Programming Assignment 2

Mech 510

Due: Monday, November 9

In this assignment, you will write a program to solve the wave equation numerically. Specifically, your code should solve

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} = 0$$

$$0 \le x \le x_{\max} \qquad 0 \le t$$

$$u = 2$$

using second-order upwind flux evaluation and two-stage Runge-Kutta time advance. Use $x_{\max} = 1$, $T(x,0) = -\sin(2\pi x)$ as an initial condition and $T(0,t) = \sin(4\pi t)$ as a boundary condition. The exact solution to this problem is

$$T(x,t) = \sin(2\pi(2t - x)).$$

1. *Verification.* Implement the upstream boundary condition as follows:

   - For $i = \frac{3}{2}$, compute the flux using second-order upwind using linear extrapolation to find $\overline{T}_0$.
   - For $i = \frac{1}{2}$, use the analytic boundary condition to find the flux.

   Compute the solution at $t = 1$ using a CFL number of $\frac{u\Delta t}{\Delta x} = 0.4$. At this time, your code should have propagated the wave completely across the domain twice. Plot the error versus x for a mesh with 20 control volumes and a mesh with 40 control volumes (on one plot). Are you satisfied that your program is correct and has the order of accuracy that you expected? For what mesh size does the L_2 error norm drop below 10^{-3}? Below 10^{-4}? (You should be able to determine these within a couple of cells using simulations with no more than 10 different mesh sizes.)

2. *Stability.* Extend your domain to $x_{\max} = 20$; use an initial condition of

$$T(x,0) = \begin{cases} -x & 0 \le x \le 1 \\ 0 & \text{elsewhere} \end{cases}$$

   and the same boundary condition as in part 1. Compute the solution at $t = 8$ on a mesh with 500 control volumes. By using a range of time steps, experimentally identify the maximum stable time step (to within ± 0.005 for CFL number). On the same graph, plot the solution at $t = 8$ for one case that is definitely stable, one that is slightly unstable, and one that you think is exactly on the borderline. Explain in your report what evidence in your solution convinces you which cases are stable versus unstable. What is the smallest number of time steps (largest Δt) that you believe is stable? Is this consistent with analysis of this space and time discretization?

3. *More Scheme Combinations.* Return to a domain size of $x_{\max} = 1$. Modify your code so that you can also run the explicit Euler time advance method and the first-order upwind flux scheme. This will give you four combinations of space-time discretization: each of first- and second-order upwind, with each of explicit Euler and RK2.[1] For each space-time scheme, use a CFL number that is 80% of the maximum value for the scheme to be stable.

   Compare these four schemes by:

   (a) Plotting, on a single graph, the L_1 and L_∞ norms of error versus t for 80 control volumes from $t = 0$ to $t = 1$.

   (b) Plotting, on a single graph, the convergence of the L_1 and L_∞ norms of error at $t = 1$ with mesh refinement (and therefore time step reduction).

   Discuss the effect on accuracy of changing the space and time discretization schemes, and the implications this has for real CFD simulations.

[1] This will be *much* easier to do if your code is modular enough that you can invoke a single time step of a given time advance scheme as a subroutine, and if you have your flux integral written as a separate subroutine that your time advance scheme can call.