

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 Silica-on-Silicon Waveguide” on page 258](#), 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 Plane Strain application mode does not cover the bending in the z direction, so you must make modifications to the plane strain equations at the equation-system level.

Note: This model requires the RF Module and the Structural Mechanics Module.

Generalized Plane Strain

One possible extension of the plane strain formulation is to assume that the strain in the z direction has the form

$$\varepsilon_z = e_0 + e_1x + e_2y$$

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-degree terms in x and y .)

One way of validating the result of this simulation is to plot the ε_x and ε_z strains on a cross section on the symmetry line at $x = 0$. Because you can expect an identical bending in the x and z directions, these strains should be identical and vary linearly.

EXTENSION OF THE PLANE STRAIN EQUATIONS

The objective is to extend the current Plane Strain application mode with additional PDE terms that represent the generalized plane strain deformation state. In COMSOL Multiphysics, the coefficients e_0 , e_1 , and e_2 in the expression for the ε_z strain is modeled with coupling variables

associated with a point geometry object (zero dimensional object). This is a convenient way to include degrees of freedom that are constant throughout a subdomain. In order to expand the equations for the Plane Strain application mode, the new ε_z strain contributions need to be added as weak terms to the ordinary plane strain equations. The reason for this is that for COMSOL Multiphysics to be able to solve the equations as a linear problem, the assembler requires that all contributions from coupling variables are entered as weak terms.

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

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

where σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{xz} are the components of the stress tensor, ε_x , ε_y , ε_z , γ_{xy} , γ_{yz} , and γ_{xz} are the components of the strain tensor, α is the thermal expansion coefficient, T is the operating temperature, and T_{ref} is the reference temperature corresponding to the manufacturing temperature. The 6-by-6 matrix D (tensor) has entries that contain expressions in Young's modulus, E , and Poisson's ratio, ν . The matrix D is presented in the *Structural Mechanics Module User's Guide*.

For generalized plane strain (and ordinary plane strain), assume that $\gamma_{yz} = \gamma_{xz} = 0$ and two of the equations vanish

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \end{bmatrix}$$

The matrix D shrinks accordingly.

The PDE solved is Navier's equation

$$-\nabla \cdot \boldsymbol{\sigma} = \mathbf{K}$$

where

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} = c \nabla \mathbf{u} - \boldsymbol{\gamma}$$

in the case of generalized plane strain and $\mathbf{u} = (u, v, w)$ are the unknown displacement variables solved for.

The coefficient c above is a tensor containing the elements of D in a pattern that is listed in the *Structural Mechanics Module User's Guide* and

$$\gamma = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) & 0 & 0 \\ 0 & (1+\nu)\alpha(T-T_{\text{ref}}) & 0 \\ 0 & 0 & (1+\nu)\alpha(T-T_{\text{ref}}) \end{bmatrix}$$

contains the contributions from the thermal strains.

In the case of ordinary plane strain the upper left 2-by-2 submatrix of σ , γ , and D is used.

It is assumed here that the variables T and T_{ref} are constants. For information on how the temperature variables enter Navier's equation as unknowns in a heat transfer problem, see the *Structural Mechanics Module User's Guide*.

To expand the plane strain equation to a generalized plane strain formulation one needs to consider the weak form of Navier's equation. The weak form of a system of equations is a scalar equation involving integrals. See "Weak Form Modeling" on page 347 in the *COMSOL Multiphysics Modeling Guide* for more information on the appearance of the weak form for systems of equations.

To get the weak form of Navier's equation, multiply by a test function $\mathbf{v} = [u_{\text{test}}, v_{\text{test}}, w_{\text{test}}]$ and integrate

$$\int_{\Omega} \mathbf{v}(-\nabla \cdot \boldsymbol{\sigma}) d\Omega = \int_{\Omega} \mathbf{v} \cdot \mathbf{K} d\Omega$$

