Exercise 1: Intro to COMSOL (revised)

Transport in Biological Systems

Fall 2015

Overview

In this course, we will consider transport phenomena in biological systems. Generally speaking, biological systems of interest have interesting geometries and we are interested in things happening in time and space. This means that analytical solutions can be challenging to find. Therefore, a modeling and simulation approach is often useful. Towards this, we will employ COMSOL in this course. It takes a little bit to get up to speed but it's a very powerful tool.

You will work on this in class (AC318) on September 3rd, the first day of class. Alisha will not be present, but you should all work as a group on this. Please try to complete the first 3 training items before class.

Training

1. *Install* COMSOL on your computer if you haven't (http://wikis.olin.edu/it/doku.php?id=comsol).

2. *Read:* http://cdn.comsol.com/documentation/5.1.0.180/IntroductionToCOMSOLMultiphysics.pdf

3. *Watch:* https://www.comsol.com/blogs/how-to-use-comsol-multiphysics-a-tutorial-video/

4. *Attend* the COMSOL training from 9:30-12:30 on September 17th. (Note: I know a few of you have class before this one. Please come as soon as you can. If it is acceptable for your other class to leave early on that day, it would be great, but I also understand if you can't.)

Exercises

1. Make a smiley face (Figure 1, which I cannot get LaTex to put here and seems to be at the end!). Hint: think about using ellipses and intersections, not lines for the smile.

2. Make a hollow sphere with holes in it (Figure 2). Don't worry about dimensions, just the general concept.
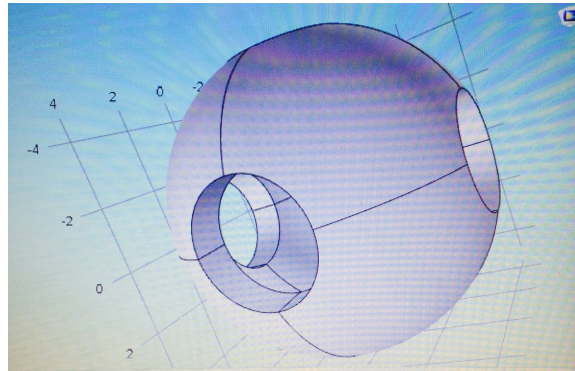

Figure 2: **A hollow sphere with holes.**

3. Do the 2 exercises attached at the end of this document. (credit: COMSOL)

Objectives and Deliverables

The objective of this course is for you to apply concepts and skills in modeling and simulation to problems in biological transport. The objectives of this particular exercise are to gain practice using COMSOL. Complete the exercises and provide figures showing the final results for the exercises you have completed. It doesn't have to be anything pretty. Your results are due by September 21st.

Cooling and Solidification of Metal

Introduction

This example is a model of a continuous casting process. Liquid metal is poured into a mold of uniform cross section. The outside of the mold is cooled and the metal solidifies as it flows through the mold. When the metal leaves the mold, it is completely solidified on the outside but still liquid inside. The metal then continues to cool and eventually solidify completely, at which point it can be cut into sections. This tutorial simplifies the problem somewhat by not computing the flow field of the liquid metal and assuming there is no volume change at solidification. It is also assumed that the velocity of the metal is constant and uniform throughout the modeling domain. The phase transition from molten to solid state is modeled via the apparent heat capacity formulation. Issues of convergence and mesh refinement are addressed for this highly nonlinear model.

The Continuous Casting application is similar to this one, except that the velocity is computed from the Laminar Flow interface instead of being considered constant and uniform. For a detailed description of the application, see Continuous Casting.
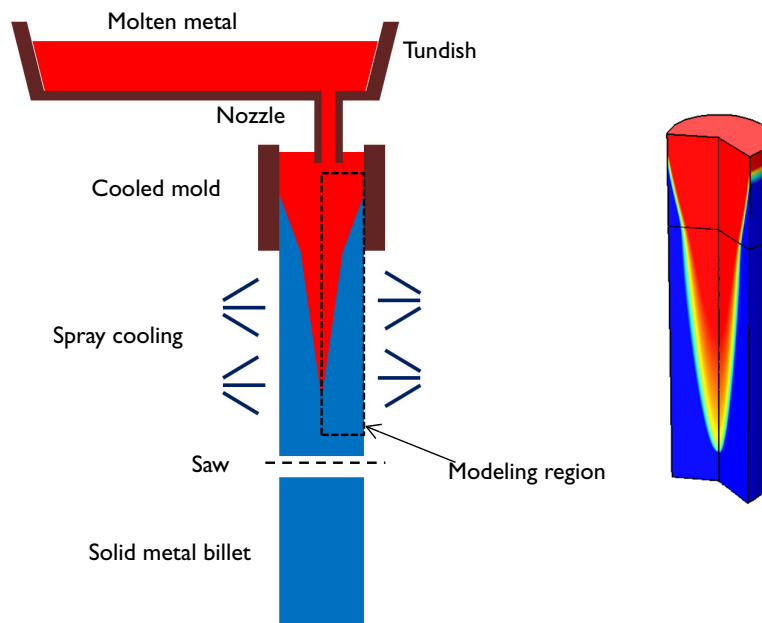
Figure 1: A continuous casting process. The section where the metal is solidifying is being modeled.

Model Overview

The model simplifies the 3D geometry of the continuous casting to a 2D axisymmetric model composed of two rectangular regions: one representing the strand within the mold, and one the spray cooled region outside of the mold, prior to the saw cutoff. In the second section, there is also significant cooling via radiation to the ambient. In this region it is assumed that the molten metal is in a hydrostatic state, that the only motion in the fluid is due to the bulk downward motion of the strand. This simplification allows the assumption of bulk motion throughout the domain.

Since this is a continuous process, the system can be modeled at steady state. The heat transport is described by the equation:

$$\nabla \cdot (-k \nabla T) = -\rho C_p \mathbf{u} \cdot \nabla T$$

where k and C_p denote thermal conductivity and specific heat, respectively. The velocity, $\mathbf{u}$, is the fixed casting speed of the metal in both liquid and solid states.

As the metal cools down in the mold, it solidifies. During the phase transition, a significant amount of latent heat is released. The total amount of heat released per unit mass of alloy during the transition is given by the change in enthalpy, ΔH. In addition, the specific heat capacity, C_p, also changes considerably during the transition. The difference in specific heat before and after transition can be approximated by

$$\Delta C_p = \frac{\Delta H}{T}$$

As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several Kelvin, in which a mixture of both solid and molten material co-exist in a "mushy" zone. To account for the latent heat related to the phase transition, the Apparent Heat capacity method is used through the Heat Transfer with Phase Change domain condition. The objective of the analysis is to make ΔT, the half-width of the transition interval small, such that the solidification front location is well defined.

Table 1 reviews the material properties in this tutorial.

