Question 1

In the class we mentioned that the Rayleigh-Taylor instability can potentially be supressed by surface tension. The bigger the disturbance, the less able surface tension is to supress it. The maximum size of a disturbance is the width of the container, d. The other factors are the surface tension, γ, gravity, g and the two densities, ρ_h and ρ_l. You can ignore viscosity as a factor as that is only relevant once the system begins to move.

a) How many dimensionless groups are required to characterise the onset of the instability?
b) One dimensionless group can be the ratio of densities (or, equivalently, the Atwood number). What other dimensionless group/s do you propose?
c) These dimensionless group/s represent ratios of forces. Can you identify the ratios?

Question 2

We wish to study the heat transfer in a radiator and, in particular how the amount of energy transferred depends on the size of the radiator, the temperature difference and the fluid and transfer properties.

Symbol	**Description**	**Units**
A	The surface area of the radiator	m^2
Q	Rate of energy transfer to the fluid	W (J/s)
Cp	Specific heat capacity	$J/K/kg$
h	External heat transfer coefficient	$W/m^2/K$
ρ	Fluid density	kg/m^3
ΔT	Temperature different	K
V	Volumetric fluid flow rate	m^3/s

(i) How many dimensionless groups are required to describe this system? Show working (clue: What are the base dimensions of energy?).

(ii) Find a suitable set of dimensionless groups. You may wish to manipulate the groups into the most appropriate ones. Why did you decide upon these particular groups?

(iii) Dimensionless groups typically involve ratios of quantities or effects, what do the ones that you have found represent?

Show your working, but you can use Python to invert any required matrices.

Question 3

If you have a model for a process it is often useful to non-dimensionalise the equation. One area that I work in is the behaviour of foams. The vertical motion of liquid in a foam can be described by the following two equations:

$$v_z = -\frac{\rho g}{3\mu C_{PB}} A - \frac{\gamma\sqrt{3-\frac{\pi}{2}}}{6\mu C_{PB}} \frac{1}{\sqrt{A}} \frac{\partial A}{\partial z} \quad \text{and} \quad \frac{\partial}{\partial t}(A\lambda) + \frac{\partial}{\partial z}(A\lambda v_z) = 0$$

Where: A is the cross-section area of the Plateau borders (the drainage channels within the foam). v_z is the vertical liquid velocity. ρ is the liquid density. μ is the liquid viscosity. γ is the surface tension. g is the acceleration due to gravity. C_{PB} is an already dimensionless drag coefficient.

λ is the length of Plateau borders per volume of foam and can be used to non-dimensionalise the Plateau border area, A, by turning it into a liquid content, ϕ:

$$\phi = \lambda A \qquad \text{and} \qquad \lambda = \frac{k_\lambda}{{d_b}^2}$$

Where: d_b is the bubble diameter and k_λ is a dimensionless geometric constant.

a) Substitute the liquid velocity into the continuity equation and expand to obtain a governing equation. You may assume that only A varies with respect to either time or space (i.e. all other variables can be assumed constant). You can also eliminate A in order to make the liquid content (which is dimensionless) the dependent variable.

The next step is to non-dimensionalise the resulting governing equation by using a dimensionless position, z^*, and time, t^*.

b) Produce a dimensionless time, t^*, and a dimensionless position, z^*. While you could use the bubble diameter, I would recommend using the physical parameters, ρ, g, μ and γ. Note that there is more than one way to achieve this. I would recommend a Bond number like non-dimensionalising of the position and to introduce the viscosity into the non-dimensional time (the speed of a process is typically viscosity dependent).
c) Non-dimensionalise the governing equation using dimensionless quantities derived above. Remember to group the physical variables into dimensionless groups. What are these dimensionless groups called?

Optional

Now that you have obtained a dimensionless version of the governing equation you can attempt to solve it numerically. You can use a modified version of the code that I gave you in class.

d) Produce a finite difference approximation of the governing equation using the following approximations:

$$\frac{\partial \phi}{\partial t^*} \approx \frac{\phi(z^*, t^* + \Delta t^*) - \phi(z^*, t^*)}{\Delta t^*}$$

$$\frac{\partial \phi}{\partial z^*} \approx \frac{\phi(z^* + \Delta z^*, t^*) - \phi(z^* - \Delta z^*, t^*)}{2\Delta z^*}$$

$$\frac{\partial^2 \phi}{\partial z^{*2}} \approx \frac{\phi(z^* + \Delta z^*, t^*) + \phi(z^* - \Delta z^*, t^*) - 2\phi(z^*, t^*)}{\Delta z^{*2}}$$

e) Write this in the form $\phi(z^*, t^* + \Delta t^*) = f\big(\phi(z^*, t^*),\ \phi(z^* + \Delta z^*, t^*), \phi(z^* - \Delta z^*, t^*)\big)$

f) Replace the approximation for the advection diffusion equation in the code with this approximation. You can them solve this problem using $\phi = 0.2$ at the top of the foam where $z = 0.1m$ (remember to convert this to a dimensionless length) and a liquid content of $\phi = 0.3$ at the bottom of the foam. Have an initial liquid content everywhere else of $\phi = 0.01$

You can assume that the liquid is water with a surfactant in it, resulting in the following physical parameters: $\rho = 1000\ kg/m^3$, $\mu = 0.001\ Pa\ s$, $\gamma = 0.02\ N/m$ and $g = 9.81\ m/s$. The dimensionless parameters can be given the values $C_{PB} = 50$ and $k_\lambda = 7$. We will assume that the bubble diameter is $d_b = 0.005\ m$.

Because this equation is non-linear, obtaining a value for the time step to use for a given resolution is slightly trickier than for the advection diffusion equation, though we can use a similar analysis (you will do more on this in ACSE-3). Consider the governing equation in the following form:

$$\frac{\partial \phi}{\partial t^*} = -v\frac{\partial \phi}{\partial z^*} - D\frac{\partial^2 \phi}{\partial z^{*2}}$$

With the governing equation for this problem v and D are functions of ϕ (and potentially its gradient). You can obtain an appropriate time step by noting that:

$$\Delta t^* \ll min\left(\frac{\Delta z^*}{|v|_{max}}, \frac{\Delta z^{*2}}{2|D|_{max}}\right)$$

Plot the liquid content as a function of height for a range of different times. This type of foam behaviour is known as forced drainage. You should see a solitary wave moving down the foam, with some liquid being sucked into the bottom of the foam due to capillarity.

Note that you will need to do a few seconds of real time, with an output every 0.1 seconds being appropriate for this set of conditions (the number of dimensionless seconds that this represents will depend on how you have non-dimensionalised time).