INSTITUT DE FRANCE
Académie des sciences

Comptes Rendus
Mécanique

Domenico Panunzio, Maria-Adela Puscas and Romain Lagrange

FSI—vibrations of immersed cylinders. Simulations with the engineering open-source code TrioCFD. Test cases and experimental comparisons

Volume 350 (2022), p. 451-476

https://doi.org/10.5802/crmeca.126

Les Comptes Rendus. Mécanique *sont membres du*
Centre Mersenne pour l'édition scientifique ouverte
www.centre-mersenne.org
e-ISSN : 1873-7234

Comptes Rendus
Mécanique
2022, Vol. 350, p. 451-476
`https://doi.org/10.5802/crmeca.126`

Short paper / *Note*

FSI—vibrations of immersed cylinders. Simulations with the engineering open-source code TrioCFD. Test cases and experimental comparisons

Domenico Panunzio[¤, a], **Maria-Adela Puscas**[b] **and Romain Lagrange**[¤, a]

[a] Université Paris-Saclay, CEA, Service d'Etudes Mécaniques et Thermiques, 91191, Gif-sur-Yvette, France

[b] Université Paris-Saclay, CEA, Service de Thermohydraulique et de Mécanique des Fluides, 91191, Gif-sur-Yvette, France

E-mails: domenico.panunzio@cea.fr (D. Panunzio), maria-adela.puscas@cea.fr (M.-A. Puscas), romain.lagrange@cea.fr, romain.g.lagrange@gmail.com (R. Lagrange)

Abstract. In this paper, we assess the capabilities of the Arbitrary Lagrangian–Eulerian (ALE) method implemented in the open-source code TrioCFD to tackle down two fluid–structure interaction problems involving moving boundaries. To test the code, we first consider the bi-dimensional case of two coaxial cylinders moving in a viscous fluid. We show that the two fluid forces acting on the cylinders are in phase opposition, with amplitude and phase that only depend on the Stokes number, the dimensionless separation distance and the Keulegan–Carpenter number. Throughout a detailed parametric study, we show that the self (resp. cross) added mass and damping coefficients decrease (resp. increase) with the Stokes number and the separation distance. Our numerical results are in perfect agreement with the theoretical predictions of the literature, thereby validating the robustness of the ALE method implemented in TrioCFD. Then, we challenge the code by considering the case of a vibrating cylinder located in the central position of a square tube bundle. In parallel to the numerical investigations, we also present a new experimental setup for the measurement of the added coefficient, using the direct method introduced by Tanaka. The numerical predictions for the self-added coefficients are shown to be in very good agreement with a theoretical estimation used as a reference by engineers. A good agreement with the experimental results is also obtained for moderate and large Stokes numbers, whereas an important deviation due to parasitic frequencies in the experimental setup appears for low Stokes number. Still, this study clearly confirms that the ALE method implemented in TrioCFD is particularly efficient in solving fluid–structure interaction problems. As an open-source code, and given its ease of use and its flexibility, we believe that TrioCFD is thus perfectly adapted to engineers who need simple numerical tools to tackle down complex industrial problems.

Keywords. Vibration, Fluid–structure interaction, Added mass, Added damping, Stokes number, ALE method, TrioCFD.

Manuscript received 30 September 2021, revised 21 July 2022 and 21 July 2022, accepted 22 July 2022.

[¤] Corresponding authors.

ISSN (electronic) : 1873-7234 `https://comptes-rendus.academie-sciences.fr/mecanique/`

1. Introduction

The accurate estimation of the force acting on a body moving in a viscous fluid is of crucial importance for engineers, particularly those working in fields such as turbomachinery [1], heat exchangers, tube banks [2,3] or energy harvesting of flexible structures [4–7]. When an immersed body is subjected to a small amplitude of motion, Stokes [8] showed that the fluid force is the sum of two terms: an added mass term related to the body's acceleration and a damping term related to the body's velocity. The concept of added mass and damping terms can be generalized to multiple immersed bodies, introducing some self and cross-added coefficients. In such a case, the self-added coefficients relate the fluid force on a body to its motion. On their hand, the cross-added coefficients relate the forces on a body to the other bodies' motion. The determination of the added coefficients has been the topic of considerable experimental, [9–14] and theoretical studies, mostly based on a Helmholtz decomposition associated with a method of images [15–20] or a conformal transformation [21–28]. These theoretical approaches have shown their efficiency in predicting the added-coefficients in some simple configurations, but extending to the industrial context remains delicate. As a consequence, engineers developed fast and robust numerical approaches, such as the immersed body method [29], the cut-cell method [30] or the penalization method [27,31–35].

