Maths and Numerical Methods Coursework

Due: Thursday 25th January at 5pm

Note that this coursework counts 2/3 of the total mark for the Maths and Numerical Methods module, with the other 1/3 being based on the class test to be sat on Friday 26th January.

The coursework consists of 2 questions, both of which must be answered. Each question must be answered in a separate file. Each question is worth 50% of the total mark for this coursework.

<u>Question 1</u>

Consider

- Explicit vs implicit methods.
- Runge-Kutta vs linear multistep methods.
- Finite difference vs finite element methods.

For each of these three comparisons, provide your own explanation for what these terms mean, what the key similarities or differences are, what the potential advantages and disadvantages are, and why or how one would choose one approach over the alternative.

Where possible you should back up your explanations and discussion with appropriate numerical examples.

Marks will be given for answers that synthesise (i.e. combine and build upon) knowledge from lectures and for providing your own interpretations. You can use any material from lectures or homework, but note that marks will not be given for simply cutting and pasting material from lectures without demonstrating your own understanding of the material.

For each of these three comparisons limit your answer to a maximum of 1000 words.

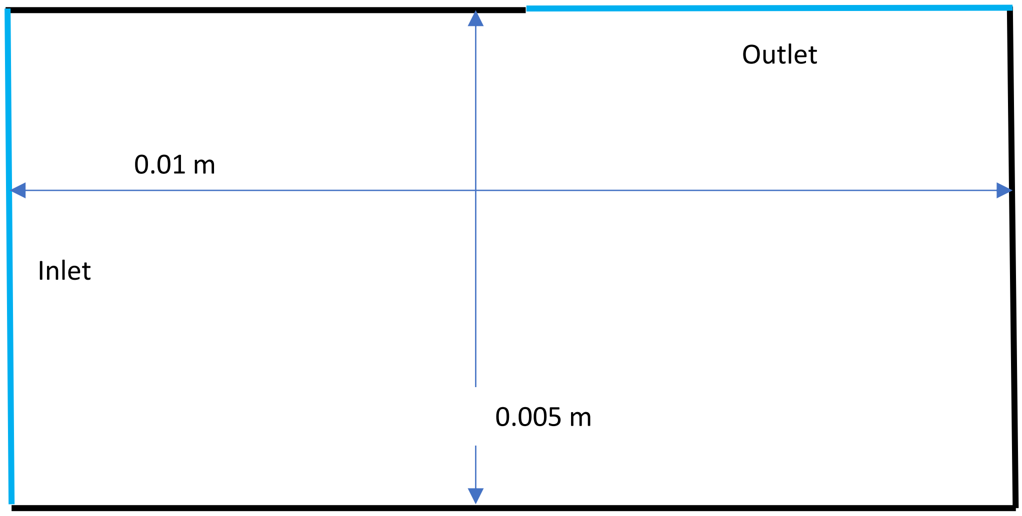
Rather than trying to cover every possible topic, higher marks will be given for a "deep dive" into one particular aspect, e.g. accuracy, generalisability, efficiency, stability, underpinning theory, ….

At the start of each of your three comparisons, give a short explanation for which aspect you have decided to focus on, and how your answer synthesises or expands upon material from the lecture.

(50 marks)

Please submit your answer as either a pdf report, or an ipynb file with all cells run and output saved. Include your github username as well as "Piggott" in the filename to differentiate it from your solution to Question 2.

Question 2



A company is designing a heating chamber to warm a liquid stream. The reactor is to consist of two parallel plates with flow between then. In order to enhance mixing the inlet will consist of the entire LHS and the outlet will be the right half of the top of the reactor. All other boundaries can be considered to be no slip.

The fluid properties are those of water ($\rho = 1000\ kg/m^3$ and $\mu = 0.001\ Pa.s$ – note be careful as to which viscosity is being used in the code – $\nu = \mu/\rho$). You can use a default pressure drop of 0.5 Pa.

a) Code a simulator to solve this problem. You may modify the code that was developed in class. Show the flow and pressure profiles for the conditions given above. Remember to check if the flow has achieved steady state (and note that for some conditions the flow may remain unsteady). (15 marks)

The flow of the heat within the water can be described using an advection-diffusion equation:

$$\frac{\partial T}{\partial t} = -\mathbf{v} \cdot \nabla T + k\nabla^2 T$$

Where T is the temperature, $\mathbf{v}$ is the liquid velocity vector, k is the thermal diffusivity. The fluid flows into the domain with a temperature of 20°C, while at the outlet the heat flows out at the same velocity as the fluid, which implies that there is zero gradient in the temperature normal to this boundary (this type of boundary is sometimes referred to as an open boundary). All other boundaries are maintained at a temperature of 100°C. Note that in reality the boundary layer near the wall is likely to result in addition heat flow resistances, but you can assume that the fluid against the wall is the same temperature as the wall. You can use a thermal diffusivity of 5e-6 m^2/s (note that this is slightly high compared to the real thermal diffusivity of heat in water, which is about 1.5e-7 m^2/s, but results in thermal Peclet numbers that are likely to produce more interesting results).

b) Write out a finite difference approximation for the advection diffusion equation given above. You can use an explicit scheme (spatial derivatives calculated at the current time step and time derivate between the current and next time step). You should using upwind approximations for the advection terms (don't assume a sign for the fluid velocities – leave them as conditions both in the derivation and the subsequent code). Note that this scheme is similar to what was derived in the lecture on Dimensional Analysis, but with two spatial dimensions rather than one. (10 marks)

c) Implement this approximation together with appropriate boundary conditions within the fluid flow simulator. This therefore takes the form of an initial value problem. In addition to the Courant number stability criterion that should already be applied for the fluid calculations, there is an additional criterion based on the diffusivity – namely that $\Delta t \ll \frac{\Delta x^2}{2k}$. Plot the results of the simulations for the base case. What is the flow average outflow temperature (average of temperature multiplied by the outflow velocity divided by the average outflow velocity)? Remember to run the simulation until the point at which it approaches steady state in terms of both the flow and the temperature. (15 marks)

d) Numerically investigate how the parameters in the models influence the results (hint: doing this in terms of dimensionless groups may help the analysis). Show trends and discuss the results. (10 marks)

Please submit your answer as either a pdf report, or an ipynb file with all cells run and output saved. Include your github username as well as "Neethling" in the filename to differentiate it from your solution to Question 1. As well as the code itself, there should be graphs and discussions of the results where appropriate.