# Solutions to the heat equation with Comsol

Comsol is a finite element software. You can use it to obtain numerical solutions to the heat equation. In some of the cases below (where the geometry is simple) these solutions exist in analytical form. We will use the software to gain intuition about the behavior of solutions to the heat equation. We will demonstrate Comsol use in class and use it somewhat throughout the course. Comsol can be pretty easy to learn and can solve a number of problems quite readily. It can be sometimes be hard to force it to do what you want. Don't get too frustrated and please seek help if you get stuck. The purpose of using finite element software in this course is to gain some insight into solutions, not to master the software.

Don't get too worried if you can't force the graphics and plots to do exactly what you want. This is one area where it can be difficult to force the software to plot what you want. If your graphs are a little clunky, you can print and annotate things by hand. If on any given problem, you find there is something more interesting to look at than what I selected here, go for it. The questions and things I ask you to explore are only suggestions. It is always best to do some simple estimates first - i.e. for a transient problem estimate how long it might take and object to cool down before solving so you know how long in time to solve for. The default for comsol is to solve a transient problem for 1 second - this will not be appropriate for most problems.

## I. IN CLASS PROBLEMS.

These problems we will work on in class. These problems are mainly to explore different situations and learn the software. You should work on these during class time, but you don't need to turn anything in for this first set of problems.

1. Solve the following problem numerically with Comsol.

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

$$T(t = 0, x) = 0; \quad T(t, x = 0) = 100; \quad T(t, x = 5cm) = 0;$$

   Solve the problem on the domain $0 < x < 5$ cm. Set the properties to that of aluminum. Adjust the time of the solution to capture the most interesting transients. Based on what we have done already, check that your answers are sound. Explore the evolution of the temperature field to gain some intuition. Try adjusting the initial and boundary conditions. Try changing the material to see if the diffusion time scale estimate $\tau \sim \alpha/L^2$ is reasonable.
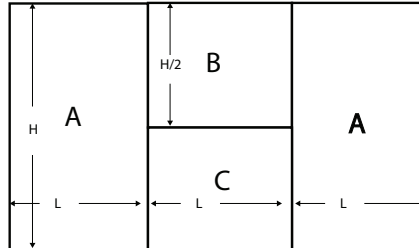
2. Solve the 2D steady state heat equation on a square of 1 by 1 cm. Set three of the boundaries to have a temperature of 0 and the other one to have a temperature of 1. Plot the heat flux vectors. Look at the temperature field as a mesh plot to explore what a zero curvature surface looks like (remember, at steady state the temperature field has no curvature). The material does not matter as the conductivity does not enter the problem.

3. Create a cube 5 cm on each side. Solve the transient heat conduction problem with a convection coefficient of 20 W/m$^2$K on all sides. Try thermal conductivities of 0.01, 1, and 100 W/mK. Compare a plot of the objects center temperature as a function of time obtained in Comsol to that obtained by the lumped approximation (where the objects temperature is assumed to be constant throughout- see book). Compute the Biot number for each case and use that dimensionless number to interpret your results.

4. The temperature of the soil as a function of depth varies throughout the year. Consider the 1D heat equation on a domain that extends sufficiently far into the earth. The boundary conditions would be

$$T(x \to \infty, t) = T_{ave}; \quad T(x = 0, t) = T_{ave} + A\sin(2\pi(t/365))$$

   where $T_{ave}$ is the average temperature throughout the year, $A$ is an amplitude that would approximate the yearly temperature swings from summer to winter, and $t$ is time measured in days. Look up reasonable estimates for $\alpha$ for soil, $T_{ave}$ and $A$ for the Boston area. Of course, the temperature is not a perfect sinusoid, but you should be able to approximate it as such within reason.

Solve the problem in Comsol. Pay attention to the units since you need to solve for 1 year. Make the domain sufficiently large such that the solution could be considered the same as you would have on an infinite domain. Plot the temperature as a function of time at various instances in time. How far down in the earth do you need to go for the temperature to be approximately constant? How does the actual result compare to an estimation based only on an argument of the units in the problem depth $\sim \sqrt{t/\alpha}$.

5. Consider heat conduction in 2D through a composite wall as shown below. Using the resistor circuit analogy, derive an expression for the total thermal resistance of the wall in terms of the geometry ($L$ and $H$) and the thermal conductivity of the three materials, $k_A$, $k_B$, and $k_c$.



In comsol, solve the steady state conduction problem for a set temperature difference on the left and right boundaries. Compute the total heat flow with Comsol and define the overall effective resistance as the ratio of the total temperature difference to the total heat flow. Compare the comsol result to the simple circuit analogy solution. Try the following sets of parameters;

- $L = 5$ cm, $H = 20$ cm, $k_A = 100$, $k_B = 10$, and $k_c = 1$ W/m K.
- $L = 20$ cm, $H = 5$ cm, $k_A = 100$, $k_B = 10$, and $k_c = 1$ W/m K.
- $L = 5$ cm, $H = 20$ cm, $k_A = 1$, $k_B = 10$, and $k_c = 100$ W/m K.

