AST 381 Homework 2: Hydrodynamics

Nina Filippova

January 27, 2022

1. Not so fast! *Use a perturbation analysis to show that the leapfrog scheme (time-centered difference),*

$$\rho_j^{n+1} - \rho_j^{n-1} = -v_0 \frac{\Delta t}{\Delta x} \left(\rho_{j+1}^n - \rho_{j-1}^n \right), \tag{1}$$

is conditionally stable, i.e., it is stable as long as $\Delta t \leq \Delta x / v_0$. Assume that v_0 is a constant velocity.

Under a Fourier transformation, $\rho_j^n = \xi_k^n e^{ikj\Delta x}$. Substituting this transformation into Eq. 1, we obtain

$$\begin{aligned}
\xi_k^{n+1} e^{ikj\Delta x} - \xi_k^{n-1} e^{ijk\Delta x} &= -v_0 \frac{\Delta t}{\Delta x} \left(\xi_k^n e^{ik(j+1)\Delta x} - \xi_k^n e^{ik(j-1)\Delta x} \right), \\
\xi_k^{n+1} - \xi_k^{n-1} &= -v_0 \frac{\Delta t}{\Delta x} \xi_k^n \left(e^{ik\Delta x} - e^{-ik\Delta x} \right), \\
\xi^2 - 1 &= -2i\xi v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x).
\end{aligned}$$

Solving this quadratic equation for ξ, we find

$$\xi = -iv_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) \pm \sqrt{1 - \left(v_0 \frac{\Delta t}{\Delta x} \right)^2 \sin^2(k\Delta x)}.$$

The magnitude of ξ – the error amplification factor after time step Δt – is

$$\begin{aligned}
||\xi|| &= \sqrt{\left[1 - \left(v_0 \frac{\Delta t}{\Delta x} \right)^2 \sin^2(k\Delta x) \right] + \left(v_0 \frac{\Delta t}{\Delta x} \right)^2 \sin^2(k\Delta x)} = 1, && v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) < 1, \\
||\xi|| &= 1, && v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) = 1, \\
||\xi|| &= \sqrt{\left(-v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) \pm \sqrt{\left(v_0 \frac{\Delta t}{\Delta x} \right)^2 \sin^2(k\Delta x) - 1} \right)^2} > 1, && v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) > 1,
\end{aligned}$$

so

$$||\xi|| \leq 1 \quad \text{if} \quad v_0 \frac{\Delta t}{\Delta x} \sin(k\Delta x) \leq 1.$$

The maximum values of $\sin(k\Delta x)$ is 1, so this numerical scheme is stable if $\Delta t \leq \Delta x / v_0$.

2. Sod it! *Using the finite volume discretization given in class, and a programming language of your choice, write your own 1D hydrodynamic solver.*

(a) *The quintessential shock benchmark problem is the "Sod Shock Tube". The problem involves setting up two discontinuous states at $t = 0$: a hot, dense gas on the left and a cool, low-density gas on the right. The result is a shock wave that travels left to right into the cold gas, and a rarefaction wave that travels into the dense gas on the left.*

Setup: *Let the domain be [0, 2], and set $\rho_1 = 1.0$, $p_1 = 1.0$ for $x \leq 0.75$ and $\rho_2 = 0.125, p_2 = 0.1$ for $x > 0.75$. Assume $\gamma = 1.4$ and use this to determine the specific energy. Set $v = 0$ everywhere initially. Use symmetry boundary conditions and $N = 200$ grid cells. Set the time step according to a CFL number of 0.5. Implement an artificial viscosity of the following form:*

$$\begin{aligned}
q_{i+1/2}^n &= \left[q_0 (u_{i+1}^n - u_i^n)^2 - q_1 (u_{i+1}^n - u_i^n)^2 \right] \frac{c_{s,i+1/2}}{\bar{\rho}}, && \left(\frac{u_{i+1}^n - u_i^n}{x_{i+1}^n - x_i^n} \right) < 0, \\
q_{i+1/2}^n &= 0, && \left(\frac{u_{i+1}^n - u_i^n}{x_{i+1}^n - x_i^n} \right) > 0,
\end{aligned}$$

where

$$\bar{\rho} = \frac{1}{2}\left(\frac{1}{\rho^{n+1}_{i+1/2}} + \frac{1}{\rho^{n}_{i+1/2}}\right), \qquad q_0 = 4, \quad q_1 = 0.5.$$

Compute the solution at $t = 0.245$ and make plots of the density, velocity, pressure, and specific energy as a function of position. Compare the shock properties (speed, density jump, pressure jump, velocity) with that predicted for these conditions.