TABLE 1: MATERIAL PROPERTIES

PROPERTY	SYMBOL	MELT	SOLID
Density	ρ (kg/m^3)	8500	8500

PROPERTY	SYMBOL	MELT	SOLID
Heat capacity at constant pressure	C_p (J/(kg·K))	531	380
Thermal conductivity	k (W/(m·K))	150	300

The melting temperature, T_m, and enthalpy, ΔH, are 1356 K and 205 kJ/kg, respectively.

This example is a highly nonlinear problem and benefits from taking an iterative approach to finding the solution. The location of the transition between the molten and solid state is a strong function of the casting velocity, the cooling rate in the mold, and the cooling rate in the spray cooled region. A fine mesh is needed across the solidification front to resolve the change in material properties. However, it is not known where this front will be.

By starting with a gradual transition between liquid and solid, it is possible to find a solution even on a relatively coarse mesh. This solution can be used as the starting point for the next step in the solution procedure, which uses a sharper transition from liquid to solid. This is done using the continuation method. Given a monotonic list of values to solve for, the continuation method uses the solution to the last case as the starting condition for the next. Once a solution is found for the smallest desired ΔT, the adaptive mesh refinement algorithm is used to refine the mesh to put more elements around the transition region. This finer mesh is then used to find a solution with an even sharper transition. This can be repeated as needed to get better and better resolution of the location of the solidification front.

In this example, the parameter ΔT is first ramped down from 300 K to 75 K, then the adaptive mesh refinement is used such that a finer mesh is used around the solidification front. The resultant solution and mesh are then used as starting points for a second study, where the parameter ΔT is further ramped down from 50 K to 25 K. The double-dogleg solver is used to find the solution to this highly nonlinear problem. Although it takes more time, this solver converges better in cases when material properties vary strongly with respect to the solution.

Results and Discussion

The solidification front computed with the coarsest mesh, and for $\Delta T = 75$ K, is shown in Figure 2. A wide transition between the molten and solid state is observed. The adaptive mesh refinement algorithm then refines the mesh along the solidification front because this is the region where the results are strongly dependent upon mesh size. This solution, and refined mesh, is used as the starting point for the next solution,

which ramps the ΔT parameter down to 25 K. These results are shown in Figure 3.

The point of complete solidification moves slightly as the transition zone is made smaller. As the transition zone becomes smaller, a finer mesh is needed, otherwise the model might not converge. If it is desired to get an even better resolution of the solidification front, the solution procedure used here should be repeated to get an even finer mesh, and further ramp down the ΔT parameter.

The liquid phase fraction is plotted along the r-direction at the line at the bottom of the mold in Figure 4, and Figure 5 shows the liquid fraction along the centerline of the strand. For smaller values of ΔT, the transition becomes sharper, and the model gives confidence that the metal is completely solidified before the strand is cut.
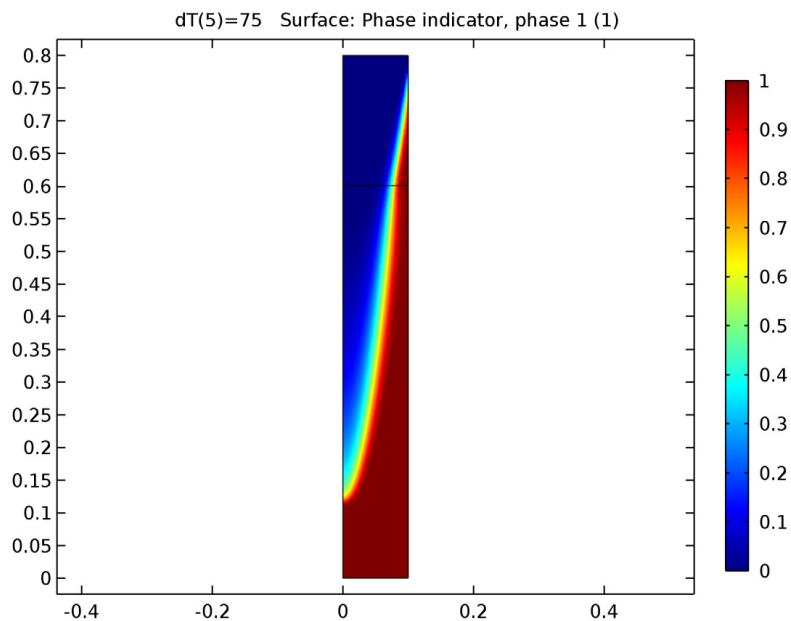
Figure 2: The fraction of liquid phase for $\Delta T = 75$ K shows a gradual transition between the liquid and solid phase.
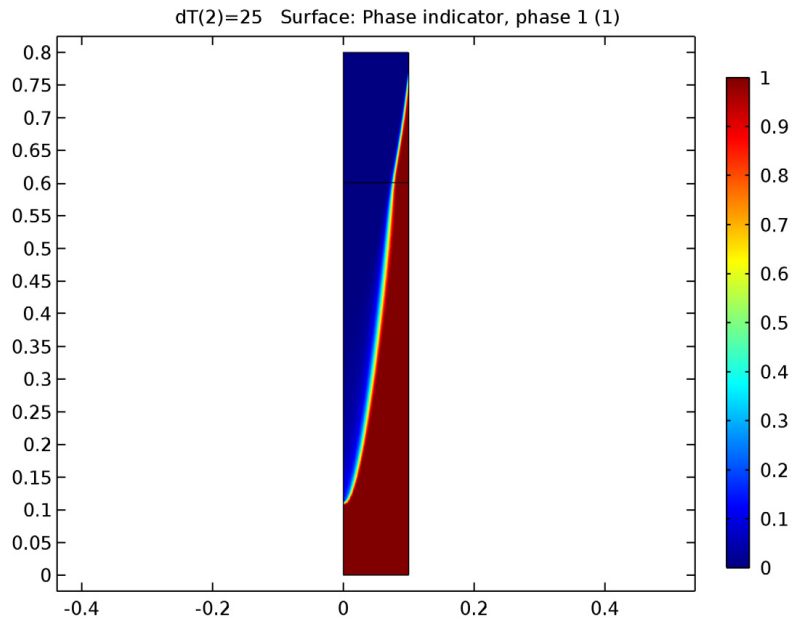
Figure 3: The fraction of liquid phase for ΔT = 25 K shows a sharp transition between the liquid and solid phase.
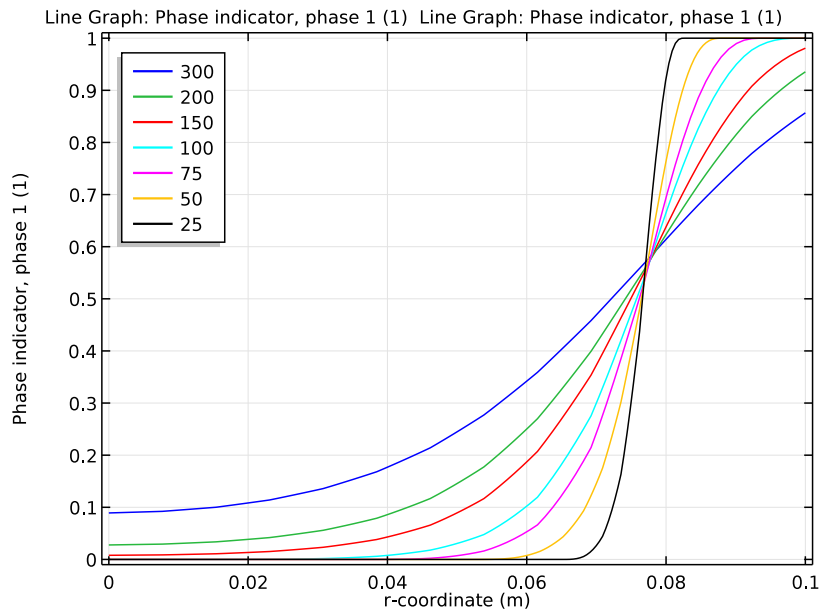
Figure 4: The fraction of liquid phase through the radius for all values of ΔT. For smaller values of ΔT, the transition is sharper.
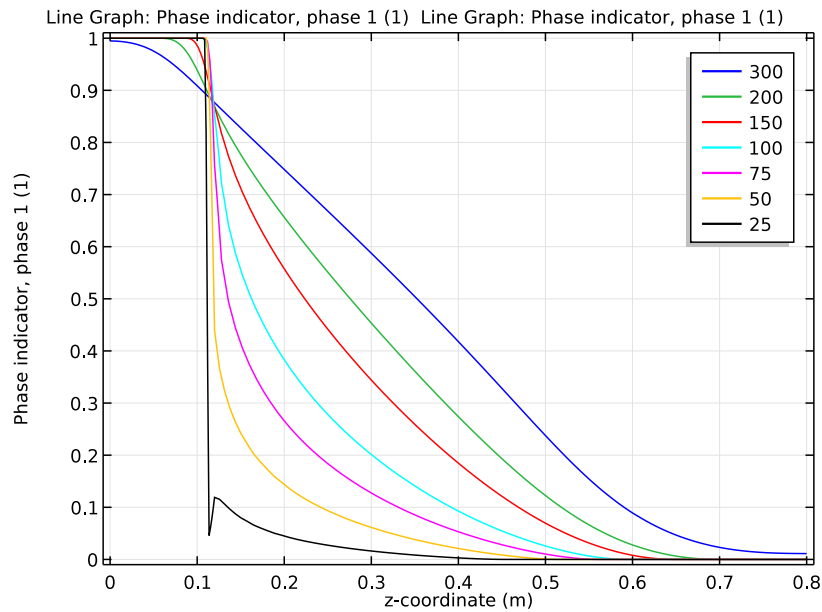
Figure 5: The fraction of liquid phase along the centerline for all values of ΔT. For smaller values of ΔT, the transition is sharper.

