*solid.sy+B2*(solid.sx+solid.sz))		Refractive index, y-component
Nz	N-(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)

Rigid Motion Suppression 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Motion Suppression**.

2 Select Domain 1 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

Si

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

2 In the **Settings** window for **Material**, type `Si` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`ESi`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nuSi`	1	Young's modulus and Poisson's ratio
Density	rho	`rhoSi`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`alphaSi`	1/K	Basic

SiO2

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	Variable	Value	Unit	Property group
Young's modulus	E	`ESiO2`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nuSiO2`	1	Young's modulus and Poisson's ratio
Density	rho	`rhoSiO2`	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`alphaSiO2`	1/K	Basic

MESH 1

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

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.

7 Select the **Maximum element size** check box. In the associated text field, type `0.2`.

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

Step 2: Mode Analysis

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

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

3 From the **Transform** list, choose **Effective mode index**.

4 In the **Search for modes around shift** text field, type `1.46`.

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

Exclude Solid Mechanics from the Mode Analysis step.

7 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Step 1: Stationary

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

4 In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

2 From the **Dataset** list, choose **Study 1/Solution Store 1 (sol2)**.

3 In the **Stress (solid)** toolbar, click **Plot**.

4 Click the **Go to Default View** button in the **Graphics** toolbar.

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

6 In the **Study** toolbar, click **Create Solution Copy**.

Study 1/Solution 1 - Plane Strain

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/Solution 1 - Copy 1 (sol3)**.

2 In the **Settings** window for **Solution**, type `Study 1/Solution 1 - Plane Strain` in the **Label** text field.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Generalized plane strain**.

DEFINITIONS

Variables 4

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
u1	u-solid.wxZ/2*Z^2	m	Displacement, x-component
v1	v-solid.wyZ/2*Z^2	m	Displacement, y-component
w1	(solid.wxZ*X+solid.wyZ*Y+solid.w0Z)*Z	m	Displacement, z-component

STUDY 1

In the **Study** toolbar, click **Compute**.

RESULTS

Stress (solid)

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

2 In the **Stress (solid)** toolbar, click **Plot**.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

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

Strain level

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**, type `Strain level` in the **Label** text field.

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

Surface 1

1 In the **Model Builder** window, expand the **Strain level** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `abs((solid.eX-solid.eZ)/solid.eZ)<0.05`.

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

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

6 In the **Strain level** toolbar, click **Plot**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Cut Line 2D 1

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

1 In the **Results** toolbar, click **Cut Line 2D**.

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

3 In row **Point 1**, set **Y** to `-10`.

4 In row **Point 2**, set **X** to `0` and **Y** to `10`.

5 Click **Plot**.

6 Right-click **Cut Line 2D 1** and choose **Duplicate**.

Cut Line 2D 2

1 In the **Model Builder** window, click **Cut Line 2D 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 - Plane Strain (sol3)**.

1D Plot Group 3

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

Line Graph 1

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

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `Nx-Ny`.

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

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

7 Right-click **Line Graph 1** and choose **Duplicate**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Line 2D 2**.

1D Plot Group 3

1 In the **Model Builder** window, click **1D Plot Group 3**.

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

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

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Y`.

7 Select the **y-axis label** check box. In the associated text field, type `Nx-Ny`.

8 In 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.

9 Click the **Show More Options** button in the **Model Builder** toolbar.

10 In the **Show More Options** dialog box, in the tree, select the check box for the node **Results>Views**.

11 Click **OK**.

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 `-10`.

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

5 In the **y minimum** text field, type `-10`.

6 In the **y maximum** text field, type `10`.

2D Plot Group 4

1 In 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 **Effective mode index** list, choose **1.4514**.

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

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Energy and power>Power flow, time average (spatial frame) - W/m²>ewfd.Poavz - Power flow, time average, z-component**.

3 In 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).

Global Evaluation 1

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

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Global>ewfd.neff - Effective mode index - 1**.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`ewfd.neff`	`1`	`Effective mode index (Generalized plane strain)`

4 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

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

2 Right-click **Global Evaluation 1** and choose **Duplicate**.

RESULTS

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 - Plane Strain (sol3)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`ewfd.neff`	1	`Effective mode index (Plane strain)`

5 Click **Evaluate**.

Create a table plot to visualize the shift of the effective mode indices.

1D Plot Group 5

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

Table Graph 1

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

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Plot columns** list, choose **Manual**.

4 From the **x-axis data** list, choose **Row index**.

5 In the **Columns** list, choose **Effective mode index (Generalized plane strain) (1)** and **Effective mode index (Plane strain) (1)**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

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

8 Click to expand the **Legends** section. Select the **Show legends** check box.

1D Plot Group 5

1 In the **Model Builder** window, click **1D Plot Group 5**.

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

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

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

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

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

7 In the **x-axis label** text field, type `Mode number`.

8 In the **y-axis label** text field, type `Effective mode index`.

9 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

10 From the **Position** list, choose **Bottom**.

11 In the **Number of rows** text field, type 2.

12 In the **1D Plot Group 5** toolbar, click **Plot**.

Finally, visualize the out-of-plane warping of the waveguide.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

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

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

4 Locate the **Extrusion** section. In the **z maximum** text field, type d.

5 In the **z variable** text field, type Z.

3D Plot Group 6

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Number Format** section.

3 Select the **Manual grid settings** check box.

4 In the **Precision** text field, type 4.

Surface 1

1 Right-click **3D Plot Group 6** and choose **Surface**.

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

3 In the **Expression** text field, type v1.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **x-component** text field, type u1.

4 In the **y-component** text field, type v1.

5 In the **z-component** text field, type w1.

6 Locate the **Scale** section.

7 Select the **Scale factor** check box. In the associated text field, type 1.

8 In the **3D Plot Group 6** toolbar, click **Plot**.

9 Click the **Go to Default View** button in the **Graphics** toolbar.

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