Analytic shock properties were obtained using the Python `sodshock` library. Assuming a shock width of a few Δx, the density jump, velocity jump, and pressure jump were computed using the values of the grid points several points to the left and right of the shock position (indicated as dashed vertical lines in the plots below).

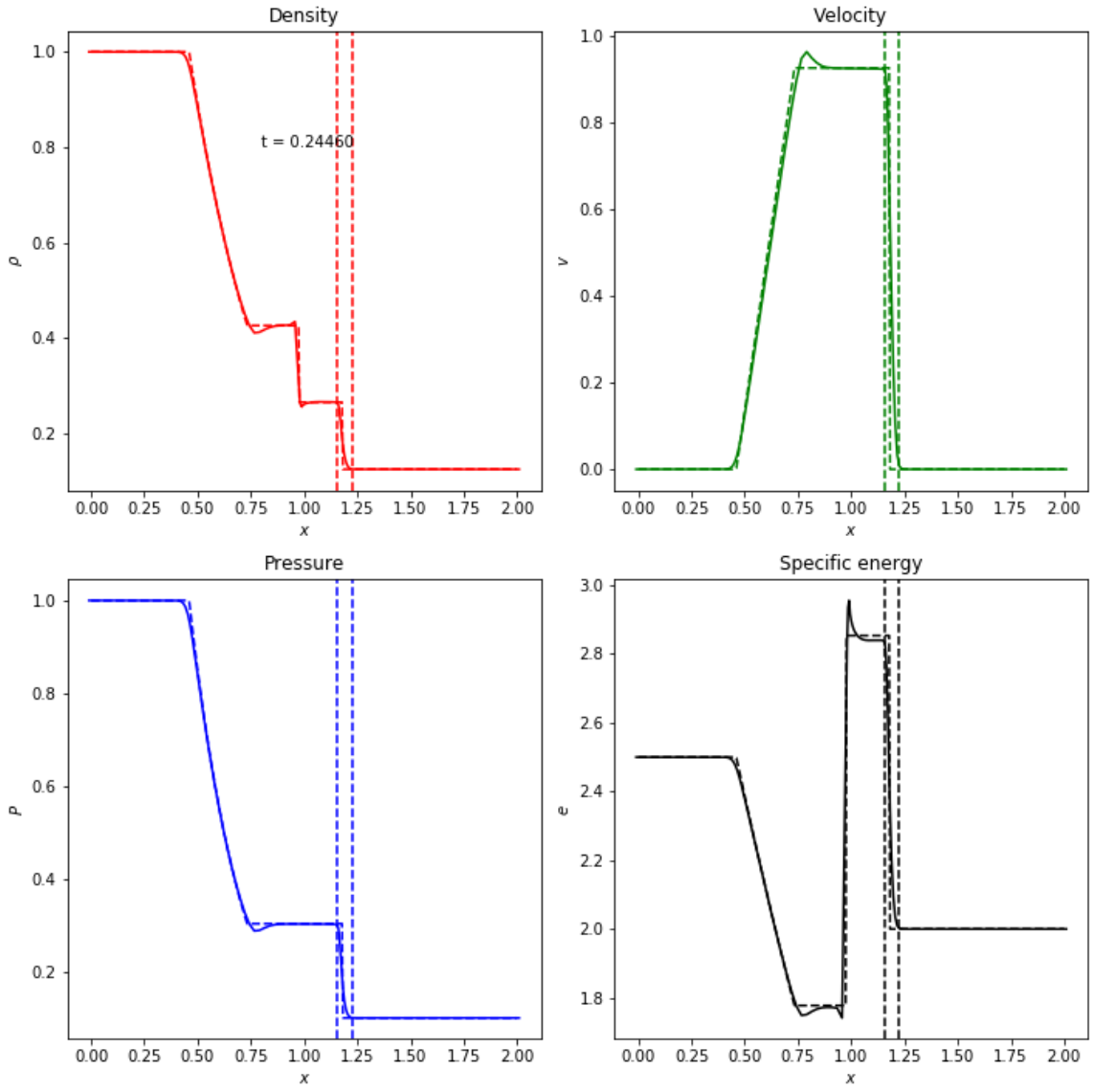

Figure 1: Density, velocity, pressure, and specific energy for $N = 200$, $q_0 = 4.0$, $q_1 = 0.5$, and $C = 0.5$ at $t = 0.245$.

