Created in COMSOL Multiphysics 6.2

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 average, maximum, and minimum temperatures, as well as the peak temperature difference across the wafer, are stored at every calculation step. The temperature distribution across the entire wafer is stored at a specified number of output time steps.

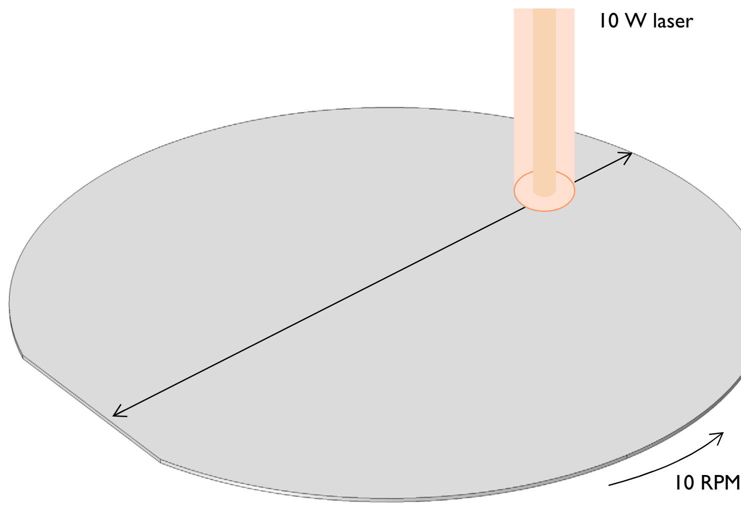


Figure 1: A silicon wafer is heated with a laser that moves back and forth. The wafer is also being rotated about its axis.

Model Definition

A 2-inch silicon wafer, as shown in Figure 1, is heated for one minute by a 10 W laser that moves back and forth, while the wafer rotates on its stage. Assuming good thermal insulation 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 heat source is modeled as a heat source moving across the surface of the spinning wafer. To model the rotation of the wafer, use the **Moving Mesh>Rotating Domain** feature. Use a Waveform function and a set of variables to define the Gaussian distribution of the laser heat load around the focal point, as it moves back and forth across the spinning structure.

In the results visualization of the temperature profile across the wafer, the results can be visualized in either the spatial frame or the material frame, representing the point of view of an outside observer or an observer moving with the rotation of the wafer, respectively.

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. Also, the solver relative tolerance is slightly lowered to better capture the effect of the moving heat load. A finer mesh and tighter solver tolerances 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 shows the probe plots of the maximum, minimum, and average temperatures of the wafer, while Figure 3 shows the probe plot of 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.

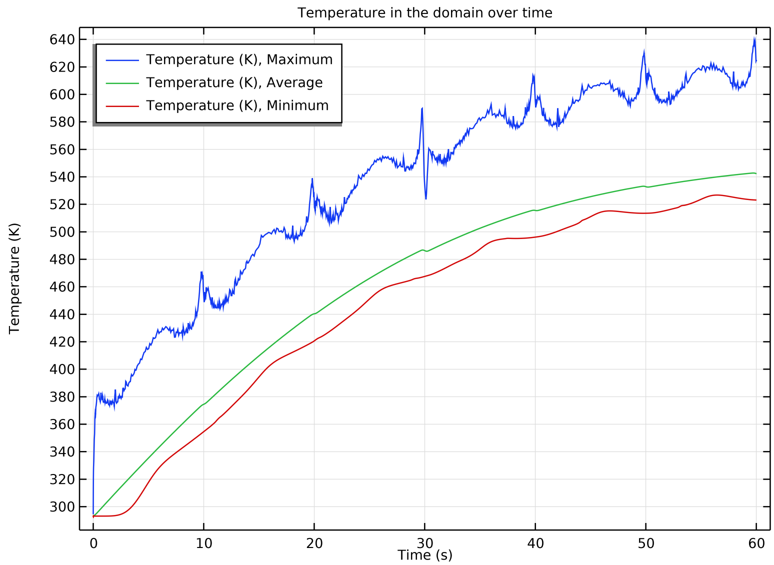


Figure 2: Maximum, minimum, and average temperatures of the wafer as functions of time.

