Project LMECA2660: external flow past a cylinder and an airfoil

B. Hardy, P. Billuart and L. Bricteux

March 26, 2020

In this project, it is proposed to simulate the external flow past a cylinder and then past an airfoil using Finite Differences (FD). The physical domain (drawn in Fig. 1) will extend from the wall body to a specified distance very far from the wall. Such external flows can be entirely characterized by the Reynolds number defined as $Re = \frac{U_\infty L_c}{\nu}$, with U_∞ the upstream velocity, ν the fluid kinematic viscosity and L_c the characteristic length of the body (set to the diameter D of the cylinder or the chord length c of the airfoil). A no-slip and no-through flow condition has to be enforced at the body wall ($\mathbf{v} = 0$), whereas the infinite upstream velocity will be enforced on the exterior boundary ($u = U_\infty$) combined with an unbounded condition ($\omega = 0$). This last condition is valid provided that the external boundary remains sufficiently far away from the vortical structures shed by the body ($d \gg L_c$). In the following, we will denote H the distance between the surface of the body and the outer boundary.

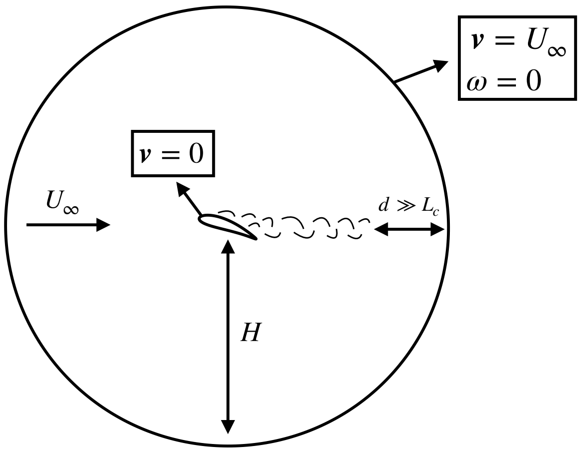

Figure 1: Physical domain for the airfoil.

Equations of the problem

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

$$\nabla \cdot \mathbf{v} = 0 \tag{1}$$

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

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

Flow solver

In order to integrate Eqs. 1 ans 2 in time, we use a two-step projection scheme combined with a MAC mesh. The convective term is integrated using the second order Adams-Bashforth scheme (with the explicit Euler scheme for the first step) while the diffusive term is handled with the Euler explicit scheme. Hence, the numerical scheme is:

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

$$\nabla_h^2 \Phi = \frac{1}{\Delta t} \nabla_h \cdot \mathbf{v}^* \tag{4}$$

$$\frac{(\mathbf{v}^{n+1} - \mathbf{v}^*)}{\Delta\, t} = -\nabla_h \Phi \tag{5}$$

$$P^{n+1} = P^n + \Phi \tag{6}$$

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

The Poisson equation (4) appearing in the projection step will be solved using the PETSc library. To install this library and get more details about how to solve this equation, refer to the "*PETSc installation notes*" document.

The time step Δt will be chosen so that the stability of the problem is ensured. It will involve the Fourier number $r = \frac{\nu \Delta t}{h_{min}^2}$ (diffusive part) and the $CFL = \frac{(|u|+|v|)\,\Delta t}{h_{min}}$ (convective part), with h_{min} the minimum mesh size in the **physical domain**.

Mapping In order to correctly model the shape of the body, a body-fitted (or conformal) grid is used in the physical domain. However, the discretization of the N-S equations in cartesian coordinates using FD schemes is no longer feasible. Therefore, a mapping has to be defined (see Fig. 2) between a rectangular computational domain $\boldsymbol{\xi} = (\xi_1, \xi_2)$ and the physical domain $\boldsymbol{x} = (x, y)$. The N-S equations are then discretized in the computational domain through a mapping which relates the physical and the computational coordinates, i.e. $\mathbf{x} = f(\boldsymbol{\xi})$. As a consequence, the differential operators in the computational domain will have to be modified in accordance with the mapping operation.