Integration by parts gives

$$0 = \int_{\partial\Omega} \mathbf{v} \boldsymbol{\sigma} \mathbf{n} d\Gamma - \int_{\Omega} \nabla \mathbf{v} \cdot \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{v} \cdot \mathbf{K} d\Omega$$

or in more detail, with $\mathbf{K} = \mathbf{0}$,

$$0 = \int_{\partial\Omega} \begin{bmatrix} u_{\text{test}} & v_{\text{test}} & w_{\text{test}} \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix} d\Gamma$$

$$- \int_{\Omega} \begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \\ v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \\ w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} d\Omega$$

where $u_{y, \text{test}} = \partial u_{\text{test}} / \partial y$ and similarly for the other test function derivatives. The dot product in the domain integral is interpreted as a dot product operating on the rows of the matrices so that

$$\int_{\Omega} \begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \\ v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \\ w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} d\Omega$$

$$= \int_{\Omega} \left(\begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \end{bmatrix} \right.$$

$$+ \begin{bmatrix} v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \tau_{yx} & \sigma_y & 0 \end{bmatrix}$$

$$\left. + \begin{bmatrix} w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

where

$$\sigma_x = \frac{E}{(1+\nu)(1-2\nu)} ((1-\nu)\epsilon_x + \nu\epsilon_y + \nu\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\sigma_y = \frac{E}{(1+\nu)(1-2\nu)} (\nu\epsilon_x + (1-\nu)\epsilon_y + \nu\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)} (\nu\epsilon_x + \nu\epsilon_y + (1-\nu)\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\tau_{xy} = \frac{E}{(1+\nu)(1-2\nu)} \left(\frac{1-2\nu}{2} \gamma_{xy} \right)$$

and

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

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

$$\varepsilon_z = \frac{\partial w}{\partial z}$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

Note that the displacement variable w in the z direction is not available in the Plane Strain application mode. However, it is not needed since the generalized formulation assumes a certain fix shape of the deformation in the z direction.

The same relations hold for the test functions,

$$\varepsilon_{x, \text{test}} = u_{x, \text{test}}$$

$$\varepsilon_{y, \text{test}} = v_{y, \text{test}}$$

$$\varepsilon_{z, \text{test}} = w_{z, \text{test}}$$

$$\gamma_{xy, \text{test}} = u_{y, \text{test}} + v_{x, \text{test}}$$

In COMSOL Multiphysics, the test functions u_{test} and v_{test} are denoted `u_test` and `v_test`, respectively. Their derivatives $u_{x,\text{test}}$, $u_{y,\text{test}}$, $v_{x,\text{test}}$, and $v_{y,\text{test}}$ are denoted `ux_test`, `uy_test`, `vx_test`, and `vy_test`, respectively. Note that the z components of the test functions are not available in the Plane Strain application mode.

Using this notation, the weak form of Navier's equation becomes (with $\mathbf{K} = \mathbf{0}$)

$$0 = \int_{\partial\Omega} \mathbf{v} \mathbf{\sigma} \mathbf{n} d\Gamma - \int_{\Omega} (\varepsilon_{x, \text{test}} \sigma_x + \varepsilon_{y, \text{test}} \sigma_y + \varepsilon_{z, \text{test}} \sigma_z + \gamma_{xy, \text{test}} \tau_{xy}) d\Omega$$

This expression for the weak equation shows which terms you must add as weak terms to generalize the plane strain formulation.

To see the difference compared to the ordinary plane strain formulation, the equation for σ_z can be separated out according to

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} - \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) \\ (1+\nu)\alpha(T-T_{\text{ref}}) \\ 0 \end{bmatrix} + \begin{bmatrix} \nu\varepsilon_z \\ \nu\varepsilon_z \\ 0 \end{bmatrix}$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)} (\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T-T_{\text{ref}}))$$

where D has been reduced to

$$D = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

The ordinary plane strain equation is obtained if $\varepsilon_z = 0$.

To expand the Plane Strain application mode, you must add weak contributions from ε_z need to the stress components σ_x and σ_y according to

$$-\frac{E}{(1+\nu)(1-2\nu)}(\varepsilon_{x, \text{test}}\nu\varepsilon_z + \varepsilon_{y, \text{test}}\nu\varepsilon_z)$$

In addition, σ_z needs to be included in the weak equation from being merely a postprocessing quantity in the original Plane Strain application mode,

$$-\varepsilon_{z, \text{test}}\sigma_z$$

where

$$\varepsilon_{z, \text{test}} = e_{0, \text{test}} + e_{1, \text{test}}x + e_{2, \text{test}}y$$

and σ_z was presented earlier.

