LMECA2660 - Project 2022

P. Balty, G. Poncelet and G. Winckelmans

In this project, it is proposed to study numerically the 2D flow past an obstacle for various cases: fixed obstacle and longitudinally oscillating obstacle.

The obstacle is a rectangular box of size $L_{box} \times H_{box}$, where $L_{box} = 5\, H_{box}$. It is located inside a rectangular computational domain Ω, which has a height of $H = 5\, H_{box}$ and a length of $L = 15\, H_{box}$. The distance between the obstacle and the inflow is $d_{in} = 3\, H_{box}$ and there is a separation of $d_{bottom} = 2\, H_{box}$ between the lower side of the obstacle and the lower limit of the domain; hence we have the same distance for other side: $d_{top} = d_{bottom}$. The sketch of the situation is represented in Fig. 1.

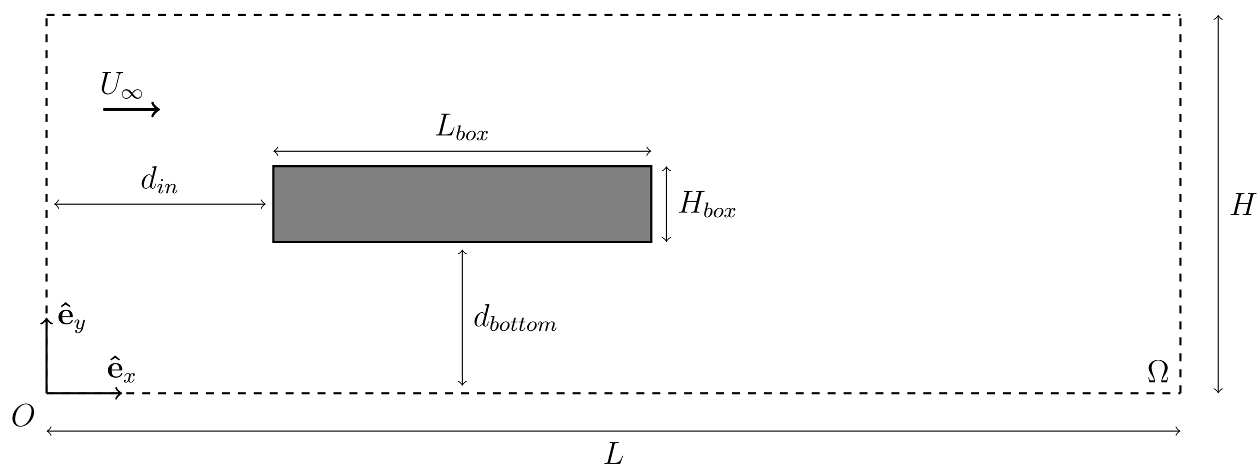

Figure 1: Description of the case studied.

The flow with fixed obstacle can be entirely characterised by the Reynolds number defined as $Re = \frac{U_\infty\, H_{box}}{\nu}$, with U_∞ the upstream velocity, ν the fluid kinematic viscosity and H_{box} the characteristic length of the obstacle. In this project, this Reynolds number is taken as $Re = 500$. A no-slip condition and no-through flow condition has to be enforced at the obstacle wall ($\mathbf{v} = 0$), whereas the far upstream velocity will be enforced at the inflow of the domain ($u = U_\infty$ and $v = 0$). The lateral boundaries will be considered as inviscid, hence with no-through flow condition ($v = 0$) and zero vorticity ($\omega = 0$). The latter is valid provided that the domain boundaries remain sufficiently far away from the vortical structures shed by the obstacle. A special care will be given to the outflow condition: a "natural" outflow condition will be imposed there. This condition is expressed as:

$$\frac{\partial u}{\partial t} + u_c \frac{\partial u}{\partial x} \simeq \frac{\left(u^*_{i+1/2,j} - u^n_{i+1/2,j}\right)}{\Delta t} + u_c \frac{\left(u^n_{i+1/2,j} - u^n_{i-1/2,j}\right)}{\Delta x} = 0 \; , \tag{1}$$

