

Laser Heating of a Silicon Wafer

Introduction

A silicon wafer is heated up by a laser that moves radially in and out over time while the wafer itself rotates on its stage. Modeling the incident heat flux from the laser as a spatially distributed heat source on the surface, the transient thermal response of the wafer is obtained. The peak, average, and minimum temperatures during the heating process are computed, as well as the temperature variations across the wafer.

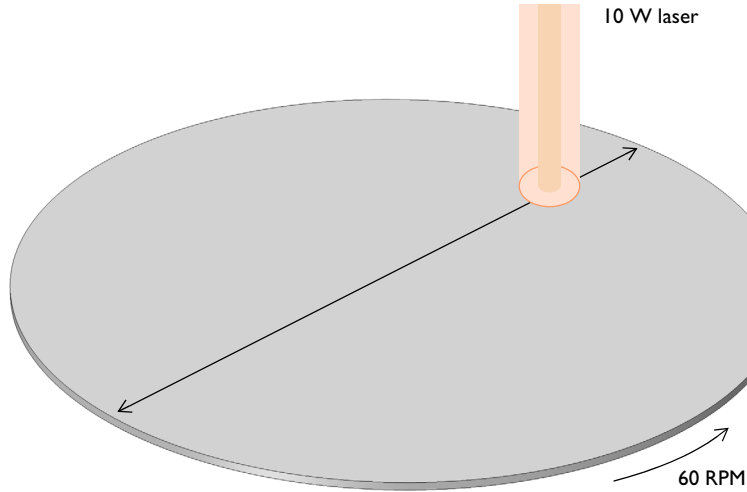


Figure 1: A silicon wafer is heated with a laser that moves back and forth. The wafer is also being rotated about its axis.

Model Definition

A 2-inch silicon wafer, as shown in [Figure 1](#), is heated for one minute by a 10 W laser that moves radially inwards and outwards, while the wafer rotates on its stage. Assuming good thermal isolation from the environment, the only source of heat loss is from the top surface via radiation to the processing chamber walls, which are assumed to be at a fixed temperature of 20 °C.

The laser beam is modeled as a heat source in the plane with Gaussian profile. To set up the heating profile the model uses the built-in Gaussian Pulse functions, which enforce that the integral under the curve equals unity. The focal point is moved by using a

triangular waveform to define its position along the x -axis over time. The wafer is assigned a bulk rotational velocity in the governing heat transfer equation.

The emissivity of the surface of the wafer is approximately 0.8. At the operating wavelength of the laser, it is assumed that absorptivity equals emissivity. The heat load due to the laser is thus multiplied by the emissivity. Assuming also that the laser is operating at a wavelength at which the wafer is opaque, no light is passing through the wafer. Therefore, all of the laser heat is deposited at the surface.

The wafer is meshed using a triangular swept mesh. Swept meshing allows for only a single thin element through the thickness, and still maintains reasonable element size in the plane. A finer mesh would give slightly more accurate predictions of the peak temperature, but predictions of average and minimum temperature would not be greatly affected.

Results and Discussion

[Figure 2](#) plots the maximum, minimum, and average temperatures of the wafer, while [Figure 3](#) shows the difference between the maximum and minimum temperature. The temperature distribution across the wafer is plotted in [Figure 4](#).

The heating profile does introduce some significant temperature variations, because the laser deposits the same amount of heat over a larger total swept area when it is focused at the outside of the wafer. An interesting modification to this example would be to investigate alternative heating profiles for smoother heating.

