FEM Project

Consider an ionic fluid trapped between two flat parallel electrodes of opposite electric sign. The cations (positive ions) within the fluid are propelled by the positive electrode (cathode) and attracted by the negative one (anode) whereas the anions (negative ions) are propelled by the anode and attracted by the cathode. The aim of this project is to determine the concentration of cations at any point in space between the two electrodes. The flow of cations can be considered to be one dimensional in the direction perpendicular to the plates (x-direction), and the concentration of cations *C* at any point x between the two parallel plates is governed by two physical phenomena: Diffusion and Convection. The former depends on the diffusivity of the fluid *D* whereas the latter depends on the velocity of the flow **u**. Hence, the differential equation governing the concentration of cations between the two plates is known as the "convection-diffusion" equation, and it reads:

$$\underbrace{-\nabla\cdot\left(D\,\nabla C\right)}_{Diffusion}+\underbrace{\mathbf{u}\cdot\nabla C}_{Convection}=0$$

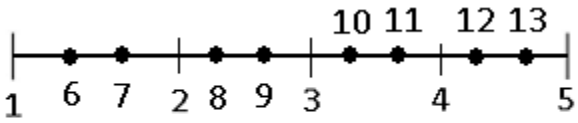
In 1D this equation simplifies to:

$$-\frac{\partial}{\partial x}\left(D\frac{\partial C}{\partial x}\right)+u\frac{\partial C}{\partial x}=0 \qquad for \quad 0<x<L$$

Where *D* and *u* are known constants. In addition, the concentrations of cations at the two electrodes are known, which leads to the following boundary conditions:

$$\begin{cases} C=0\ mol/m^3 & at\ x=0 \\ C=1\ mol/m^3 & at\ x=L \end{cases}$$

Take *L*=1m, *D*=2 m^2/s and *u*=10m/s.



(Sample Mesh)

1) Write the weak form Galerkin formulation for this problem.
2) Develop a numerical code (MATLAB, JAVA, C++…) to solve the problem using the finite element method. The total number of elements N is specified by the user as an input to your program. The remaining components of the code are specified below:

- **Mesh size:** The mesh size is directly deduced from the number of elements N, specified by the user as an input to the code.
- **Order of interpolation:** Cubic elements (3rd order)
- **Node numbering:** The elements are connected in series and the nodes are numbered as shown in the sample mesh above.
- **Integration Method:** You have to use "Gauss Quadrature" numerical integration method.
- **Matrix Structure:** Use a sparse matrix structure (if you are using MATLAB, you can use its built-in sparse matrix library).
- **Assembly:** The assembly procedure must be carried out using a connectivity matrix.
- **Resolution:** Use a direct sparse linear system solver (if you are using MATLAB, you can use its built-in sparse solver "\").

N.B.: The CPU time required by your program to run will be used as a measure of its efficiency. A well programmed and organized code runs in less time… Hence your job consists not only in building a code, but also in optimizing it and removing any unnecessary operations **(Avoid using MATLAB's symbolic variables which are very computationally demanding and would lead to a slow operation of your program).**

3) Run a convergence analysis with respect to the mesh size and provide an appropriate figure showing your convergence results as a function of the number of elements N in the mesh.

4) Build the same model using COMSOL and compare your results with those provided by COMSOL in order to check their validity. A comparison figure showing the superposition of the two results is to be provided.

Deadline and important information (Please read carefully):

The deadline for this project is set to **Sunday April 10th 2022 at 11:59pm**.

You are expected to send your project by email to wassim.habchi@lau.edu.lb prior to this deadline. Any project received after the deadline will not be considered. Below are some guidelines for your project submission:

1- Make sure that the subject of your email is "FEM Project" because I have installed a filter on my mailbox that would filter your projects based on the subject of the email. If you do not put the right title in the subject of your email, it will not be filtered...

2- You should attach a compressed (zipped) folder to your email having your name as a folder name (e.g.:Wassim_Habchi), and this folder should contain the following:

   a. Your report which should not exceed 5 pages (excluding the front page). Please make sure that you name your report "FEM_Report.doc" or "FEM_Report.docx" or "FEM_Report.pdf"

b. The COMSOL file you have developed in order to validate your results. It should also be named "FEM_Project.mph"

c. The source code including all the functions needed in your main program. Your main program should absolutely be named "Main.m" (in MATLAB) or “Main.cpp” (in C++), etc.

3- The next step after sending your project by email is to hold a meeting with the instructor for project assessment. Every student will be assigned a time slot of 15min in the week following the project submission. You will be choosing your timeslot according to your availability. For this, a timesheet will be shared with you to reserve a timeslot.

Good Luck !!!