Allan Wei
PHYS 4250
Dr. Jason Fiege
Delica Leboe-McGowan
March 10,2021

Assignment 2 report

Abstract

In this simulation, we use Sod's shock tube to test the accuracy of Roe's algorithm. In computational fluid dynamics, Godunov's' scheme is a finite volume method solving Riemann problem at each intercell boundary. Roe's method, based on Godunov's scheme, involves finding intercell flux passing through two cells boundary. In the result, we compared exact solution algorithm with Roe's algorithm, as well making some changes in Roe's algorithm to see how it behaves.

Theory (brief)

Exact solution

The analytic derivation of Sod's shock tube is dividing the shock tube into 5 regions, region I and V are the regions having the characteristic of initial state of shock tube, region II, III and IV are the rarefaction wave, the contact discontinuity and shock wave respectively. The pressure for example, of region II can be found as follows:

$$P_2 = P_1\left[\frac{2}{\gamma+1} + \frac{\gamma-1}{cs_1(y+1)}\left(u_1 - \frac{x-x_0}{t}\right)\right]^{\frac{2y}{\gamma-1}}$$

Pressure in region III however, requires root finder of the equation

$$(P_3 - p_R)\sqrt{\frac{A}{P_3+B}} + \frac{2Cs_1}{\gamma-1}\left(\left(\frac{P_3}{P_1}\right)^{\frac{\gamma-1}{2\gamma}} - 1\right) + u_R - u_1 = 0$$

Other primitive variables can also be found analytically.

Roe's method

Roe's method is based on Godunov's scheme

$$U_i^{n+1} = U_i^n + \frac{\Delta t}{\Delta x}\left(F_{i-\frac{1}{2}} - F_{i+\frac{1}{2}}\right)$$

Where the intercell flux is given by

$$F_{i+\frac{1}{2}} = \frac{1}{2}(F_L + F_R) - \frac{1}{2}\sum_{j=1}^{3}\tilde{\alpha}_j\left|\widetilde{\lambda_j}\right|\tilde{k}_j$$

With

$$F_L = \begin{pmatrix}\rho_L u_L \\ \rho_L u_L^2 + P_L \\ u_L(E_L + P_L)\end{pmatrix} \text{ and } F_R = \begin{pmatrix}\rho_R u_R \\ \rho_R u_R^2 + P_R \\ u_R(E_R + P_R)\end{pmatrix}$$

Details in theory can be found in A2_shocktube pdf.

MATLAB interpretation

parameters

For the base case, we have our shock tube of length one, with left chamber initial condition of ρ_L=1.0, u_L=0.75 and P_L=1.0, right chamber ρ_R=0.125, u_R=0.0 and P_R=0.1, separate by x_0=0.5. The specific heat ratio we use is γ=1.4.

Functions

pStart solver is a root finder being used to find the pressure in region III, with tolerance of 0.01.

Results

Base case

Below we have the plots of density, velocity and pressure using the exact solution algorithm and Roe's method with the initial set up discussed above.

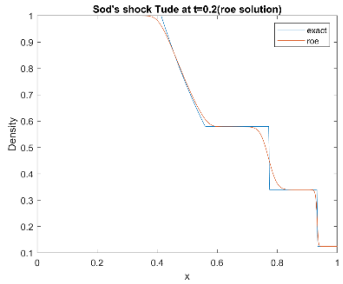


image 1 Density plot (roe vs exact) at t=0.2

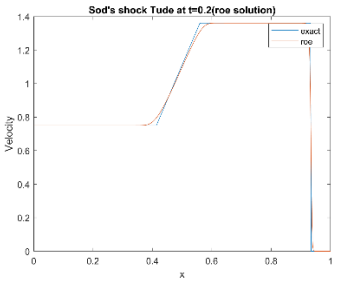


image 2 Velocity plot (roe vs exact) at t=0.2

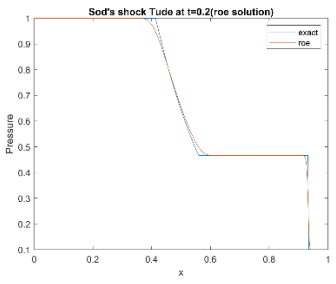


image 3 Pressure plot (roe vs exact) at t=0.2

(images showing non-continues lines when importing to word, exact image please see attached photo in the folder)

We can see that Roe's method is a very good approximation of the exact solution, discontinuous jumps in region III and IV are smoothed out in the simulation.