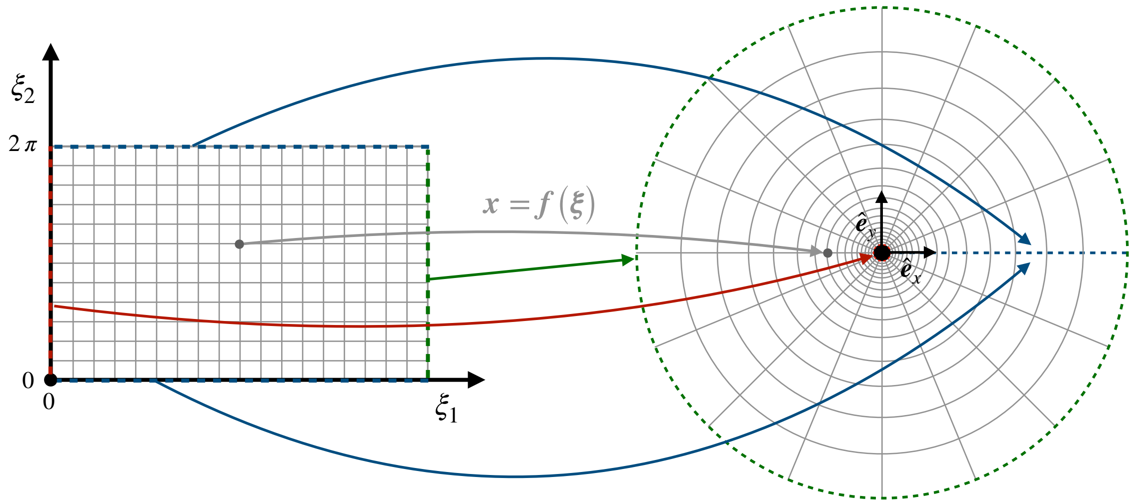

Figure 2: Mapping from the computational to the physical domain in the case of the cylinder.

After the mapping, the mesh obtained in the physical domain is orthogonal, i.e. its intersecting edges are still perpendicular to each other. A stretching in the normal direction is also integrated in the mapping in order to refine the grid close to the wall (i.e. in the boundary layer region, where the velocity gradients are important) while it coarsens progressively in the wake region and far from the region of interest. The unknown velocity field $\mathbf{v}$ will be expressed in terms of its normal (u_n) and tangential (u_θ) components. The mesh is staggered (MAC mesh) to avoid a decoupling between the pressure and the velocity fields. An example of the staggered orthogonal mesh is shown in Fig. 3.

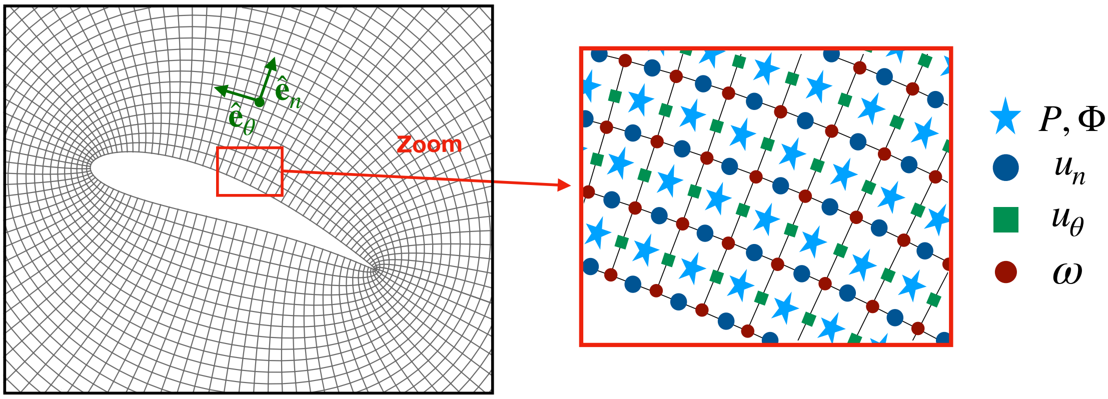

Figure 3: Example of a (coarse) orthogonal staggered mesh as handled by your FD solver.