Application Library path: `Heat_Transfer_Module/Thermal_Processing/`
`cooling_solidification_metal`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select study** tree, select **Preset Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

First, set up parameters and variables used in the continuous casting process model analysis.

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 Click **Load from File**.

4 Browse to the application's Application Library folder and double-click the file `cooling_solidification_metal_parameters.txt`.

GEOMETRY 1

Create two rectangles representing the strand within the mold, and spray cooled region outside of the mold.

Rectangle 1 (r1)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

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

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.6`.

Rectangle 2 (r2)

1 On the **Geometry** toolbar, click **Primitives** and choose **Rectangle**.

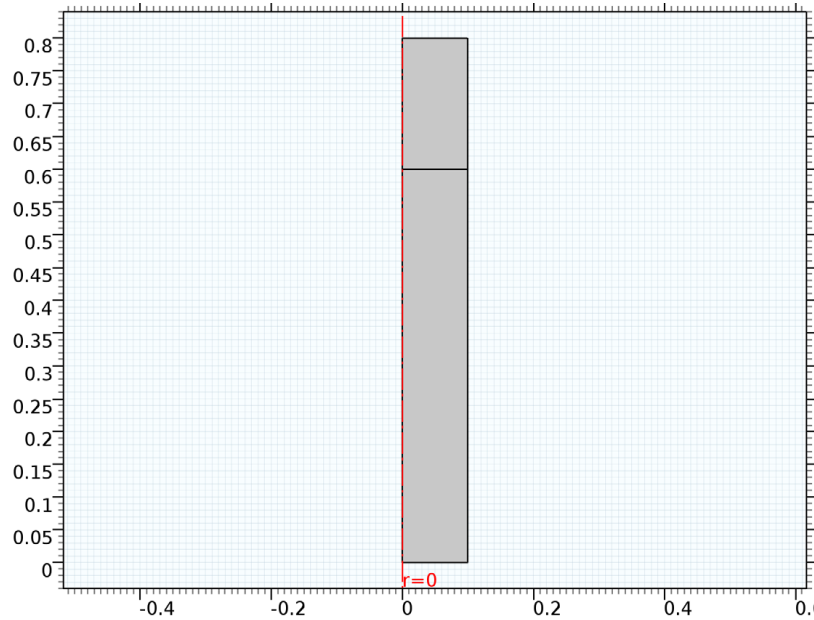**2** In the **Settings** window for Rectangle, locate the **Size** section.

3 In the **Width** text field, type `0.1`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **z** text field, type `0.6`.

6 Click the **Build All Objects** button.

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

MATERIALS

Material 1 (mat1)

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

2 In the **Settings** window for Material, type Solid Metal Alloy in the **Label** text field.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	300	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_s	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Material 2 (mat2)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for Material, type Liquid Metal Alloy in the **Label** text field.

3 Select Domains 1 and 2 only.

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

Property	Name	Value	Unit	Property group
Thermal conductivity	k	150	W/(m·K)	Basic
Density	rho	8500	kg/m³	Basic
Heat capacity at constant pressure	Cp	Cp_l	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Set up the physics.

HEAT TRANSFER IN FLUIDS (HT)

Initial Values 1

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

2 In the **Settings** window for Initial Values, locate the **Initial Values** section.

3 In the T text field, type T_in.

Heat Transfer with Phase Change 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Transfer with Phase Change**.

