MAE3187 Computer Project
Numerical Solution of Two-Dimensional, Transient Temperature Distribution in a Rectangular Domain

Due Date: April 5, 2012

Purpose

The purpose of this assignment is to apply the finite difference numerical solution technique to solve the transient heat conduction equation in two dimensions, given by

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{g}{k} = \frac{1}{\alpha}\frac{\partial T}{\partial t} \tag{1}$$

where T is the temperature, x and y are the physical coordinates of the domain, t is time, and α is the thermal diffusivity of the material (in m^2/s). Note that $\alpha = k/\rho C_p$, where k is thermal conductivity, ρ is mass density, and C_p is specific heat of the material subjected to heating. Equation (1) represents a linear, second-order, parabolic partial differential equation (PDE). Note that internal volumetric heat generation (g) would, for example, be caused by passing electric current through the material. Also note that a common mistake is not to include the heat generation rate in the proper units (recall that it represents energy per unit volume). In the numerical solution g is known and can be different at different locations of the block. For our case, we assume g to be a constant and same at every location.

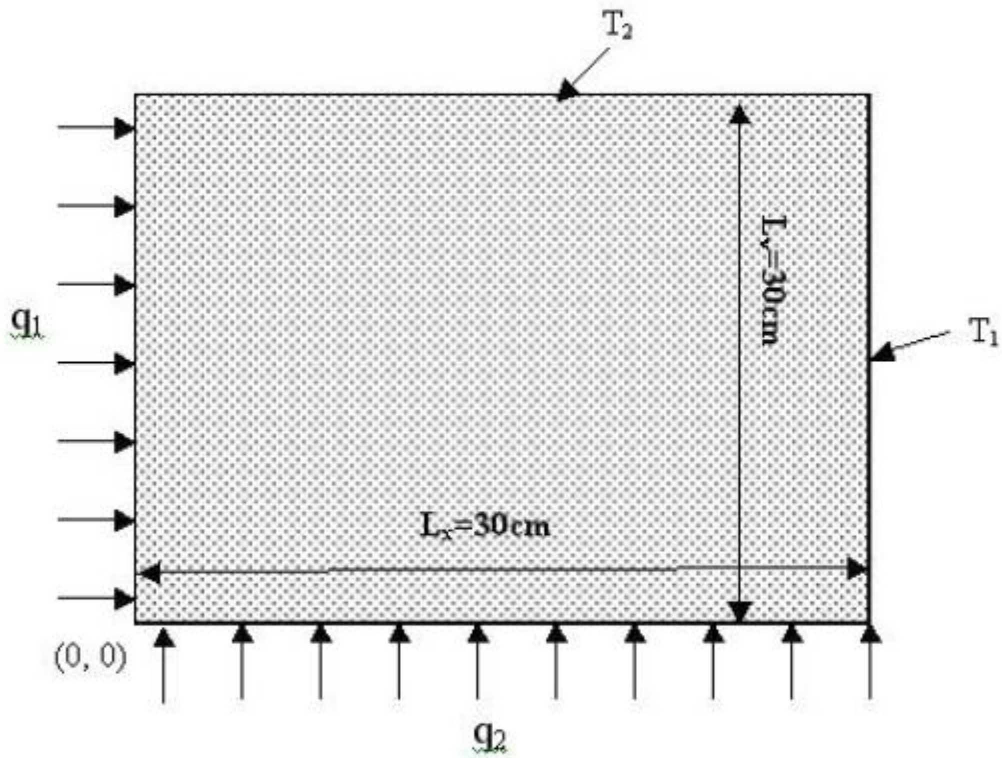
Please use an **explicit** time differencing approach to find the temperature distribution at different time values. Such explicit schemes introduce the concept of numerical instability to computer solutions. That is, if the time step of the computations is not chosen properly, the error in numerical calculations become unbounded and gives unrealistic answers, eventually causing the program execution to abort due to numerical overflows. After the completion of the general programming, two cases must be conducted by varying the time step in the problem:

- A stable solution will be obtained for the transient temperature distribution in a rectangular domain as described in the problem below.
- A simulation for a deliberately unstable case will be performed to demonstrate the effect of improper time step choice.

Problem Definition

The physical problem to be solved in this assignment is as follows:
An aluminum plate (30 cm x 30 cm x 2 cm) is being cooled along two adjacent sides with chilled water. You may assume that the temperature of these sides is being maintained at T_1 and T_2 (choose reasonable values). The other two sides are being heated with strip heaters as shown in the figure below. Initially, the aluminum plate is at room temperature, T_i before these boundary conditions are imposed. As the plate has thermal inertia, the temperature distribution in the plate will change with time and eventually

reach steady state. Find the temperature distribution after 90 seconds, 360 seconds and at steady state. Use q_1 = 100W/ cm^2 and q_2 = 50 W/cm^2 and g =1 W/cm^3.