How well does the circuit analogy holds. When is the analogy good and when does it break down?

## II. COMPARING COMSOL TO DATA

The following two problems you should do some analysis and turn something in.

1. Imagine a thin copper bar with thickness $b$, width $w$, length $L$, and thermal conductivity $k$. One end is attached to a wall at a temperature of $T_{hot}$. The bar is exposed to air with a temperature $T_\infty$ and and a convection coefficient, $h$. We will assume, for now, that we are in a regime that $b << w$ and $b << L$. Since the bar is thin and made of a good conductor of heat, it is probably a good assumption that the temperature at any axial location is constant across the bar's cross section. The temperature, however, must change quite significantly as a function of axial distance along the bar away from the wall.

Consider the steady state temperature distribution as a function of distance, $x$. In the text, I derive an equation to describe the temperature as a function of $x$ for a cylindrical rod. Modify the derivation to show that for the rectangular cross section you obtain a similar equation,

$$\frac{d^2T}{dx^2} = \frac{hP}{kA}(T - T_\infty),$$

where $P$ is the perimeter around the cross sectional area $A$. In this case $P = 2b + 2w$ and $A = bw$.

To make the equation dimensionless, you have two choices. One (which we will follow) is that you can scale $x$ by the length of the rod. This would result in a single dimensionless parameter in the equation and the domain for the equation would extend from $0 < x < 1$. The equation is

$$\frac{d^2\Theta}{dx^2} = m^2\Theta.$$

where $m^2 = hPL^2/kA$ is a system parameter. To solve the equation we require boundary conditions. At the hot end, we assume the temperature is known. At the far end, (in the text) we assume that convection is active

at the end such that in dimensional form $k\partial T/\partial x = h(T - T_\infty)$. Create a plot for different values of $m$ on the same plot to confirm the result in the text.

In many cases it is reasonable to assume that the end is insulated, i.e. $\partial T/\partial x = 0$ at $x = L$. Rework the solution in the text for the insulated end condition and compare the two solutions for different parameters values. Can you say something about when the insulated end approximation is good?

Now, plot your solution (with the insulated end condition) and the experimental data (posted on the website) for this problem. You will need to transform the experimental data to follow the dimensionless form above. The predictions should be in reasonable agreement with the model. You will need to fit a value of $h$ such that the data and the model agree with each other. The fins are at different temperatures and thus it is possible that a different value of $h$ will fit each experiment well. A number on the order of 10 W/m$^2$K is a reasonable first guess.

Now, we will use comsol to confirm that the assumption of constant temperature across the fin above is valid. Create a 3D rectangular fin with the dimensions of the experiment. Set the temperature of one end to be fixed, the other surfaces to have a convection coefficient with a value of 10 W/m$^2$K (or whatever value fit your experiment). Set up the problem in dimensional form, setting the same parameters as the experiment. Solve the steady state conduction problem in comsol and explore the temperature difference across the thickness of the fin. Plot the temperature as a function of distance and compare to the analytical solution that you derived above. How good was your assumption in the analytical solution of no heat flux leaving the end of the fin? How good does the simple 1D model approximate the full 3D solution? Generally, the point of this whole exercise is to look at how a simplified 1D analytical model compares to the detailed simulation of comsol and how both compare to a simple experiment. Explore the comparison between the full and simple model in whatever way you deem appropriate.

2. Go to Aavid Thermalloy's website and find part number 72545 for a heat sink; `http://www.aavid.com/products/extrusion-heatsinks/72545` The website will provide detailed information on the heat sink geometry and its performance. Included are charts showing the amount of heat dissipated vs. the temperature above ambient as well as the overall thermal resistance.

Your job is to model the steady state behavior of the heat sink. Put the dimensional geometry into Comsol along with appropriate material constants. Set up Comsol to only solve the steady state behavior. As an approximation, you can solve the problem in 2D and not worry about 3D effects. On the "hot" edge you should apply a constant temperature boundary condition of 50 C. On the edges exposed to air you need to apply a convection condition. Initially, assume the convection coefficient is constant everywhere and equal to approximately, $h = 10 \ W/m^2K$.

Compute the thermal resistance from Comsol and compare to the data sheet for natural convection. Vary the hot temperature and see how your modeled resistance changes with applied temperature. To compute the overall thermal resistance, you will need to figure out how to integrate the total heat flux through the hot surface to get a total number of Watts/m (not Watts since this is 2D). Note on the heat sink website that you choose a length and then get the thermal resistance in terms of C/W. You should convert this number to a per unit length number to compare to your model. You will need to be a little careful at this stage.

The main cause of the difference between the model and reality is that the convection coefficient may vary in space, and importantly with temperature. The hotter the air, the more buoyant hot air will be and therefore there will be greater flow and more vigorous convection. Let us continue to assume $h$ remains a constant along the boundary of the heat sink, but lets see how it varies with temperature. For a few (at least 3) data points on measured performance curves, "fit" a value of $h$ which provides reasonable (not perfect) agreement. Plot the estimated value of $h$ vs. temperature. How strongly does $h$ vary with temperature?