2 Select Domains 1 and 2 only.

3 In the **Settings** window for Heat Transfer with Phase Change, locate the **Model Inputs** section.

4 Specify the **u** vector as

0	r
-v_cast	z

5 Locate the **Phase Change** section. In the $T_{\text{pc},1 \to 2}$ text field, type T_m.

6 In the $\Delta T_{1 \to 2}$ text field, type dT.

7 In the $L_{1 \to 2}$ text field, type dH.

8 Locate the **Phase 1** section. From the **Material, phase 1** list, choose **Solid Metal Alloy (mat1)**.

9 Locate the **Phase 2** section. From the **Material, phase 2** list, choose **Liquid Metal Alloy (mat2)**.

Temperature 1

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

2 Select Boundary 5 only.
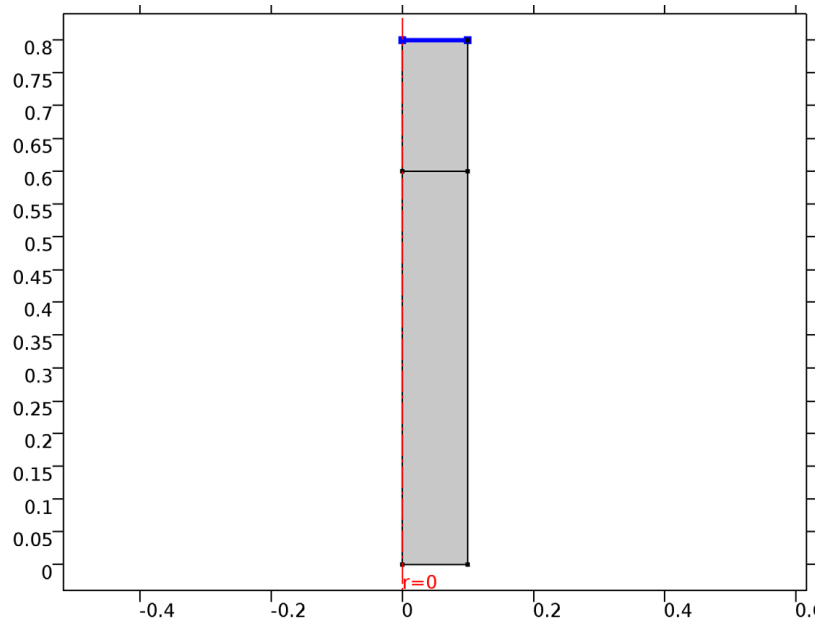
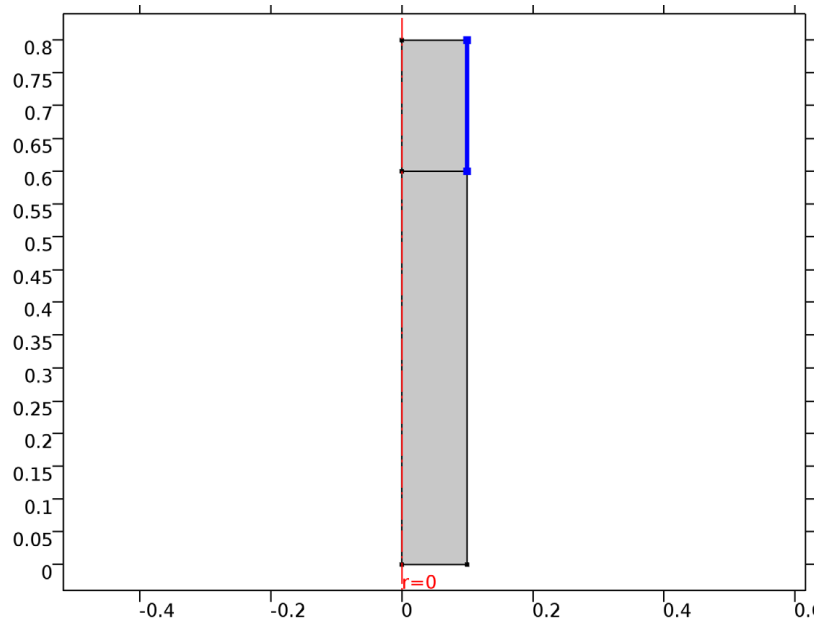

3 In the **Settings** window for Temperature, locate the **Temperature** section.

4 In the T_0 text field, type T_in.

Heat Flux 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 7 only.

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

4 Click the **Convective heat flux** button.

5 In the h text field, type h_mold.

6 In the T_{ext} text field, type T0.

Heat Flux 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 Select Boundary 6 only.

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

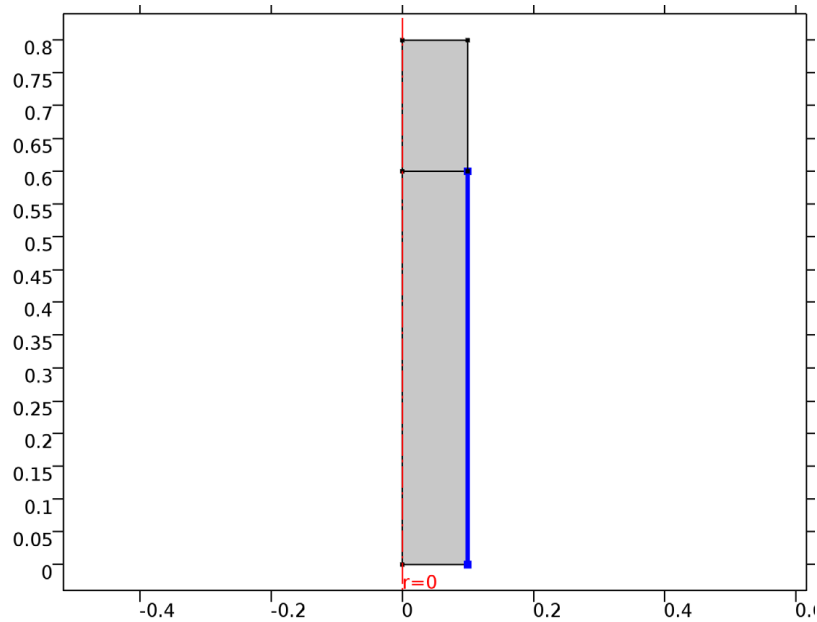**4** Click the **Convective heat flux** button.

5 In the h text field, type h_spray.

6 In the T_{ext} text field, type T0.

Diffuse Surface 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Diffuse Surface**.

2 Select Boundary 6 only.

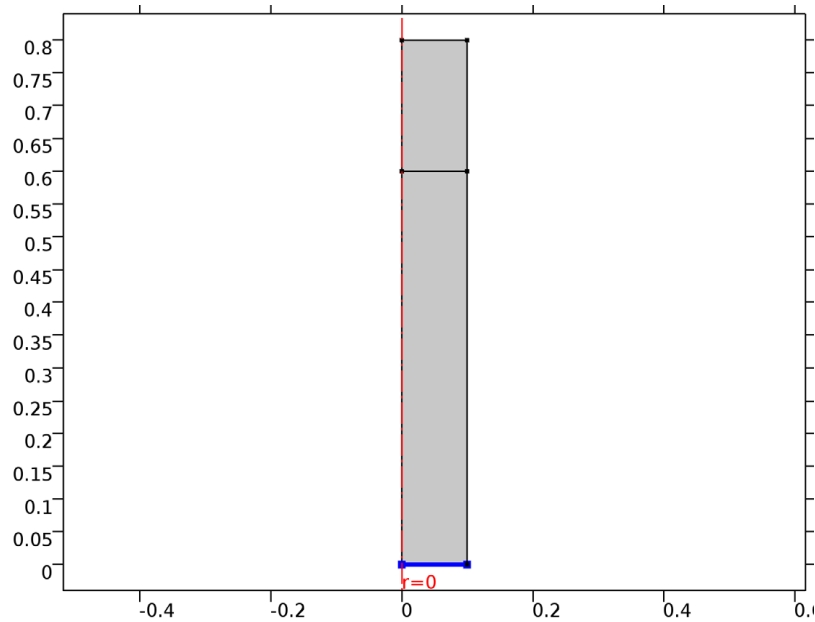**3** In the **Settings** window for Diffuse Surface, locate the **Surface Emissivity** section.