In COMSOL Multiphysics you model the coefficients e_0 , e_1 , and e_2 in the expression for the ε_z strain with coupling variables as weak contributions to the ordinary plane strain equations. These coefficients represent three additional unknown variables that prevail throughout the modeled domain. In practice, these variables need to be associated with an arbitrary point of the geometry. Using coupling variables in this way specifies a map from this point onto the entire domain on which the generalized plane strain equations hold. COMSOL Multiphysics creates the test functions corresponding to e_0 , e_1 , and e_2 automatically.

Model Library path:

RF_Module/Optics_and_Photonics/stress_optical_generalized

Modeling Using the Graphical User Interface—Plane Strain Analysis

Many of the modeling steps are similar to the model [“Stress-Optical Effects in a Silica-on-Silicon Waveguide” on page 258](#), but some steps are made in a different order, and there are obviously differences when generalizing the plane strain problem.

MULTIPHYSICS SETTINGS

- 1** In the **Model Navigator**, select **2D**.
- 2** Select the **Structural Mechanics Module>Plane Strain>Static analysis** application mode.
- 3** Click **Multiphysics** then **Add** to add this application mode to the model.
- 4** Next select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves> Mode analysis** application mode. Add this mode to the model.
- 5** For convenience select **Free space wavelength** from the **Specify wave using** list. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency.
- 6** Click **OK**.

The unknown parameters ϵ_0 , ϵ_1 , and ϵ_2 should be constant within the whole modeling domain. You can achieve this by adding a new geometry with only a single point to the model and define ϵ_0 , ϵ_1 , and ϵ_2 as the dependent variables at this point. Later on you define coupling variables that map these dependent variables from the point to the whole model.

- 1** From the **Multiphysics** menu, select the **Model Navigator**. Click the **Add Geometry** button to add a 2D geometry (2D is the default space dimension).
- 2** Select the **COMSOL Multiphysics>PDE Modes>Weak Form, Point** application mode from the application mode tree.
- 3** Type ϵ_0 ϵ_1 ϵ_2 in the **Dependent variables** text field.
- 4** Click **Add** to add the Weak Form, Point application mode to the second geometry. This application mode contains the three degrees of freedom for the parameters ϵ_0 , ϵ_1 , and ϵ_2 that describe the z strain.
- 5** Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves (rfwv)** application mode. This makes this application mode specify the interpretation of the parameters to the eigenvalue solver given in the **Solver Parameters** dialog box.
- 6** Click **OK** to close the **Model Navigator**.

The Weak Form, Point application mode becomes the active application mode when you close the Model Navigator.

GEOMETRY MODELING

The three degrees of freedom ϵ_0 , ϵ_1 , and ϵ_2 must be associated with a point. The location of this point is insignificant. This means that the added geometry could equally well have been a 1D or a 3D geometry. Using the programming language, there is also an option of using a 0D geometry.

Add a point at any position, for example, the origin (0, 0), on the **Geom2** page in the drawing area.

OPTIONS AND SETTINGS

In the **Constants** dialog box enter the following names, expressions, and descriptions (optional); when done, click **OK**. They are identical to the constants defined on [page 261](#).

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nBuf	1.445	Refractive index, silica (SiO ₂)
nClad	1.445	Refractive index, silica
deltan	0.0075	Relative index difference: $\Delta = \frac{(n_{\text{core}}^2 - n_{\text{cladding}}^2)}{2n_{\text{core}}^2}$
nCore	nClad/sqrt(1-2*deltan)	Refractive index, core
alphaSi	2.5e-6 [1/K]	Thermal expansion coefficient, silicon
alphaSiO2	0.35e-6 [1/K]	Thermal expansion coefficient, silica
ESi	110 [GPa]	Young's modulus, silicon
ESiO2	78 [GPa]	Young's modulus, silica
nuSi	0.19	Poisson's ratio, silicon
nuSiO2	0.42	Poisson's ratio, silica
B1	4.2e-12 [m ² /N]	First stress optical coefficient
B2	0.65e-12 [m ² /N]	Second stress optical coefficient
T1	20 [degC]	Operating temperature
T0	1000 [degC]	Reference temperature

GEOMETRY MODELING

- 1 Switch to **Geom1** by clicking its tab.
- 2 Draw rectangles corresponding to the different regions.