Q1

The exact solution algorithm for Sod's Shock Tube behaves normally within the time range 0< t < 0.23, while keeping the initial setup the same.

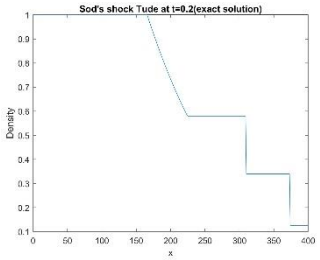


image 4 Density plot (exact solution) at t=0.2

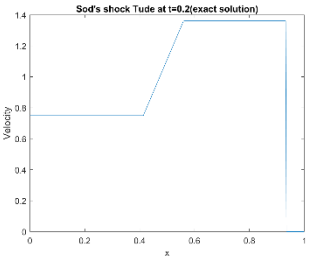


image 5 Velocity plot (exact solution) at t=0.2

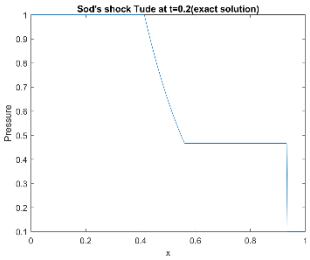


image 6 Pressure plot (exact solution) at t=0.2

When time go higher than 0.23, we notice that the right boundary of region 4 exceeds the cell limit, since region 4 is defined to be $x_0 + t * v_{shock}$,which causes the plot to fail. If we insist to plot the graph, one can simply remove 'xCoords' from the plot input.

However, the graph does not correctly represent the gas in the shock tube. Since we code the range of region 5 to be region4end:400, such indexing will give an empty double while region4end>400. Hence region 5 will be incorrectly empty.

```
Error using plot
Vectors must be the same length.

Error in shockTube (line 22)
a1=plot(xCoords,exactDensity);M1='exact';
```

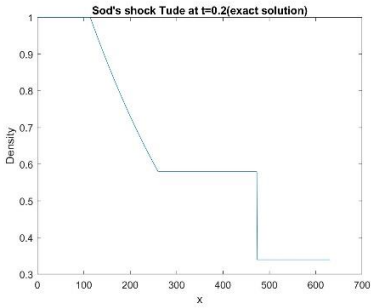


image 7 error at t=0.24

image 8 Incorrect density plot (exact solution) at t=0.5

Changing initial setup for left chamber

Now we keep time fix at t=0.2 while changing the initial velocity of left chamber to be above 1.3, the algorithm will fail for the same reasoning (region 4 exceed max cells). Similarly, when the initial density of left chamber goes higher than 1.6, the algorithm will also fail.

```
Error using plot
Vectors must be the same length.

Error in shockTube (line 22)
a1=plot(xCoords,exactDensity);M1='exact';
```

image 9 error with par.vxL =1.4

```
Error using plot
Vectors must be the same length.

Error in shockTube (line 22)
a1=plot(xCoords,exactDensity);M1='exact';
```

image 10 error with par.densL =1.7

However, the algorithm handles well under large initial density (in reality, the pressure should also go up, but for the purpose of testing we keep initial pressure unchanged). This is because, when looking in the formula for finding regions, region 1 is larger with larger initial density, while contracting region 2-4. However, region 1 is large up to the limit of $x_0 + (u_1 - CS_1)t$, where CS_1 approach 0 with large density, hence it will not exceeds the max number of cells.