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

5 Locate the **Ambient** section. In the T_{amb} text field, type T0.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 2 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

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

3 From the **Element size** list, choose **Extremely fine**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** check box.

Step 1: Stationary

Set up an auxiliary continuation sweep for the dT parameter.

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

2 In the **Settings** window for Stationary, click to expand the **Study extensions** section.

3 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	300 200 150 100 75	

6 Select the **Adaptive mesh refinement** check box.

Solution 1

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

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

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1>Stationary Solver 1** node, then click **Adaptive Mesh Refinement**.

4 In the **Settings** window for Adaptive Mesh Refinement, locate the **Mesh Refinement** section.

5 From the **Refinement method** list, choose **Mesh initialization**.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1>Stationary Solver 1** click **Fully Coupled 1**.

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

8 Locate the **Method and Termination** section. From the **Nonlinear method** list, choose **Double dogleg**.

9 On the **Study** toolbar, click **Compute**.

RESULTS

2D Plot Group 1

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type `Solid and Liquid Phases (Adaptive Mesh)` in the **Label** text field.

Solid and Liquid Phases (Adaptive Mesh)

1 Right-click **Results>Solid and Liquid Phases (Adaptive Mesh)** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 On the **Solid and Liquid Phases (Adaptive Mesh)** toolbar, click **Plot**.

The reproduced figure describes the fraction of liquid phase for $\Delta T = 75$ K.

ADD STUDY

1 On the **Home** 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**.

4 In the **Select study** tree, select **Preset Studies>Stationary**.

5 Click **Add Study** in the window toolbar.

STUDY 2

Step 1: Stationary

1 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

2 In the **Model Builder** window, click **Study 2**.

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

4 Clear the **Generate default plots** check box.

5 In the **Model Builder** window, click **Step 1: Stationary**.

6 In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.

7 Locate the **Values of Dependent Variables** section. Select the **Initial values of variables solved for** check box.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 1, Stationary**.

10 From the **Solution** list, choose **Adaptive Mesh Refinement 1**.

11 From the **Parameter value (dT)** list, choose **75**.

12 Click to expand the **Mesh selection** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT	50 25	

Solution 3

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

2 In the **Model Builder** window, expand the **Solution 3** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 3>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for Fully Coupled, locate the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Double dogleg**.

6 On the **Study** toolbar, click **Compute**.

RESULTS

2D Plot Group 2

1 On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for 2D Plot Group, type Solid and Liquid Phases in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 3**.

Solid and Liquid Phases

1 Right-click **Results>Solid and Liquid Phases** and choose **Surface**.

2 In the **Settings** window for Surface, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Heat Transfer in Fluids>Phase change>ht.theta1 - Phase indicator, phase 1**.

3 On the **Solid and Liquid Phases** toolbar, click **Plot**.

This shows the fraction of liquid phase for $\Delta T = 25$ K.

1D Plot Group 3

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

2 In the **Settings** window for 1D Plot Group, type Phase Indicator at Symmetry Axis in the **Label** text field.

Phase Indicator at Symmetry Axis

1 On the **Phase Indicator at Symmetry Axis** toolbar, click **Line Graph**.

2 Select Boundaries 1 and 3 only.

3 In the **Settings** window for Line Graph, locate the **y-axis data** section.

4 Click **ht.theta1 - Phase indicator, phase 1** in the upper-right corner of the section. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type z.

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

7 Right-click **Results>Phase Indicator at Symmetry Axis>Line Graph 1** and choose **Duplicate**.

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

9 From the **Data set** list, choose **Study 2/Solution 3**.

10 On the **Phase Indicator at Symmetry Axis** toolbar, click **Plot**.

Compare the resulting plot with Figure 5 showing the fraction of liquid phase through the centerline for all values of ΔT.

Phase Indicator at Symmetry Axis 1

1 In the **Model Builder** window, right-click **Phase Indicator at Symmetry Axis** and choose **Duplicate**.

2 In the **Settings** window for 1D Plot Group, type Phase Indicator through Radius in the **Label** text field.

Phase Indicator through Radius

1 In the **Model Builder** window, expand the **Results>Phase Indicator through Radius** node, then click **Line Graph 1**.

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

3 Click **Clear Selection**.

4 Select Boundary 4 only.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type r.

6 On the **Phase Indicator through Radius** toolbar, click **Plot**.

7 In the **Model Builder** window, under **Results>Phase Indicator through Radius** click **Line Graph 2**.

8 In the **Settings** window for Line Graph, locate the **Selection** section.

9 Click **Clear Selection**.

10 Select Boundary 4 only.

11 Locate the **x-Axis Data** section. In the **Expression** text field, type r.

12 In the **Model Builder** window, click **Phase Indicator through Radius**.

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

14 From the **Position** list, choose **Upper left**.

15 On the **Phase Indicator through Radius** toolbar, click **Plot**.

Compare the resulting plot with Figure 4 showing the fraction of liquid phase through the radius for all values of ΔT.

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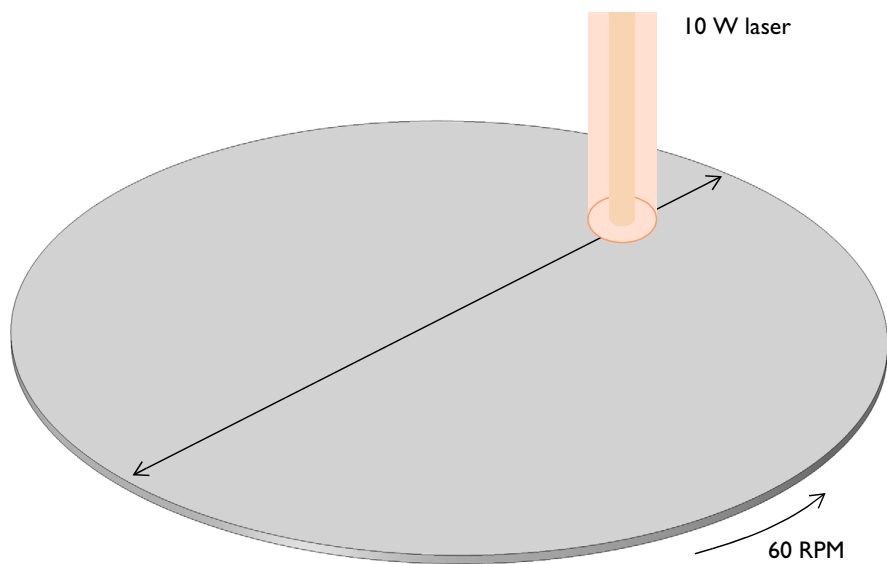
Figure 1: A silicon wafer is heated with a laser that moves back and forth. The wafer is also being rotated about its axis.