$$\frac{\partial \omega}{\partial t} + u_c \frac{\partial \omega}{\partial x} \simeq \frac{\left(\omega^*_{i+1/2,j+1/2} - \omega^n_{i+1/2,j+1/2}\right)}{\Delta t} + u_c \frac{\left(\omega^n_{i+1/2,j+1/2} - \omega^n_{i-1/2,j+1/2}\right)}{\Delta x} = 0 \tag{2}$$

where u_c is the convective outflow velocity, here chosen to be the upstream velocity U_∞ and the superscript $*$ refers to the prediction step of the flow solver (see section below). We recall that, on the MAC mesh, $h\,\omega_{i+1/2,j+1/2} = \left(v_{i+1,j+1/2} - v_{i,j+1/2}\right) - \left(u_{i+1/2,j+1} - u_{i+1/2,j}\right)$. The first equation is used to update the value of $u^*_{i+1/2,j}$ on the boundary (value which is also equal to $u^{n+1}_{i+1/2,j}$), and the second equation is used after the fist one to set the ghost value $v^*_{i+1,j+1/2}$.

Equations of the problem

The equations to solve are the Navier-Stokes equations for an incompressible flow:

$$\nabla \cdot \mathbf{v} = 0 \tag{3}$$

$$\frac{D\mathbf{v}}{Dt} = -\nabla P + \nu \nabla^2 \mathbf{v} \tag{4}$$

with $\nu = \dfrac{\mu}{\rho}$ the kinematic viscosity and $P = \dfrac{(p - p_{ref})}{\rho}$ the kinematic pressure (p_{ref} being any chosen reference pressure).

Flow solver

We recall that Eq (3) is not an evolution equation, and hence it cannot be integrated in time; it is a constraint that must be satisfied at all locations and at all times; something that the pressure P will achieve. In order to integrate Eq. (4) in time and also guarantee that Eq (3) is satisfied at the end of each time step, a two-step projection scheme combined with a staggered MAC mesh is used. The convective term is integrated using the explicit and second order Adams-Bashforth scheme (explicit Euler scheme for the first time step) while the diffusive term is handled using the explicit and first order Euler scheme. The numerical scheme is:

$$\frac{(\mathbf{v}^* - \mathbf{v}^n)}{\Delta t} = -\frac{1}{2}\left(3\,\mathbf{H}_h^n - \mathbf{H}_h^{n-1}\right) - \nabla_h P^n + \nu\,\nabla_h^2 \mathbf{v}^n \tag{5}$$

$$\nabla_h^2 \Phi = \frac{1}{\Delta t} \nabla_h \cdot \mathbf{v}^* \tag{6}$$

$$\frac{(\mathbf{v}^{n+1} - \mathbf{v}^*)}{\Delta t} = -\nabla_h \Phi \tag{7}$$

$$P^{n+1} = P^n + \Phi \tag{8}$$

where $\mathbf{H}_h$ is the discretised convective term of the momentum equation.

The Poisson equation (6) appearing in the projection step will be solved using the PETSc library. To install this library and get more details about how we solve this equation, we refer to the "*PETSc installation notes*" document.

A mesh with $\Delta x = \Delta y = h$ will be used, where h fulfils a constraint on the precision of the solution, that is to say that the "mesh Reynolds number", $Re_h = \frac{(|u|+|v|)\,h}{\nu}$, must be "sufficiently moderate". To also properly capture the boundary layers, the "mesh Reynolds number based on the vorticity", $Re_{h,\omega} = \dfrac{|\omega|\,h^2}{\nu}$, must also be "sufficiently moderate".

The time step Δt must be chosen so as to ensure the stability of the problem. It involves the "Fourier number" $r = \frac{\nu \Delta t}{h^2}$ (diffusive part) and the "Courant-Friedrichs-Lewy" number $CFL = \beta = \frac{(|u|+|v|)\,\Delta t}{h}$ (convective part).

Cases studied

Different cases will be considered in this project:

In a **first step**, the obstacle is at rest. You are asked to study the 2D flow around a static rectangular obstacle and with a symmetric vortex shedding.

