The convection-diffusion equation can model, for instance, the evolution in time of the concentration of a pollutant in a river. To discuss the treatment of numerical oscillations in convection-dominated problems, the steady 1D equation is considered first

$$au_x - \nu u_{xx} = s, \quad x \in (0,1) \tag{1}$$

with source term $s = 1$, convection velocity $a = 1$ and diffusion ν. The problem is complemented with homogeneous Dirichlet boundary conditions $u(0) = u(1) = 0$.

a) Write down the weak form of the steady problem with a Galerkin formulation (i.e. weighted residuals with the same space for approximation of the solution and for weighting functions).
Use the 1D integration by parts formula $\int_0^1 fg'\,dx = -\int_0^1 f'g\,dx + f(1)g(1) - f(0)g(0)$.

b) Write down the expression of the matrix and the RHS of the system corresponding to the discretization of the Galerkin formulation in terms of the basis functions N_i.

The Matlab code available at the course intranet implements the finite element solution of the steady problem with linear finite elements and a Galerkin formulation.

c) Run the code with a uniform mesh with mesh size $h = 0.1$ (i.e. 10 elements), for $\nu = 0.5, 0.05, 0.01, 10^{-3}$. Compute the Péclet number $Pe = ah/(2\nu)$ for each value of ν. Is the method behaving as expected?

The problem is to be solved for $\nu = 10^{-3}$. Two different strategies are considered to attenuate the numerical oscillations: (i) refine the mesh to have small enough Pe, (ii) use a Stream-line Upwind Petrov Galerkin (SUPG) stabilized formulation.

d) Select a mesh size h to compute the solution with $\nu = 10^{-3}$ and a Galerkin formulation. Run the code with the selected uniform mesh size. What is the Pe? Is the solution behaving as expected?

e) Considering such a fine mesh in all the domain is not an efficient strategy, specially for 2D or 3D computations. Do we really need to use a fine mesh in all the domain? Propose a non-uniform mesh and use it in the Matlab code to get a solution without numerical oscillations. Write down here the Matlab instructions for the definition of the nodes coordinates X. How many nodes does the non-uniform mesh have?

f) Write down the weak form of the SUPG formulation for this problem.

g) Write down the expression of the matrix and the RHS of the system corresponding to the discretization of the SUPG formulation in terms of the basis functions N_i.

h) Modify the Matlab code to implement the SUPG formulation. Propose a value of the stabilization parameter τ for solving the problem with mesh size $h = 0.1$ and diffusion $\nu = 10^{-3}$, and justify your selection. Run the code and check if the numerical solution is behaving as expected.
Write down here the lines modified in the code, and the numerical value of τ.

In practice both strategies are used: a finer mesh is used in regions with sharp fronts, to get a better representation of the front, and a stabilized formulation is used if the mesh is still not fine enough to avoid numerical oscillations.

The transient 1D convection-diffusion equation, with no source term, $u_t + a u_x - \nu u_{xx} = 0$, and homogeneus boundary conditions, is considered next. The initial condition is a Gaussian hill centered at $x = 0.4$.

i) Run the code with 100 elements ($h = 0.01$), velocity $a = 1$, final time $T = 0.6$ and time step $\Delta t = 0.001$ (i.e 600 time steps), for $\nu = 10^{-2}$ and $\nu = 10^{-3}$. When do numerical oscillations appear? Why? How could these numerical oscillations be alleviated?