

Finite Element Models for Heat Transfer Modeling and Control (EDS)

Introduction

This week you will experiment with COMSOL finite element modeling software for a variety of diffusion experiments. We will start with four tutorial case studies for practice, incrementally building your ability to model heat transfer problems with finite element techniques. After that you will analyse real heat sink designs.

Objectives

1. Learn how to use COMSOL Multiphysics 3.4 to extend your ability to model heat transfer.
2. Become a good consumer of finite element models. Learn to ask simple questions to verify the sanity of your models.
3. Connect what COMSOL is doing to your work with close-form solutions and circuit models.
4. Build the capability to model real-world situations, specifically two particular heat sink designs.

Tutorial

Read the Flippin' Manual (RTFM)

Typically this is a sign of desperation for engineers, but don't be afraid to read the manual. Under the "Quick Start" help section in COMSOL there is a short section on "Diffusion". This section might help you understand what is under the hood and show you how the finite element formulation relates to the close-form solutions we've been working on in labs one and 2. In the standard COMSOL installation you can find this documentation at... `file:///C:/COMSOL34/doc/multiphysics/wwhelp/wwhimpl/js/html/wwhelp.htm?context=multiphysics&file=html_quickintro.1.1.html`

Case Study 1: 1-D Transient Diffusion

Physical Setup

Consider the scaled diffusion problem

$$\frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2} \quad (1)$$

with **boundary and initial conditions**,

$$T(x = 0, t > 0) = 0 \quad (2)$$

$$T(x = 1, t > 0) = 0 \quad (3)$$

$$T(0 < x < 1, t = 0) = 1 \quad (4)$$

Solve the problem using COMSOL. Select the transient heat transfer problem in 1-dimension. Draw your domain as a line between 0 and 1. Set the constants in the "physics; sub-domain settings" such that you are solving the diffusion equation with unity constant. Note that Comsol has several additional terms in the heat transfer model that are not part of the standard heat equation. Constants associated with these terms should remain zero (as they will by default). The constants that should be one are related to the heat capacity, density, and thermal conductivity. Set the initial temperature to T=1.0. You can check the equation at the top of the window to confirm that your constants are set to solve the heat diffusion equation as written above. Under "Physics; boundary conditions" set the left and right boundary to have a temperature of T=0.

Solution

Explore the meshing criteria. You must define an initial mesh and then you can repetitively refine spatial mesh. How many elements are required for a good solution? Run the simulation. Examine the “Solver Parameters” to make sure you understand how the time-steps are selected.

Post-Processing

Under “Post-Processing” run the animation and plot the solutions at various times. You may need to adjust the “Solver Parameters” to output the solution at relevant times.

Some comments on behavior

This activity is meant to guide you through a tutorial experience that many engineers find helpful. This behavior might be described as solving a trivial problem with non-trivial means. The point of this exercise is not to solve solve this problem! We probably all know what happens when you attach cold ends to a hot beam—it cools down. However, because we know the answer, we know the truth we can make sure we understand the tool (COMSOL). You should play with this new tool in this simple environment. A term for this “play” is *sensitivity analysis*. It is typically important to discover the sensitivity of your solution to changes in the model parameters, i.e., how do the inputs affect the outputs? Qualitatively we can consider some of the solution parameters and see how they affect the solution.

- How does your choice of meshing parameters affect the solution?
- Examine the “Solver Parameters”. How do these parameters affect the solution?

With such a simple problem, nearly anything you try will result in a “good” solution. We will find examples later that require more care.

Case Study 2: 1D Diffusion With New Boundary Conditions

Consider 1-D heat conduction in a domain that is 100 mm wide. The left half ($0 < x < 50$) is made of copper, $k = 400$ W/mK, and the right half ($50 < x < 100$) is made of glass, $k = 1.5$ W/mK (use silica glass if you use the material libraries or just set the thermal conductivity). Initially, the entire system is at room temperature, $T = 25^\circ\text{C}$. Suddenly, the left edge is raised to 100°C and the right edge is lowered to 0°C .