REGION	LOWER LEFT CORNER	UPPER RIGHT CORNER
Silicon Wafer	(-16e-5, -1e-4)	(16e-5, -1.7e-5)

Buffer	(-16e-5, -1.7e-5)	(16e-5, -3e-6)
Cladding	(-16e-5, -3e-6)	(16e-5, 1.3e-5)
Core	(-3e-6, -3e-6)	(3e-6, 3e-6)

PHYSICS SETTINGS

Point Settings and Boundary Conditions

All regions have free boundaries, which also is the default boundary condition. However, these conditions do not suffice in creating a unique solution because the computational domain can move and rotate freely; the problem is ill-posed. The problem becomes well-posed by adding constraints at points to keep the domain fixed.

- 1 Select the **Plane Strain (smpn)** application mode from the **Multiphysics** menu.
- 2 From the **Physics** menu, choose **Point Settings**.
- 3 In the **Point Settings** dialog box, choose Point 1 and, on the **Constraint** page, select the **R_x** and **R_y** check boxes to constrain the displacements in the x - and y -direction to zero in the lower-left corner. This keeps the domain from moving (translational move).
- 4 Select Point 9 and then select the **R_y** check box to constrain the y direction only in the lower-right corner, keeping the domain from rotating but allowing it to slide in the x direction.
- 5 When done, click **OK**.

Subdomain Settings

The air domain need not be part of the plane strain model.

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 Specify the subdomain settings for the Plane Strain application mode according to the following table.

	SUBDOMAIN 1				SUBDOMAINS 2-4			
Page	Material		Loads		Material		Loads	
	E	ESi			E	ESiO2		
	v	nuSi	Temp	T1	v	nuSiO2	Temp	T1
	α	alphaSi	Tempref	T0	α	alphaSiO2	Tempref	T0

Before entering the temperatures on the **Load** page, you must select the **Include thermal expansion** check box. It is not necessary to specify the density ρ , because it does not enter the equation for static problems.

- 3 Click **OK**.

EXPRESSION VARIABLES

- 1 From the **Options** menu, choose **Expressions>Subdomain Expressions**.
- 2 In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables; when done, click **OK**.

NAME	SUBDOMAIN SELECTION	EXPRESSION
E	1	ESi
	2-4	ESiO2
nu	1	nuSi
	2-4	nuSiO2
alpha	1	alphaSi
	2-4	alphaSiO2
ex	1-4	ux
ey	1-4	vy
ez	1-4	e00+e11*x+e22*y
sx	1-4	sx_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sy	1-4	sy_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sz	1-4	E/((1+nu)*(1-2*nu))*(nu*ex+nu*ey+(1-nu)*ez-(1+nu)*alpha*(T1-T0))
N	1	nSi
	2	nBuf
	3	nCore
	4	nClad
Nx	1	N
	2-4	N-B1*sx-B2*(sy+sz)
Ny	1	N
	2-4	N-B1*sy-B2*(sx+sz)
Nz	1	N
	2-4	N-B1*sz-B2*(sx+sy)

The variables ex , ey , and ez are the strains ϵ_x , ϵ_y , and ϵ_z , and the variables sx , sy , and sz are the stresses σ_x , σ_y , and σ_z . The definitions of the refractive index variables Nx , Ny , and Nz differ slightly compared to the previous model. In this case they are expressed in the generalized stress variables sx , sy , and sz . Because of the use of integration coupling

variables (e_{00} , e_{11} , and e_{22}), COMSOL Multiphysics cannot determine the unit for other variables that directly or indirectly depend on these integration coupling variables. This causes the warnings for inconsistent units here and in the specification of the refractive index. You can disregard these warnings.

COUPLING VARIABLES

The unknown parameters e_0 , e_1 , and e_2 should be constant within the whole modeling domain. You can achieve this by defining coupling variables that map these dependent variables from the point to the entire model:

- 1 Switch to **Geom2** by clicking its tab.
- 2 Open the **Point Integration Variables** dialog box by selecting **Integration Coupling Variables>Point Variables** from the **Options** menu.
- 3 Enter the following names and expressions, each on a separate row in the table on the **Source** page.

NAME	EXPRESSION
e_{00}	e_0
e_{11}	e_1
e_{22}	e_2

The variables e_{00} , e_{11} , and e_{22} take the values of e_0 , e_1 , and e_2 , respectively, in the entire model.