Differential operators The differential operators can be expressed in the orthogonal system of coordinates as:

$$\mathbf{H} = \mathbf{v} \cdot \nabla \mathbf{v} = \begin{bmatrix} (\mathbf{v} \cdot \nabla \mathbf{v}) \cdot \hat{\mathbf{e}}_n \\ (\mathbf{v} \cdot \nabla \mathbf{v}) \cdot \hat{\mathbf{e}}_\theta \end{bmatrix} = \begin{bmatrix} \frac{u_n}{h_1}\frac{\partial u_n}{\partial \xi_1} + \frac{u_\theta}{h_2}\frac{\partial u_n}{\partial \xi_1} + \frac{u_\theta}{h_1\, h_2}\left(u_n \frac{\partial h_1}{\partial \xi_2} - u_\theta \frac{\partial h_2}{\partial \xi_1}\right) \\ \frac{u_n}{h_1}\frac{\partial u_\theta}{\partial \xi_1} + \frac{u_\theta}{h_2}\frac{\partial u_\theta}{\partial \xi_2} + \frac{u_n}{h_1\, h_2}\left(u_\theta \frac{\partial h_2}{\partial \xi_1} - u_n \frac{\partial h_1}{\partial \xi_2}\right) \end{bmatrix} \tag{7}$$

$$\nabla P = \begin{bmatrix} \nabla P \cdot \hat{\mathbf{e}}_n \\ \nabla P \cdot \hat{\mathbf{e}}_\theta \end{bmatrix} = \begin{bmatrix} \frac{1}{h_1}\frac{\partial P}{\partial \xi_1} \\ \frac{1}{h_2}\frac{\partial P}{\partial \xi_2} \end{bmatrix} \tag{8}$$

$$\nabla^2 \Phi = \frac{1}{h_1 h_2}\left[\frac{\partial}{\partial \xi_1}\left(\frac{h_2}{h_1}\frac{\partial \Phi}{\partial \xi_1}\right) + \frac{\partial}{\partial \xi_2}\left(\frac{h_1}{h_2}\frac{\partial \Phi}{\partial \xi_2}\right)\right] \tag{9}$$

$$\nabla \cdot \mathbf{v} = \frac{1}{h_1 h_2}\left[\frac{\partial}{\partial \xi_1}(h_2 u_n) + \frac{\partial}{\partial \xi_2}(h_1 u_\theta)\right] \tag{10}$$

$$\omega = \nabla \times \mathbf{v} = \frac{1}{h_1 h_2}\left[\frac{\partial}{\partial \xi_1}(h_2 u_\theta) - \frac{\partial}{\partial \xi_2}(h_1 u_n)\right] \tag{11}$$

$$\nabla^2 \mathbf{v} = -\nabla \times \omega + \nabla(\nabla \cdot \mathbf{v}) = \begin{bmatrix} (\nabla^2 \mathbf{v}) \cdot \hat{\mathbf{e}}_n \\ (\nabla^2 \mathbf{v}) \cdot \hat{\mathbf{e}}_\theta \end{bmatrix} = \begin{bmatrix} -\frac{1}{h_2}\frac{\partial \omega}{\partial \xi_2} \\ \frac{1}{h_1}\frac{\partial \omega}{\partial \xi_1} \end{bmatrix} \tag{12}$$

with $h_1 \triangleq |\frac{\partial \mathbf{x}}{\partial \xi_1}|$, $h_2 \triangleq |\frac{\partial \mathbf{x}}{\partial \xi_2}|$ the form factors, and $\hat{\mathbf{e}}_n \triangleq \frac{1}{h_1}\frac{\partial \mathbf{x}}{\partial \xi_1}$ and $\hat{\mathbf{e}}_\theta \triangleq \frac{1}{h_2}\frac{\partial \mathbf{x}}{\partial \xi_2}$ the local orthogonal base vectors.

