Model created in COMSOL Multiphysics 6.4

# Thermal Heating of a Semiconductor Saturable Absorber Mirror (SESAM)

## *Introduction*

Semiconductor Saturable absorber mirrors (SESAMs) are devices used for shaping optical pulses and used in laser cavities to form mode locked lasers. The pulse shaping performance from the SESAM comes from a nonlinear absorbance profile in the material that has a higher absorption rate for lower intensity light and a lower absorption rate for higher intensity light. At higher powers, the optical absorption in the SESAM results in an increase in temperature and, in turn, thermal expansion, which deforms the surface of the SESAM. This deformation affects the reflected beam shape, changing the focal properties of the mirror. This can cause instabilities in mode locked laser cavities as the SESAM warms during operation and changes the shape of the reflected beam. This model demonstrates the thermal heating of a SESAM and the effect this has on the reflected beam.

A SESAM typically consists of a layered Bragg reflector mounted to a substrate. Within the Bragg reflector, there are one or more layers of quantum wells (QWs). These are made of a semi-conductor material that is highly absorptive at the design wavelength, for example, GaAs-InAs. At higher incident optical intensities, the free electrons in the valence band that absorb each photon are depleted, resulting in saturated absorption, so a higher proportion of the high-intensity light is reflected. The timescale of the optical pulse to the recovery time of the SESAM carriers determines if the behavior is modeled as fast or slow saturable absorption. This model demonstrates fast saturable absorption.

## *Model Definition*

### GEOMETRY

The SESAM in this model is represented by a 2D cross-section. The laser beam is represented by a Gaussian beam incident on the SESAM through an air domain, as shown in Figure 1. The SESAM is on a GaAs substrate that is mounted on a heat sink, which is represented by a fixed temperature boundary. The SESAM is an anti-resonant design consisting of a 30-layer Bragg reflector with a QW, as shown in Figure 2.

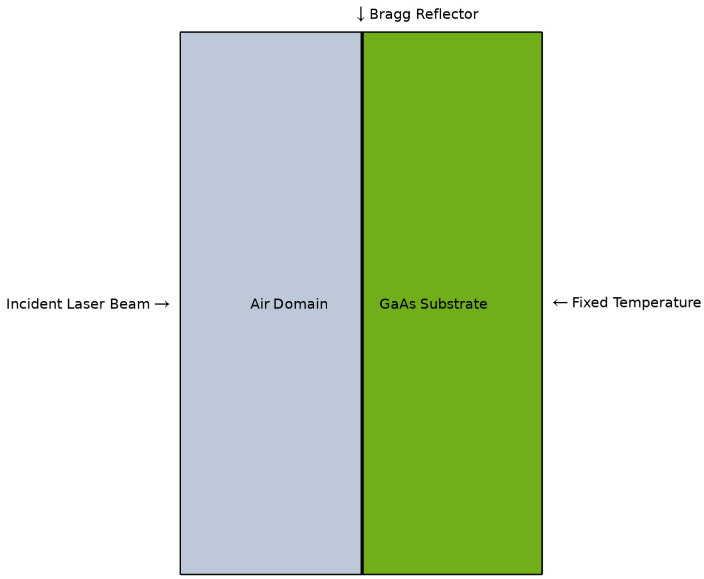



*Figure 1: Cross-section of SESAM with a laser beam excited from the boundary on the left side of the air domain. The SESAM is grown on a GaAs Substrate attached to a heat sink, which is represented by a fixed temperature on the right boundary.*

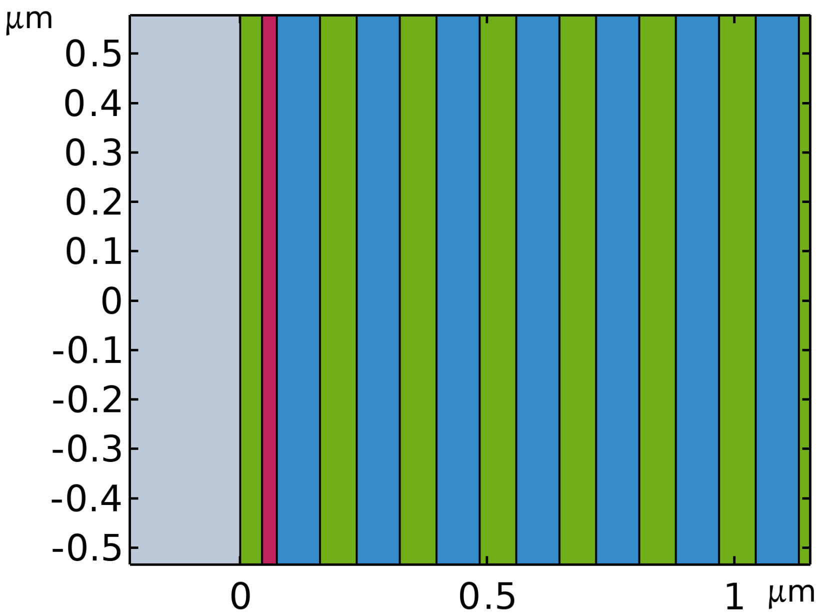



*Figure 2: Closeup of the Bragg reflector geometry: green — GaAs layers, blue — AlAs layers, red — GaInAs QW.*

The layers of the Bragg reflector are alternating high index, $n_H$, GaAs and lower index, $n_L$, AlAs. The layer thickness is such that $n_H/t_H = n_L/t_L = \lambda_0/4$. Further details can be found in the Ray Optics Module Application Library model *Distributed Bragg Reflector*. The QW is added at the interface between the first GaAs and AlAs layers. The layers form an anti-resonant SESAM with the E-field being a minimum at the SESAM surface. The first peak of the E-field coincides with the QW, shown in Figure 3.

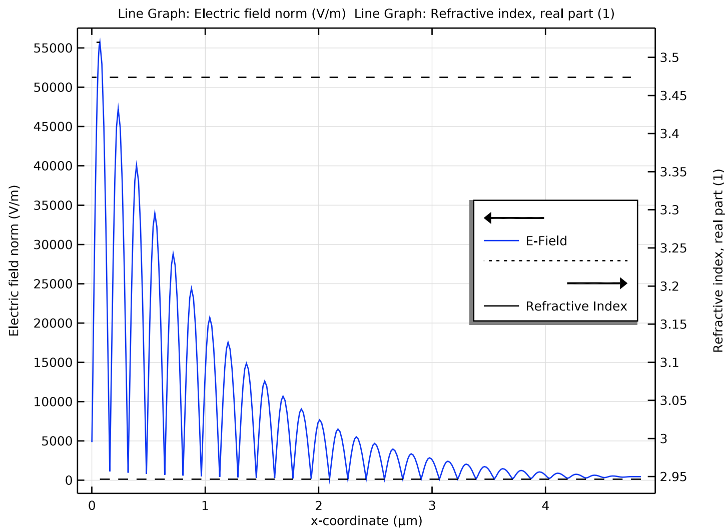



*Figure 3: Electric field within the Bragg reflector. The black line shows the layers of high and low refractive index corresponding to the GaAs and AlAs layers, as well as the higher index GaInAs of the QW. The blue line shows the E-field in each layer. The maximum coincides with the QW.*

## MATERIALS

The optical absorption in the materials is set in the extinction coefficient, that is, the imaginary part of the refractive index, $ki$. The material properties have a linear absorption by default. To model the nonlinear and saturating behavior of the semiconductors, $ki$ is modified to include an extra factor $d$, which is dependent on the optical intensity and denotes fast saturable absorption.

$$\delta_q = \frac{1}{\left(1 + \frac{I_{\text{pulse}}}{I_0}\right)}$$

where $I_{\text{pulse}}$ is the incident pulse intensity and $I_0$ is the Saturation intensity of the SESAM.

