Fluid-rigid body interaction solver 2D (FRiBoIS2D): general notes

Alessandro Nitti

August 2, 2024

1 Governing equations

The governing equations of a rigid body subjected to an incompressible, Newtonian flow are solved in the following dimensionless form. They entail the Newton law of motion for the body center of mass, the cotinuity equation and the Navier-Stokes equation:

$$\rho \mathbf{A} \frac{\partial^2 \mathbf{x}}{\partial t^2} + \mathbf{C} \frac{\partial \mathbf{x}}{\partial t} + \mathbf{K} \Delta \mathbf{x} = \mathbf{c} + \mathbf{g}\,, \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0\,, \tag{2}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\, \mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} + \mathbf{f}_{\mathrm{IB}}\,. \tag{3}$$

The rigid body can undergo plane motion. Therefore, its dynamics is fully described by the 3 degrees of freedom of the center of mass $\mathbf{x} = [z/L, y/L, \alpha]$, where L is the reference length scale used to build the dimensionless problem. The other reference parameters are the fluid density ρ_f, the farfield velocity U, and the fluid viscosity ν. The fluid motion is solved for the dimensionless velocity $\mathbf{u} = \tilde{\mathbf{u}}/U$ and pressure field $p = \tilde{p}/(\rho_f)U^2$.
The other parameters appearing in the rigid body equation are thus obtained after scaling with the reference quantities:
ρ_s denotes the solid density, J_0 the polar moment of area with respect to the rotation center, F_z, F_y

Dimensionless parameter	Symbol	Definition
density ratio	ρ	$\rho = \rho_s/\rho_f$
dim.less area	$\mathbf{A}$	$\mathbf{A} = \left[A/L^2, A/L^2, J_0/L^4\right]$
dim.less damping	$\mathbf{C}$	$\mathbf{C} = \left[b_z/(\rho_f U L), b_y/(\rho_f U L), b_\alpha/(\rho_f U L^3)\right]$
dim.less stiffness	$\mathbf{K}$	$\mathbf{K} = \left[k_z/(\rho_f U^2), k_y/(\rho_f U^2), k_\alpha/(\rho_f U^2 L^2)\right]$
force coefficient	$\mathbf{c}$	$\mathbf{c} = \left[F_z/(\rho_f U^2 L), F_y/(\rho_f U^2 L), T/(\rho_f U^2 L^2)\right]$
Reynolds number	Re	Re$= UL/\nu$

Table 1: Summary of applied boundary conditions.

and T are the resultant hydrodynamic forces/moment over the body surface. Conversely, $\mathbf{g}$ denotes the gravity force array.

2 Methodology

The coupled fluid–structure system is simulated by means of the immersed boundary framework described by [3] and [6]. The flow equations are integrated by a classical fractional step method based on conservative second-order accurate centered differences over a staggered grid [4]. Time integration is performed by a semi-implicit scheme, where the convective terms are advanced by an explicit third-order Runge–Kutta scheme and the diffusive terms by an implicit Crank–Nicholson scheme. The large penta-diagonal systems arising from the semi-implicit integration of the momentum equation are solved by means of an approximate factorization, whereas the Poisson problem formulated to enforce continuity is solved with a direct solver to ensure mass conservation to a tight tolerance. Further details of the numerical method for the fluid phase can be found in [7]. Following the approach presented in [9] the forcing term necessary to fulfill the no-slip condition is computed on Lagrangian markers laying on the immersed surface in the form of a volume force field, and then transferred to the Eulerian nodes. The information at the Lagrangian marker location is interpolated by means of a moving-least squares approach [10], which is recognized to attenuate spurious oscillations of hydrodynamic loads for moving interfaces while preserving second-order accuracy in space. The present solver has been validated and verified extensively for a broad variety of stationary and moving boundary problems in earlier works [3, 5]. The rigid body equation of motion is transformed into two first-order ordinary differential equations by a state-space formulation, which in turn are integrated by a fourth-order explicit Runge–Kutta scheme. The fluid and structural sub-systems are solved in a sequential fashion since the involved phase density ratios do not compromise the stability and the accuracy of the loosely coupled approach. The reader can find a numerical evidence for this assumption in the work by Borazjani, Ge and Sotiropoulos [1]. A Dirichelet velocity boundary condition is used at the domain inlet, and a radiative outflow boundary condition at the domain outlet. Free-slip condition are specified for the upper end lower boundaries. The relative spacing between adjacent Lagrangian markers is set equal to $0.5\Delta x$, with Δx being the local Eulerian grid spacing, as numerical trade-off between accuracy of the interface condition and computational expense [6]. Given the present IB treatment, the flow field across the surface presents a smooth transition layer whose thickness takes at most two Eulerian cells, as shown by de Tullio and Pascazio [3] with numerical experiments. Therefore, viscous and pressure loads are evaluated by interpolating the field variables at a probe created along the outward-pointing normal from the surface [11]. The probe length is selected as the local averaged cell size.