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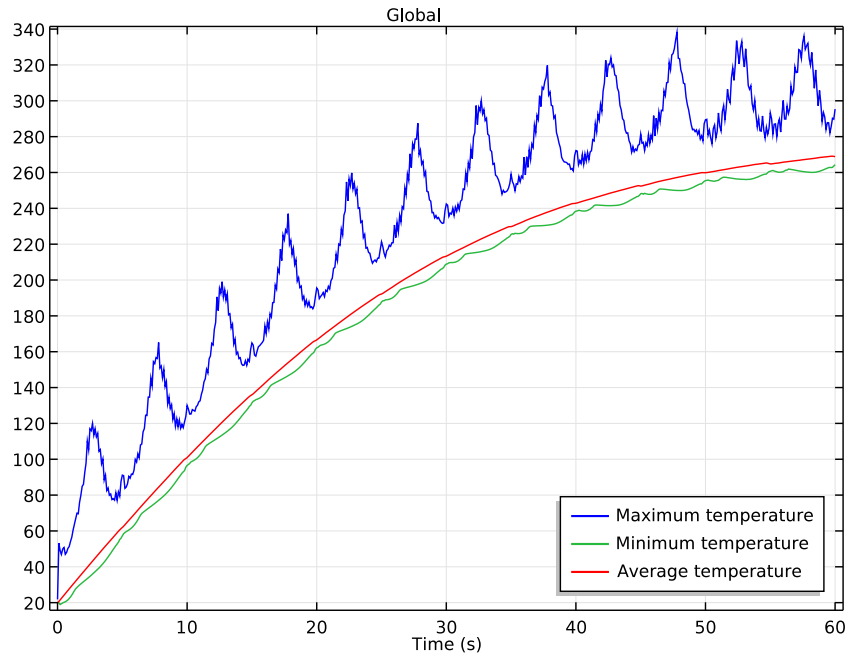
Figure 2: Maximum, minimum, and average temperatures of the wafer as functions of time.
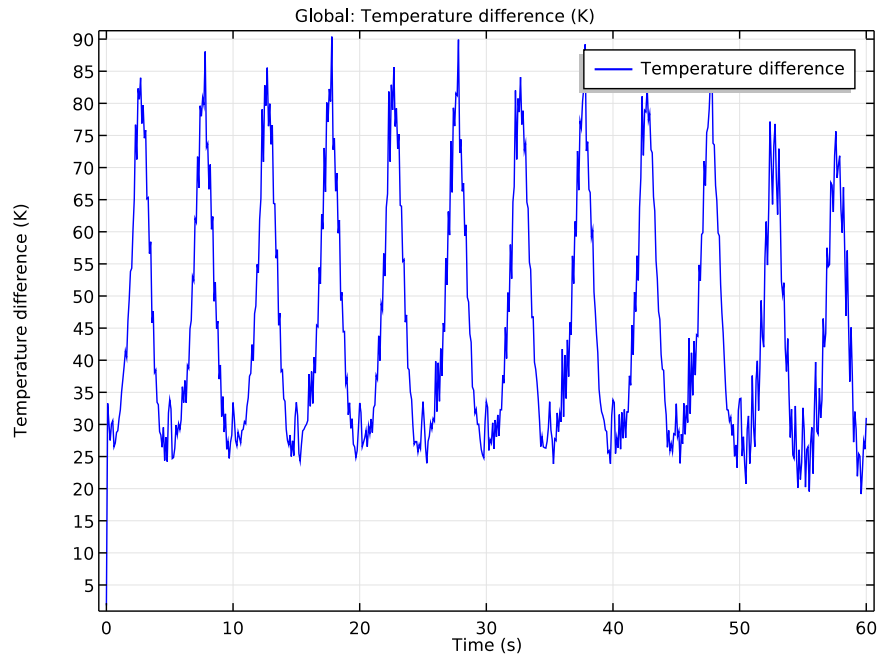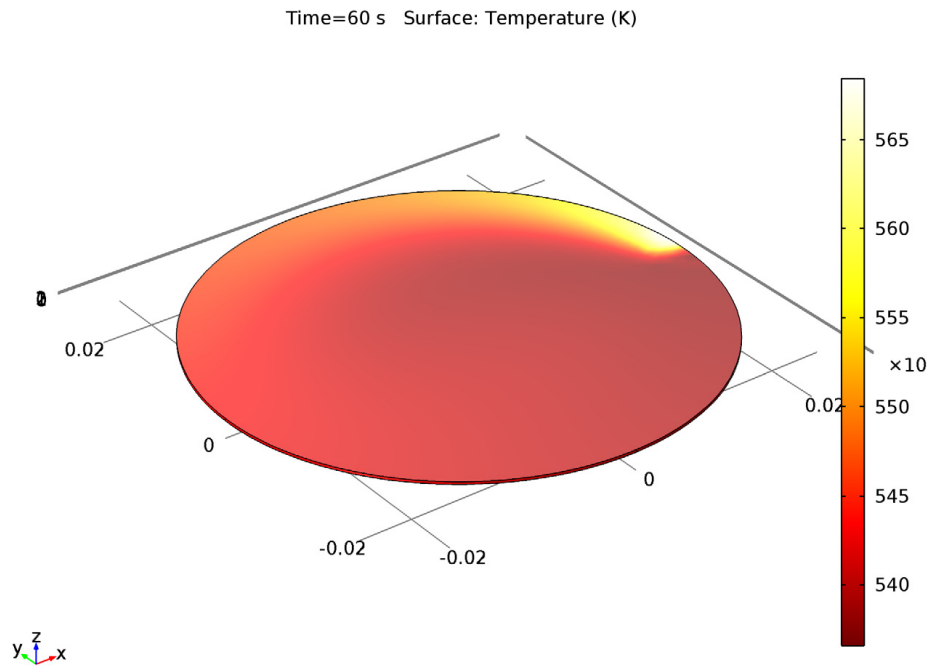

Figure 3: Difference between maximum and minimum temperatures on the wafer.

Time=60 s   Surface: Temperature (K)

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

1 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 Hz	Rotational speed
angular_v	2*pi*v_rotation	6.283 Hz	Angular velocity
period	10[s]	10 s	Time for laser to move back and forth
r_spot	2.5[mm]	0.0025 m	Radius of laser spot size
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.
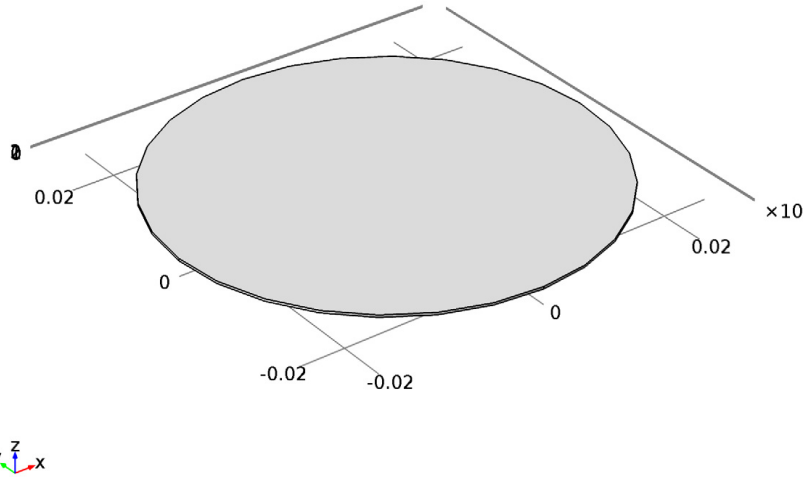
Cylinder 1 (cyl1)

1 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 the **Build All Objects** button.