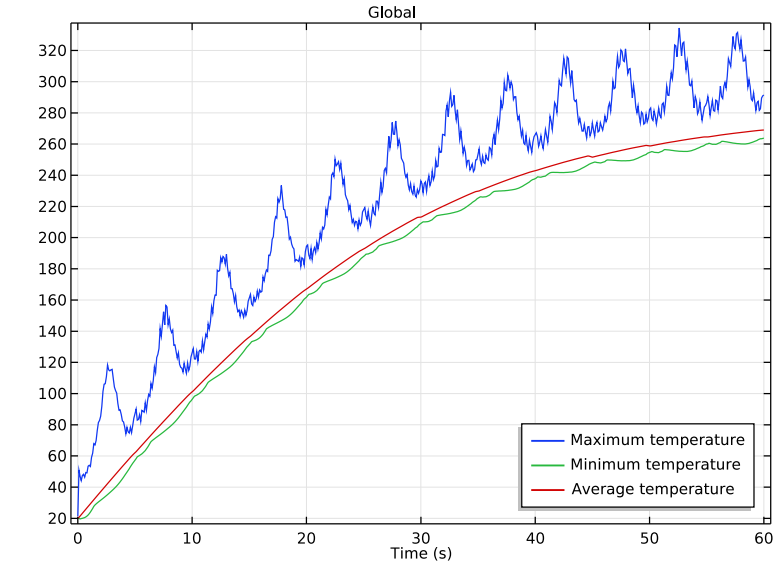


Figure 2: Maximum, minimum, and average temperatures of the wafer as functions of time.

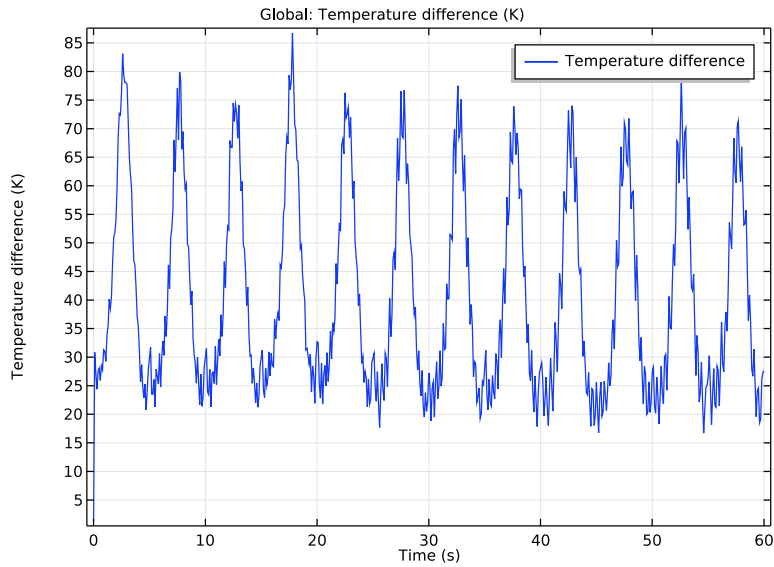


Figure 3: Difference between maximum and minimum temperatures on the wafer.

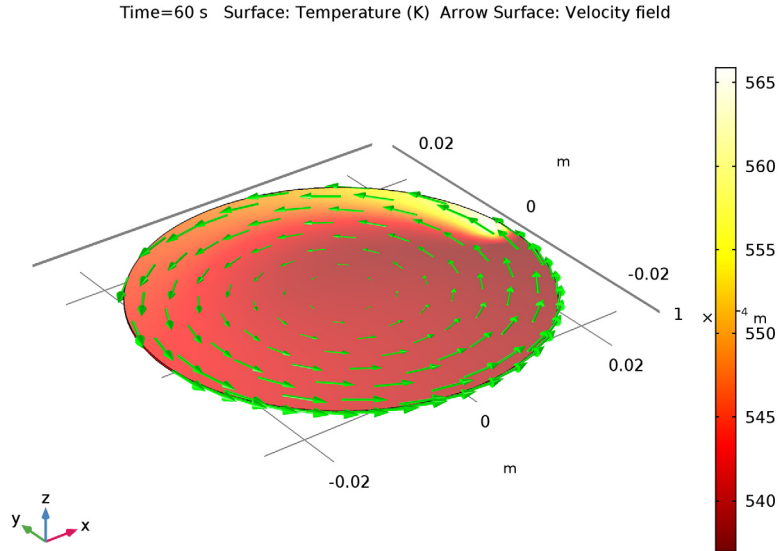


Figure 4: Temperature variation across the wafer. The arrow plot describes the velocity of the wafer.

