

# ME-474 Numerical Flow Simulation

Assessment #1

Fall 2025

Answer the questions below and write a short report (one single pdf file per group). The report should be clear and complete, but also concise. Make sure the figures are readable, which means that you must put axis labels and choose suitable ranges, scales, fonts etc.

Consider a fluid flowing through a plane channel between horizontal walls located in  $y = 0$  and  $y = H$ . The temperature field  $T(x, y)$  is solution of the steady heat equation,

$$\text{div}(\rho \mathbf{u} T) = \text{div}(\Gamma \text{grad}(T)),$$

where the density  $\rho$ , velocity field  $\mathbf{u}$  and thermal properties are known ( $\Gamma = k/c_p$  is the ratio of thermal conductivity  $k$  to specific heat capacity  $c_p$ ). Assume that  $T$  is a passive scalar, i.e. the fluid properties and the velocity field are independent of temperature. The two-dimensional numerical domain is a simple rectangle such that  $0 \leq x \leq L$  and  $0 \leq y \leq H$ .

1. Assume that the velocity field is laminar and fully developed, i.e.  $\mathbf{u}(y)$  depends on  $y$  only. Write the expression of the velocity field  $\mathbf{u} = (u_x, u_y)$ , as a function of the mean streamwise velocity  $u_{mean} = \int_0^H u_x(y) dy$ . What is the expression as function of the maximum streamwise velocity  $u_{max} = \max_y (u_x(y))$ ?

2. Consider a uniform structured mesh where all the inner control volumes (i.e. all the CVs except those touching at least one boundary) have the same width  $\Delta x$  and the same height  $\Delta y$ . Integrate the heat equation over one of the inner CVs and use the divergence theorem.

3. Carefully discretize the diffusion term using central differencing (CD).

Next, carefully discretize the convective term using upwind differencing (UD).

4. Combine the results of steps 3 to obtain the algebraic equation to be solved in each inner CV. Check that all the terms have the same physical unit.

5. As the numerical mesh is structured, it is possible to assign indices to each CV, say a row index  $i$  and a column index  $j$ . Choose a convention for these indices and explain it with a sketch.

What are the indices of the CVs touching each of the four boundaries?

Given an arbitrary inner CV, what are the indices of its neighbors?

6. Once discretized, the unknown  $\mathbf{T}$  is a vector of  $N$  values. Choose a convention for the numbering of these  $N$  degrees of freedom (DOFs).

What is the relationship between the DOF index  $n$  and the mesh's row and column indices  $i$  and  $j$ ? In particular, what is the DOF index  $n$  for the CVs touching each of the four boundaries?

Given an arbitrary inner CV of DOF index  $n$ , what are the DOF indices of its neighbors?

Now, start writing your code, defining the matrix  $\mathbf{A}$  and the right-hand side vector  $\mathbf{b}$  for all the equations corresponding to inner CVs.

7. Next, you must take care of the boundary conditions. The fluid enters the channel at the inlet at a temperature  $T = T_{in}$ . For simplicity, let the nodes in the inlet column of CVs sit exactly on the inlet. What is the (discrete) equation to be satisfied by the DOFs of these CVs? Implement it in your code.
8. At the outlet, assume the temperature is fully developed and impose a Neumann boundary condition  $\partial T/\partial x = 0$ . For simplicity, let the nodes in the outlet column of CVs sit exactly on the outlet. What is the (discrete) equation to be satisfied by the DOFs of these CVs? For consistency, discretize using CD. Implement this equation in your code.
9. For the moment, consider Dirichlet boundary conditions  $T = T_{wall}$  on the walls. For simplicity, let the nodes in the first and last rows of CVs sit exactly on the upper and lower walls. What is the (discrete) equation to be satisfied by the DOFs of these CVs? (Note that you have two options for each of the 4 corner CVs, which can be considered to belong either to the walls or to the inlet/outlet.) Implement that equation in your code.
10. Consider the following arbitrary numerical values for the fluid's physical properties:  $\rho = 1 \text{ kg/m}^3$ ,  $c_p = 10 \text{ J/(K.kg)}$ ,  $k = 0.12 \text{ W/(m.K)}$ , and  $\nu = 10^{-2} \text{ m}^2/\text{s}$ . What is the value of  $\Gamma$ ?  
Consider a channel of length  $L = 10 \text{ m}$  and height  $H = 1 \text{ m}$ . We want to study heat transfer at a rather low value of the Péclet number,  $\text{Pe} = \rho u_{mean}/(\Gamma/(2H)) = 16.5$ . What is the value of  $u_{mean}$ ? What is the corresponding Reynolds number defined as  $\text{Re} = 2Hu_{mean}/\nu$ ? Can the flow be expected to be laminar, as assumed earlier?
11. The inlet and wall temperatures are  $T_{inlet} = 50^\circ\text{C}$  and  $T_{wall} = 100^\circ\text{C}$ . Assemble the system of equations on a coarse mesh with  $(n_x, n_y) = (50, 5)$  CVs in the  $x$  and  $y$  directions, respectively.  
For now, solve the linear system using the backslash command (also called mldivide) of Matlab,  $\mathbf{T} = \mathbf{A} \setminus \mathbf{b}$ . Check visually that the solution looks physically reasonable.
12. Produce suitable plots demonstrating *a posteriori* (from the solution) that all the boundary conditions have been correctly implemented.
13. Plot the 2D solution  $T(x, y)$ , as well as the outlet temperature  $T_o(y) = T(L, y)$ , the centerline temperature  $T_c(x) = T(x, H/2)$ , and the velocity-weighted mean temperature