## PHYSICS

This model includes four physics interfaces with two Multiphysics couplings. The main simulation is conducted in the first Frequency-Stationary Study. This study type is suited to handle the frequency solution from the Electromagnetic Waves, Beam Envelopes interface and the stationary solution from Heat Transfer in Solids, which are coupled through Electromagnetic Heating. The deformation of the SESAM is simulated in the same study by adding a Solid Mechanics interface that couples to Heat Transfer in Solids through the Thermal Expansion Multiphysics.

A separate Global ODEs and DAEs interface and a second Stationary study are added to provide curve fitting to the electric field produced from the first study and to quantify the results.

### MESH

To ensure precise modeling of the optical effects, a fine mesh that is subwavelength in element size is required. However, the thermal and structural properties of the model do not require such a fine mesh, so these regions can be set with a coarser mesh for computational efficiency.

## *Results and Discussion*

The saturable nature of the SESAM means that the model does not have a linear response to the incident laser power, as the proportion of energy absorbed depends on the pulse parameters and intensity. The absorption within the SESAM is shown in Figure 4. The complex refractive index is much lower in the regions with a higher electric field amplitude.

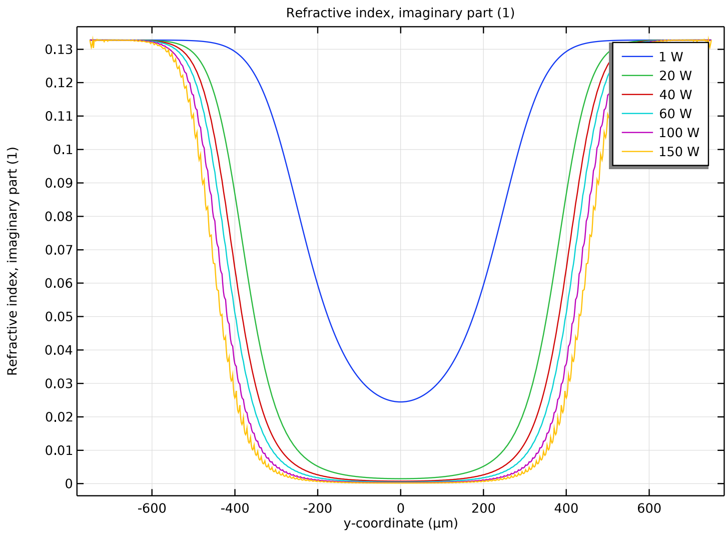



*Figure 4: Reduction of the extinction coefficient in the QW as the SESAM is saturated at higher incident powers.*

Figure 5 shows the resultant temperature distribution within the model. The fixed room temperature boundary at the back of the substrate sets the lower bound of the temperature distribution on that side. The material properties then set how the SESAM deforms at those temperatures. The expansion of the SESAM surface is shown in Figure 6. This is a small fraction of the wavelength, but it has a measurable impact on the phase and wavefront curvature of the reflected beam. This curvature is shown for different laser incident powers in Figure 8.

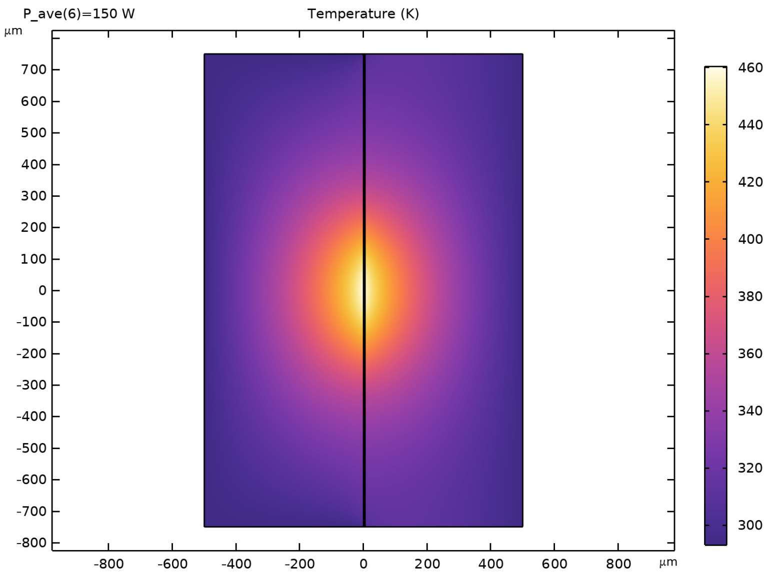



*Figure 5: Temperature distribution in both the air and substrate domains.*

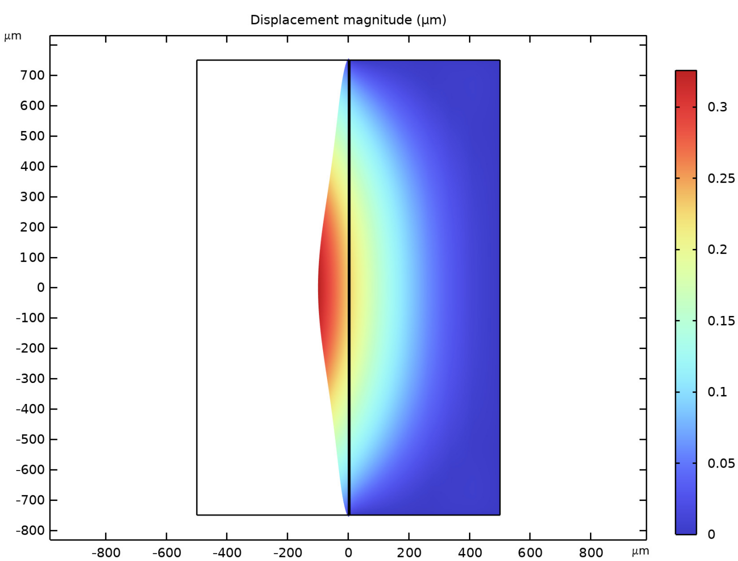



*Figure 6: The deformation of the SESAM due to thermal expansion.*

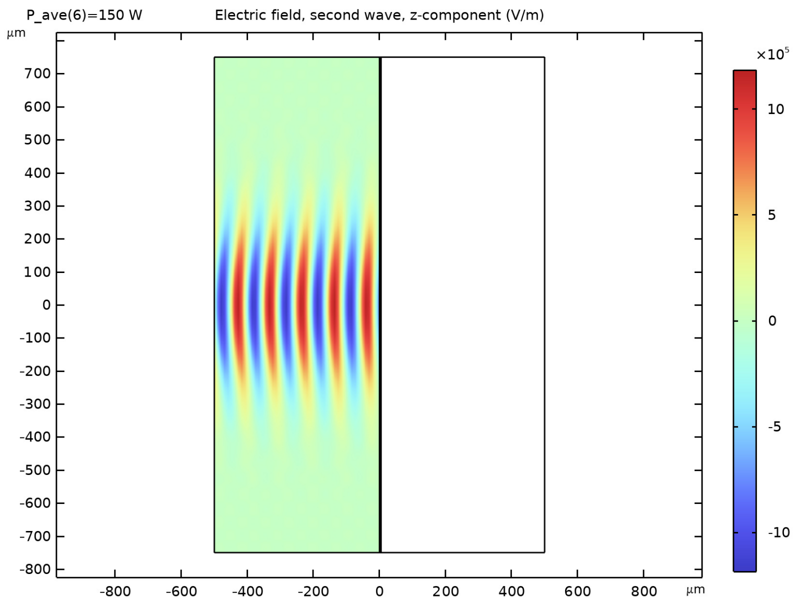



*Figure 7: The electric field of the reflected beam showing the wavefront curvature from the deformed SESAM surface.*