Initial Condition

$t=0$ $\quad T(x,y,0)=T_i$

Boundary Condition along the four sides

$x=0$ $\quad q_1=-k\dfrac{\partial T}{\partial x}(0,y,t)$

$y=0$ $\quad q_2=-k\dfrac{\partial T}{\partial y}(x,0,t)$

$x=L$ $\quad T(L,y,t)=T_1$

$y=L$ $\quad T(x,L,t)=T_2$

Notes

For the present problem, one needs the initial temperature distribution at the beginning of the time domain due to the parabolic nature of the transient term. Your initial condition will affect your transient solution but not the steady state solution. The thermal diffusivity for the material should also be specified. Remember to use the same units for time and thermal diffusivity. As the time increases in a transient problem, the temperature approaches the corresponding steady-state solution. This is obvious from experience, as everyone knows that a hot frying pan eventually cools down to room temperature (the boundary condition) after removing it from the stove. You will utilize this fact to find the steady state solution.

Solution Procedure

The computer program will show the variations of the temperature within the plate as a function of time. It should be noted that the structure of the program will use the explicit procedure of marching in the time domain. Thus each step will represent distinct temperature solutions in time. The basic procedure in computing the temperature distribution is to find the temperature at each node by making use of the temperature values at the neighboring nodes as well as that of the same node at the previous time level *p*. The expression for an internal node for the case when $g = 0$ is given by

$$T_{i,j}^{p+1} = T_{i,j}^{p} + \alpha \Delta t \left(\frac{T_{i+1,j}^{p} - 2T_{i,j}^{p} + T_{i-1,j}^{p}}{(\Delta x)^2} + \frac{T_{i,j+1}^{p} - 2T_{i,j}^{p} + T_{i,j-1}^{p}}{(\Delta y)^2} \right) \tag{2}$$

where p represents time *t*, and *p*+1 represents time at the next level *t*+*dt*. Note that you will have to store temperatures in two arrays. One for the temperatures at time *p* and one at time *p*+1. As mentioned earlier, the explicit scheme represented by Equation (2) introduces a question of numerical stability. That is, improper choice of the time step Δt will result in unbounded numerical error. To ensure numerically stable solutions, the magnitude of the time step for any node not on the boundary must be limited by

$$\Delta t \le \frac{1}{2\alpha} \left(\frac{1}{\frac{1}{(\Delta x)^2} + \frac{1}{(\Delta y)^2}} \right) \tag{3}$$

However, note that in the case of convective and insulated boundary conditions, additional stability restrictions may arise. Hence it is sensible to evaluate the restriction on time step on the boundaries and use the most limiting value. For the stable case, use a time step which is 90% or less of the limiting value. This provides a margin of safety to prevent problems from round-off and truncation errors. For the unstable case, use a time step that is at least twice the limiting value.

Submit

1. Justification for performing two-dimensional analysis for a three dimensional object.
2. Your equations for the four boundaries, at corners if applicable and for the interior nodes.
3. Restriction on time step due to the boundary conditions using energy balance.
4. One way to verify that your numerical solution is correct is to refine your mesh and see if your solution converges. Hence first use the number of nodes in *x*- and *y*- direction to be M=N=31 and then re-run your program for M and N =61. In both cases plot the graphical temperature distribution after 90 and 360 seconds when $g = 0$ and for $g = 1$ W/cm^3. Comment on your results. Note that when you

decrease your *dx* and *dy*, you need to reduce your time step accordingly to meet the stability requirement.

5. The mesh refinement will only tell you about convergence but how can you ensure that it is converging to the correct physical solution? One way to verify this is conduct a global energy balance. Do this on your steady-state solution for g=0 and g=1 W/cm^3. The approach to this is to take the temperature distribution that you obtain from your numerical solution for steady state and conduct energy balance for the entire block by calculating the energy coming in from the boundaries (You already know the q for two boundaries as that is the boundary condition and as you know the temperature distribution -you can use the Fourier law to find the energy leaving or entering from the other two boundaries where a constant temperature BC is applied). The sum of all this has to be equal to the energy produced within the block under steady state conditions. List the error for each case for both meshes.
6. Graph the steady state temperature distribution and the minimum time it takes to reach the steady state for g=0. This can be done by allowing the stable solution to proceed for a sufficient amount of time such that the temperatures from one time step to the next are essentially the same.
7. Demonstration of the numerical instability (and the concept of garbage in-garbage out) by choosing a large time step and printing out the temperature distribution at t = 80 seconds for g = 0.