SUBDOMAIN SETTINGS

Finally, enter the additional terms in the generalized plane strain problem.

- 1 Switch to **Geom1** by clicking its tab.
- 2 Select **Equation System>Subdomain Settings** from the **Physics** menu.
- 3 On the **Weak** tab in the **Subdomain Settings - Equation System** dialog box, add $-E/((1+\nu)*(1-2*\nu))*(e_{x_test}*\nu*e_z+e_{y_test}*\nu*e_z)-e_{z_test}*s_z$ in the second row of the **weak** term for all subdomains. The two first rows correspond to the plane strain problem. You can enter this expression in any of these two fields, because the software adds them together when assembling the problem.
- 4 Click **OK**.

MESH GENERATION

- 1 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.
- 2 Refine it twice by clicking the **Refine Mesh** button on the Main toolbar twice.
- 3 Switch to **Geom2** by clicking its tab.

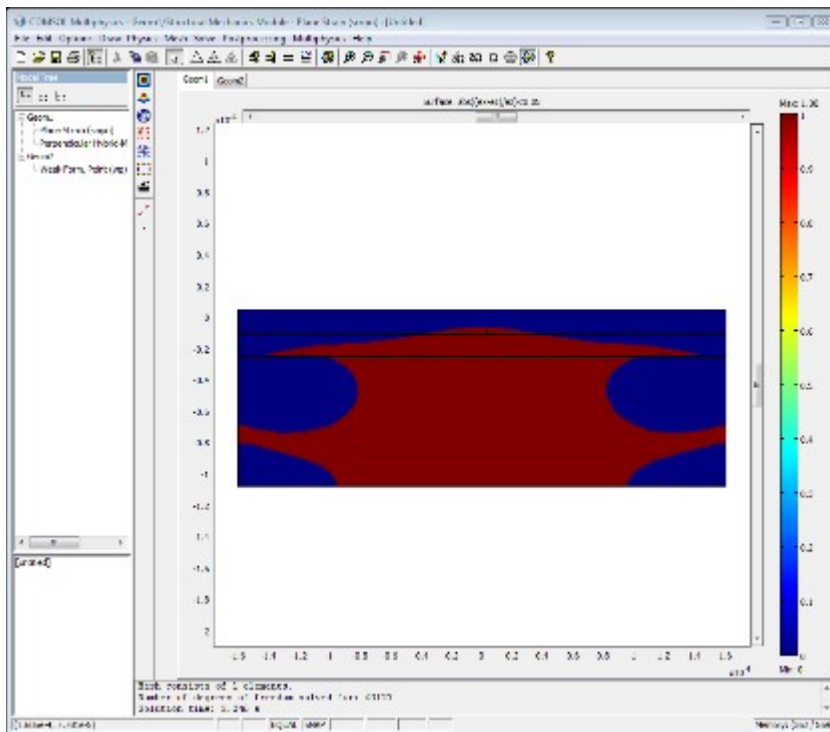
- 4 Initialize the mesh in this geometry too.

COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box select the **Stationary** solver. When done, click **OK**.
- 3 From the **Solve** menu, choose **Solver Manager**.
- 4 On the **Solve For** tab in the **Solver Manager** dialog box, select only the **Plane Strain** and the **Weak Form, Point (wp)** application modes. This excludes the hybrid-mode waves application mode from the equations to solve. When done, click **OK**.
- 5 Click the **Solve** button, then click **OK**.