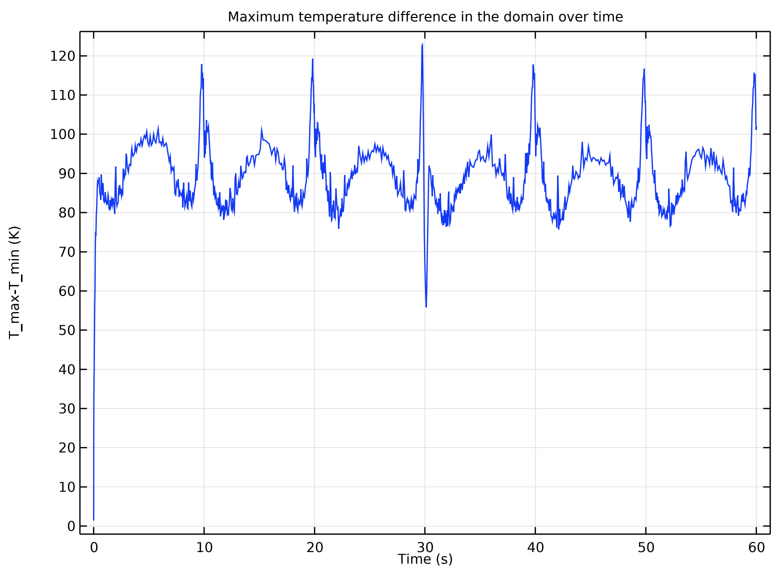


Figure 3: Difference between maximum and minimum temperatures on the wafer.

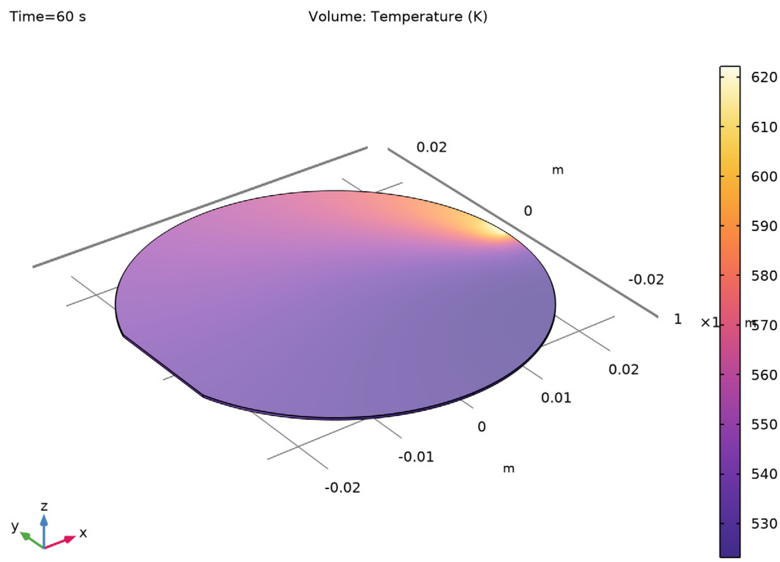


Figure 4: Temperature variation across the wafer.

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 **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

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

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 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	10[rpm]	0.16667 1/s	Rotational speed
period	20[s]	20 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 1

Create a cylinder for the silicon wafer.

Cylinder 1 (cyl1)

1 In 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`.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `2*r_wafer`.

4 In the **Depth** text field, type `2*r_wafer`.

5 In the **Height** text field, type `thickness`.

6 Locate the **Position** section. In the **x** text field, type `-0.95*r_wafer`.

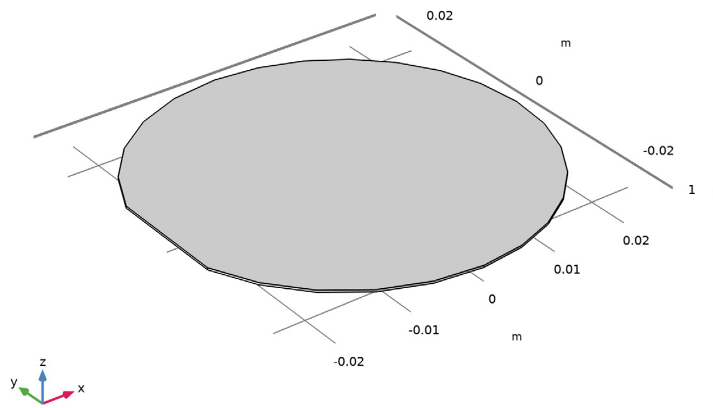
7 In the **y** text field, type `-r_wafer`.

Intersection 1 (int1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Intersection**, click **Build All Objects**.



DEFINITIONS

Define the functions to be used 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*Triangle(t/period)		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)		distance from focal point
Flux	((2*p_laser)/(pi*r_spot^2))*exp(-(2*r_focus^2)/r_spot^2)		laser heat flux, Gaussian profile

Waveform 1 (wv1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Waveform**.