$$T_{mean}(x) = \frac{\int_0^H u_x(y)T(x, y) dy}{\int_0^H u_x(y) dy}.$$

It is also interesting to characterize the length of the entrance region; determine for example the location  $x_e$  such that  $T_c(x_e)$  reaches 90% of  $T_{wall}$ .

14. Now, implement a successive over-relaxation algorithm to solve the linear system. Start from a suitable initial guess  $\mathbf{T}^0$ . At each iteration  $k$ , monitor the relative iteration error  $\|\mathbf{T}^k - \mathbf{T}^{k-1}\|_2/\|\mathbf{T}^{k-1}\|_2$  and the normalized residual  $\|\mathbf{A}\mathbf{T}^k - \mathbf{b}\|_2/\|\text{diag}(\mathbf{A}) \mathbf{T}^k\|_2$ , and stop when both are smaller than the tolerance  $10^{-5}$ .  
Plot the variation of these two quantities with  $k$ . Compare two different values of the relaxation factor, for example  $\omega = 1$  (pure Gauss-Seidel) and  $\omega = 1.5$  (over-relaxation). In each case, how many iterations did you need to reach convergence?
15. In heat transfer problems involving a solid and a fluid that flows either due to an external driving mechanism (forced convection) or spontaneously due to buoyancy (natural convection), an important dimensionless number is the Nusselt number  $\text{Nu}$ . This number compares, at the fluid-solid boundary, the total heat transfer (due to solid-fluid conduction and fluid convection) to the purely conductive heat transfer (due to solid-fluid conduction only, as if there was no flow). Interestingly, for laminar flows in plane channels,  $\text{Nu}(x)$  decreases in the entrance region but, far enough downstream of the inlet, approaches a constant value independent of the Péclet number. See for instance the book

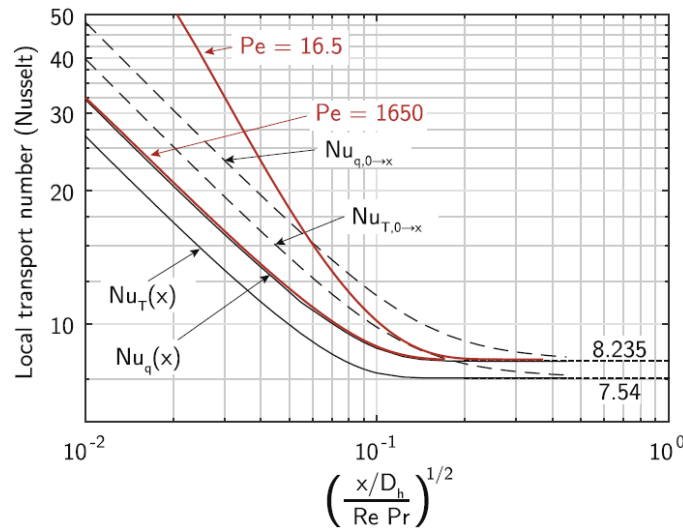
*Convection Heat Transfer* (Bejan, 2103) and the figure below, reproduced from an article by Rohlf's and Lienhard (Int. J. Heat Mass Transfer, 2016). In that figure, solid lines show

$$\text{Nu} = \frac{(2H)q}{k\Delta T},$$

where  $q = k\partial T/\partial n$  is the heat flux through the wall (defined positive from the wall to the fluid, with  $\partial/\partial n$  the normal derivative, and with the normal vector pointing from the fluid to the solid), and where  $\Delta T = T_w - T_{mean}$ , is the difference between the fluid temperature at the wall and the velocity-weighted mean temperature defined earlier. In other words:

$$\text{Nu} = \frac{\partial T/\partial n}{\Delta T/(2H)}.$$

Black solid lines in the figure correspond to the limit of large Péclet number.  $\text{Nu}_T$  is for uniform wall temperature and  $\text{Nu}_q$  for uniform heat flux. Red solid lines correspond to  $\text{Nu}_q$  from accurate numerical simulations performed at  $\text{Pe} = 16.5$  and  $\text{Pe} = 1650$ . For any Péclet number,  $\text{Nu}_T(x) \rightarrow 7.54$  and  $\text{Nu}_q(x) \rightarrow 8.24$  downstream. Note the logarithmic scales. Note also that  $D_h = 2H$  and  $\text{Re Pr} = \text{Pe}$ .



From your simulation, compute and plot  $\text{Nu}_T(x)$  along the lower wall. Does it tend to a limit? Does that value compare well with the expected limit given above?

Do you observe something unexpected near the outlet? Suggest a possible explanation.

16. Compute the solution on increasingly finer meshes with  $(n_x, n_y) = (100, 11)$ ,  $(200, 21)$  and  $(400, 41)$ . Compute quantities of interest for the solutions obtained on the 4 meshes considered so far: outlet  $T_o(y)$ ; centerline  $T_c(x)$  and mean  $T_{mean}(x)$ ; entrance length  $x_e$ ; lower-wall  $\text{Nu}(x)$ . Report 4 plots that compares these results. Comment.
17. In this question only, switch from UD to QUICK for the discretization of the convective term. Note that the equation in the leftmost column of CVs does not change (inlet boundary condition); however, in the second column, QUICK cannot be applied as is, so you can use UD in this column. Compute the solution on the 4 meshes used so far. Plot  $x_e$  for UD and QUICK on the 4 meshes. Plot the relative variation with respect to the value obtained on the finest mesh, for UD and QUICK. Comment.
18. The 4 meshes used so far have inner CVs that are approximately square, i.e.  $\Delta x \approx \Delta y$ . It may be possible and more efficient to use different mesh sizes: this would yield a similar accuracy at a lower cost, or a better accuracy at a similar cost. Compute the solution on meshes with  $(n_x, n_y) = (400, 5)$ ,  $(400, 11)$  and  $(400, 21)$ , as well as  $(n_x, n_y) = (100, 41)$ ,  $(200, 41)$  and  $(400, 41)$ . Produce a plot with two curves comparing a relevant scalar quantity (e.g. the entrance length  $x_e$  or the minimum value of  $\text{Nu}(x)$ ) as a function of  $\Delta y$  for  $n_x = 400$ , and as a function of  $\Delta x$  for  $n_y = 41$  (including relevant solution(s) from previous questions). Comment.

19. In this question only, compute and plot the solution  $T(x, y)$  at larger values of the Péclet number:  $Pe = 50$  and  $Pe = 100$ . Deduce why it is numerically challenging to tackle large-Pe problems.
20. From now on, consider Neumann boundary conditions on the walls:

$$k \frac{\partial T}{\partial n} = q_{wall}.$$

What is the (discrete) equation to be satisfied by the DOFs of the uppermost and lowermost CVs? (Carefully consider the special case of the 4 corner CVs too.) Implement that equation in your code. Solve the problem for  $q_{wall} = 10 \text{ W/m}^2$ . Check visually that the solution looks physically reasonable. Check *a posteriori* (from the solution) that the new boundary conditions are correctly implemented. Compute and plot  $Nu_q(x)$  along the lower wall. Is it in good agreement with the corresponding curve in the above figure?

21. Is thermal energy conserved globally in your numerical simulation? To answer, first evaluate the heat flowing in or out through each outer boundary, then compute the global balance, and conclude.
22. Consider non-uniform wall boundary conditions, aiming to mimic local heating applied externally: positive heat flux  $q_{wall} = k\partial T/\partial n$  imposed in the region  $2 \leq x \leq 5$  of the lower wall; insulated/adiabatic walls ( $k\partial T/\partial n = 0$ ) everywhere else. Based on numerical simulations, find the values of  $q_{wall}$  needed to reach an outlet temperature of  $60^\circ\text{C}$ ,  $70^\circ\text{C}$  and  $80^\circ\text{C}$  ( $+10^\circ\text{C}$ ,  $+20^\circ\text{C}$  and  $+30^\circ\text{C}$  compared to the inlet). Comment.