POSTPROCESSING AND VISUALIZATION

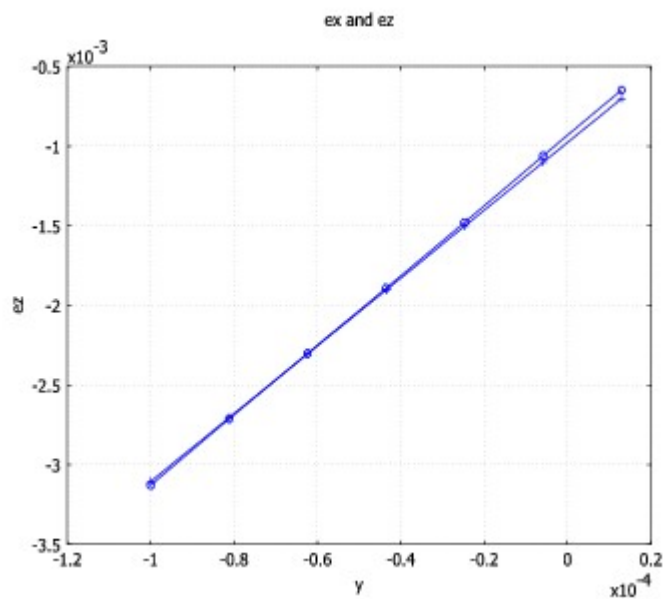
- 1 Switch to **Geom1** by clicking its tab.
- 2 From the **Postprocessing** menu, choose **Plot Parameters**.
- 3 Click the **Surface** tab on the **Plot Parameters** dialog box. Change the string in the **Expression** edit field on the **Surface Data** page to $\text{abs}((\epsilon_x - \epsilon_z) / \epsilon_z) < 0.05$. Clear the **Smooth** check box. Click **OK** to visualize the area where the relative difference between the x and z strain 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.



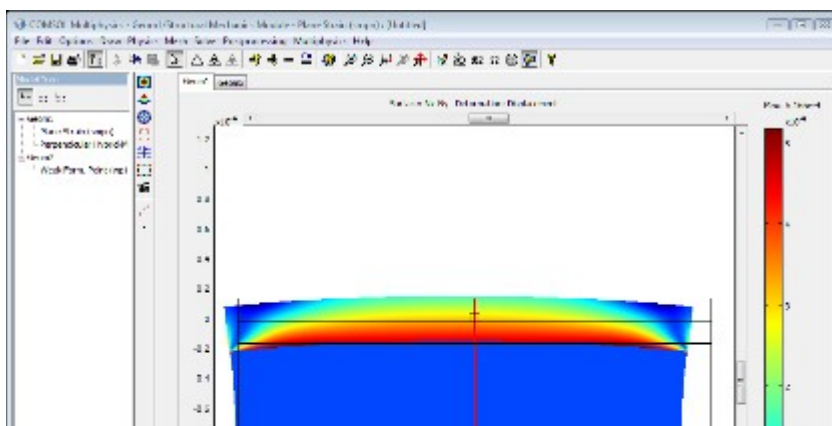
- 4 For symmetry reasons the strains ϵ_x and ϵ_z should be equal. To see how well this model has achieved this, make a cross-section plot of them. Open the **Cross-Section Plot Parameters** dialog box from the **Postprocessing** menu. Enter the **Title** ϵ_x and ϵ_z by

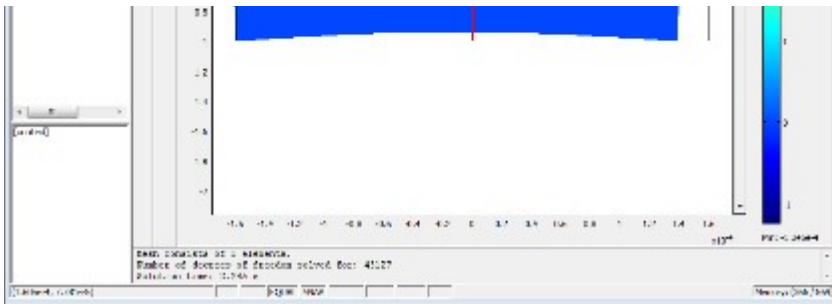
using the **Title/Axis** dialog box on the **General** page. On the **Line/Extrusion** tab, set **x0** to 0, **x1** to 0, **y0** to $-1e-4$, and **y1** to $1.3e-5$ to make a vertical line across the structure. Enter the **y-axis data expression** e_x , and set the **Line resolution** to 25. Set the **x-axis data** to y . Click the **Line Settings** button and select **Circle** as **Line marker**; then click **OK**. Click **Apply** to plot e_x .

- 5 On the **General** tab check **Keep current plot** to be able to plot e_z in the same figure window.
- 6 Then change the **Expression** to e_z in **y-axis data**. Use **Plus sign** as **Line marker** for this plot. Click **OK** to make the plot.