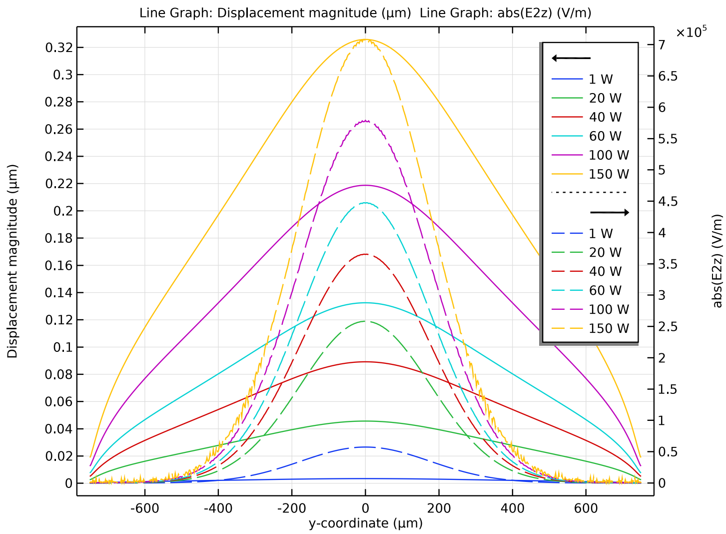



*Figure 8: The curvature of the SESAM surface under different incident laser powers (solid lines). The Electric field of the reflected beam (dashed lines).*

The radius of curvature of the SESAM surface decreases with laser power as the surface deforms further. This can be seen in Figure 9, which shows the resultant change in the curvature of the mirror from the fitted curves.

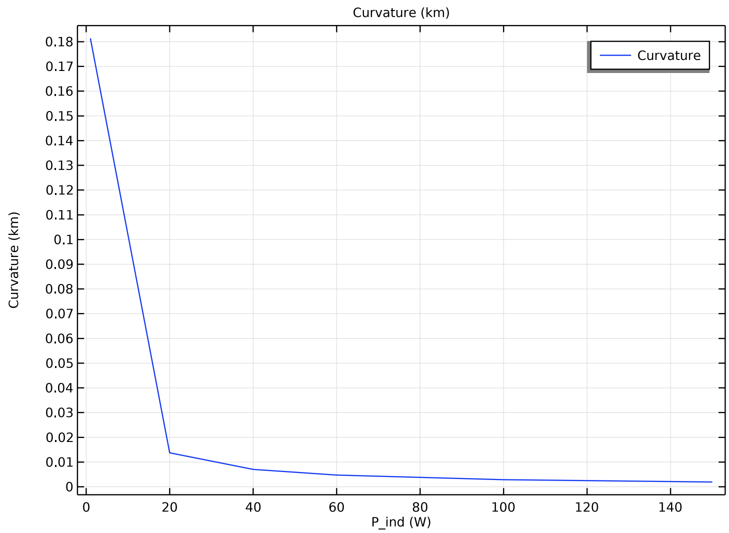



*Figure 9: Change in mirror curvature as a function of incident laser power.*

---

**Application Library path:** `Wave_Optics_Module/Couplers_Filters_and_Mirrors/sesam_laser_heating`

---

## *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 **Heat Transfer** > **Electromagnetic Heating** > **Laser Heating**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Frequency–Stationary**.

6 Click **Done**.

## 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**.

## GLOBAL DEFINITIONS

### *SESAM Parameters*

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

2 In the **Settings** window for **Parameters**, type `SESAM Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

### *Beam Parameters*

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

2 In the **Settings** window for **Parameters**, type `Beam Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

## GEOMETRY 1

### *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 `thickness_air+thickness`.

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

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

6 In the **y** text field, type `-height/2`.

*GaAs*

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

2 In the **Settings** window for **Rectangle**, type GaAs in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type thickness_GaAs.

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

5 Locate the **Position** section. In the **y** text field, type -height/2.

*GaAs Layers*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type GaAs Layers in the **Label** text field.

3 On the object **r2**, select Domain 1 only.

*AlAs*

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

2 In the **Settings** window for **Rectangle**, type AlAs in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type thickness_AlAs.

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

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

6 In the **y** text field, type -height/2.

*AlAs Layers*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type AlAs Layers in the **Label** text field.

3 On the object **r3**, select Domain 1 only.

*Array 1 (arr1)*

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the objects **r2** and **r3** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type N_layers.

5 Locate the **Displacement** section. In the **x** text field, type thickness_AlAs+thickness_GaAs.

*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 `thickness_QW`.

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

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

6 In the **y** text field, type `-height/2`.

This puts the QW layer over the first interface between the GaAs and AlAs layers. This places them in the antinode of the E-field in the Bragg reflector.

*QW*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `QW` in the **Label** text field.

3 On the object **r4**, select Domain 1 only.

*Union 1 (uni1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 From the **Input objects** list, choose **All objects**.

4 Click **Build Selected**.

*Substrate*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Substrate` in the **Label** text field.

3 On the object **uni1**, select Domain 63 only.

*GaAs Domains*

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `GaAs Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog, in the **Selections to add** list, choose **GaAs Layers** and **Substrate**.

5 Click **OK**.

*SESAM Surface*

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `SESAM Surface` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **uni1**, select Boundary 4 only.

*Point 1 (pt1)*

In the **Geometry** toolbar, click **Point**.

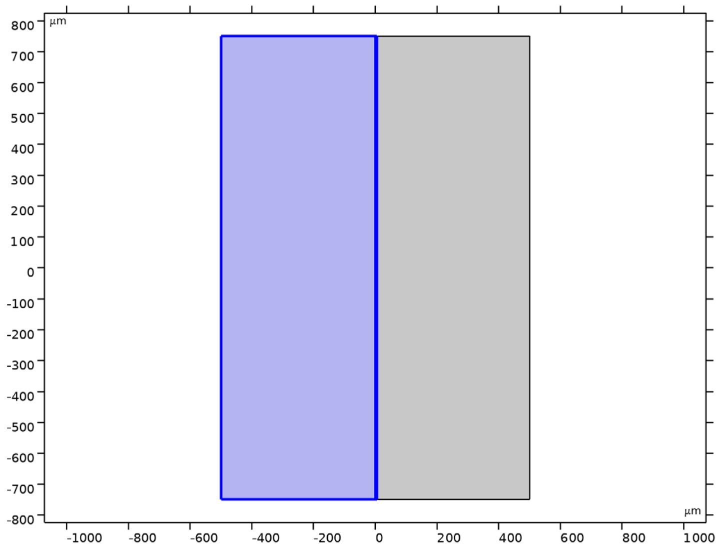
*Form Union (fin)*

Click **Build All**.

### ELECTROMAGNETIC WAVES, BEAM ENVELOPES (EWBE)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Beam Envelopes (ewbe)**.

**2** Select Domains 1–62 only.




**3** In the **Settings** window for **Electromagnetic Waves, Beam Envelopes**, locate the **Components** section.

**4** From the **Electric field components solved for** list, choose **Out-of-plane vector**.

*Scattering Boundary Condition 1*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

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

**3** Select Boundary 1 only.

**4** In the **Settings** window for **Scattering Boundary Condition**, locate the **Scattering Boundary Condition** section.