Changing initial setup for right chamber

The algorithm fails when initial density goes below 0.08. The reason is again region 4 exceeds the max cell. For initial velocity, algorithm behaves normally under the range -7 to 0. Going lower than -7 will result in empty region 2. since region 2 index from x1end:x2end where x2end is negative when velocity is smaller than -7, hence the indexing returns empty double. However, the algorithm is functioning under arbitrarily small density.

```
Array indices must be positive integers
or logical values.

Error in exactSolution (line 130)
density(x3range) = dens_3;

Error in shockTube (line 16)
[exactDensity, exactVelocity,
exactPressure] = exactSolution(par);
```

image 11 error with vxR = -8.0

```
Error using plot
Vectors must be the same length.

Error in shockTube (line 22)
a1=plot(xCoords,exactDensity);M1='exact'
```

image 12 error with densR =0.07

There are total of 12 different initial setups to try (3 from each side,6 in total and either large or small for each variable), though some might not have physical meaning (i.e. left density low while the pressure is higher than right chamber pressure). One can test more with different initial conditions, but from above we can conclude that exact solution algorithm can only handle conditions that various little amount from the initial setup.

Q2

Roe's solution algorithm works wider range of time and initial conditions than exact solution algorithm.

At t=0.24 exact solution algorithm fails; however, roe solution is still functional.

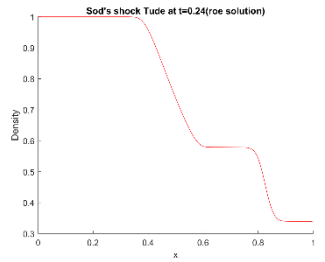


image 13 Density plot (roe solution) at t=0.24

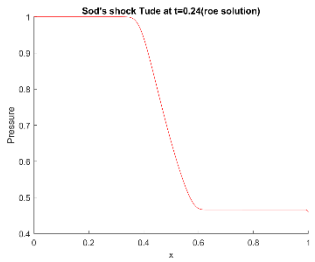


image 14 Density plot (roe solution) at t=0.24

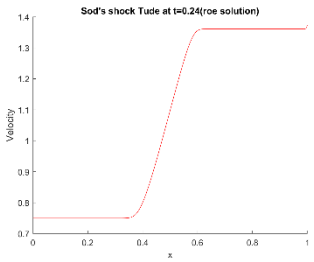


image 15 Density plot (roe solution) at t=0.24

We can also see the behavior of the plot from t=0.2 to 0.6. For value larger than 0.6 the plot reaches a steady state and no further changes are found.

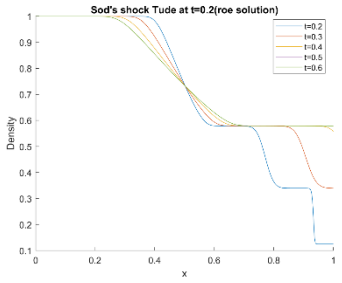


image 16 Density plot (roe solution) range from t=0.2 to 0.6

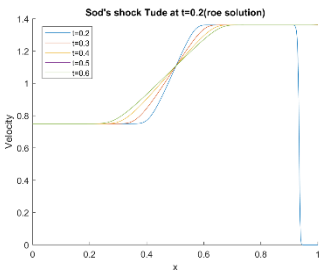


image 17 Velocity plot (roe solution) range from t=0.2 to 0.6

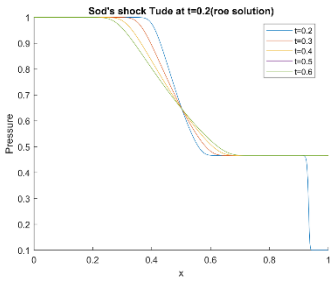


image 18 Pressure plot (roe solution) range from t=0.2 to 0.6

Changing initial condition of left chamber and right chamber

The plots below show the initial condition where exact solution fails but works in roe solution (only density plots because limit spacing).

