Geology 131: Physical Volcanology
Fall 2015

Lab 13 Thermal Modeling of Dikes

Since the primary means of magma transport in the shallow crust involves movement through dikes and sills, it becomes very important to understand them—how much magma can reach that reservoir, how much lava will erupt onto the surface, how explosive might that eruption become—key questions like these all depend on the rate and state of magma delivery! You have already learned a bit about some of the basic mechanics of dike propagation, but because magma in a dike is usually much hotter than the surrounding host rock there are obvious questions:

- When dike injection occurs, what sort of thermal perturbation does it induce in the surrounding host rock?
- How rapidly does magma in the dike cool, elevating the viscosity to choke off magma ascent or lateral transport, and what physical factors dictate this?
- What does it take to keep dike magma in a state that permits it to flow? There are clear examples of dikes transporting magma across 1000's of kms, for instance…

Addressing questions like these is difficult, and a great deal of energy has gone into research investigations seeking answers to them. New papers come out regularly that refine our understanding, yet the core elements of our knowledge have been in place for some time, and this basic thermal framework is where we will focus our attention today.

This week you will begin by constructing a basic thermal model of a dike in COMSOL, thereby augmenting the mechanical grounding you acquired earlier in the semester. The time-dependent process of dike cooling can be assessed for simple situations using high-precision analytical solutions; however, the process of working through these solutions is time intensive (requiring the use of Fast Fourier Transforms; we cover this currently in Geophysics; for today I will ask you to simply take my word for it that the numerical solutions calibrate beautifully against the analytical ones). In addition, the numerical models provide distinct advantages once you master their operation, as a wide array of plausible, real-world scenarios can be examined which existing analytical solutions can't really address.

Once we have the basic model constructed you will examine and think through the implications for different situations: instantaneous injection, sustained flow, and a flow that varies over time.

Augmenting this, next time you will expand your understanding of dike behavior by using your models to explore how long material flowed in a CRFB feeder dike, a situation constrained by a combination of thermal modeling and mineralogical constraints.

For next class, in the assigned paper:

- Carefully read the Introduction and sections starting on pg. 2307 with "Thermal history and implications of diking," paying special attention to Fig. 14.
- Strive to identify the conditions you'll require for your own model of this situation!
- Skim the remainder for context.

Okay, time to get started!

Start-up Guide: Designing Steady State and Time-dependent Thermal Models to Examine Dike Cooling in COMSOL 5.1

This guide was prepared for use with COMSOL v5.1 and introduces you in a first order fashion to modeling thermal cooling problems with finite element models. It is intended to augment our discussion and work together, and is thus a somewhat detailed reminder of basic steps (rather than a comprehensive guide to modeling in COMSOL). The initial steps will be similar to what you encountered before in the uplift/rupture modeling process, but several new things will be introduced as well.

START COMSOL! (It's on the Mac side of the computer…)

1. **MODEL WIZARD –** tell the software what kind of model you plan to run.
   a. Select **File > New** and then click the **Model Wizard** icon
   b. Select **2D**. This tells COMSOL that we will be examining a single slice through a body that extends a long distance into and out of the plane of the screen. When is this a suitable approximation for a dike?
   c. On the *Select Physics* page, select **Heat Transfer > Heat Transfer in Solids (ht)**; click **Add**, then once Solid Mechanics has been added to the "added physics interfaces" box proceed to the next page by clicking the green **Study** arrow (i.e., Next up: Study settings…)
   d. On the *Studies* page, select **Preset Studies > Stationary**. This tells COMSOL that we will performing a static computation, meaning we are interested in the final outcome, not all the intervening processes.
      i. We will adjust this later on in lab to examine time-dependent variations.
   e. Now that you have told COMSOL what kind of problem style you'll be working with, click **Done** to begin building the specific model itself.

*As we did previously, we'll now move into the Model Builder process. Again, if you ever get too enthusiastic moving windows around and need to reset them, choose **Windows > Desktop Layout > Reset Desktop**.*

2. BUILD YOUR STARTING MODEL—A SIMPLE GEOTHERMAL GRADIENT

We will begin by simply defining a host rock subjected to a geothermal gradient. This will let us see the basic elements of a static thermal model. Once we have these pieces under our belts, we will add a dike to the model and examine time-dependent situation.