3 Test cases

Vortex-Induced vibrations of a cyrcular cylinder The first test consists of a circular cylinder undergoing vortex-induced vibrations. The cylinder has 1 degree of freedom, i.e., it is allowed to oscillate in the y direction. With zero damping, the governing equations yield:

$$\rho A \frac{\partial^2 y}{\partial t^2} + \xi_y \Delta y = c_y \,, \tag{4}$$

where $\xi_y = k_y/\rho_f U^2 L$ and $c_y = F_y/\rho_f U^2 L$. The net hydrodynamic loading is measured via the reduced velocity, which can be expressed as a function of the dimensionless parameters as: $U_r = 2\pi\sqrt{\rho A/\xi_y}$. In this test, the parameters are chosen to let the cylinder undergo the lock-in regime: $\rho = 10$, $A = \pi/4$, $U_r = 5.5$, $\mathrm{Re} = 100$. Under lock-in the vortex shedding frequency remains locked at the system natural frequency when increasing the hydrodynamic loading conditions, and the body exhibits large amplitude oscillations. Results are compared to those proposed by Bourguet and LoJacono [2].

Galloping of a rectangular wing The second test entails the rotational galloping of a rectangular wing with nonzero damping. The simulated rectangle is only allowed to rotate about its center of mass. Thus, the only governing equation is:

$$\rho J_0 \frac{\partial^2 \alpha}{\partial t^2} + \mu_\alpha \frac{\partial \alpha}{\partial t} + \xi_\alpha \Delta \alpha = c_T \,, \tag{5}$$

where $\mu_\alpha = b_\alpha/(\rho_f U L^3)$, $\xi_y = k_\alpha/(\rho_f U^2 L^2)$ and $c_T = T/(\rho_f U^2 L^2)$. The reduced velocity is analogously defined as $U_r = 2\pi\sqrt{\rho J_0/\xi_\alpha}$, whereas the critical damping parameter can be computed as $\zeta = b_\alpha/(2\sqrt{\xi_\alpha \rho J_0})$

The chosen parameters are taken from the original work by Robertson et al. [8], who first proposed this investigation: $\rho = 40$, $J_0 = 2.5$, $U_r = 40$, $\zeta = 0.5$, $\mathrm{Re} = 250$. The rectangle has aspect ratio 3.

References

[1] Iman Borazjani, Liang Ge, and Fotis Sotiropoulos. Curvilinear immersed boundary method for simulating fluid structure interaction with complex 3d rigid bodies. *Journal of Computational physics*, 227(16):7587–7620, 2008.

[2] Rémi Bourguet and David Lo Jacono. Flow-induced vibrations of a rotating cylinder. *Journal of Fluid Mechanics*, 740:342–380, 2014.

[3] Marco D de Tullio and Giuseppe Pascazio. A moving-least-squares immersed boundary method for simulating the fluid–structure interaction of elastic bodies with arbitrary thickness. *Journal of Computational Physics*, 325:201–225, 2016.

[4] John Kim and Parviz Moin. Application of a fractional-step method to incompressible navier-stokes equations. *Journal of computational physics*, 59(2):308–323, 1985.

[5] A Nitti, G De Cillis, and MD de Tullio. Cross-flow oscillations of a circular cylinder with mechanically coupled rotation. *Journal of Fluid Mechanics*, 943:A30, 2022.

[6] Alessandro Nitti, Josef Kiendl, Alessandro Reali, and Marco D de Tullio. An immersed-boundary/isogeometric method for fluid–structure interaction involving thin shells. *Computer Methods in Applied Mechanics and Engineering*, 364:112977, 2020.

[7] Paolo Orlandi. *Fluid flow phenomena: a numerical toolkit*, volume 55. Springer Science & Business Media, 2000.

[8] I Robertson, L Li, SJ Sherwin, and PW Bearman. A numerical study of rotational and transverse galloping rectangular bodies. *Journal of fluids and structures*, 17(5):681–699, 2003.

[9] Markus Uhlmann. An immersed boundary method with direct forcing for the simulation of particulate flows. *Journal of computational physics*, 209(2):448–476, 2005.

[10] Marcos Vanella and Elias Balaras. A moving-least-squares reconstruction for embedded-boundary formulations. *Journal of Computational Physics*, 228(18):6617–6628, 2009.

[11] Shizhao Wang, Marcos Vanella, and Elias Balaras. A hydrodynamic stress model for simulating turbulence/particle interactions with immersed boundary methods. *Journal of Computational Physics*, 382:240–263, 2019.