In the present work, we aim to analysis the efficiency of the Arbitrary Lagrangian–Eulerian (ALE) method [36–38] implemented in the open-source code TrioCFD [39, 40], developed by the CEA for the nuclear industry. TrioCFD is a C++ object-oriented parallel software designed for calculations of unsteady laminar or turbulent fluid flows. The calculations are performed on structured (parallelepipeds) and non-structured (tetrahedrons) meshes of several millions of control volumes (hybrid finite volume element, see [39–41]). The code structure is flexible, allowing the user to choose the discretization method, the convection and time schemes, as well as the turbulence model.

This article is organized as follows. Section 2 presents the ALE method and the main equations solved by TrioCFD. In Section 3, we test the code, considering the vibration of two coaxial cylinders separated by a fluid layer. The added coefficients are extracted from the numerical predictions of the fluid forces, and the results are compared with those of the literature. In Section 4, we challenge the code, considering the vibration of a cylindrical tube located in the central position of a square tube array. Our numerical results for the fluid coefficients are compared with a phenomenological estimation used by engineers of the nuclear industry and with some experimental measurements that we have performed on a new experimental set-up built at CEA. Finally, Section 5 summarizes our findings.

2. Numerics

The numerical simulations of the Navier–Stokes equations have been conducted on the 1.8.0 version of TrioCFD, a programmable CFD code based on the TRUST platform and a C++ language architecture, see [39,40]. This open source code allows the user to choose a wide range of options and parameters, among which the discretization schemes in space and time, the turbulence models (RANS or LES approaches) or the boundary conditions. Given its flexibility, the code is widely used in the nuclear industry for massive parallel and high performance calculations.

To determine the flow of a fluid, it is necessary to describe the kinematics of all its material particles throughout time. To do so, one can adopt either an Euler description of motion, in which a fluid particle is identified by its instantaneous position, or a Lagrange description of motion, in which a fluid particle is identified by its initial position. Both descriptions are totally equivalent, leading to different forms of the Navier–Stokes equations that can be discretized on a stationary mesh grid (Euler) or a mesh grid that follows the motion of the fluid particles (Lagrange).

In both cases, the mesh grids do not account for the motion of the boundaries, which makes the numerical simulations of the related Navier–Stokes equations delicate.

To overcome this problem, several approaches, such as the immersed boundary methods [42–44], or the ALE method [36–38] have been developed.

In the ALE approach, a fluid particle is identified by its position relative to a frame moving with a nonuniform velocity v_{ALE}. In this new frame of reference, the Navier–Stokes equations are expressed as

$$\begin{cases} \nabla\cdot v = 0, & \text{(a)} \\ \dfrac{\partial J v}{\partial t} = J\left(\nu\Delta v - \nabla\cdot((v - v_{ALE})\otimes v) - \dfrac{1}{\rho}\nabla p\right), & \text{(b)} \end{cases} \tag{1}$$

with J the Jacobian of the transformation between the ALE and the Lagrange descriptions. The ALE method is actually a hybrid description between the Euler and the Lagrange descriptions, both of them corresponding to the particular cases $v_{ALE} = 0$ and $v_{ALE} = v$, respectively. In the ALE framework, the choice of v_{ALE} is arbitrary as long as the deformation of the mesh grid remains under control. For moderate deformations, v_{ALE} is usually defined as the solution of an auxiliary Laplace problem, see [45].

In case of unstructured meshes (triangles in 2D or tetrahedrons in 3D), TrioCFD uses Finite Volume-Element (FVE) approach to solving discretized Navier–Stokes equations. These methods combine the finite element method with the finite volume method, gathering the advantages of each approach for incompressible Navier–Stokes problems [40, 41].

Applying the numerical FVE discretization on (1), along with the Forward Euler scheme for the time discretization, results in the following discrete system:

$$\begin{cases} [M]\dfrac{J^{n+1}V_h^{n+1} - J^n V_h^n}{\Delta t} = J^{n+1}\left([A]V_h^{n+1} - [L(V_h^n)]V_h^{n+1} + [L(V^n)]V_{h,ALE}^{n+1} - [G]P_h^{n+1}\right), & \text{(a)} \\ [D]V_h^{n+1} = 0, & \text{(b)} \end{cases} \tag{2}$$

where V_h and P_h are the unknowns vectors of the discrete fluid flow velocity vector and pressure, Δt is the time step, $[M]$ is the mass matrix operator, $[A]$ is the discrete diffusion operator, $[L(V_h)]$ is the non-linear discrete convection operator, $[G]$ is the discrete gradient operator, and $[D]$ is the discrete divergence operator. The superscripts n and $n+1$ indicate the time step at which the variable is computed.