In the *Model Builder* window (left hand side) and associated *Settings* windows, proceed as follows, drawing on your experience last week…

a. **Establish Global Definitions**

   i. **RC** (right-click) **Global Definitions** and add **Parameters** then **Variables**. We will return to these and enter whatever is necessary as we progress.

b. **Define Initial Geometry**

   i. **RC Geometry 1 > “Rectangle”** to add a rectangle to the model. This will be our slice of host rock, and we want it to be 10 km wide, 5 km tall, with the surface of the Earth located at y=0 (note: y is negative downwards) and the model will be centered at x=0.

      1. For **Parameters** define **Wth** and **Hgt** and set them to **10[km]** and **5[km]** respectively. Use parameters Wth and Hgt to define your initial rectangle’s physical dimensions.
      2. **Position> Base > Center**, with **x = 0** and **y = -Hgt/2**
      3. **Build All Objects**. Is the Earth’s surface at y=0 and centered around x=0?

c. **Mesh**

   i. For now we will simply use the default mesh settings.

d. **Materials**

   i. **RC Materials > Add Material > Search** for **Granite**

   ii. In the *Add Material* window, open **Built-In**, then **RC Granite > Add to Component 1**.

      1. Verify that Granite has now been added under Materials in the *Model Builder*.
      2. **Close** the *Add Material* window.

   iii. Under Materials > Granite > Basic we want to make the thermal conductivity a parameter we control.

      1. Create a **Parameter** called **thcon** and set it equal to 2.9287[W/m/K] (Watts per meter per Kelvin)
      2. In the *Settings* window, enter **thcon** as the **Expression** for **Thermal conductivity** (replacing a fixed value of 2.9 W/m/K).

e. Establish Heat Transfer Boundary Conditions

The starting default is to have all four walls as ‘thermal insulators’ (i.e., no heat flux across the boundary) and an initial value for the material of 293.15[K]. We will need to change these settings to create a thermal gradient.

   i. First, establish the thermal gradient conditions.

      1. We will assume a gradient of **30 K/km**; create an associated **Parameter** called **grad** and assign it this value.

2. Create a new **Variable**, **tgrad**, and define it using the expression **293.15[K]-grad*y**. This will ensure that temperature increases from 293.15 K at the surface (y=0) to larger temperature values with depth.

ii. **RC** on **Heat Transfer in Solids (ht) > Temperature**. Then, **do so again**. You should now have Temperature 1 and 2 in the list under Heat Transfer in Solids (ht).

1. Select **Temperature 1**, which we'll use to set our surface temperature.

a. Under *Settings*, activate the Boundary Selection, then **select the top surface** of the rectangle (#3).

b. We'll leave the temperature of this surface as the default 293.15[K]

2. Select **Temperature 2**, and assign it to the base of the model (#2). Now, though, we need an elevated temperature as the geotherm has increased with depth.

a. Delete the 293.15[K] and replace it with **tgrad**. What does this do, and why are the signs the way they are?

iii. Select **Heat Transfer in Solids > Initial Values** and in the Settings window enter *tgrad* for the temperature, replacing the default value of 293.15[K].

f. **Save** your model, then **Run** it to examine the results.

i. If everything is correct, you'll see a plot of temperature appear on the right hand part of the COMSOL window! The default colors are a little non-intuitive, but you can change them easily if you wish to do so…

ii. Your plot should show a 150 degree variation from top to bottom, as expected since we have a thermal gradient of 30 K/km and went down 5 km!

iii. Note that there are many other things we could plot, we'll see a few later.

Again, save your geotherm model. The result isn't that exciting, but we're going to build on it soon in ways that should prove more interesting!

Now it is time to add a dike to the model. We'll start by redefining things a little bit so that we can have the dike in a host of uniform temperature—directly akin to an analytical FFT analysis (and in this case y=0 is not the 'free surface of the Earth' it is just an arbitrary depth boundary)—but then we will quickly move beyond this to explore conditions that an analytical approach, like the FFT, really can't provide insight into…

3. **DIKE IN A UNIFORM TEMPERATURE HOST MATERIAL**

   a. **Eliminate Thermal Gradient in Host Rock.**

      i. **Heat Transfer in Solids > Temperature 1 > RC** and select **Disable**

      ii. **Repeat** process for **Temperature 2**

      iii. **Heat Transfer in Solids > Initial Values 1** > set back to **293.15[K]**

   b. **Add Dike to Model**

      i. Create a new feature, a dike that extends from the top of the model region to the bottom, and make it 100 m thick.