2 In the **Settings** window for **Waveform**, type `Triangle` in the **Function name** text field.

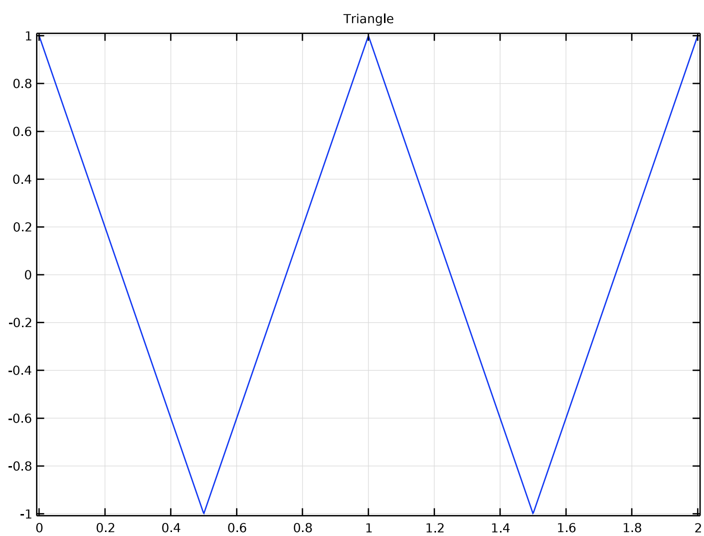
3 Locate the **Parameters** section. From the **Type** list, choose **Triangle**.

4 Clear the **Smoothing** check box.

5 In the **Period** text field, type `1`.

6 In the **Phase** text field, type `pi/2`.

7 Click **Plot**.



Maximum

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2 In the **Settings** window for **Domain Probe**, type `Maximum` in the **Label** text field.
3 In the **Variable name** text field, type `T_max`.
4 Locate the **Probe Type** section. From the **Type** list, choose **Maximum**.

Average

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2 In the **Settings** window for **Domain Probe**, type `Average` in the **Label** text field.
3 In the **Variable name** text field, type `T_average`.

Minimum

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2 In the **Settings** window for **Domain Probe**, type `Minimum` in the **Label** text field.
3 In the **Variable name** text field, type `T_min`.
4 Locate the **Probe Type** section. From the **Type** list, choose **Minimum**.

Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.
2 In the **Settings** window for **Global Variable Probe**, type `T_diff` in the **Variable name** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `T_max-T_min`.

COMPONENT 1 (COMP1)

Rotating Domain 1

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Rotating Domain**.
2 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.
3 From the **Rotation type** list, choose **Specified rotational velocity**.
4 From the **Rotational velocity expression** list, choose **Constant revolutions per time**.
5 In the *f* text field, type `v_rotation`.

HEAT TRANSFER IN SOLIDS (HT)

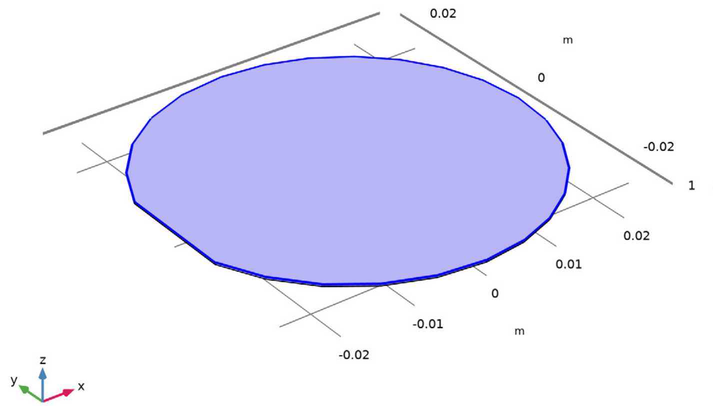
Set up the physics. First, include the wafer's rotational velocity in the governing heat transfer equation.

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

Heat Flux 1

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

Surface-to-Ambient Radiation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Surface-to-Ambient Radiation**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Surface-to-Ambient Radiation**, locate the **Surface-to-Ambient Radiation** section.

4 From the ε list, choose **User defined**. In the associated text field, type `emissivity`.

ADD MATERIAL

1 In 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 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

Use a fine triangular swept mesh.