(b) *Simple numerical schemes are prone to certain artifacts near the shock interface. What deviations from the analytic solution do you notice? What numerical improvements would you apply to address them?*

The numerical density, velocity, pressure, and specific energy over/undershoot the analytic solution, though this is more visible at the positions of the initial contact discontinuity and the boundary of the rarefaction region. At the position of the shock wave, the shock front is somewhat smeared. A scheme with lower truncation error may reduce the smearing of the shock wave.

(c) *Check the shock solution for $N = 100$, 200, and 400 grid cells. How quickly is the solution converging? Is this consistent with your expectations?*

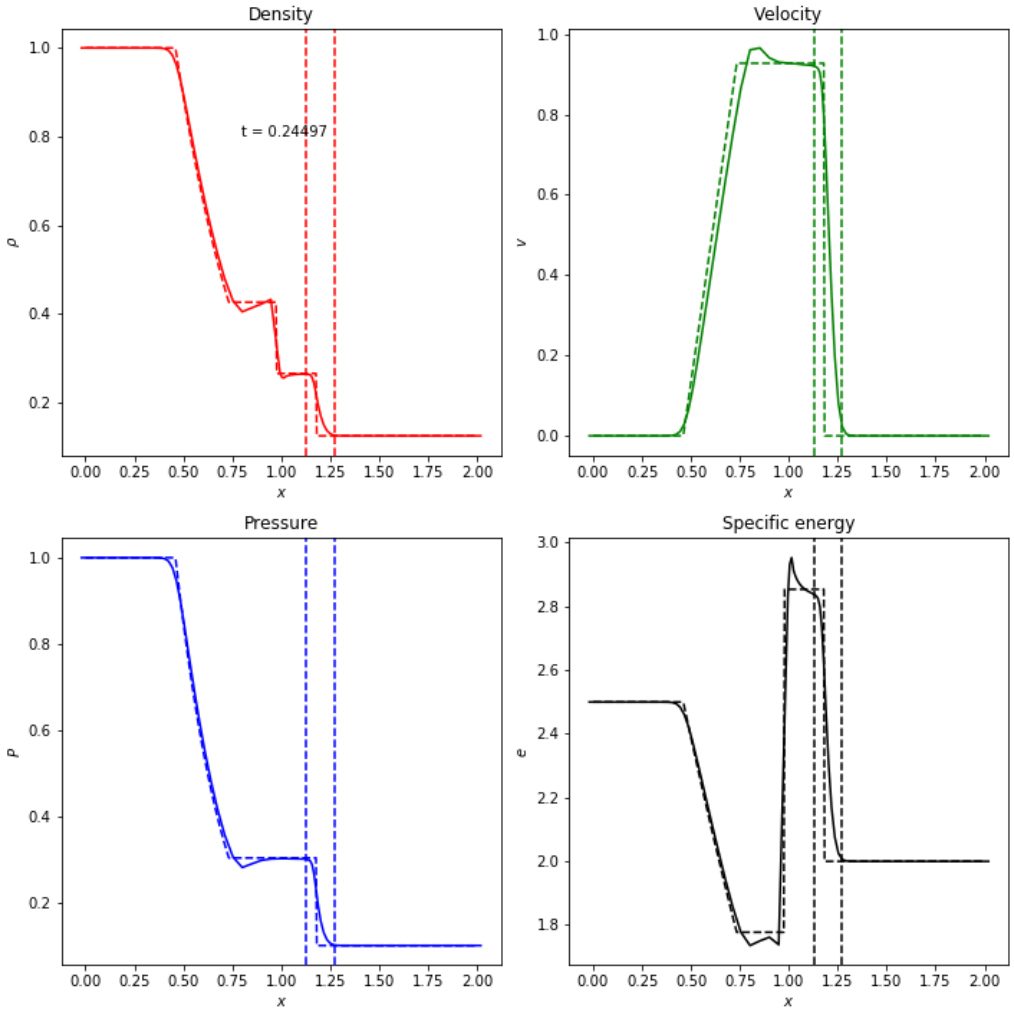

Figure 2: Density, velocity, pressure, and specific energy for $N = 100$, $q_0 = 4.0$, $q_1 = 0.5$, and $C = 0.5$ at $t = 0.245$.