**5** From the **Incident field** list, choose **Gaussian beam**.

6 In the $w_0$ text field, type `w0`.

7 From the **Input quantity** list, choose **Power**.

8 In the $P$ text field, type `P_ave/w0`.

9 Specify the $\mathbf{E}_{g0}$ vector as

| | |
|---|---|
| 1 | z |

*Scattering Boundary Condition 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Scattering Boundary Condition**.

2 Select Boundary 188 only.

This scattering boundary condition absorbs any remaining optical power assuming it is dissipated into the substrate without having to include that domain in the optical modeling.

## HEAT TRANSFER IN SOLIDS (HT)

*Temperature 1*

Next, set a fixed temperature at the backface of the substrate to emulate an attached heat sink. The fixed temperature around the air domain maintains ambient room temperature.

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundaries 1–3 and 191 only.

## ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

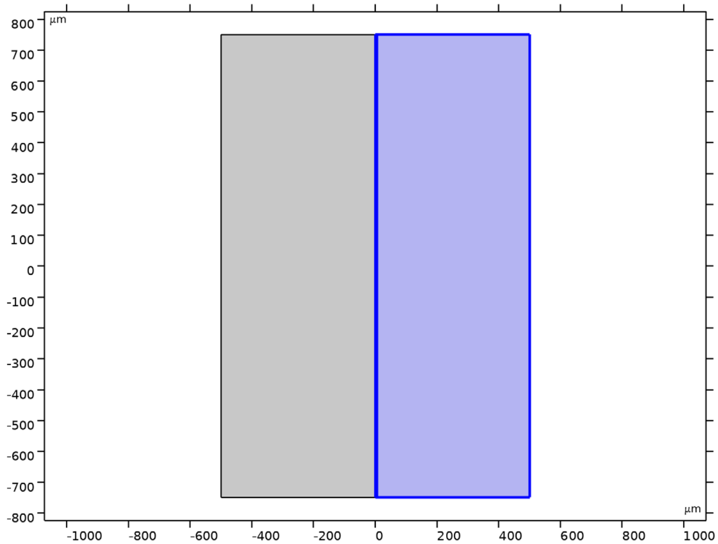
3 In the tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

4 Click the **Add to Component 1** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

#### SOLID MECHANICS (SOLID)

Select Domains 2–63 only.




*Fixed Constraint 1*

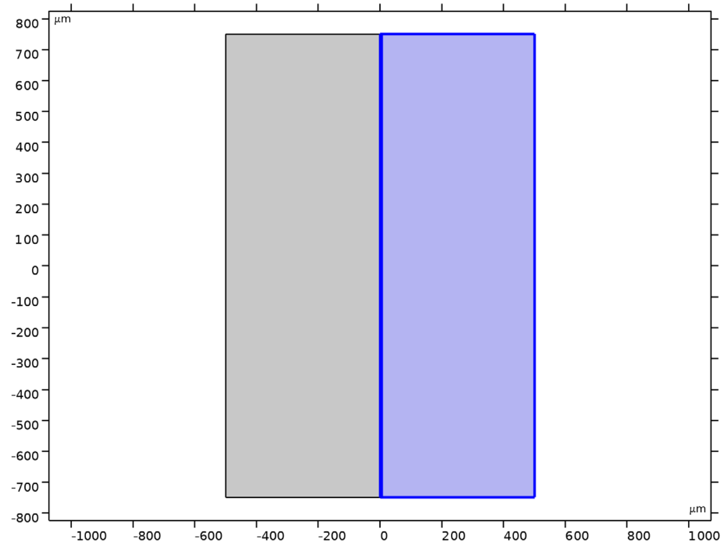
1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. Click the **Select Box** button in the **Graphics** toolbar.
3. Select Boundaries 189–191 only.

#### MULTIPHYSICS

*Thermal Expansion 1 (te1)*

1. In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain** > **Thermal Expansion**.
2. Click the **Select Box** button in the **Graphics** toolbar.

**3** Select Domains 2–63 only.




## MESH 1

The nature of this model requires a specific mesh to solve efficiently as the optical domains require a much finer mesh then that needed to model thermal effects.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3. From the list, choose **User-controlled mesh**.
4. In the **Mesh** toolbar, click **Clear Sequence**.

### *Distribution 1*

1. Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Distribution**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `floor(0.12*height/lda0)`.

### *Distribution 2*

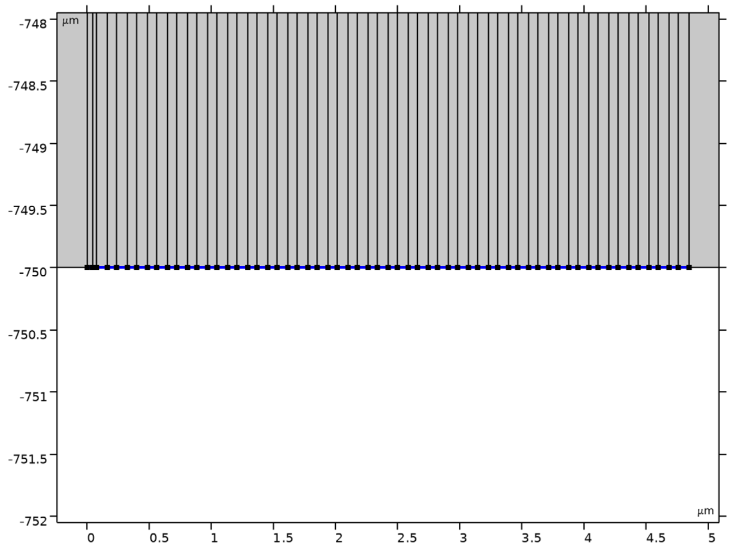
1. In the **Model Builder** window, right-click **Mesh 1** and choose **Distribution**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `floor(0.1*thickness_air/lda0)`.

*Distribution 3*

1 Right-click **Mesh 1** and choose **Distribution**.

2 Click the **Select Box** button in the **Graphics** toolbar.

3 Select Boundaries 5, 9, 12, 15, 18, 21, 24, 27, 30, 33, 36, 39, 42, 45, 48, 51, 54, 57, 60, 63, 66, 69, 72, 75, 78, 81, 84, 87, 90, 93, 96, 99, 102, 105, 108, 111, 114, 117, 120, 123, 126, 129, 132, 135, 138, 141, 144, 147, 150, 153, 156, 159, 162, 165, 168, 171, 174, 177, 180, 183, and 186 only.




*Distribution 4*

1 Right-click **Mesh 1** and choose **Distribution**.

2 Select Boundary 189 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type `floor(0.03*thickness/lda0)`.

6 In the **Element ratio** text field, type `0.1`.

7 From the **Growth rate** list, choose **Exponential**.

*Mapped 1*

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, click to expand the **Reduce Element Skewness** section.

3 Select the **Adjust edge mesh** checkbox.

**4** Click **Build All**.

#### DEFINITIONS

*Variables SESAM*

**1** In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

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

These variables define the saturation properties of the SESAM and will be applied in the material properties.

**3** In the **Settings** window for **Variables**, type `Variables SESAM` in the **Label** text field.

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

| Name | Expression | Unit | Description |
|---|---|---|---|
| `I_pulse` | `ewbe.Poav1x/(Rrep*tau_p)` | W/m² | `Pulse intensity` |
| `del_q` | `1/(1+I_pulse/I_0)` | | `Change in absorption` |

*Fitting Variables*

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

These variables will be used later when fitting the curvature of the SESAM.