You will notice that all differential operators involve the form factors and/or their derivatives. All these terms should be computed only once at the beginning of the simulation and further stored to be used at each time step. We also observe that the diffusive term can be computed as the opposite of the rotational of the vorticity field. We really advice you to use this expression. Another possible expression for the Laplacian can be retrieved using the orthogonal mapping properties but it requires the computation of a dozen of terms (much less convenient). As the Laplacian of the velocity field is computed by discretizing a quantity twice, the second order accuracy will be lost even if you use second order schemes at each step. In order to preserve the second order accuracy of the Laplacian operator, you should implement a third order scheme (at least) for the discretization of Eq. (11) and a second order scheme for Eq. (12).

Flow past a cylinder The mapping proposed for the cylinder is the following :

$$\begin{bmatrix} x \\ y \end{bmatrix} = f(\xi_1, \xi_2) = \begin{bmatrix} r(\xi_1)\,\cos(\xi_2) \\ r(\xi_1)\,\sin(\xi_2) \end{bmatrix}$$

with $r(\xi_1) = \frac{D}{2} + \beta\left(e^{\alpha\,\xi_1} - 1\right)$ and ξ_1 and ξ_2 discretized.

In this mapping, α, β and $\Delta\xi_1$ are the unknowns. Those can be fixed using three constraints:

1. We want the outer boundary of the domain $r(\xi_1) = \frac{D}{2} + H$ to end up in $\xi_1 = 1$, i.e. $H = \beta\left(e^{\alpha} - 1\right)$. It follows that

$$\alpha = \log\left(\frac{H}{\beta} + 1\right). \tag{13}$$

2. A second equation can be obtained from the the definition of the stretching factor defined as the size ratio between two adjacent cells in the normal direction: $\gamma \triangleq \frac{h_n^{(i+1)}}{h_n^{(i)}}$. Expressing $h_n^{(i+1)}$ and $h_n^{(i)}$ from the mapping function, we obtain:

$$\begin{aligned} h_n^{(i+1)} &= \beta\left(e^{\alpha\,(i+1)\,\Delta\xi_1} - e^{\alpha\,i\,\Delta\xi_1}\right), \\ h_n^{(i)} &= \beta\left(e^{\alpha\,i\,\Delta\xi_1} - e^{\alpha\,(i-1)\,\Delta\xi_1}\right). \end{aligned}$$

   Injecting those expressions in the definition of γ and after doing some algebra, we get that

$$\gamma = e^{\alpha\,\Delta\xi_1},$$

   which provides an equation for $\alpha\,\Delta\xi_1$:

$$\alpha\,\Delta\xi_1 = \log\left(\gamma\right). \tag{14}$$

3. Finally, we fix the first cell size at the wall in the normal direction $h_n^{(1)}$:

$$h_n^{(1)} = \beta\left(e^{\alpha\Delta\xi_1} - 1\right).$$

   Injecting Eq. (14) into this last expression, we obtain the third equation:

$$\beta = \frac{h_n^{(1)}}{\gamma - 1}. \tag{15}$$

Using Eq. (15), you can directly obtain β from $h_n^{(1)}$ and γ. Knowing β and from H, we can next compute α using Eq. (13). And finally knowing α and using Eq. (14), we obtain $\Delta\xi_1$ and therefore also the number of cells n_{ξ_1} in the normal direction as $n_{\xi_1} = 1/\Delta\xi_1$. This last number is probably not an integer. Round it up to the nearest integer: $n_{\xi_1} = ceil\,(1/\Delta\xi_1)$ so that the extent of the domain is equal to or slightly larger than the prescribed value H. The computational domain will then also be slightly larger than 1 along the ξ_1 axis.