         1. Create **Parameters dw** (dike width), **dh** (dike height) – what should their values be? – and **dtemp** (set to 1293.15[K]).

         2. Create second **rectangle** under Geometry, with appropriate geometry (it will be the dike) and placement (centered on x=0, extends same height as host rock).

      ii. Under **Heat Transfer in Solids (ht)**, add a second **Initial Value** and use it to set the starting temperature of the dike using **dtemp** (be sure to apply only to the dike domain, #2).

   c. **Add Time-Dependent Physics**

   Now, since we are interested in how the dike cools we need to look at the temperature distribution in the dike and host rock as a function of time. A Stationary solution will no longer be sufficient, we need to introduce some time-dependent physics.

      i. **Save** your progress thus far before proceeding.

      ii. On the **icon menu bar** at the top of your *COMSOL* window you'll see a drop-down menu called **Study 1**, and **<u>next to it</u>** a symbol that looks like a **pair of eyeglasses** (at least to me!). If you hover over the latter you'll see "Add Study" – **click on this icon to add a new Study**.

         1. In the *Add Study* window that opens select **Time Dependent** and then click **+ Add Study**.

            a. In the *Model Builder*, you should now see a new item called **Study 2**.

            b. With this in place, we won't need Study 1 any longer, so **RC** on **Study 1** and choose **Delete**.

            c. **Re-save** your file, but **give it a new name**… just in case!

      iii. Now, we need to pick some times to solve for.

         1. Open Study 2 and select **Step 1: Time Dependent**.

         2. In the *Settings* window

a. Set the **Times** as **range(0,10[yr],100[yr])** – this expression says we will calculate solutions at t =0, t = 10 yrs, t = 20 yrs, … t = 100 yrs.

iv. **Run** Study 2

d. Examine Results—Surface Plot

i. The result should be a plot of temperature within the model area. Note now, however, you can plot the data for any particular time you requested.

1. Pick a time (years, expressed as seconds unfortunately), then Re-plot. By doing this as a function of time you can see how the thermal perturbation from the dike spreads over time.

ii. If you want to fix this display so that it reports years instead of seconds

1. go back to **Study 2 > Step 1** and **set Time unit to "a"** (for annum)
2. eliminate the [yr] markers after the range times (so this becomes **range(0,10,100)**) then rerun the model.
3. Now when you view the results, you'll see values expressed as years… more logical and easier to keep track of too!

e. Examine Results—Graphing the Data

i. **RC Results > 1D Plot Group**. Then, **RC** the new **1D Plot Group > Line Graph**.

ii. Before we can plot anything, we need to tell COMSOL where our data are located. In case there are edge effects, we'll need to create a line for our measurement purposes that slices left to right across the middle of the dike. The best way to do this is to add a new line along which nodes can be placed by COMSOL.

1. **RC Geometry > Bezier Polygon**.
2. Under **Polygon Segments**, select **Add Linear**, then use the Control points to define the ends of the line, which should run from, for Control Point 1 x and y values (**–Wth/2**, **-Hgt/2**), to Control Point 2 x and y values (**Wth/2**, **-Hgt/2**). When you have entered these points, click **Build Selected** to verify that the line plots where expected.
3. **Run** the model.

iii. Under **Results** select **Line Graph 1**

1. In the Line Graph Settings window, add the three line segments from the Bezier Polygon you drew to the Selection box (Activate, etc… should be segments 4,9,14).

2. Leave y-axis data as the variable T (temperature), change the **x-axis** data from Arc-Length to **Expression**, then set the expression to **x**.

iv. Now, **Plot** the data.

1. You will want to add a **Legend**! Experiment with the plot settings until you discover how to do this, or ask me if you're stuck.
2. Experiment as well with plotting whichever years you want to see!