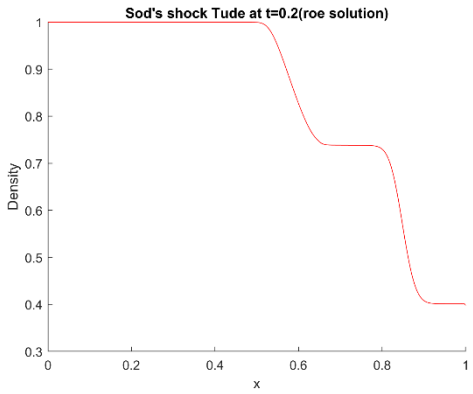


image 19 Density plot (roe solution) with vxL =1.4

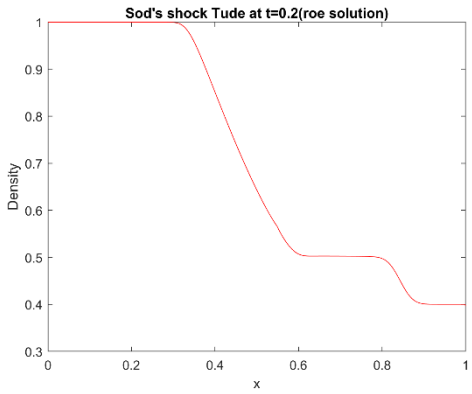


image 20 Density plot (roe solution) with presL =1.7

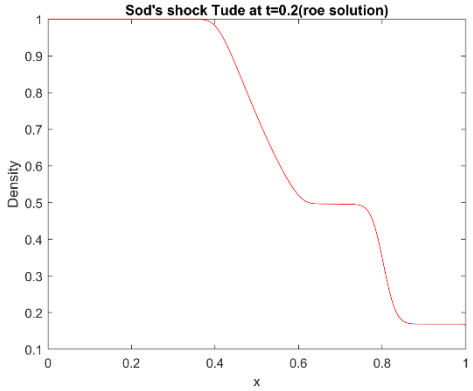


image 21 Density plot (roe solution) with densR =0.07

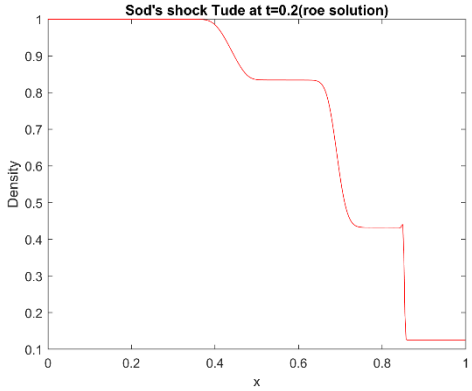


image 22 Density plot (roe solution) with vxR =-1

One can see that roe solution is more robust and efficient than exact solution. When looking at exact solution, errors are caused by deciding the regions, where it could lead to exceeding max number of cells. However, roe solution algorithm does calculation on each individual cell, without deciding the regions.

Q3

Image below shows density plots of roe solution by decreasing cell size of 50 each time. The first noticeable different is observed when number of cells equal to 300. However, one can always zoom into the image and see a noticeable difference even when the number of cells is decreased by the amount of 1. We can see that, as expected, higher number of cell numbers result in better approximation of the Sod's shock tube.

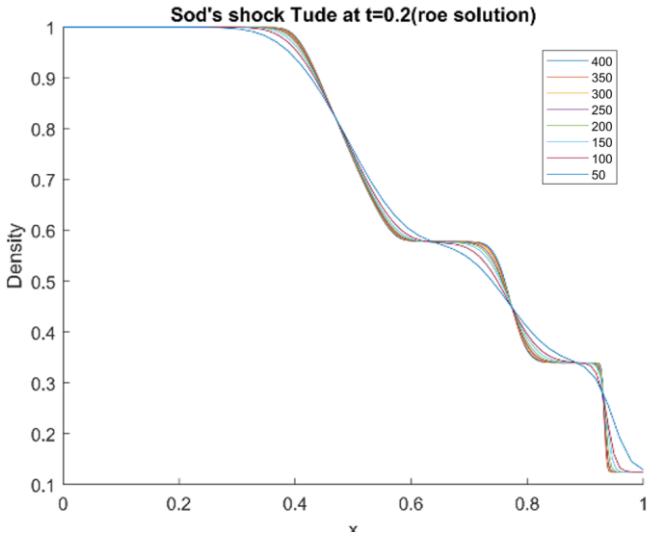


image 23 density plot (roe solution) of cells value 400 to 50(blue to dark blue respectively)

Also below are the velocity plot and pressure plot from left to right respectively.