In order to solve the velocity–pressure coupling, a multi-step (projection–correction) technique [46, 47] is employed, where an intermediate (predicted) velocity V_h^* is computed:

$$J^{n+1}\left(\frac{1}{\Delta t}[M] - [A] + [L(V_h^n)]\right)V_h^* = \frac{1}{\Delta t}[M]J^n V_h^n + J^{n+1}[L(V_h^n)]V_{h,ALE}^n - J^{n+1}[G]P_h^n, \tag{3}$$

and the mass conservation is then enforced by solving a Poisson equation for pressure:

$$[D][M]^{-1}[G]P_h' = \frac{1}{\Delta t}[D]V_h^*. \tag{4}$$

Finally, the velocity is updated using the predicted velocity V_h^* and the pressure increment P_h':

$$V_h^{n+1} = V_h^* - \Delta t[M]^{-1}[G]P_h', \quad P_h^{n+1} = P_h^n + P_h'. \tag{5}$$

In the present work, the linear systems (3) is solved by the iterative solver GMRES from the PETSc library [48].

3. Study 1. Vibrations of two coaxial cylinders in a viscous fluid

In this section, we test the capabilities of the code TrioCFD along with the ALE approach, considering the 2D case of two coaxial cylinders vibrating in a viscous fluid. The goal of this preliminary work is to introduce all the important concepts that will be used in the study of the

Figure 1. Configuration with two concentric cylinders. The fluid in the annulus region is homogeneous with mass density ρ and kinematic viscosity ν.

vibrations of a tube in a square bundle, considered in Section 4. In what follows, we introduce the problem, present the numerical setup and finally discuss our numerical results in comparison with the theoretical estimations of [49, 50].

3.1. Presentation of the problem and governing equations

Let C_1 and C_2 be two concentric cylinders with diameters D_j and boundaries ∂C_j, $j \in \{1, 2\}$, see Figure 1. One cylinder, either C_1 or C_2, oscillates in the (x, y) plane with a simple harmonic motion of angular frequency Ω and a displacement amplitude U. The fluid in the annulus region is Newtonian, homogeneous, with mass density ρ and kinematic viscosity ν. The fluid flow generated by the oscillation of one cylinder is assumed as incompressible and two-dimensional.

The Navier–Stokes equations and the boundary conditions for the fluid flow (v, p) are:

$$\begin{cases} \nabla\cdot v = 0, & \text{(a)} \\ \dfrac{\partial v}{\partial t} + (v\cdot\nabla)v + \dfrac{1}{\rho}\nabla p - \nu\Delta v = 0, & \text{(b)} \\ v - \dfrac{dU}{dt} = 0 \quad \text{on } \partial C_j,\ j \in \{1, 2\}. & \text{(c)} \end{cases} \tag{6}$$

The (6c) ensures the continuity of velocities at the cylinder boundaries ∂C_j. The fluid force acting on C_j is the sum of a pressure and a viscous term

$$F_j = -\int_{\partial C_j} p\, n_j\, dL_j + \rho\nu \int_{\partial C_j} [\nabla v + (\nabla v)^T]\cdot n_j\, dL_j, \tag{7}$$

where n_j is the outward normal unit vector to ∂C_j, see Figure 1, and dL_j is an infinitesimal line element of integration.

Picking $D_1/2$ and Ω^{-1} as a characteristic length and time, the dimensionless quantities t^*, u^*, v^*, p^* and f_j^* are defined as

$$t = \Omega^{-1} t^*, \quad U = U u^*, \quad v = U\Omega v^*, \quad p = \rho U \frac{D_1}{2}\Omega^2 p^*, \quad F_j = \rho U \left(\frac{D_1}{2}\Omega\right)^2 f_j^*. \tag{8}$$

To reduce the number of parameters of the problem, the following rescaled quantities are also defined

$$\varepsilon = \frac{D_2}{D_1}, \quad KC = \frac{U}{D_1}, \quad Sk = \frac{D_1^2(\Omega/2\pi)}{\nu}, \tag{9}$$

as the dimensionless separation distance, the Keulegan–Carpenter number and the Stokes number, respectively. Introducing (8) in (6) yields the dimensionless Navier–Stokes equations

$$\begin{cases} \nabla\cdot v^* = 0, & \text{(a)} \\ \dfrac{\partial v^*}{\partial t^*} + 2KC(v^*\cdot\nabla)v^* + \nabla p^* - \dfrac{2}{Sk}\Delta v^* = 0, & \text{(b)} \\ v^* - \dfrac{du^*}{dt^*} = 0 \quad \text{on } \partial C_j,\ j = \{1,2\}. & \text{(c)} \end{cases} \tag{10}$$

The dimensionless fluid force acting on C_j is

$$f_j^* = -\int_{\partial C_j} p^* n_j \, dl_j + \frac{2}{Sk}\int_{\partial C_j} [\nabla v^* + (\nabla v^*)^T]\cdot n_j \, dl_j, \tag{11}$$

with $dl_j = 2dL_j/D_1$.

