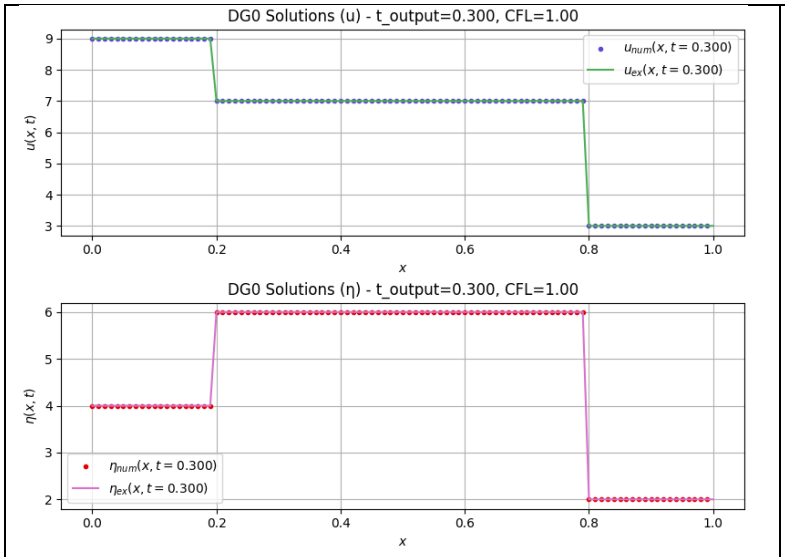
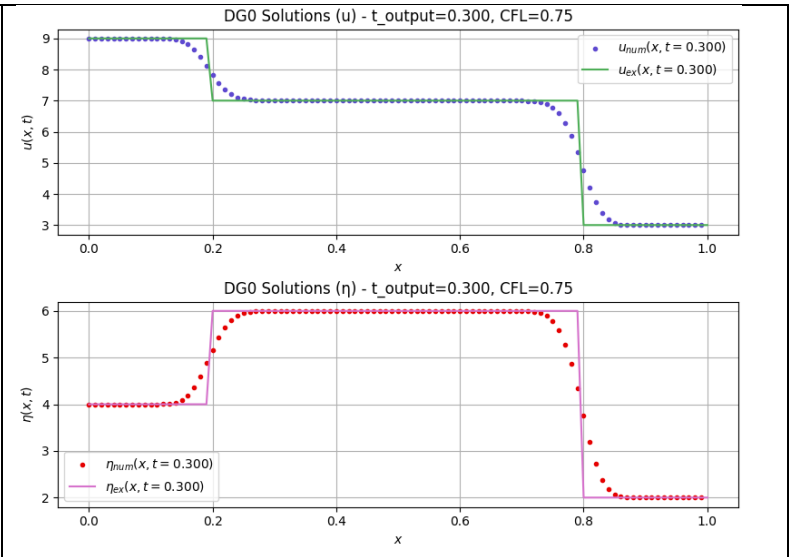
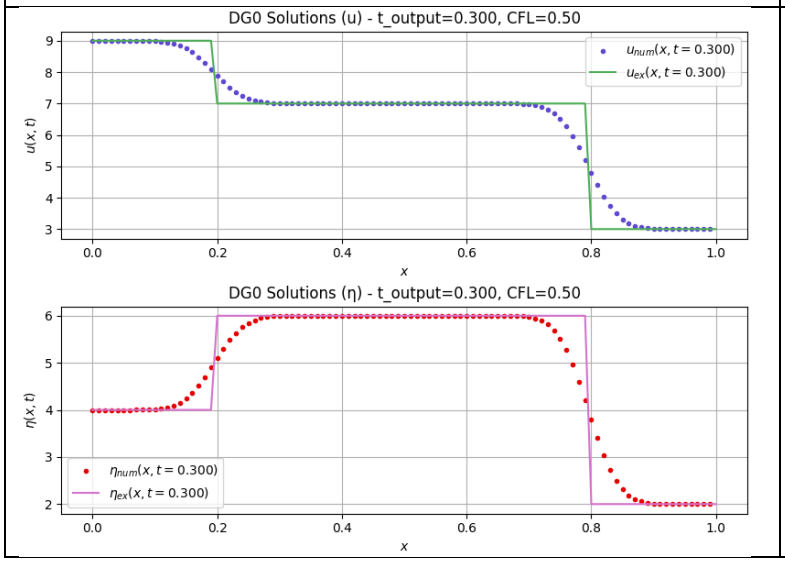
Question 4 (Numerics Ex2)