We also ask you to initialize the pressure and velocity fields to a potential flow. A potential flow is the solution obtained when you assume the flow as inviscid (i.e. irrotationnal). This flow has an analytical expression:

$$
\begin{aligned}
u_n &= U_\infty \left(1 - \frac{(D/2)^2}{r^2}\right) \cos(\theta) \\
u_\theta &= -U_\infty \left(1 + \frac{(D/2)^2}{r^2}\right) \sin(\theta)
\end{aligned}
$$

As a potential flow satisfies the Bernouilli principle, the initial reduced pressure can be obtained as:

$$
P = -\frac{|\mathbf{v}|^2}{2}.
$$

Accuracy When a numerical simulation aims to capture boundary layers, a good diagnostic for its accuracy at the wall is the dimensionless mesh size of the first point in the normal direction: $y^+ = \frac{\sqrt{\tau_w}\, h_n^{(1)}}{\nu}$, with the wall shear stress $\tau_w = \nu \left(\frac{\partial u_\theta}{\partial n}\right)\Big|_w = \nu\,(\omega \times \boldsymbol{n})\,|_w$. Usually, the first normal mesh size $h_n^{(1)}$ is fixed so that $y^+ < 1$. Due to the stretching, the mesh resolution will get coarser when moving away form the body. We ask you to set the stretching factor and the resolution in the tangential direction in such a way to maintain the mesh Reynolds number based on the vorticity $Re_w = \frac{|\omega| h_{max}^2}{\nu} < 40$ in the $12\,D$ region surrounding the cylinder. The extent of the domain will be set to $H = 50\,D$.

Flow past an airfoil The mapping function for the airfoil is much more complex. A routine giving the form factors and their derivatives for a regularized Joukowsky airfoil at 20° of angle of attack is provided. In this case, the stretching factor, first normal mesh size, tangential resolution and domain extent will be fixed for you. A routine initializing the velocity and pressure field is also provided.

You are asked to :

1. Produce a numerical code able to simulate external flows on any type of orthogonal meshes (e.g for any type of orthogonal mapping).
2. Simulate the flow around the cylinder at $Re = \frac{U_\infty D}{\nu} = 550$ up to a dimensionless time $\frac{t_{\text{end}}\, U}{D} = 50$, and provide:
   (a) The lift and drag coefficients as a function of the dimensionless time.

(b) The vorticity field for the dimensionless times $\frac{tU}{D} = 1, 2, 3, 4, 6, 10, 15, 20, 40, 50$. Zoom only on the 10 D surrounding the cylinder to produce those images.

(c) The Strouhal number based on the lift coefficient when the shedding is developped.

In order to trigger the vortex shedding, we ask you to add a sinusoidal perturbation on the vertical component of the velocity over one Strouhal period. The amplitude of this perturbation should be around 25% of the upstream velocity. Strouhal number for this Reynolds is about 0.19.

3. Simulate the flow around the Joukowsky airfoil at 20° of angle of attack and $Re = \frac{U_\infty c}{\nu} = 1000$ until a dimensionless time of $\frac{t_{end} U}{c} = 15$, and provide:

   (a) The lift and drag coefficients as a function of the dimensionless time.

   (b) The vorticity field for the dimensionless times $\frac{tU}{c} = 0.1, 0.25, 0.5, 1, 2, 3, 5, 8, 10, 15$. Zoom only on the 7 c surrounding the airfoil to produce those images.

4. Comment your results

This project can be accomplished in pairs or alone. Your pdf report must be uploaded by **Friday 15/05/2020, 6 pm** on the Moodle website of the course, together with your C source code. Please compress your code and your report into one single archive file before uploading it.