For $KC \ll 1$, the dimensionless Navier–Stokes equations are linear. It follows that the fluid forces are linear combinations of the cylinder velocity du^*/dt^* and acceleration d^2u^*/dt^{*2}

$$f_j^* = -\left([M_j]\frac{d^2u^*}{dt^{*2}} + [C_j]\frac{du^*}{dt^*}\right), \tag{12}$$

with

$$[M_j] = \begin{pmatrix} m_{self}^{(j)} \\ m_{self}^{(j)} \end{pmatrix} \quad \text{and} \quad [C_j] = \begin{pmatrix} c_{self}^{(j)} \\ c_{self}^{(j)} \end{pmatrix} \quad \text{if } C_j \text{ is moving,} \tag{13a}$$

$$[M_j] = \begin{pmatrix} m_{cross} \\ -m_{cross} \end{pmatrix} \quad \text{and} \quad [C_j] = \begin{pmatrix} c_{cross} \\ -c_{cross} \end{pmatrix} \quad \text{if } C_j \text{ is stationary.} \tag{13b}$$

The self-added mass and damping coefficients $m_{self}^{(j)}$ and $c_{self}^{(j)}$ relate the fluid force on the moving cylinder to its own motion. The cross-added mass and damping coefficients m_{cross} and c_{cross} relate the fluid force on the stationary cylinder to the motion of the other cylinder. The fluid added coefficients are functions of the dimensionless separation distance ε and the Stokes number Sk.

For a harmonic displacement $u^* = \sin(t^*)e_x$, the dimensionless fluid forces (12) reduce to

$$f_j^* = \left(m_{self}^{(j)}\sin(t^*) - c_{self}^{(j)}\cos(t^*)\right)e_x, \quad \text{if } C_j \text{ is moving,} \tag{14a}$$

$$f_j^* = (m_{cross}\sin(t^*) - c_{cross}\cos(t^*))e_x, \quad \text{if } C_j \text{ is stationary,} \tag{14b}$$

or similarly $f_j^* = h_j \sin(t^* + \varphi_j)e_x$, with

$$h_j = \sqrt{\left(m_{self}^{(j)}\right)^2 + \left(c_{self}^{(j)}\right)^2} \quad \text{and} \quad \varphi_j = -\arctan\left(\frac{c_{self}^{(j)}}{m_{self}^{(j)}}\right) \quad \text{if } C_j \text{ is moving,} \tag{15a}$$

$$h_j = \sqrt{m_{cross}^2 + c_{cross}^2} \quad \text{and} \quad \varphi_j = -\arctan\frac{c_{cross}}{m_{cross}} \quad \text{if } C_j \text{ is stationary.} \tag{15b}$$

Yeh et al. [50] derived a few analytical expressions of the fluid added coefficients. Introducing $\Re$ and $\Im$ the real and imaginary part operators, these expressions are as follows

$$m_{self}^{(1)} = \Re(a_{11}), \quad m_{self}^{(2)} = \varepsilon^2\Re(a_{22}), \quad m_{cross} = \varepsilon\Re(a_{12}), \tag{16a}$$

$$c_{self}^{(1)} = \Im(-a_{11}), \quad c_{self}^{(2)} = \varepsilon^2\Im(-a_{22}), \quad c_{cross} = \varepsilon\Im(-a_{12}), \tag{16b}$$

with $a_{11} = -(1+2a)$, $a_{22} = 2a/\varepsilon$, $a_{12} = 1 - 2a/\varepsilon^2$ and

$$a = \begin{vmatrix} 1 & 1 & J_1(\beta_1) & Y_1(\beta_1) \\ 0 & 1 & \varepsilon^{-1}J_1(\beta_2) & \varepsilon^{-1}Y_1(\beta_2) \\ 2 & 2 & \beta_1 J_0(\beta_1) & \beta_1 Y_0(\beta_1) \\ 0 & 2 & \beta_1 J_0(\beta_2) & \beta_1 Y_0(\beta_2) \end{vmatrix} \begin{vmatrix} 1 & 1 & J_1(\beta_1) & Y_1(\beta_1) \\ \varepsilon^{-2} & 1 & \varepsilon^{-1}J_1(\beta_2) & \varepsilon^{-1}Y_1(\beta_2) \\ 0 & 2 & \beta_1 J_0(\beta_1) & \beta_1 Y_0(\beta_1) \\ 0 & 2 & \beta_1 J_0(\beta_2) & \beta_1 Y_0(\beta_2) \end{vmatrix}^{-1}. \tag{17}$$

In (17), J_n and Y_n are the Bessel functions of the first and second kind, with arguments $\beta_1 = (1-i)\sqrt{2}/2\sqrt{Sk}$ or $\beta_2 = \varepsilon\beta_1$.