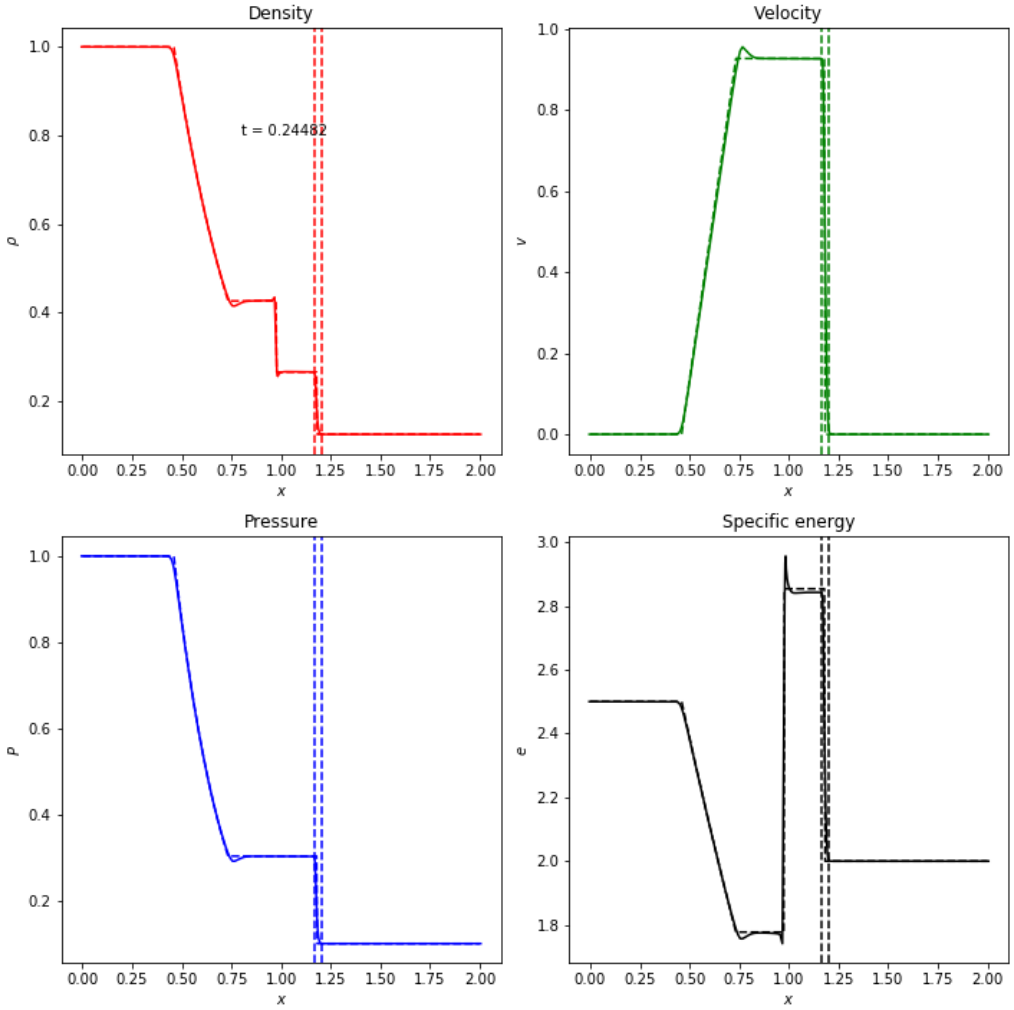

Figure 3: Density, velocity, pressure, and specific energy for $N = 400$, $q_0 = 4.0$, $q_1 = 0.5$, and $C = 0.5$ at $t = 0.245$.

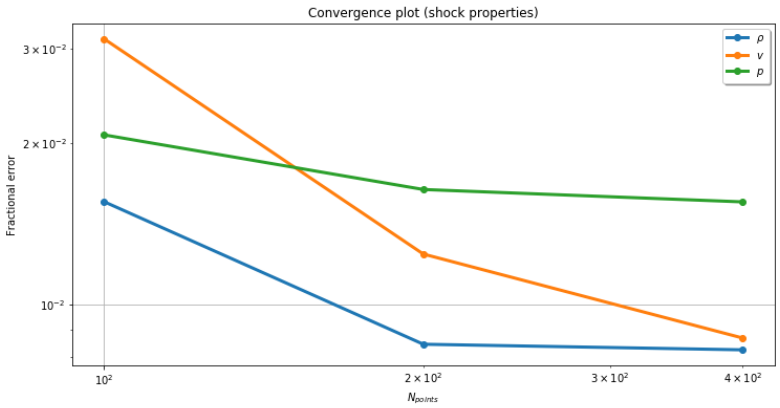

Figure 4: Convergence plots for $N = 100$, 200, and 400.