Use Comsol to solve this problem numerically. Observe the development of the temperature field as the system evolves to steady state. Plot the steady state temperature distribution. Could you have found this steady state without solving the entire transient problem numerically? When the temperature distribution is steady, compute the heat flux at the left ($x = 0$) and right ($x = 1$) edge of the domain (by hand and in Comsol).

Use the “Postprocessing-Plot Parameters” menu to explore the solution. Under the “Line” tab you can define what is plotted in your animation or static plot. Next, move to the “General” tab to make a plot. You can select a time instant with the “Solution at time:” combo box.

Case 3: 2D Transient Analysis

Physical Setup

Solve a two-dimensional transient heat diffusion problem using Comsol. Draw a square box between -1 and 1 in both the x and y coordinate directions. Set the material constants in the “physics; sub-domain settings” so you are solving the diffusion equation with unity constants. Set the initial condition to $T=0$. Set one horizontal wall to have a boundary condition of $T=1$. Set one vertical wall to have a boundary condition of $T=0$. Set the other two walls to be insulating.

Solution

Mesh and solve the problem. You will need to select appropriate resolution in both time and space to achieve a stable solution. Note that under solver parameters you will want to save only 10-20 snapshots of the data or else your animator is likely to crash¹.

Post Processing

Using the post-processing, plot animations of the development of the temperature field. Make an AVI movie showing the time-evolution of both the temperature profile (as color contours) and heat flux (as arrows).

Also make a single plot of the heat flux vectors at the final time. Is this the steady-state heat flux?

Case 4: 2D Steady-State Analysis

This problem is constructing a simple model of heat transfer in a composite material. Solve a two-dimensional steady state heat transfer problem using Comsol. Create 3 by 3 grid of square boxes that are 50 cm on each side. The total region will be 150 cm by 150 cm, however the 50 cm boxes should each be separate regions.

Under “physics; sub-domain settings”, set the material (using the load button) to be concrete or copper. Set the boxes such that you have a checkerboard of concrete and copper. Set the initial temperature to 298 Kelvin. Set the boundary conditions such that the left is held fixed at 400 K and the right is held fixed at 298 K. Solve the problem. You should notice that the inner boundaries are not selectable. You can leave the upper and lower boundaries to be thermally insulated.

Plot the temperature field and the heat flux vectors on the same plot. You can export the plot as an image to document the process.

Another note on behavior

We have come along way from the first simple 1D transient model. You should still be able to use your intuition for this case. Make sure that you understand the model output. Can you explain the resulting plots from what we’ve learned about diffusion?

Ask questions, what would happen if you swapped the checkerboard pattern, moving the copper blocks to the concrete blocks and vice-versa? What would happen if all the blocks were copper? What if all were concrete?

Heat Sink Analysis

Go to <http://www.aavidthermalloy.com>. Search for the part number 62560 to obtain its data sheet. 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.

Start Simple

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. Just like in design, with modeling we should start simple. *Everything should be made as simple as possible, but not simpler* [A. Einstein].

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, just as you did analytically in the fin lab. Initially, assume the convection coefficient is constant and equal to approximately, $h = 10 \text{ W/m}^2\text{K}$.

¹You might consider saving the setup before you start working on the solution!

Compute the thermal resistance from Comsol and compare to the data sheet for natural convection. Vary the hot temperature and see how your resistance changes with 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. If you change the temperature of the hot edge, how does the overall thermal resistance change in your model.

Parameter Identification

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 it will be and therefore will act to dissipate more heat. 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 (about 5) 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, i.e. linearly, square, square root?

Deliverable

Your lab report should document your heat sink model. **You do not need to document working through the tutorial problems.**

You should be able to compare your model results to the empirical data from the data sheets. Do you believe the model? How could you validate your results with more accuracy? What assumptions have you made that might affect the efficacy of your model? The goal is to communicate that you have achieved the objectives of the experience. Be a good consumer of these models by providing discussion of the qualitative and quantitative validation process.