**2** In the **Settings** window for **Variables**, type `Fitting Variables` in the **Label** text field.

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

| Name | Expression | Unit | Description |
|---|---|---|---|
| `delx_para` | `-1/Radius*y^2+Peak` | | `Parabolic approximation` |
| `Para_fit` | `intop1((delx-delx_para)^2)` | | `Error function` |
| `delx` | `withsol('sol3', solid.disp, setval(P_ave, P_ind))` | m | `Numerical Solution` |
| `delx_0` | `withsol('sol3', maxop1(solid.disp))` | | `Maximum displacement` |

*Integration 1 (intop1)*

**1** In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

**2** In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

**4** From the **Selection** list, choose **SESAM Surface**.

*Integration 2 (intop2)*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Boundary 1 only.

*Maximum 1 (maxop1)*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **SESAM Surface**.

*Integration 3 (intop3)*

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Point 4 only.

## ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Air**.

4 Click the **Add to Component** button in the window toolbar.

5 In the tree, select **Optical** > **Inorganic Materials** > **As - Arsenides** > **Models and simulations** > **GaAs (Gallium arsenide) (Adachi 1989: n,k 0.207-12.4 um)**.

6 Click the **Add to Component** button in the window toolbar.

## MATERIALS

*GaAs (Gallium arsenide) (Adachi 1989: n,k 0.207-12.4 um) (mat2)*

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

2 From the **Selection** list, choose **GaAs Domains**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | `55[W/(m*K)]` | W/(m·K) | Basic |
| Density | rho | `5318[kg/m^3]` | kg/m³ | Basic |
| Heat capacity at constant pressure | Cp | `330[J/(kg*K)]` | J/(kg·K) | Basic |
| Young's modulus | E | `85.5[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.31` | 1 | Young's modulus and Poisson's ratio |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | `5.73e-6` | 1/K | Basic |

## ADD MATERIAL

**1** Go to the **Add Material** window.

**2** In the tree, select **Optical** > **Inorganic Materials** > **As - Arsenides** > **Models and simulations** > **AlAs (Aluminium arsenide) (Rakic and Majewski 1996: n,k 0.221-2.48 um)**.

**3** Click the **Add to Component** button in the window toolbar.

## MATERIALS

*AlAs (Aluminium arsenide) (Rakic and Majewski 1996: n,k 0.221-2.48 um) (mat3)*

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

**2** From the **Selection** list, choose **AlAs Layers**.

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

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | `90[W/(m*K)]` | W/(m·K) | Basic |
| Density | rho | `3720[kg/m^3]` | $kg/m^3$ | Basic |
| Heat capacity at constant pressure | Cp | `330[J/(kg*K)]` | J/(kg·K) | Basic |
| Young's modulus | E | `160[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.31` | 1 | Young's modulus and Poisson's ratio |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | `2.6e-6` | 1/K | Basic |

## ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Optical** > **Miscellaneous** > **Semiconductor alloys** > **GaAs-InAs (Gallium indium arsenide, GaInAs) (Adachi 1989: n,k 0.207-12.4 um)**.

3 Click the **Add to Component** button in the window toolbar.

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

## MATERIALS

*GaAs-InAs (Gallium indium arsenide, GaInAs) (Adachi 1989: n,k 0.207-12.4 um) (mat4)*

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

2 From the **Selection** list, choose **QW**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | `46[W/(m*K)]` | W/(m·K) | Basic |
| Density | rho | `5500[kg/m^3]` | $kg/m^3$ | Basic |
| Heat capacity at constant pressure | Cp | `330[J/(kg*K)]` | J/(kg·K) | Basic |
| Young's modulus | E | `85.5[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.31` | 1 | Young's modulus and Poisson's ratio |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | `5.73e-6` | 1/K | Basic |
| Refractive index, imaginary part | ki_iso ; kiii = ki_iso, kiij = 0 | `ni(c_const/freq)*del_q` | 1 | Refractive index |

## COMPONENT 1 (COMP1)

Add a deforming mesh to the air domain.

### *Deforming Domain 1*

**1** In the **Physics** toolbar, click **Moving Mesh** and choose **Free Deformation**.

**2** In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.

**3** Click **Clear Selection**.

**4** Select Domain 1 only.

## STUDY 1

### *Step 1: Frequency–Stationary*

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

**2** In the **Settings** window for **Frequency–Stationary**, locate the **Study Settings** section.

**3** In the **Frequency** text field, type `c_const/lda0`.

### *Parametric Sweep*

1 In 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 |
|---|---|---|
| P_ave (Average power of laser) | 1 20 40 60 100 150 | W |

A Frequency domain study step is used to obtain initial values for the Electromagnetic Waves physics.

### *Step 2: Frequency Domain*

1 In the **Study** toolbar, click **Frequency Domain**.

2 Drag and drop below **Parametric Sweep**.

3 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

4 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Moving Mesh**.

5 Locate the **Study Settings** section. In the **Frequencies** text field, type `c_const/lda0`.

The nonlinear nature of this model requires some adjustments to the default solver settings. Namely, using a segregated solver to alternately solve for the electromagnetic problem in one step and then the structural and thermal problems in the second step.

### *Solution 1 (sol1)*

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** node.

4 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** and choose **Segregated**.

5 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **Segregated 1** node, then click **Segregated Step**.

6 In the **Settings** window for **Segregated Step**, locate the **General** section.

7 In the **Variables** list, choose **Spatial Mesh Displacement (comp1.spatial.disp)**, **Temperature (comp1.T)**, and **Displacement Field (comp1.u)**.

8 Under **Variables**, click **Delete**.

9 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** right-click **Segregated 1** and choose **Segregated Step**.

10 In the **Settings** window for **Segregated Step**, locate the **General** section.

11 Under **Variables**, click **Add**.

12 In the **Add** dialog, in the **Variables** list, choose **Spatial Mesh Displacement (comp1.spatial.disp)**, **Temperature (comp1.T)**, and **Displacement Field (comp1.u)**.

13 Click **OK**.

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

## RESULTS

Add two cut lines for evaluating behavior at different points in the model.

### *Cut Line 2D 1*

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **Cut Line 2D**.

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

4 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

5 Locate the **Line Data** section. In row **Point 2**, set **x** to `thickness`.

6 Click **Plot**.

### *Study 1/Parametric Solutions 1 (4) (sol3)*

1 In the **Model Builder** window, under **Results** > **Datasets** right-click **Study 1/ Parametric Solutions 1 (sol3)** and choose **Duplicate**.

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

3 From the **Frame** list, choose **Material (X, Y, Z)**.

### *Cut Line 2D 2*

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (4) (sol3)**.

4 Locate the **Line Data** section. In row **Point 1**, set **X** to `thickness_GaAs-thickness_QW/2`.

5 In row **Point 1**, set **Y** to `-height/2`.

6 In row **Point 2**, set **Y** to `height/2`.

7 In row **Point 2**, set **X** to `thickness_GaAs-thickness_QW/2`.

8 Click **Plot**.

*Global Evaluation 1*

The reflectivity can be evaluated by using the outcoupling efficiency on the Scattering Boundary Condition with the input wave.

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol3)**.