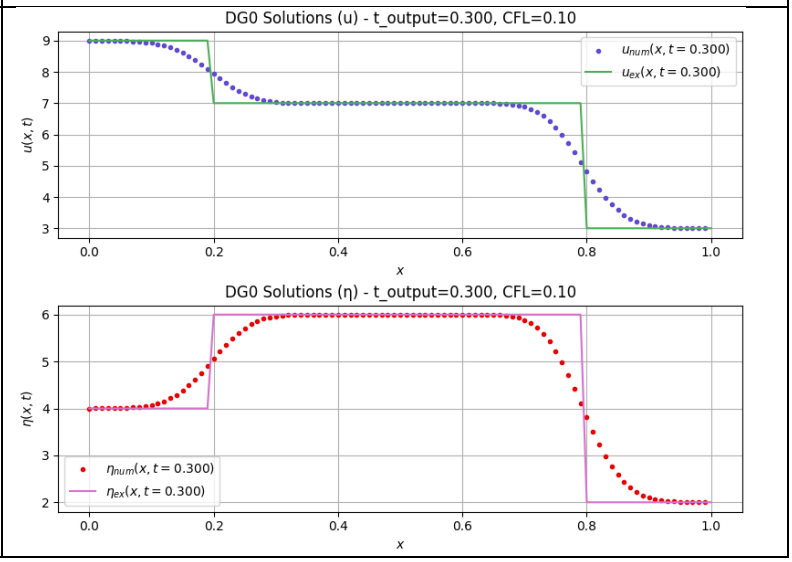
The Godunov scheme was implemented as in Num2Q4.py. This code was based on the given code LAE_DG0.py. The solutions were verified against the exact Riemann solutions as given in Question 2 and 3, shown as u_ex, η_ex in the graphs below. We used initial conditions u_l = 9, u_r = 3, η_l = 4, η_r = 2– perhaps on the larger side, but they show up well in the graphs. The spatial refinement was Δx = 100. Unit values for g, H0 were used.

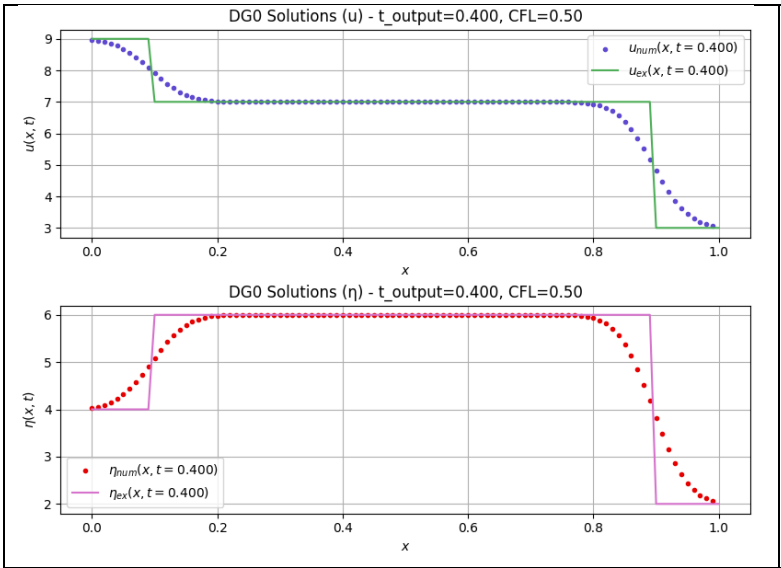
Note that the finite volume scheme used is conservative as the inflow flux function is equal to the outflow flux function, that is, the outflow from volume k equals the inflow to volume k+1. Out of interest, one of my initial errors in my code was using the ICs at the left boundary for the inflow flux (and hence violated conservation) which gave incorrect solutions.