Swept 1

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

2 In the **Settings** window for **Swept**, click to expand the **Sweep Method** section.

3 From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Distribution 1

1 Right-click **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.

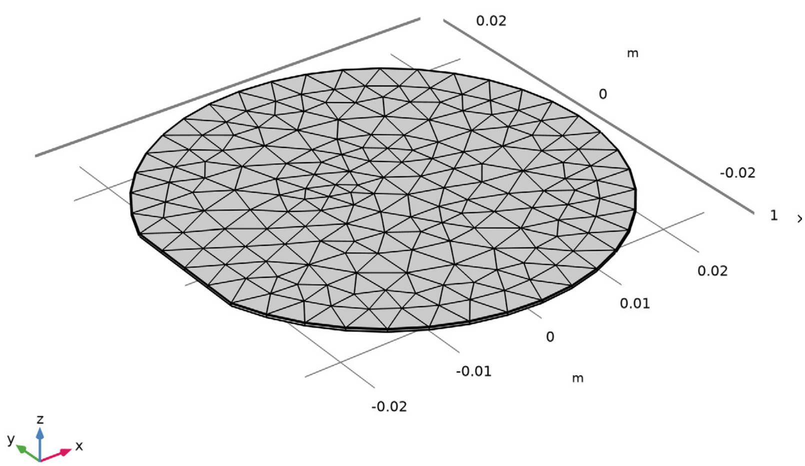
Size

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 **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,1,60)`.

Tighten the relative tolerance to better capture the effect of the moving heat load.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-3`.

6 In the **Home** toolbar, click **Compute**.

RESULTS

Study 1/Solution 1 (sol1)

Change the frame to **Spatial** in order to visualize the wafer displacement.

Probe Temperature in the Domain

The first default plot shows the change of temperature in time.

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Results>Probe Plot Group 1**.

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Temperature in the domain over time`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Temperature (K)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Temperature in the Domain** node, then click **Probe Table Graph 1**.

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

3 In the **Columns** list, choose **Temperature (K), Maximum**, **Temperature (K), Average**, and **Temperature (K), Minimum**.

4 In the **Probe Temperature in the Domain** toolbar, click **Plot**.

Temperature (ht)

The second default plot shows the temperature distribution in the computational domain.

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

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Spatial (x, y, z)**.

4 In the **Temperature (ht)** toolbar, click **Plot**.

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

Compare the temperature variation with that shown in Figure 4.

Add a predefined plot showing the temperature isosurfaces.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

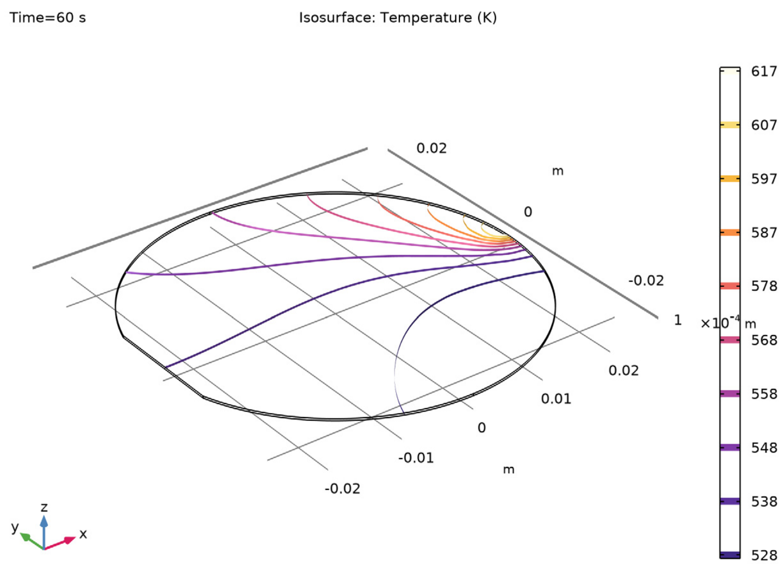
2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids>Isothermal Contours (ht)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Isothermal Contours (ht)



Maximum Temperature Difference in the Domain

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Maximum Temperature Difference in the Domain` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Manual**.
4. In the **Title** text area, type `Maximum temperature difference in the domain over time`.

Probe Table Graph 1

1. Right-click **Maximum Temperature Difference in the Domain** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, type `Probe Table Graph 1` in the **Label** text field.
3. Locate the **Data** section. From the **Plot columns** list, choose **Manual**.
4. In the **Columns** list, select **T_max-T_min (K)**.
5. In the **Maximum Temperature Difference in the Domain** toolbar, click **Plot**.