In general, the fractional error between the numerical solution and the analytic solution decreases with increasing N for the density, velocity, and pressure jumps across the shock. A more rigorous definition of the shock width would help with comparing the convergence of the solution.

(d) *Decrease the CFL number to 0.25 and increase it to 1.1. What happens to the shock solution in each case?*

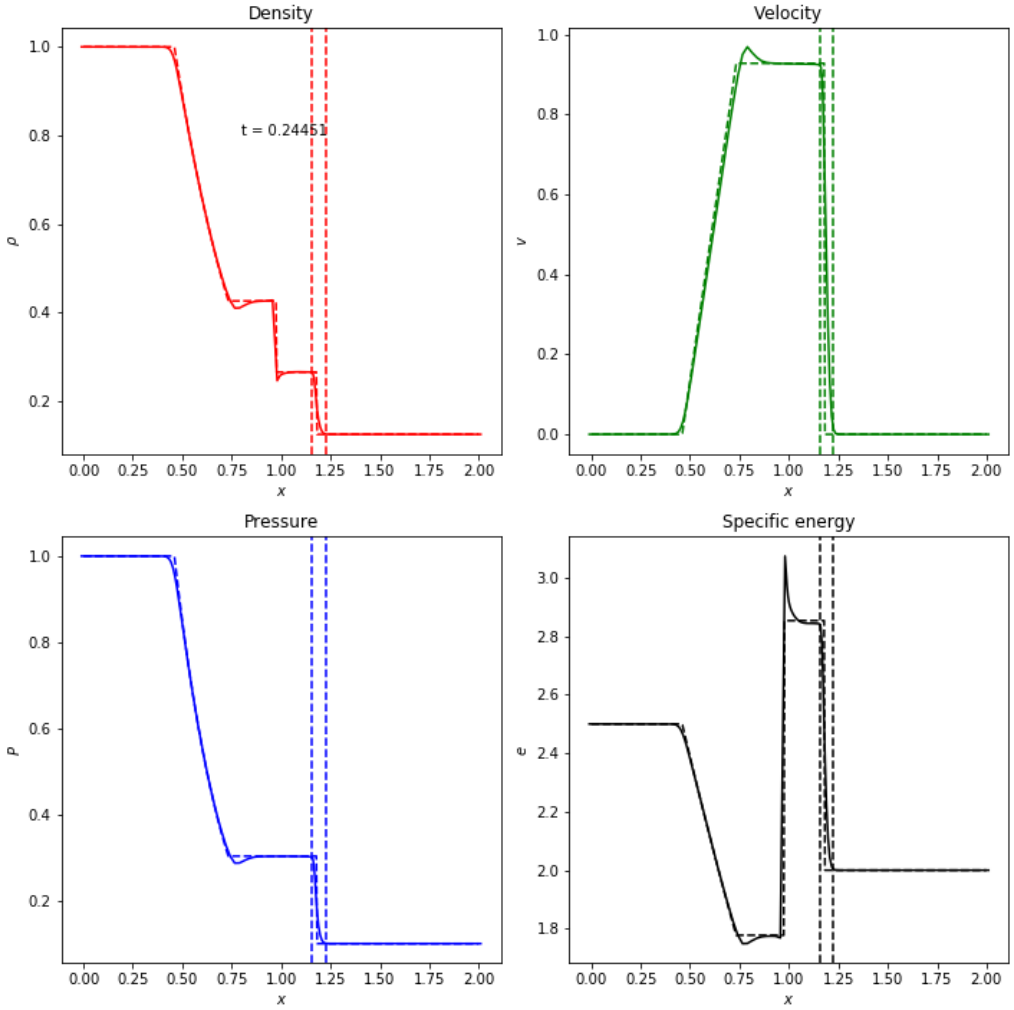

Figure 5: Density, velocity, pressure, and specific energy for $N = 200$, $q_0 = 4.0$, $q_1 = 0.5$, and $C = 0.25$ at $t = 0.245$.