Application Library path: COMSOL_Multiphysics/Heat_Transfer/
laser_heating_wafer

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 **3D**.
- 2** In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.
- 3** Click **Add**.

- 4 Click **Study**.
- 5 In the **Select Study** tree, select **Preset Studies>Time Dependent**.
- 6 Click **Done**.

GLOBAL DEFINITIONS

Start by defining parameters for use in the geometry, functions, and physics settings.

Parameters

- 1 On the **Home** toolbar, click **Parameters**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
r_wafer	1[in]	0.0254 m	Wafer radius
thickness	275[um]	2.75E-4 m	Wafer thickness
v_rotation	60[rpm]	1 l/s	Rotational speed
angular_v	2*pi*v_rotation	6.283 l/s	Angular velocity
period	10[s]	10 s	Time for laser to move back and forth
r_spot	2[mm]	0.002 m	Laser beam radius
emissivity	0.8	0.8	Surface emissivity of wafer
p_laser	10[W]	10 W	Laser power

Here, the unit 'rpm' is revolution per minute.

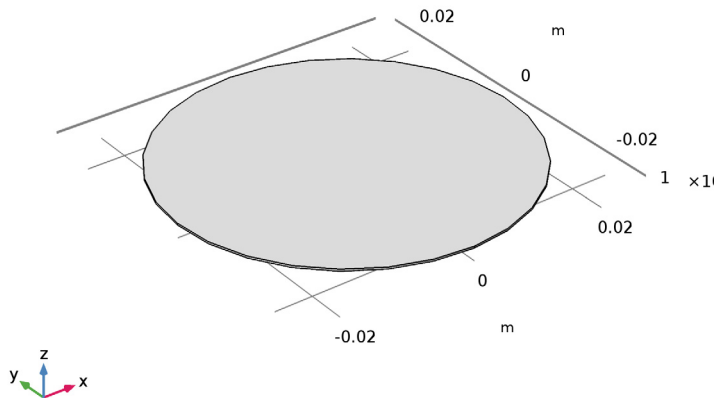
GEOMETRY I

Create a cylinder for the silicon wafer.

Cylinder 1 (cyl1)

- 1 On the **Geometry** toolbar, click **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type r_wafer.
- 4 In the **Height** text field, type thickness.

5 Click **Build All Objects**.



DEFINITIONS

Define functions for use before setting up the physics.

Variables 1

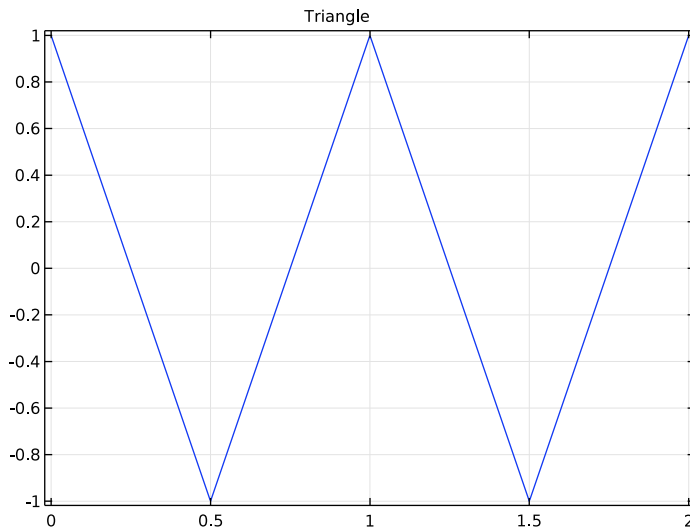
- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:

Name	Expression	Unit	Description
x_focus	$r_wafer * \text{Triangle}(t / \text{period})$	m	x-location of laser focal point
y_focus	0[m]	m	y_location of laser focal point
r_focus	$\sqrt{(x - x_focus)^2 + (y - y_focus)^2}$	m	distance from focal point
Flux	$((2 * p_laser) / (\pi * r_spot^2)) * \exp(-(2 * r_focus^2) / r_spot^2)$	W/m ²	laser heat flux, Gaussian profile

Waveform 1 (wv1)

- 1 On the **Home** toolbar, click **Functions** and choose **Local>Waveform**.

