

Instruction: The task can be carried out individually or in groups of two (recommended), and it should be presented orally (code demonstration) along with submission of code and a compact report.

Attention! Plagiarism check (including usage of AI tools) will be performed on all submissions.

In Assignment-2 you worked with the diffusion equation. Now we add the convective terms (LHS, see Eq. 1) and move the diffusion terms to the RHS. Equation 1 is the convection-diffusion equation, or the transport equation for temperature. You should write a computer program to solve Eq. 1 in two dimensions using FVM. Use the **hybrid differencing scheme**, see Eq. 5.43 in Versteeg & Malalasekara [1]. Note that the continuity error should be $\Delta F = 0$. It is recommended that you use MATLAB. The velocity field and the grid are given; there is an m-file (velocity.m) by which you can read and plot the vector field of the velocity. The two-dimensional transport equation for temperature reads:

$$\frac{\partial}{\partial x}(\rho UT) + \frac{\partial}{\partial y}(\rho VT) = \frac{\partial}{\partial x}\left(\Gamma \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma \frac{\partial T}{\partial y}\right) + S \quad (1)$$

where, $\Gamma = k/c_p$. The algebraic equation system should be solved using **TDMA (recommended)** and **Gauss-Seidel**.

Tips to proceed:

- Fetch the grid and the velocity field from the given files.
- Plot the velocity field using the given m-file. From the supplied velocity field you can find out the location and extent of the inlet and outlet.
- Start to write your program to solve Eq. 1
- You may use a coarse grid when you are writing/debugging the program. When the program works you can also use a fine grid.

Note that when prescribing a heat flux you must divide by c_p since Eq. 1 is an equation for temperature rather than for energy. Use $\partial T/\partial \eta = 0$ (where, η denotes the coordinate normal to the wall) at walls if no boundary condition is specified (see figure 1).

Convergence:

It is very important to verify that a converged solution has been obtained. At each iteration compute the residual as,

$$\epsilon = \frac{1}{f} \left(\sum_{all \ cells} |a_E T_{i+1,j} + a_W T_{i-1,j} + a_N T_{i,j+1} + a_S T_{i,j-1} + S - a_P T_{i,j}| \right) \quad (2)$$

where f is a temperature flux used to normalize the residual. The temperature flux f should be representative of the total flux in the domain. In the present task, it is suitable to take f as the inlet mass flux multiplied by the difference of the temperature flux at inlet and outlet, i.e. $f = (\rho U h)_A \Delta T$, where ΔT is the temperature difference between inlet and outlet. The solution is considered as converged when $\epsilon < 0.001$.

Presentation/Report:

The computational domain and boundary conditions are as shown in figure 1. Discuss the results both from physical and numerical point of view. Present the results as contour plots (wherever applicable). The presentation/report must include the following:

1. Sensitivity to boundary conditions. Make (an interesting) change of one boundary condition.
2. Sensitivity to convergence. Does the temperature change if you use another convergence criterion? Say $\epsilon = 0.01$ and $\epsilon = 0.0001$.
3. Check if you have global conservation, i.e. does the heat flux through all boundaries (inlet, outlet and walls) sum up to zero (as it should if you don't have any source term).

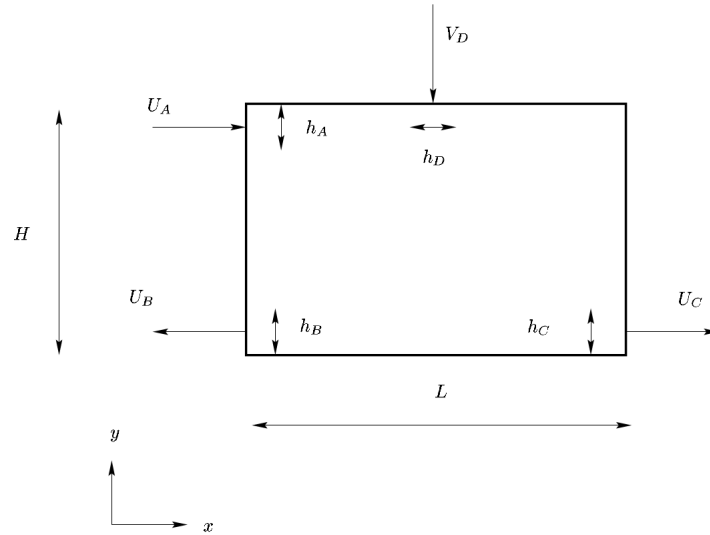


Figure 1: Computational domain. The extent in the third coordinate direction is 1. Physical data: $\rho = 1, k/c_p = 1/50, h_A/H = h_C/H = 0.068$. Boundary conditions: $U_A = 1, U_B = 0, U_C = 1, V_D = 0, T_A = 20^\circ C$. At $x = L$ (other than outlet), $T = 50^\circ C$

References

- [1] H. Versteeg and W. Malalasekera. *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*. Longman Scientific & Technical, Harlow, England, 1st edition, 1995.