DEFINITIONS

Define functions for use before setting up the physics.

Gaussian Pulse 1 (gp1)

1 On the **Home** toolbar, click **Functions** and choose **Local>Gaussian Pulse**.

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

3 In the **Standard deviation** text field, type r_spot/3.

The laser beam radius is equal to three standard deviations of the Gaussian pulse. These three standard deviations account for 99.7% of the total laser energy.

4 Click the **Plot** button.
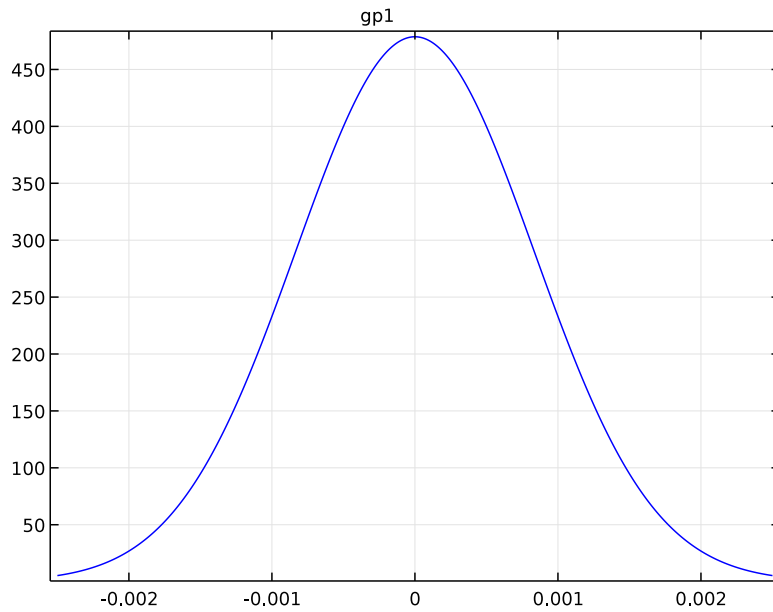
Waveform 1 (wv1)

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

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

3 From the **Type** list, choose **Triangle**.

4 Clear the **Smoothing** check box.

5 In the **Angular frequency** text field, type `2*pi/period`.

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

7 In the **Amplitude** text field, type `r_wafer`.

8 Click the **Plot** button.
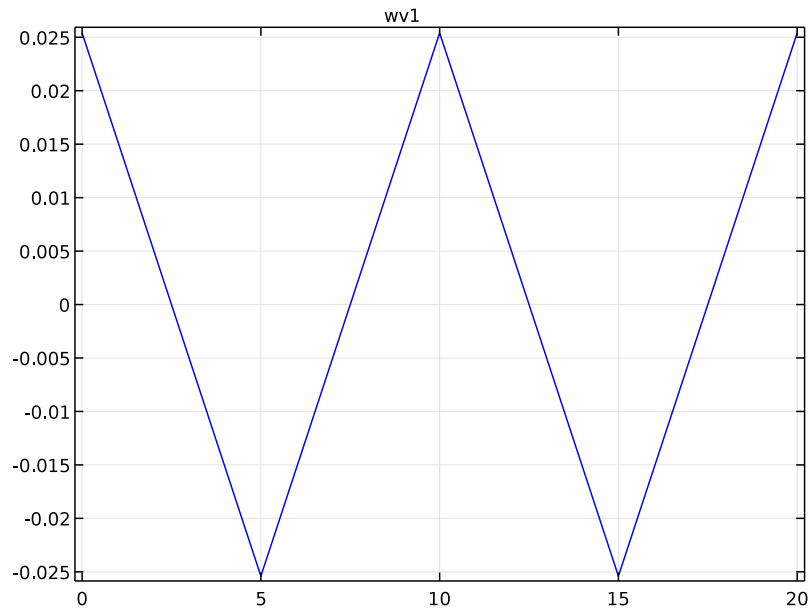
Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type
`p_laser*gp1(x-wv1(t))*gp1(y)`.

4 In the **Arguments** text field, type `x, y, t`.

5 Locate the **Units** section. In the **Arguments** text field, type `m, m, s`.

6 In the **Function** text field, type `W/m^2`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
x	-r_spot	r_spot
y	-r_spot	r_spot
t	period/4	period/4

8 Click the **Plot** button.
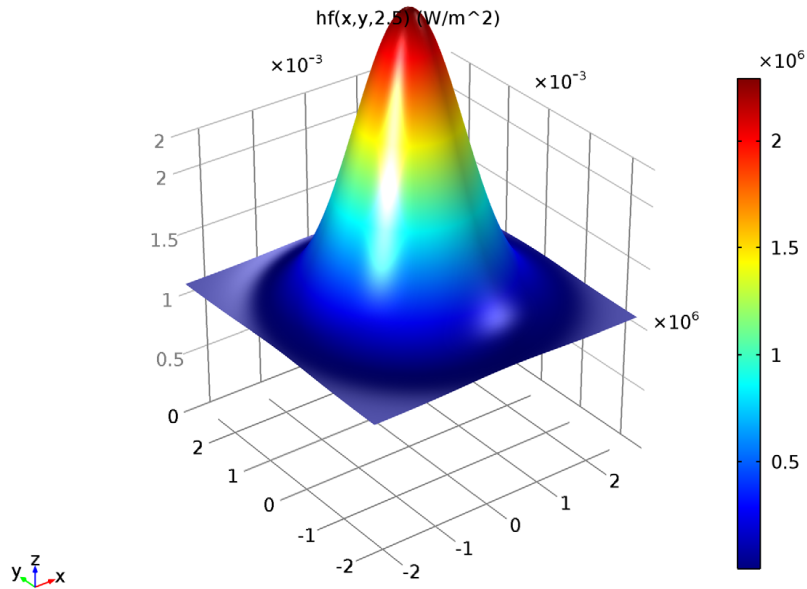
Next, add coupling operators for the result analysis.

Average 1 (aveop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

2 Select Domain 1 only.

Maximum 1 (maxop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Maximum**.