- 2 In the **Settings** window for **Waveform**, type Triangle in the **Function name** text field.
- 3 Locate the **Parameters** section. From the **Type** list, choose **Triangle**.
- 4 Clear the **Smoothing** check box.
- 5 In the **Angular frequency** text field, type 2π .
- 6 In the **Phase** text field, type $\pi/2$.
- 7 In the **Amplitude** text field, type 1.
- 8 Click **Plot**.



Average I (aveopI)

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Maximum I (maxopI)

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

Minimum I (minopI)

- 1 On the **Definitions** toolbar, click **Component Couplings** and choose **Minimum**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all domains.

HEAT TRANSFER IN SOLIDS (HT)

Set up the physics. First, include the wafer's rotational velocity in the governing heat transfer equation.

Solid 1

In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Solids (ht)** click **Solid 1**.

Translational Motion 1

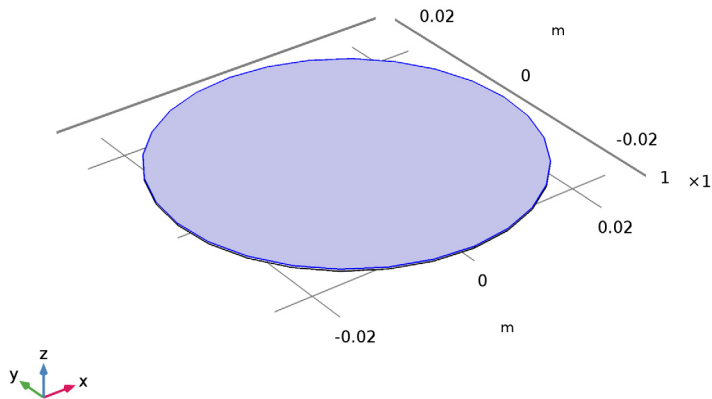
- 1 On the **Physics** toolbar, click **Attributes** and choose **Translational Motion**.
- 2 In the **Settings** window for **Translational Motion**, locate the **Translational Motion** section.
- 3 Specify the $\mathbf{u}_{\text{trans}}$ vector as

$-y \cdot \text{angular_v}$	x
$x \cdot \text{angular_v}$	y
0	z

Next, add heat flux and surface-to-ambient radiation on the wafer's top surface.

Heat Flux 1

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
- 2 Select Boundary 4 only.



- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

- 4 In the q_0 text field, type `emissivity*Flux`.

Diffuse Surface I

- 1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.
- 2 Select Boundary 4 only.
- 3 In the **Settings** window for **Diffuse Surface**, locate the **Surface Emissivity** section.
- 4 From the ϵ list, choose **User defined**. In the associated text field, type `emissivity`.

ADD MATERIAL

- 1 On the **Home** toolbar, click **Add Material** to open the **Add Material** window.
- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-In>Silicon**.
- 4 Click **Add to Component** in the window toolbar.
- 5 On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH I

Use a fine triangular swept mesh.

Swept I

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Swept**.
- 2 In the **Settings** window for **Swept**, click to expand the **Sweep method** section.
- 3 Locate the **Sweep Method** section. From the **Face meshing method** list, choose **Triangular (generate prisms)**.