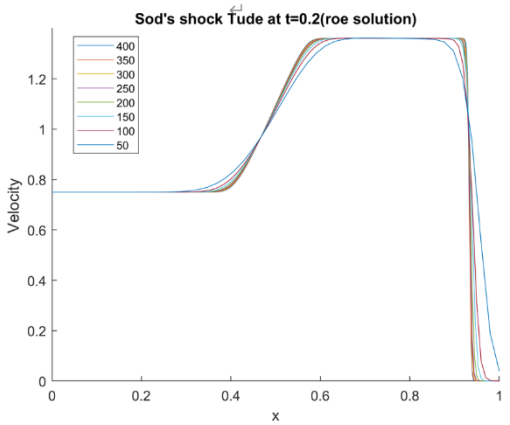


image 24 velocity plot (roe solution) of cells value 400 to 50(blue to dark blue respectively)

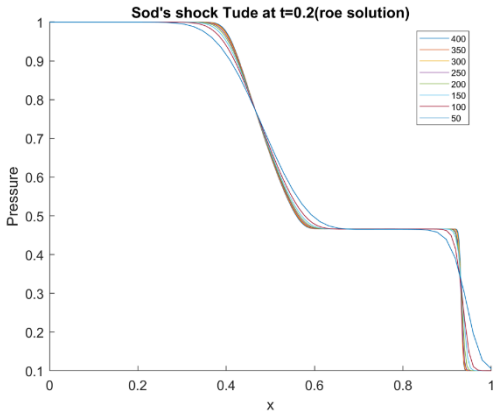


image 25 pressure plot (roe solution) of cells value 400 to 50(blue to dark blue respectively)

Q4

Below are the results of density, velocity and pressure when cfl=0.1 with cycle =1000.

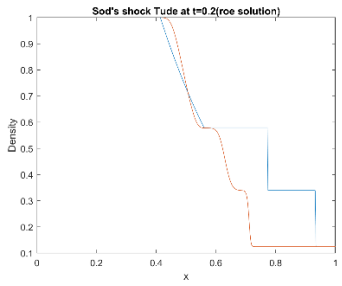


image 26 density plot (roe [orange] and exact [blue] solution with cfl=0.1)

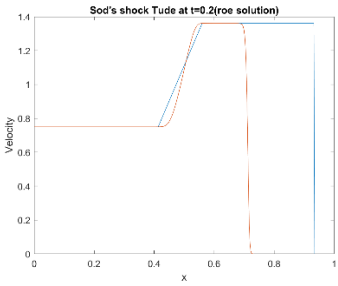


image 27 velocity plot (roe [orange] and exact [blue] solution with cfl=0.1)

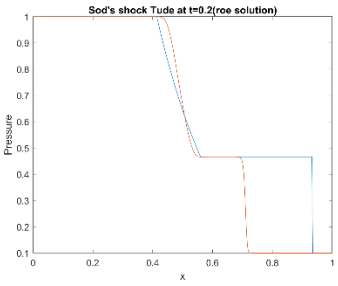


image 28 pressure plot (roe [orange] and exact [blue] solution with cfl=0.1)

However, by default our max cycle is 1000. With cfl=0.1, our total time has not yet

reach 0.2, hence the roe plot is far off the exact plot. If we increase the max cycle to 10000, we have the following plot:

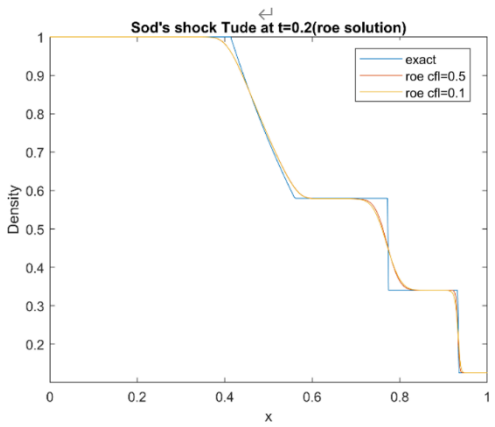


image 29 density plot (exact, roe cfl=0.5, roe cfl=0.1)