In a **second step**, you are asked to introduce a vertical perturbation at the start of the simulation so as to trigger an asymmetric vortex shedding. To do so, you have to impose an oscillating vertical velocity to the entire mesh. This trick (applicable only when the

the body's boundaries are aligned with the grid points) allows us to accurately capture the physics around a moving object, without relying on other methods (penalisation, immersed methods, etc). In terms of equations, the velocity of the mesh is accounted for in the convective term of the momentum equation, Eq. (9), using

$$\mathbf{H}_h = ((\mathbf{v} - \mathbf{v}_{mesh}) \cdot \nabla)\, \mathbf{v} = \nabla \cdot (\mathbf{v}\, (\mathbf{v} - \mathbf{v}_{mesh})) \ . \tag{9}$$

You must hence also update the boundary conditions on the obstacle boundary so as to enforce the no-slip boundary condition. The vertical perturbation velocity given to the mesh will be taken as

$$\frac{v_{mesh}}{U_\infty}(t^*) = \frac{1}{10} \sin (2\pi\, t^*) \tag{10}$$

where t^* is the dimensionless time, $t^* = \frac{t\, U_\infty}{H_{box}}$. After one period, the perturbation velocity of the mesh must be reset to zero.

In a **third step**, you are asked to impose an horizontally oscillating movement to the obstacle and maintain it. The displacement of the obstacle is harmonic and taken as

$$\frac{x_{mesh}}{H_{box}}(t^*) - \frac{x_{mesh}}{H_{box}}(0) = \kappa\ (1 - \cos (2\pi\, S_t\, t^*)) = \frac{\alpha}{2\pi\, S_t}\ (1 - \cos (2\pi\, S_t\, t^*)) \tag{11}$$

where S_t is the "Strouhal number" of the forcing, defined as $S_t = \dfrac{H_{box}\, f}{U_\infty}$ (with f the frequency). The velocity of the mesh is thus

$$\frac{u_{mesh}}{U_\infty}(t^*) = \alpha\ \sin (2\pi\, S_t\, t^*) = \kappa\, (2\pi\, S_t)\ \sin (2\pi\, S_t\, t^*) \ . \tag{12}$$

The convective velocity used for the outflow boundary condition must then also be modified using

$$\frac{u_c}{U_\infty}(t^*) = 1 - \frac{u_{mesh}}{U_\infty}(t^*) \ . \tag{13}$$

We will here consider only one case: the case $S_t = \dfrac{1}{3}$ and $\alpha = \frac{1}{2}$ (thus $\kappa \simeq 0.2387$).

Finally, in a **fourth and last step**, you are asked to also add the vertical perturbation initially so as to break the symmetry of the previous simulation and produce an asymmetrical vortex shedding.

Presentation of the results

You are asked to:

1. Produce a numerical code to study the four situations. In each simulation, the solution must be computed until a dimensionless time of $t^* = 50$ is reached.

2. Plot contours of the dimensionless vorticity field $\omega^* = \frac{\omega\, H_{box}}{U_\infty}$ for each case and at a few different dimensionless times. For the cases where the obstacle is moving, the body must be shown at its correct location for each time shown (i.e., you must also take into account its displacement, x_{mesh}).

3. Plot the dimensionless streamlines $\psi^* = \frac{\psi}{U_\infty\, H_{box}}$ (i.e. using iso-ψ^* lines) of the flow at $t^* = 12.5$ and 25 for the first case. Also measure the length of the recirculation region that forms behind the obstacle at each of those times.

4. Plot the dimensionless streamlines $\overline{\psi}^*$ and vorticity $\overline{\omega}^*$ of the "time-averaged flow" for the second case. Be careful that the time averaging must be done using solely the fields with a well established vortex shedding (here you can start averaging at $t^* = 20$). Measure the length of the recirculation region.

5. Plot the evolution of :

   - the maximum mesh Reynolds number, Re_h,
   - the maximum mesh Reynolds number based on the vorticity, $Re_{h,\omega}$,
   - the drag and lift coefficients of the obstacle.

6. Comment your results.

This work can be done in a team of two students. It can of course also be done alone. The code and report smust be uploaded on the Moodle website of the course by Friday May 6, 2022. Upload your source code in C and your report in .pdf format. Make sure that you have compressed your code and your report into one single archive file before uploading it.