The plot shows the temperature distribution calculated for the dike. You'll see that there are some numerical errors right at the wall of the dike (temperatures drop below the background 293.15[K] level) – these occur in the analytical solutions as well and stem from the fact that (a) we are looking at equations that define high flux (high gradients across distances smaller than elements causes problems), and (b) we are solving a continuous equation by breaking it up into discrete elements, and can only calculate the values at the nodes of the mesh elements to determine a value at the center. The coarser the mesh, the more significant (and persistent) these will be. To see this effect, select **Mesh**, and set the resolution to **Extremely Fine**, then re-**run the model**. Note that this means there are a lot more elements and hence calculations (we went from **~2000** degrees of freedom **to ~26,000**), so it takes longer, but it should still be fairly quick. Then, re-examine the line graph. Note now that fewer times are showing that numerical error, that it is less drastic, etc. By the 100 yr mark, the temperature within the interior of the dike has dropped to roughly 725 K.

The situation you have just examined is one of "instantaneous injection" and then cooling of dike material. The magma intrudes very rapidly into the host and then ceases to advect heat in with new magma flow through the dike, so the magma then cools progressively toward the background temperature. It is a simple (and geologically plausible) scenario, one well handled analytically, but what if we wanted something other than a uniform background temperature, say one that intruded instantaneously into a rock affected by a geothermal gradient? Suddenly the analytical solutions become difficult or even intractable—yet we can simulate it quickly and easily using numerical methods…

4. Thermal Cooling of a Lateral Dike with Geothermal Gradient

a. Redefine Dike Geometry

i. Return to the geometry of the dike and redefine it so that Rectangle 2 is only 2000 m high but with the same center (0, -Hgt/2) and width; you can think of your geometry now as a simplified slice through a laterally propagating blade-like dike.

b. Reinstate Geothermal Gradient

i. **Enable Temperature 1 and 2**, reapply Selections so that only the surface of the Earth is selected for Temperature 1 and the base of the model for Temperature 2.

  1. Each time you add new geometric elements to a model, or remove them, expect that the line segments etc. get renumbered, so check what you are plotting, what you are applying boundary conditions to, etc.!

ii. Redefine **Initial Values 1** as **tgrad**, and **Initial Values 2** as **dtemp**, ensuring that they are applied to the correct domains (host rock for 1, dike for 2)

iii. **Save**, then **Run** Study 2.

  1. The impact of the thermal gradient is fairly small given the size of the dike, but if you run the model for sufficient time you'll see that **the thermal signature is decidedly asymmetric**, reflecting the fact that the base of the dike is adjacent to warmer host rock than the top of the dike.
  2. Experiment with the time range until you feel you have a good handle on how the dike is going to cool as a function of time.

Okay, now that you have the idea, and have seen how your numerical model can quickly be adapted to examine situations difficult or impossible to address analytically, let's look at a more realistic dike, say one that is 5 m across (still pretty big!), and figure out how well our numerical model performs.

5. Refining Your Numerical Solution and Improving Visualization Control

a. Eliminate Geothermal Gradient

i. Begin by stripping out the geothermal conditions once again; we want here to duplicate conditions that can be addressed in analytical model solutions in order to see how well the model actually does.

  1. **Disable Temperature 1 and 2**, reset **dh** to **Hgt**, redefine **Initial Value 1** as **293.15[K]**. B model to verify that you have things set as expected.

b. Re-size model

i. Now, **reduce** the **height** and **width** of the model plus the **width of the dike** by a **factor of 20** (adjust your **Parameters**).

c. Reset Times Examined

i. Under **Step 1: Time Dependent** set your **Time Unit** to "**d**" (days; look at your other choices) and the times as **range(0,5,60)**.

d. Reset Mesh

i. Set the Mesh resolution back to **Normal**.

e. **Save**, then **Run** Model

f. Initial Visualization of Experimental Data Results

i. Go to your **1D Plot Group**, pick **30** days (1 month) from the list as your time selection, then **Plot**.

ii. What you'll see is that the dike temperature maxes out just a little below 700°C (you can **change the y-axis unit on the plot** to make this easier to see, figure out how!), a few degrees below the actual value of ~703°C obtained from analytical models.

iii. Thinking about what you learned above, why might this number be lower than the analytical result? **Discuss your ideas with those near you, then with me before proceeding.**

g. Improving Data Visualization

Let's experiment a bit with different ways to control visualization of the data your model is providing. This is usually worth taking a bit of time to do, as it can enhance your ability to think about what you are seeing significantly. There are lots of "tricks" one can employ here, so these are just examples, and as you continue to model different problems over time you may discover others.

i. <u>Fix Color Range</u>

1. Click **Results > Temperature** and **vary the Time (d) plotted**, observing the depiction in the *Graphics* window.

   a. You'll notice that the range of colors remains constant, but as time advances the values represented by these colors changes.

      i. At 5 days the darkest red depicts a value a little in excess of 1300K, but by 30 days it represents a bit more than 960K instead.

      ii. This might be just how you want to see things, but what if you want to keep the colors steady, so each color is assigned to a fixed temperature?