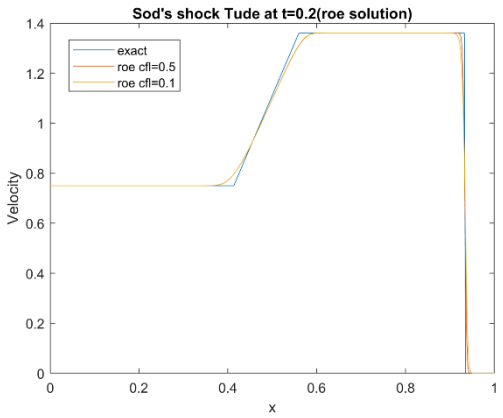


image 30 velocity plot (exact, roe cfl=0.5, roe cfl=0.1)

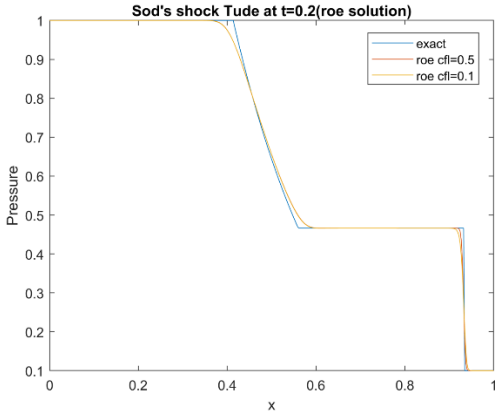
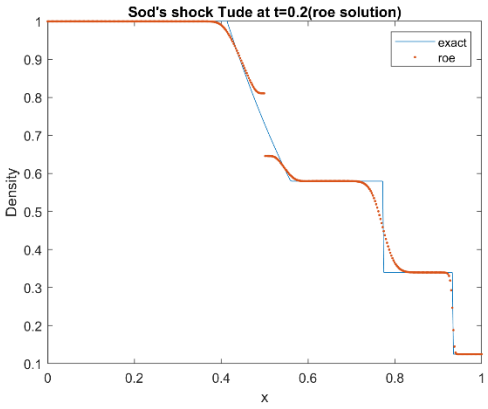


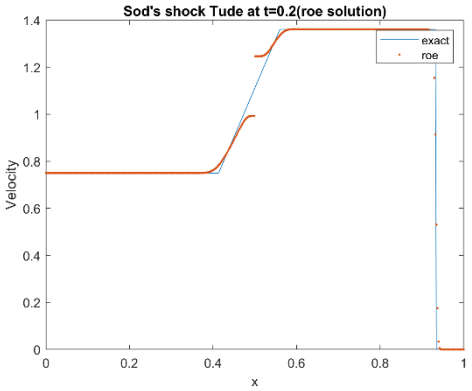
image 31 pressure plot (exact, roe cfl=0.5, roe cfl=0.1)

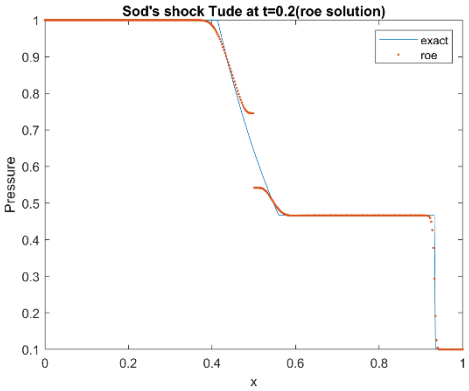
We can see that when we simulate using cfl=0.5, the result is closer to exact solution than using cfl=0.1. This result is counter-intuition, as we expect higher accuracy when decreasing the time step. The reason behind using larger Courant-Friedrichs-Lewy number results in better approximation than using smaller value is caused by numerical diffusion.

Q5

Below are the plots of roe solution when enthalpy fix is removed. We can see there are discontinues jumps in region 2 in above plots. Unlike shock and contact waves, rarefaction waves have the property of continues change, which is spread out and decaying. Without the help of enthalpy fix, rarefaction wave in the simulation is unphysical.


Sod's shock Tude at t=0.2(roe solution)
exact
roe
Density
x


Sod's shock Tude at t=0.2(roe solution)
exact
roe
Velocity
x


Sod's shock Tude at t=0.2(roe solution)
exact
roe
Pressure
x