ME-474 Numerical Flow Simulation

Exercise: 1D steady incompressible flow; pressure-velocity coupling

Fall 2022

This is a "pen-and-paper" exercise that will help you understand how to solve a fluid flow, with one specific method (here, the SIMPLE algorithm on a staggered grid). It is also a preparation for assessment #1, which will focus on the numerical implementation with Matlab.

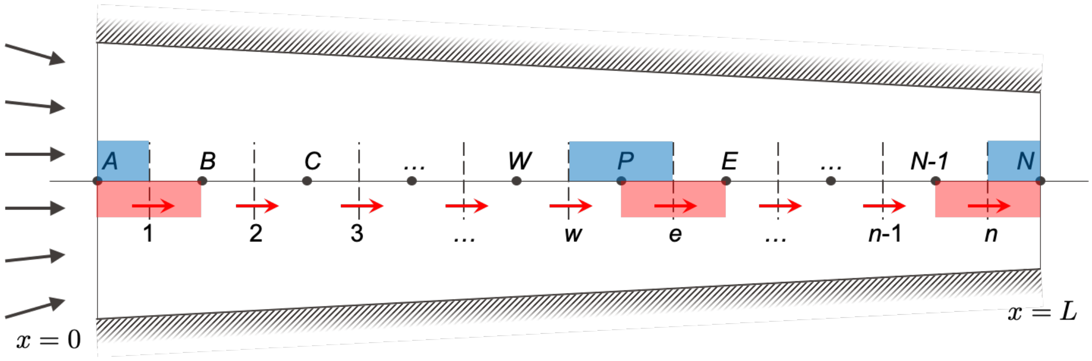
We consider the steady incompressible flow through a 2D planar, converging nozzle. We assume that the flow is frictionless, i.e. we neglect viscosity. We will therefore solve the inviscid Navier-Stokes equations (also called Euler equations). This assumption allows us to neglect the boundary layers that would develop along the walls if the flow was viscous. If we further assume that the flow is unidirectional along the x direction, and that all flow variables are uniform (i.e. constant) throughout every cross-section perpendicular to the flow direction, we can reduce the problem from 2D to 1D, and develop a set of 1D governing equations.

Data and guidelines:

- Constant density ρ.
- Inlet cross-section area A_{in}. Outlet cross-section area A_{out}. The area $A(x)$ varies linearly with distance from the nozzle inlet (see sketch below).
- Inlet boundary condition: assume that the flow entering the nozzle is drawn from a very large chamber. Far away upstream from the nozzle inlet, the fluid has zero momentum (i.e. the fluid is at rest). The stagnation pressure is denoted p_0.
- Outlet boundary condition: static pressure $p_{out} = 0$ Pa.
- Assume the domain $x \in [0, L]$ is discretized with N equidistant pressure nodes (original grid, N control volumes) and $n = N - 1$ equidistant velocity nodes (staggered grid, $n = N - 1$ inner faces).
- Discretize the convective term of the momentum equation with the upwind differencing scheme (UD).
- The goal of this exercise is to derive explicitly the equations to be solved at the different steps of the SIMPLE algorithm. This will make the numerical implementation easier.

  In the SIMPLE algorithm, the velocity and pressure are decomposed into a guess and a correction, $u = u^* + u'$, $p = p^* + p'$. The steps of the algorithm are summarized below (see also the lecture slides):

  1. Solve the momentum equation for the guess velocity u^*, using a guess pressure p^*.
  2. Solve the pressure correction equation for p'. This equation comes from the continuity equation, where velocity is replaced by a simplified expression
  $$u_e = u_e^* + u_e' = u_e^* + (p_P' - p_E')d_e.$$
  3. Correct pressure and velocity:
  $$p = p^* + p',$$
  $$u_e = u_e^* + u_e' = u_e^* + (p_P' - p_E')d_e.$$
  4. Iterate until convergence.



Questions

1. Discretize the momentum equation on the velocity grid (staggered grid). Deduce the algebraic equation for the guess velocity u^*. For the nonlinear convective term, use Picard's method with the previous velocity u^{old}. For the pressure term, use a guess pressure p^*.

2. Express the inlet pressure p_{in} as a function of the upstream chamber stagnation pressure p_0.

3. Derive the equation to be implemented for u_1^* in the first velocity CV.

4. The equation you just found is nonlinear in u_1^*. Linearize the nonlinear term using Picard's method with the previous velocity. Next, use the deferred correction approach if needed.

5. Derive the equation to be implemented for u_n^* in the last velocity CV.

6. Discretize the continuity equation on the pressure grid (original grid). Deduce the algebraic equation for the pressure correction p'.

7. What is the equation to be implemented for p'_A in the first pressure CV?

8. What is the equation to be implemented for p'_N in the last pressure CV?

9. It is actually possible to find an analytical solution to this problem, which will be useful to assess the accuracy of the numerical solution. Find the analytical expressions of $u(x)$ and $p(x)$.