- 7 To view the stress-induced birefringence $n_x - n_y$ together with the deformation, open the **Plot Parameters** dialog box and select the **Surface** and **Deformed shape** check boxes in the **Plot type** area. On the **Surface** page, enter the $N_x - N_y$ in the **Expression** edit field on the **Surface Data** page, and select the **Smooth** check box again. In the **Deformation data** area, choose **Plane Strain (smpn) > Displacement** from the **Predefined quantities** list on the **Subdomain Data** page, which is the default selected on the **Deform** tab. Click **OK**.





Note: This model includes an extension of the plane strain PDE system. This means that some of the postprocessing variables in the Plane Strain application mode are no longer valid and give wrong results if used. We recommend that you use only the variables entered in the **Subdomain Expressions** dialog box for postprocessing.

Modeling Using the Graphical User Interface—Optical Mode Analysis

Continue with the mode analysis in the same way as in the previous model on [page 268](#).

GEOMETRY MODELING

You can reduce the computational domain significantly for the optical mode analysis because the energy of the fundamental modes is concentrated in the core region, and the energy density decays exponentially in the cladding and buffer regions.

Draw a rectangle with corners at $(-1e-5, -1e-5)$ and $(1e-5, 1e-5)$ on the **Geom1** page.

PHYSICS SETTINGS

Next, set up the parameters for the hybrid-mode wave problem. Select the **Perpendicular Hybrid-Mode Waves (rfwv)** application mode in the **Multiphysics** menu.

Scalar Variables

- 1 From the **Physics** menu, choose **Scalar Variables**.
- 2 In the **Application Scalar Variables** dialog box, set the **Free space wavelength** to $1.55e-6$. Click **OK**.

Subdomain Settings

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 Deactivate the Perpendicular Hybrid-Mode Waves application mode in all subdomains except 4, 5, and 6 (clear the **Active in this domain** check box for those subdomains). This makes sure that the solution for the electromagnetic waves only includes the newly drawn rectangle.
- 3 For all active domains, select the **Specify material properties in terms of refractive index** button. Enter the values for N_x , N_y , and N_z , respectively, as anisotropic refractive

indices by selecting **Diagonal** from the **Anisotropy** list in the extended **n** edit field and changing the entries on the diagonal in the refractive index matrix

- 4 When done, click **OK**.

EXPRESSION VARIABLES

Occasionally the associative geometry algorithm does not update all variables and settings exactly as expected when modifying the geometry. In this case the expression for **N** might have got the wrong value in domain 6. In the **Subdomain Expressions** dialog box change the value of **N** in Subdomain 6 to `nCore`.

MESH GENERATION

- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 On the **Subdomain** tab in the **Free Mesh Parameters** dialog box, set the **Maximum element size** to $2e-6$ for Subdomains 4 and 5, and to $2.5e-7$ for Subdomain 6.
- 3 Click the **Remesh** button to initialize the mesh, then click **OK**.

COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.
- 3 Enter 1.46 in the text field **Search for effective mode indices around**. The default **Desired number of effective mode indices** is set to 6 , which is fine. Click **OK**.

These settings make the eigenmode solver search for the 6 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. For propagating modes it must hold that

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

- 4 From the **Solve** menu, choose **Solver Manager**.
- 5 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves (rfwv)** application mode on the **Solve For** page. This ensures that only this application mode is part of the eigenmode computation.
- 6 On the **Initial Value** page, set the initial value to **Current solution** in the **Initial value** area. This takes the plane strain solution and uses it to evaluate the refractive indices for the mode analysis.
- 7 Click the **Solve** button, then click **OK**.

The following table contains the effective mode indices together with the effective mode indices obtained from the analysis in the previous model without the generalization of the plane strain equations.

	GENERALIZED PLANE STRAIN	PLANE STRAIN	DIFFERENCE
EFFECTIVE MODE INDEX			*1e-3
$n_{\text{eff}1}$	1.444923	1.444873	-.05
$n_{\text{eff}2}$	1.445195	1.445134	-.06
$n_{\text{eff}3}$	1.445213	1.445138	-.07
$n_{\text{eff}4}$	1.445487	1.445399	-.09
$n_{\text{eff}5}$	1.450928	1.450871	-.06
$n_{\text{eff}6}$	1.451217	1.451135	-.08

There is a systematic shift in the propagation constants when the strain in the z direction is taken into account.