2. Click **Surface 1 > Range** and you'll see that you can actually take manual control of the color assignments (**Color Range**) and the data plotted (**Data Range**). Try activating Manual <u>Color</u> Range and set the minimum to 293.15 and the maximum to 1293.15

3. Return to **Temperature** and **flip through the times again**. See the difference? This will be better for some purposes and worse for others, but the key is that you have control to make things display as needed.

ii. <u>Add Contours</u>

1. Maybe you'd like to see contours instead of just a shaded temperature plot. Under **Results**, **RC Temperature > Contour**.

    a. In *Settings*, set **Entry method** to **Levels** and enter a single value that makes sense given the temperatures displayed on your plot (try 350 or 325).

    b. Set **Coloring** to **Uniform** and **White** (or whatever works given your background colors), then **Plot**.

        i. You can also set a range of values by clicking on the button to the right of Levels and entering a Min, Max and Step size. Try it!

2. You might also decide that the contrast here with your background surface image stinks, so you'd like to change that too.

    a. If you select **Surface 1** and look at the options there you can again adjust the **color scheme**.

    b. My favorite **Color Tables** to use are the **Rainbow** and **Rainbow Light** tables, set so that, intuitively, high temps are red and low temps are blue.

iii. <u>Limit Data Displayed</u>

Perhaps you're interested in a geological phenomenon for which only parts of the data are necessary. For instance, by the time lava drops roughly 75 degrees below the liquidus, crystal content has often become so high that the elevated viscosity will make continued flow very difficult (barring unusual circumstances) in a lava flow or tabular intrusion. It would be great if we could depict just those areas where the rock is above the expected temperature. There are a lot of ways to do this using COMSOL's syntax, but here are a few examples.

1. **Results > Temperature > Surface**. For the expression, **replace T** with

    a. if(T>(dtemp-75),T,nan)

        i. This says that **if** the first part of the expression is met (i.e., if T is above dtemp-75), **then** T is plotted, **otherwise** "nan" (not a number) is plotted instead, i.e. no data.

        ii. Plot for time t=0, to see the outcome.

        iii. You may need to refine your times to get cleaner results (don't do this at present).

        iv. Refining the mesh will be needed to get really good data (again, don't do this at present).

b. T*(T>(dtemp-75))

i. The portion in parentheses is a **Boolean** expression, i.e. the expression in parentheses is a True/False check, returning a **value of 1 when true** and a **value of 0 when false**. This yields a similar result, since the temperature T is multiplied by either 1 or 0, with the former passing through the actual temperature data while all other values are zeroed out.

ii. Try using this as the y-axis expression for the 1D Line Graph, and notice how the fluid portion of the dike, the part capable of flowing, narrows quickly with time. It remains viable for somewhere between 5-10 days.

iv. <u>Explore!</u>

Set the **Mesh** to **Extemely Fine**. Then, using the display options you have learned, vary the time scales you model and try to estimate **how long it takes for the entire 5 m dike to cool to 100 degrees above background** (~400 K) temperatures (hint: long time!).

Look as well at the temperature patterns in the host rock. What is the hottest the host rock can get, and where does it happen? Why might having this temperature constraint be a problem, and what are the implications for long-term dike transport of magma? Discuss with me (or with someone who has already discussed it with me) before proceeding.