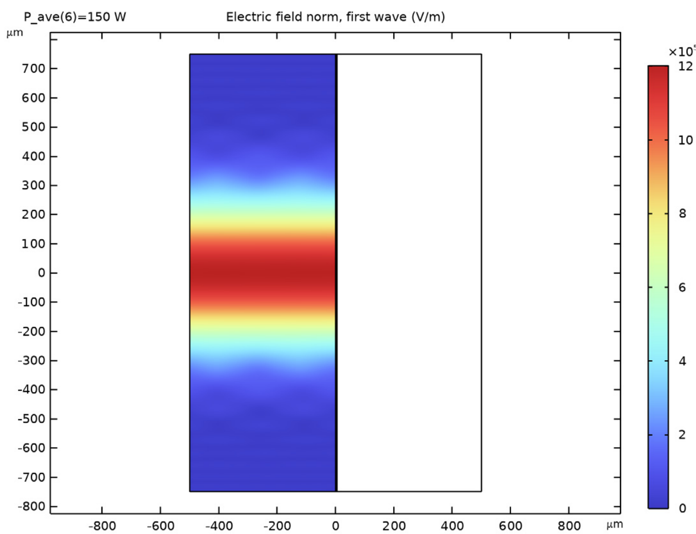
4 Locate the **Expressions** section. In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `ewbe.sctr1.etaOut` | 1 | `Outcoupling efficiency` |

5 Click **Evaluate**.

*Electric Field, First Wave (ewbe)*

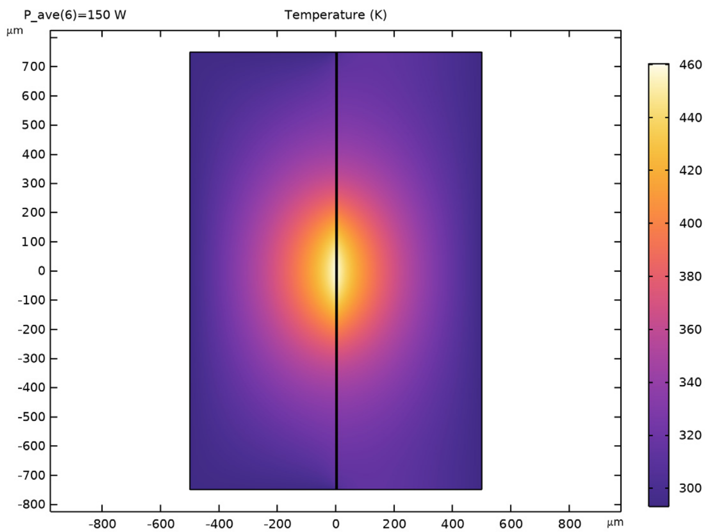
1 In the **Model Builder** window, under **Results** click **Electric Field, First Wave (ewbe)**.




*The electric field is evaluated only in the air and SESAM domains.*

### *Temperature (ht)*

1 In the **Model Builder** window, click **Temperature (ht)**.




*The temperature increase is centered at the overlap between the laser beam and the SESAM, and extends into both the substrate and surrounding air.*

### *Displacement*

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Displacement` in the **Label** text field.

### *Surface 1*

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

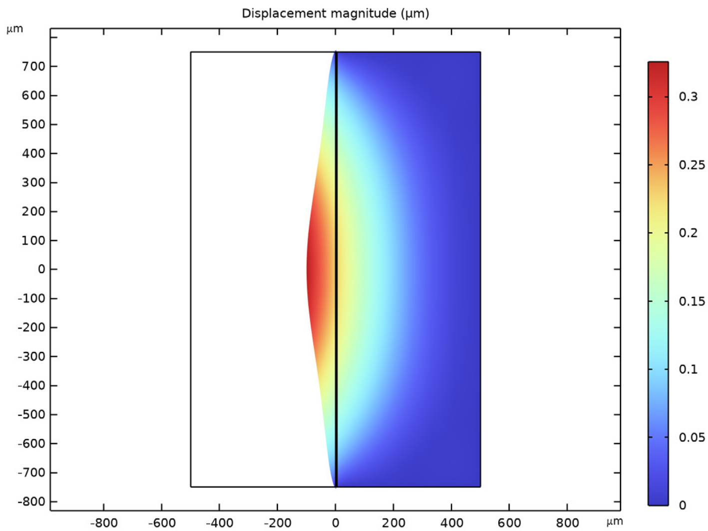
2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `solid.disp`.

4 In the **Displacement** toolbar, click **Plot**.

### Deformation 1

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




*The resulting displacement is maximum at the highest temperature.*

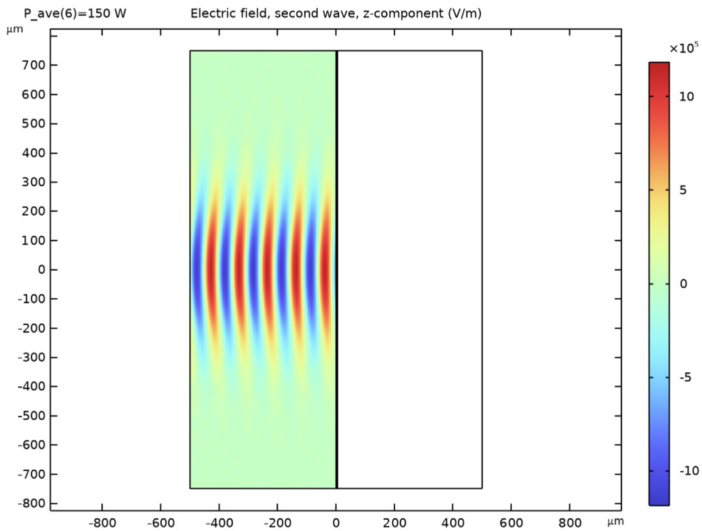
### Electric Field

**1** In the **Model Builder** window, expand the **Results** > **Electric Field, Second Wave (ewbe)** node, then click **Electric Field**.

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

**3** In the **Expression** text field, type `ewbe.E2z`.

**4** In the **Electric Field, Second Wave (ewbe)** toolbar, click **Plot**.




*The E2z component shows the change in wavefront curvature of the reflected beam.*

Next, the Cut Line data sets can be used to see more detailed behavior of the E-field within the SESAM.

### E-Field in SESAM

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

**2** In the **Settings** window for **1D Plot Group**, type `E-Field in SESAM` in the **Label** text field.

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

**4** Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

### Line Graph 1

**1** In the **E-Field in SESAM** toolbar, click **Line Graph**.

**2** In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

**3** Select the **Show legends** checkbox.

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

**5** In the **Expression** text field, type `x`.

### E-Field in SESAM

In the **E-Field in SESAM** toolbar, click **Line Graph**.

*Line Graph 2*

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

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

**3** From the **Parameter selection (P_ave)** list, choose **First**.

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

**5** Locate the **Legends** section. Select the **Show legends** checkbox.

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

**7** In the **Expression** text field, type `x`.

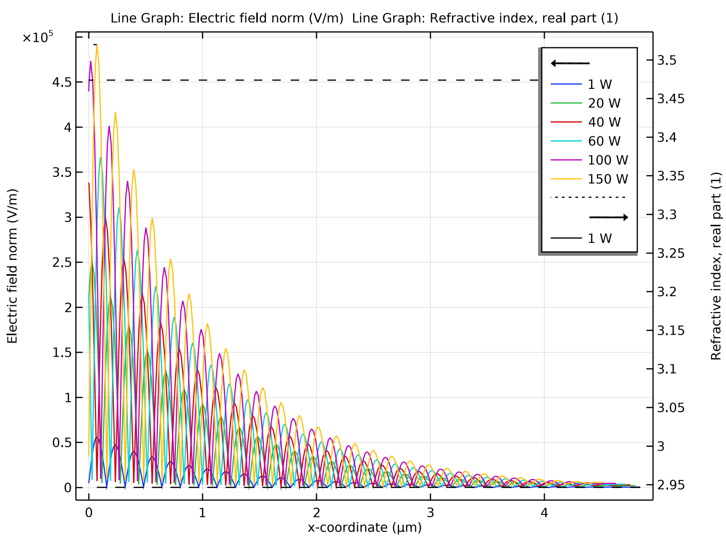
*E-Field in SESAM*

**1** In the **Model Builder** window, click **E-Field in SESAM**.

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

**3** In the table, select the **Plot on secondary y-axis** checkbox for **Line Graph 2**.

**4** In the **E-Field in SESAM** toolbar, click **Plot**.




*The E-field of the antiresonant SESAM showing negligible amplitude at the surface but peak amplitude in the quantum wells. The refractive index plot highlights the layers of the quantum wells and the layers of the Bragg reflector. The field clearly shows a reduction in amplitude with each layer until it is minimal. You can experiment with different numbers of layers to achieve different reflectivities.*

*Absorption in QW, y Direction*

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

**2** In the **Settings** window for **1D Plot Group**, type `Absorption in QW, y Direction` in the **Label** text field.