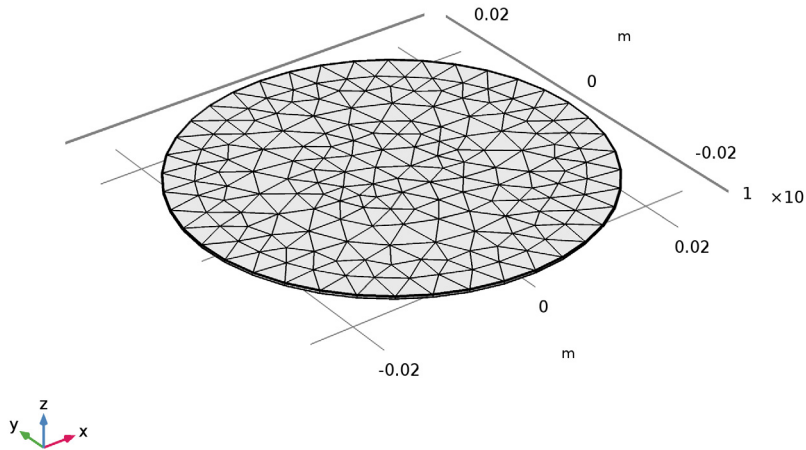
Distribution I

- 1 Right-click **Component 1 (comp1)>Mesh 1>Swept 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 1.

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 **Fine**.

4 Click **Build All**.



STUDY 1

Step 1: Time Dependent

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 In the **Times** text field, type range (0,0.1,60).
- 3 On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht)

The first default plot shows the temperature on the wafer surface. Add an arrow plot for the velocity field describing the wafer rotation.

Arrow Surface 1

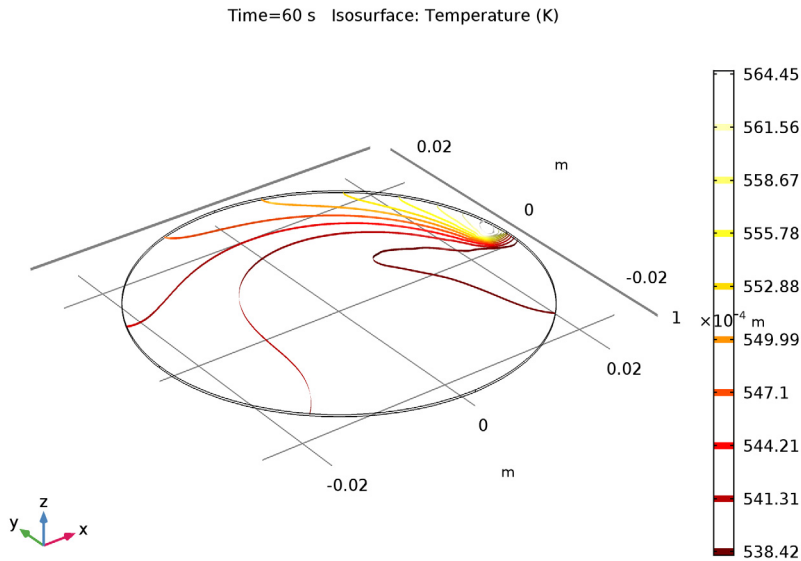
- 1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Surface**.
- 2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Solids>ht.ux,ht.uy,ht.uz - Velocity field**.

- 3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.
- 4 In the associated text field, type 0.036.
- 5 From the **Color** list, choose **Green**.
- 6 On the **Temperature (ht)** toolbar, click **Plot**.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Compare the temperature variation with that shown in [Figure 4](#).

Isothermal Contours (ht)

The second default plot shows the isosurface temperature.



ID Plot Group 3

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, click to expand the **Legend** section.
- 3 From the **Position** list, choose **Lower right**.

Global 1

- 1 Right-click **ID Plot Group 3** and choose **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)	degC	Maximum temperature
minop1 (T)	degC	Minimum temperature
aveop1 (T)	degC	Average temperature

4 On the **ID Plot Group 3** toolbar, click **Plot**.

Compare the plot showing the maximum, minimum and average temperature with [Figure 2](#).

Global 1

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

2 In the **Model Builder** window, right-click **ID Plot Group 4** and choose **Global**.

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

4 In the table, enter the following settings:

Expression	Unit	Description
maxop1 (T) - minop1 (T)	K	Temperature difference

5 On the **ID Plot Group 4** toolbar, click **Plot**.

Finish the result analysis by comparing the generated plot with [Figure 3](#).