To complete our lab work today, and prepare a model you will use next class period, we want to explore a significantly more realistic scenario: one in which magma in the dike keeps flowing for a period of time before the source of magma (and hence the pressure driving magma flow) stops advecting fresh material, hence heat, through the dike. Put another way, we will hold the dike temperature constant for some window of time during which the magma is flowing through it, after which we will let the entire system (dike and host rock) cool. Think through what you expect the temperature profile to do relative to the instantaneous injection case just examined above, and sketch a graph (schematic is fine) recording your hypothesis. Then, let's try modeling it…

6. Steady Magma Flow in Dike, then Cessation of Flow and Cooling

To date you have applied initial conditions (e.g., *dtemp*) and boundary conditions (e.g., setting the surface temperature to 293.15K), but **what is needed now is a time-dependent control** on the temperature in the dike. In essence, we'd like to keep the dike at *dtemp* for a time period, in this instance we'll say 30 days, after which hot material isn't constantly resupplied and so the magma in place will cool over time to the background temperature in the model.

a. **Enable Advanced Physics**

   i. On the *Model Builder* window, find the **icon at the top** that looks like an **eye with a line over it** (hovering over it will reveal that the button is called **Show**). **Click** this button, and **select** (enable) **Advanced Physics Options**.

      1. This will provide us with some extra flexibility when setting conditions within domains, on boundaries, etc.

   ii. **RC Heat Transfer in Solids (ht) > More** (make sure you do this for the **Domains, i.e. the filled ovals**, not for the Lines (oval with line through it) or Points; see me if you are at all uncertain about this!) and select **Pointwise Constraint**.

      1. This in essence will let us write our own equations to dictate conditions within a domain.

   iii. For the Pointwise Constraint *Settings*

      1. select the two dike segments (#3,4)
      2. For constraint expression, type: **(T–dtemp)*(t<=30[d])**.
         a. Like the Boolean expressions used earlier, this is a specialized notation specific to Pointwise Constraints.
         b. The first parentheses in essence says "**assign T the value dtemp**".
         c. The second set of parentheses again defines a conditional statement. It adds "**provided that time t is less than or equal to 30 days**".
         d. As long as the condition is met, T will be held at dtemp, but afterwards the constraint vanishes and the system is left to thermally decay on its own.
         e. So, putting together the two pieces of the expression, we hold the dike temperature fixed at dtemp for 30 days (hot magma flows through it for 30 days) and then the temperature is allowed to decay with time.

   iv. Make sure your **Time-Dependent run time exceeds 30 days**, maybe use something like range(0,1,50) with Time Unit *d*.

   v. **Save** and then **Run** the model.

   vi. Review the Line Graph of the results. **Zoom in to the higher temperatures**. If all went well, you should see temperatures holding steady at *dtemp* for about the first 30 days and then decaying thereafter.

1. Note that numerical errors will occur at short time scales. Just like the triangular mesh elements smear a solution somewhat across a space, COMSOL must pick times to calculate solutions to ensure a clean answer at the times requested, which smears things a little in time. This causes the dike temp to release a little earlier than expected, on the order of 2 days early in this run.

b. **Refine Mesh in a Targeted Fashion to Explore How to Improve Solution**

i. **RC Mesh > More Operations > Edge**.

ii. **RC Edge** and select **Size** (which will appear under it)

1. For Edge 1, in *Settings*, from the edge-to-edge horizontal line segment **select** only the part that lies within the dike (**#9**). Since that's where we need higher resolution, we'll target just that little bit specifically.

iii. **Edge > Size 1 > Custom** for **Element Size** in *Settings* window.

1. Check **Maximum element size** and **Minimum element size** and enter **dw/32** for both. This says, in essence, that elements abutting that small line segment must be 1/32 the dike width in size, a much higher resolution than the single element we had before!

iv. RC Mesh > Free Triangular

1. If necessary, **RC Edge > Move Up** (so that Free Triangular is at the bottom of the meshing stack). The final sequence should be Size, then Edge 1, then Size 1, then Free Triangular 1.

v. **Build All**, then zoom in to examine the result. Notice that the advantage of this strategy is that we don't add really high resolution everywhere, advisable since high resolution slows down our calculations immensely. We've just targeted the spot where that extra precision may be of benefit!

c. **Save**, then **Run** the model.

d. **Re-examine the graph**…

i. See how the signature at the top end has cleaned up a great deal (still stuck with the t=0 "horns" error – as is the analytical solutionI might add)?

ii. Now, look at the information recorded closer to the edges of the dike. How does what you are seeing compare with your initial hypothesis?

iii. What are the implications for the temperature in the surrounding host rock?

There is a lot more we could do, but we'll call it quits here in terms of introducing new materials. **Save your file and place it someplace accessible for next class period**, then continue exploring and asking questions until you are satisfied you have a handle on what is going on.