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

### *Line Graph 1*

**1** In the **Absorption in QW, y Direction** toolbar, click **Line Graph**.

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

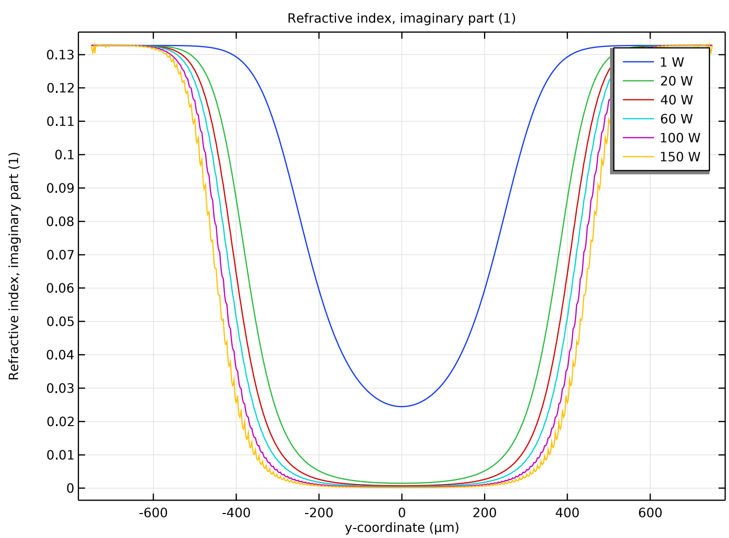
**3** In the **Expression** text field, type `ewbe.ki_iso`.

**4** Locate the **Legends** section. Select the **Show legends** checkbox.

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

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

**7** In the **Absorption in QW, y Direction** toolbar, click **Plot**.




*The absorption in the quantum well is saturated at the point of highest intensity on the incident laser beam.*

### *Temperature with Laser Power*

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

**2** In the **Settings** window for **1D Plot Group**, type `Temperature with Laser Power` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (3) (sol3)**.

### Global 1

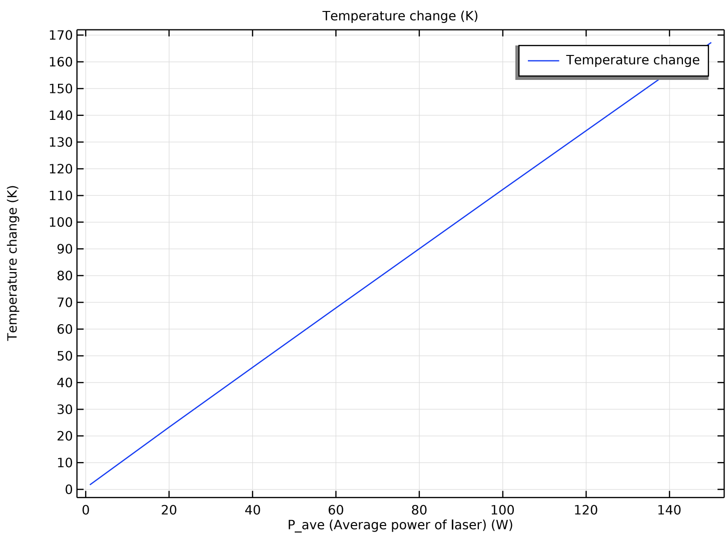
1 In the **Temperature with Laser Power** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

| Expression | Unit | Description |
| --- | --- | --- |
| `maxop1(T)-maxop1(ht.Tinit)` | `K` | `Temperature change` |

4 In the **Temperature with Laser Power** toolbar, click **Plot**.




*The peak temperature compared to the ambient initial temperature.*

### Reflectivity with Laser Power

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol3)**.

4 In the **Label** text field, type `Reflectivity with Laser Power`.

### Global 1

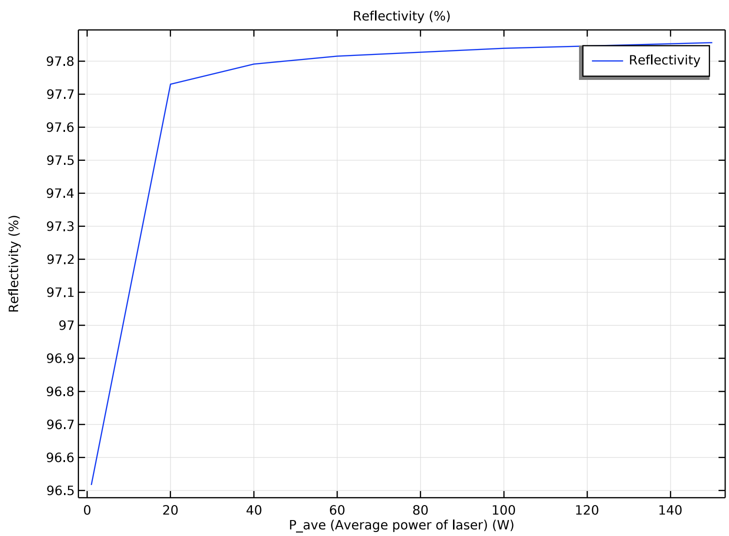
1 In the **Reflectivity with Laser Power** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

**3** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `intop2(abs(ewbe.nPoav2))/intop2(abs(ewbe.nPoav1))` | `%` | `Reflectivity` |

**4** In the **Reflectivity with Laser Power** toolbar, click **Plot**.




*The reflectivity increases with incident power.*

### SESAM Curvature

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

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

**3** From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (3) (sol3)**.

**4** In the **Label** text field, type `SESAM Curvature`.

**5** Locate the **Plot Settings** section. Select the **Two y-axes** checkbox.

### Surface Curvature

**1** In the **SESAM Curvature** toolbar, click **Line Graph**.

**2** In the **Settings** window for **Line Graph**, type `Surface Curvature` in the **Label** text field.

**3** Locate the **Selection** section. From the **Selection** list, choose **SESAM Surface**.

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

**5** Locate the **Legends** section. Select the **Show legends** checkbox.

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

7 In the **Expression** text field, type `y`.

*SESAM Curvature*

In the **SESAM Curvature** toolbar, click **Line Graph**.

*E-Field: z-Component*

1 In the **Settings** window for **Line Graph**, type `E-Field: z-Component` in the **Label** text field.

2 Locate the **Selection** section. From the **Selection** list, choose **SESAM Surface**.

3 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(E2z)`.