Increasing the Courant number to 1.1 quickly results in negative velocities, extreme over/undershoots of density and pressure, and then unphysical (i.e. negative) density and pressure regions, before running into issues with, e.g., square roots of negative quantities (therefore no plots of the numerical solution). Lowereing the Courant number to 0.25 appears to make the over/undershoots (e.g., in specific energy at $x \sim 1.0$) slightly narrower and more sharply peaked.

(e) *Evaluate how sensitive the solution is to the magnitude of the coefficient of artificial viscosity. Increase the viscosity coefficients by a factor of 2 and decrease them by a factor of 2. How does the solution change? What impact do the two different viscous components have on the solution?*

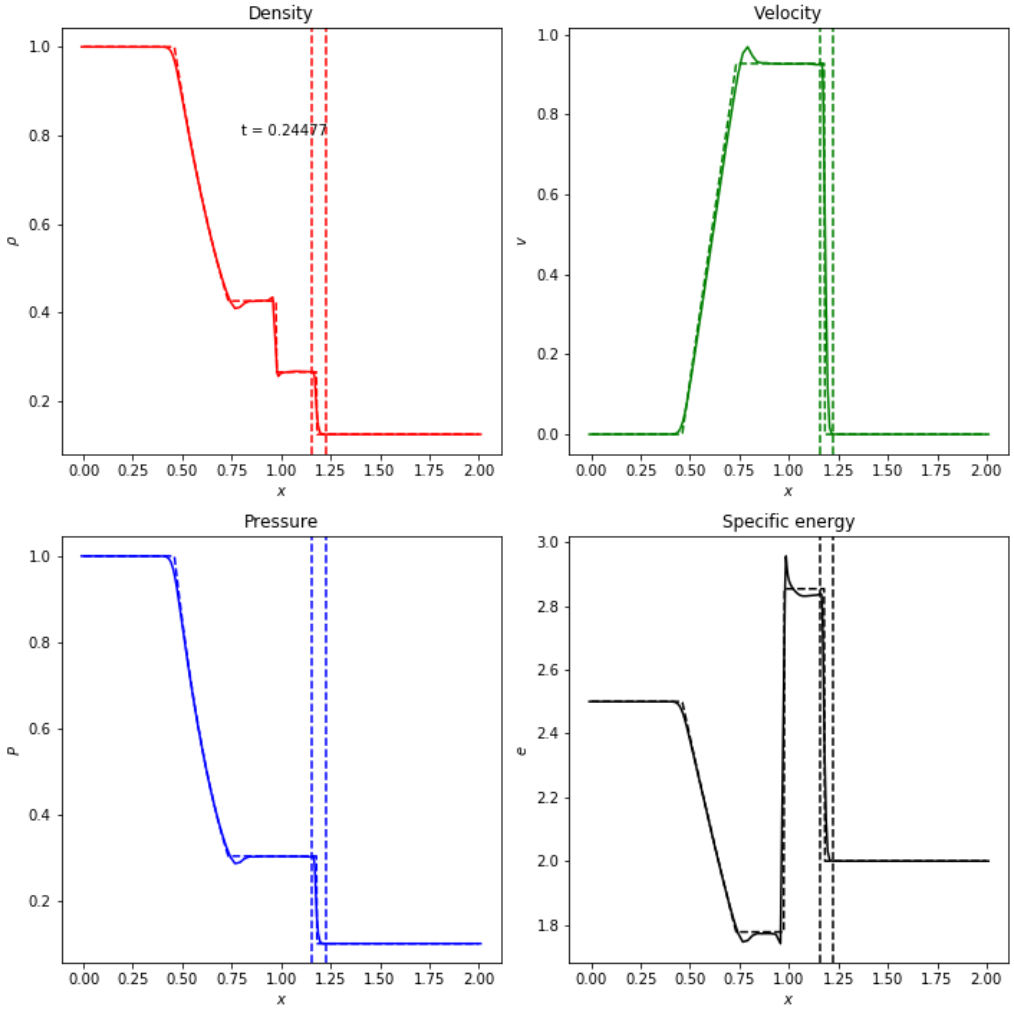

Figure 6: Density, velocity, pressure, and specific energy for $N = 200$, $q_0 = 2.0$, $q_1 = 0.25$, and $C = 0.25$ at $t = 0.245$.

Doubling the artifical velocity coefficients again leads to no numerical solution (for similar reasons as in part d). Halving the artificial viscosity coefficients seems to result in a more vertical, straighter shock front (by visual inspection).