2 Select Domain 1 only.

Minimum 1 (minop1)

1 On the **Definitions** toolbar, click **Component Couplings** and choose **Minimum**.

2 Select Domain 1 only.

HEAT TRANSFER IN SOLIDS (HT)

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

Heat Transfer in Solids 1

In the Model Builder window, under **Heat Transfer in Solids (ht)**, click **Heat Transfer in Solids 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 $\mathbf{u}_{trans}$ vector as

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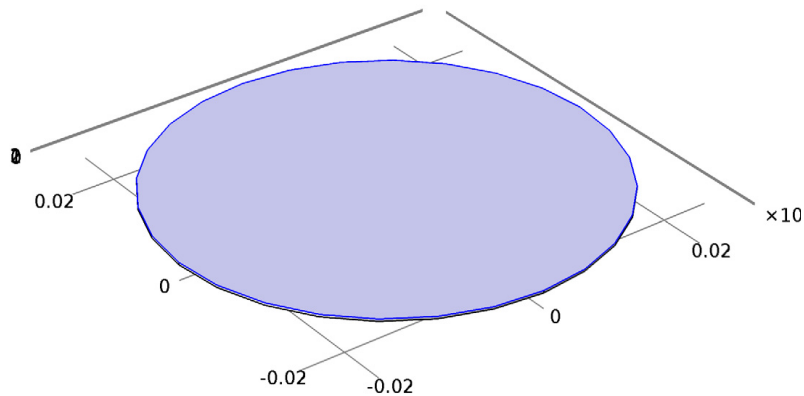
3 In the **Settings** window for Heat Flux, locate the **Heat Flux** section.

4 In the q_0 text field, type `emissivity*hf(x,y,t)`.

Diffuse Surface 1

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 1

Use a fine triangular swept mesh.

Swept 1

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 1

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 the **Build All** button.
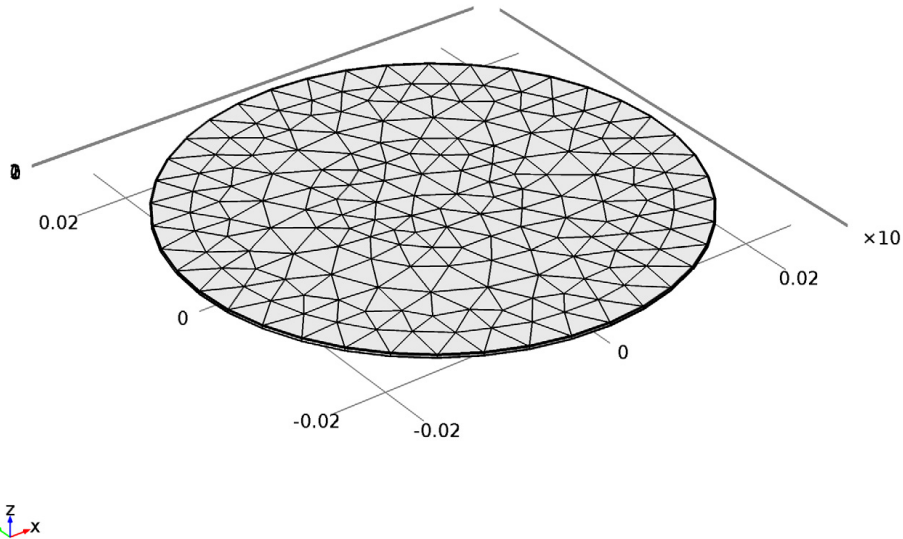
STUDY 1

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 **Times** text field, type range(0,0.1,60).

4 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.

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**.
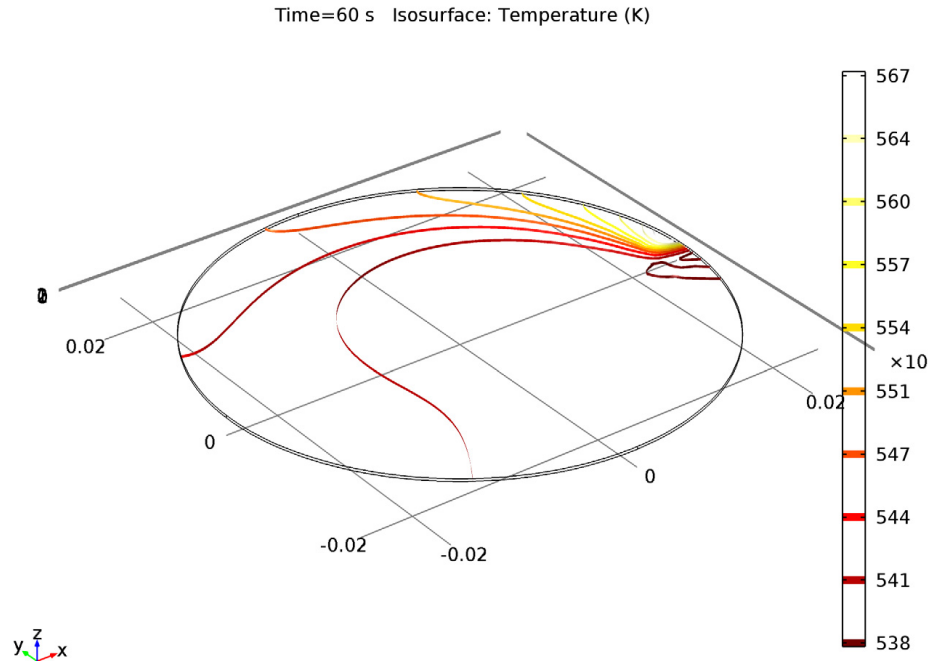
3 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

4 In the associated text field, type `0.036`.

5 From the **Color** list, choose **Green**.

Compare the temperature variation with that shown in Figure 4.

Isothermal Contours (ht)

The second default plot shows the isosurface temperature.

Time=60 s   Isosurface: Temperature (K)

1D Plot Group 3

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

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

3 From the **Position** list, choose **Lower right**.

4 On the **1D Plot Group 3** toolbar, click **Global**.

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

6 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

7 On the **1D Plot Group 3** toolbar, click **Plot**.

Compare the plot showing the maximum, minimum and average temperature with Figure 2.

1D Plot Group 4

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

2 On the **1D Plot Group 4** toolbar, click **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 **1D Plot Group 4** toolbar, click **Plot**.
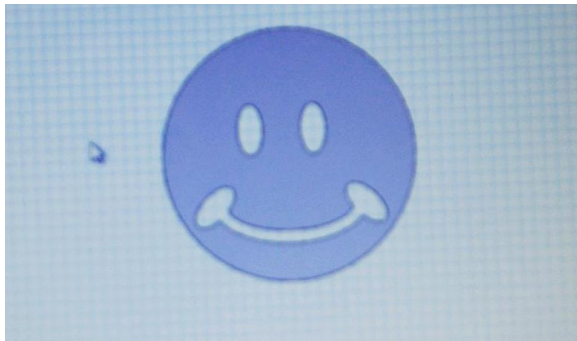
Finish the result analysis by comparing the generated plot with Figure 3.

Figure 1: **A smiley face.**