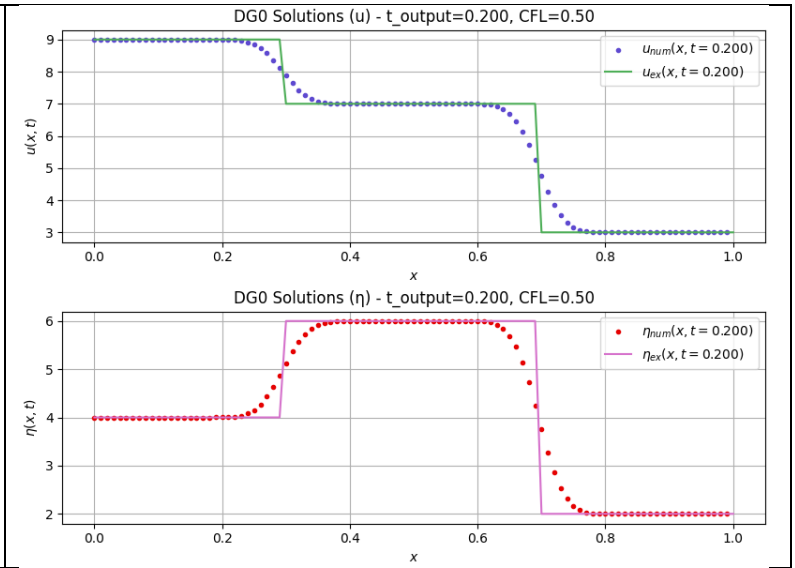


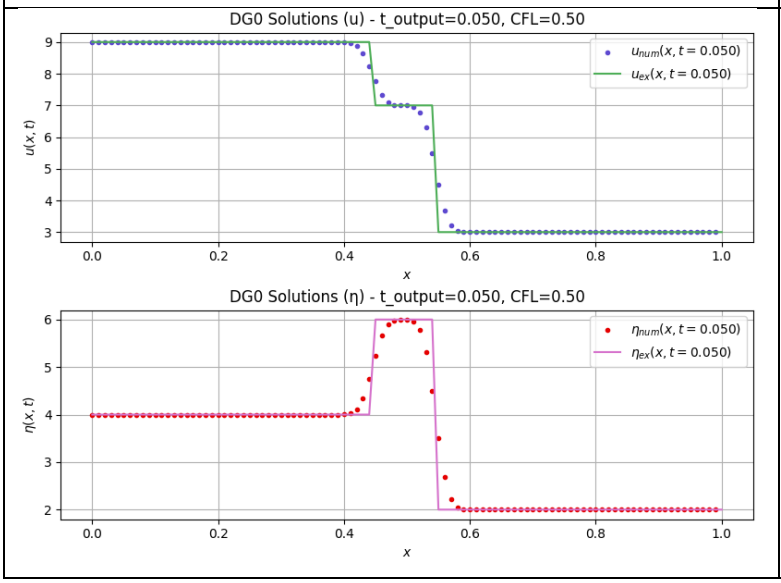


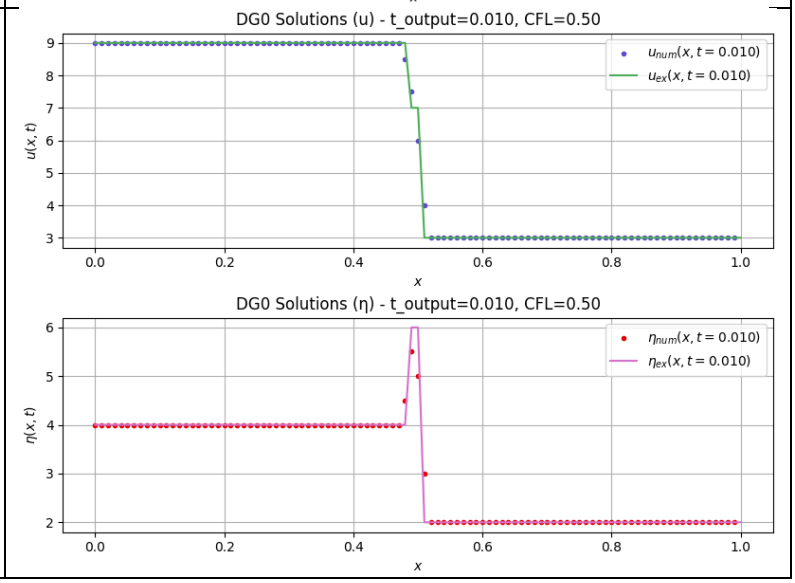












Decreasing CFL and therefore the timestep decreases accuracy around discontinous points, which is unexpected. Values of CFL < 1 seems to “smooth” the solution around these points. Decreasing the size of the time period limits accuracy acround discontinous points, although later tests (not shown) show this can be fixed by refining the mesh in the x-direction, for example, with $\Delta x = 1000$.

Overall, the exact and numerical solutions display the discontinuous step solutions we were looking for with the linear shallow water equations, as indicated by Qu2 and Qu3. The ‘smoothing’ of the solutions as CFL decreases is interesting, as while it limits accuracy, perhaps this is closer to the physical behaviour of the flow.

Notes on the code: Just out of interest, this code demonstrated to me how important the flux functions and choice of scheme are to the solver. It was simple to code in the exact solution, but my first attempts for the numerical solutions came out with extremely high/negative non-physical values, and had no ‘step’ shape, even though I could see no issues with the process/base code I used. It took a long time for me to realise my inflow and interior fluxes were slightly incorrect, and that the solver would reject physical g=9.81.