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

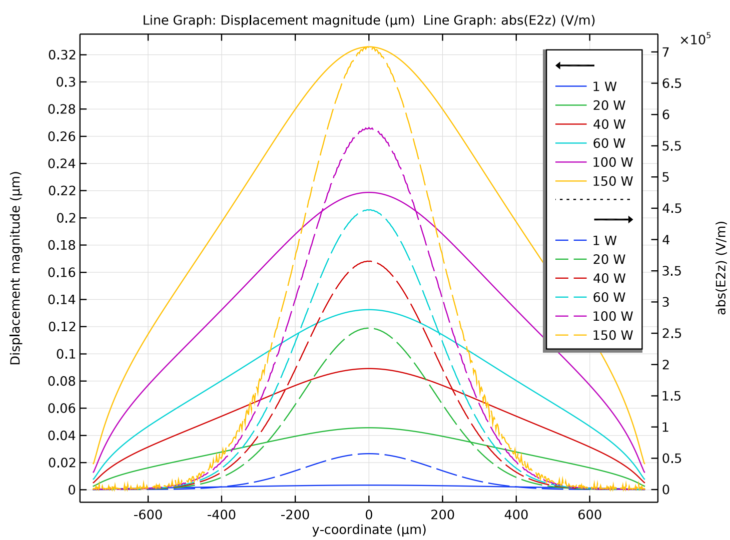
6 In the **Expression** text field, type `y`.

7 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

9 Locate the **Legends** section. Select the **Show legends** checkbox.

10 In the **SESAM Curvature** toolbar, click **Plot**.



### ADD PHYSICS

Next, add a Global ODEs and DAEs interface and a separate stationary study to fit a parabolic curve to the curvature of the SESAM surface.

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Mathematics** > **ODE and DAE Interfaces** > **Global ODEs and DAEs (ge)**.

4 Click the **Add to Component 1** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

### GLOBAL ODES AND DAES (GE)

*Global Equations 1 (ODE1)*

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

2 In the table, enter the following settings:

| Name | f(u,ut,utt,t) (1) | Initial value (u_0) (1) | Initial value (ut_0) (1/s) | Description |
|---|---|---|---|---|
| `Radius` | `d(Para_fit, Radius)` | `1 [m]` | `0` | |
| `Peak` | `d(Para_fit, Peak)` | `1 [um]` | `0` | |

3 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

4 In the **Physical Quantity** dialog, type `leng` in the text field.

5 In the tree, select **General** > **Length (m)**.

6 Click **OK**.

7 In the **Settings** window for **Global Equations**, locate the **Units** section.

8 Click **Define Source Term Unit**.

9 In the **Source term quantity** table, enter the following settings:

| Source term quantity | Unit |
|---|---|
| Custom unit | `m^2` |

### STUDY 1

*Step 1: Frequency Domain*

Disable the ODE interface in the first study so it does not affect future runs of this study.

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

2 In the **Settings** window for **Frequency Domain**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Global ODEs and DAEs (ge)**.

*Step 2: Frequency–Stationary*

1 In the **Model Builder** window, click **Step 2: Frequency–Stationary**.

2 In the **Settings** window for **Frequency–Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Global ODEs and DAEs (ge)**.

## ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces > Stationary**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

## STUDY 2

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

2 Clear the **Generate default plots** checkbox.

*Parametric Sweep*

1 In 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 |
|---|---|---|
| P_ind (Power index used for fitting study) | 1 20 40 60 100 150 | W |

*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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Heat Transfer in Solids (ht)**.

**4** In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkboxes for **Electromagnetic Heating 1 (emh1)** and **Thermal Expansion 1 (te1)**.

**5** In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Moving Mesh**.

The purpose of this study is only to evaluate the Global ODEs and DAEs interface.

*Solution 10 (sol10)*

In the **Study** toolbar, click **Show Default Solver**.

*Solution 10 (sol10)*

**1** In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 10 (sol10)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

**2** In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

**3** From the **Nonlinear method** list, choose **Constant (Newton)**.

The Constant (Newton) method is more suited to this study.

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

## RESULTS

*SESAM Curvature*

The fitted curve can be added to the previous plot for comparison.

**1** In the **Model Builder** window, under **Results** click **SESAM Curvature**.

*Fitted curve: z-Component*

**1** In the **SESAM Curvature** toolbar, click **Line Graph**.

**2** In the **Settings** window for **Line Graph**, type `Fitted curve: z-Component` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 10 (sol10)**.

**4** Locate the **Selection** section. From the **Selection** list, choose **SESAM Surface**.

**5** Locate the **y-Axis Data** section. In the **Expression** text field, type `delx_para`.

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

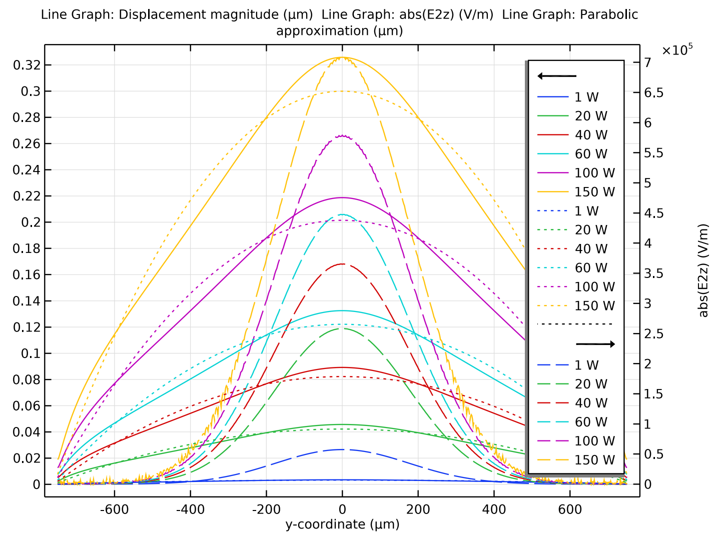
**7** In the **Expression** text field, type `y`.

**8** Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

**9** Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

**10** Locate the **Legends** section. Select the **Show legends** checkbox.

**11** In the **SESAM Curvature** toolbar, click **Plot**.




*The dotted lines show the parabolic approximation of the thermal expansion.*

### Curvature with Laser Power

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

**2** In the **Settings** window for **1D Plot Group**, type `Curvature with Laser Power` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 10 (sol10)**.

### Global 1

**1** In the **Curvature with Laser Power** toolbar, click **Global**.

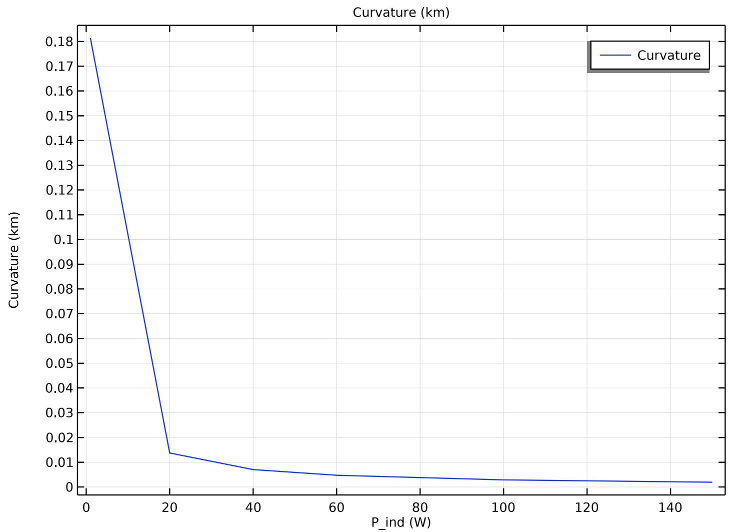
**2** In the **Settings** window for **Global**, locate the **y-Axis Data** section.

**3** Click **Clear Table**.

**4** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| Radius | km | Curvature |

**5** In the **Curvature with Laser Power** toolbar, click **Plot**.




*The radius of curvature is seen to decrease with the incident laser power.*