From an asymptotic expansion of these functions as $Sk \to \infty$, we show that $(m^{(j)}_{self}, m_{cross}, c^{(j)}_{self}, c_{cross})$ are equivalent to $(\tilde{m}^{(j)}_{self}, \tilde{m}_{cross}, \tilde{c}^{(j)}_{self}, \tilde{c}_{cross})$ with

$$\tilde{m}^{(j)}_{self} = m^{(j)pot}_{self} + \frac{4}{\sqrt{\pi}}\frac{1}{\sqrt{Sk}}, \quad \tilde{m}_{cross} = m^{pot}_{cross} - \frac{4}{\sqrt{\pi}}\frac{1}{\sqrt{Sk}}, \tag{18a}$$

$$\tilde{c}^{(j)}_{self} = \frac{4}{\sqrt{\pi}}\frac{1}{\sqrt{Sk}}\frac{\varepsilon^4+\varepsilon}{(\varepsilon^2-1)^2}, \quad \tilde{c}_{cross} = -\frac{4}{\sqrt{\pi}}\frac{1}{\sqrt{Sk}}\frac{\varepsilon^4+\varepsilon}{(\varepsilon^2-1)^2}, \tag{18b}$$

and

$$m^{(1)pot}_{self} = \frac{\varepsilon^2+1}{\varepsilon^2-1}, \quad m^{(2)pot}_{self} = \frac{\varepsilon^2(\varepsilon^2+1)}{\varepsilon^2-1}, \quad m^{pot}_{cross} = -\frac{2\varepsilon^2}{\varepsilon^2-1}. \tag{19}$$

The terms $m^{(j)pot}_{self}$ and m^{pot}_{cross} are the inviscid limits of $m^{(j)}_{self}$ and m_{cross} as $Sk \to \infty$. These terms can also be obtained from a potential theory in which the fluid forces are only due to the pressure field. Note that $c^{(j)pot}_{self} = c^{pot}_{cross} = 0$ as there are not damping effects in the inviscid framework.

In Section 3.3 the theoretical estimations (16) and (18) are compared with the numerical predictions performed with TrioCFD. Numerically, the fluid forces acting on ∂C_j are computed by the sum of the pressure and viscous terms in (7) given by TrioCFD. To extract the added coefficients from the numerical simulations of the dimensionless fluid forces, we introduce the Fourier inner product over five periods

$$\langle f(t^*), g(t^*)\rangle = \frac{1}{5}\int_0^{10\pi} f(t^*)g(t^*)\,dt^*. \tag{20}$$

From (14), it follows that the added-coefficients are

$$m^{(j)}_{self} = \frac{\langle \sin(t^*), f^*_j(t^*)\cdot e_x\rangle}{\pi} \quad \text{and} \quad c^{(j)}_{self} = -\frac{\langle \cos(t^*), f^*_j(t^*)\cdot e_x\rangle}{\pi} \quad \text{if } C_j \text{ is moving}, \tag{21a}$$

$$m_{cross} = \frac{\langle \sin(t^*), f^*_j(t^*)\cdot e_x\rangle}{\pi} \quad \text{and} \quad c_{cross} = -\frac{\langle \cos(t^*), f^*_j(t^*)\cdot e_x\rangle}{\pi} \quad \text{if } C_j \text{ is stationary}. \tag{21b}$$

3.2. Numerical setup

To discretize the fluid domain, we use an unstructured grid of triangles generated by the Gmsh platform, see [51]. Due to its adaptive and automatic algorithm, Gmsh makes it possible to choose between two different local sizes for the 2D mesh: a small local size, lc_{fine}, for elements close to the moving cylinder and a large local size, lc, for elements close to the stationary cylinder. In this way, a refined mesh is used in the regions with large gradient fields whereas a loose mesh is used in the areas with low gradient fields. In Appendix B, a mesh sensitivity analysis clearly shows a convergence of the mass coefficients as lc and lc_{fine} are changed. The convergence of the damping coefficients is less obvious, especially for high values of Sk. Physically, this is related to the thickness of the boundary layer, which tends to zero as Sk increases. It follows that a finer mesh is required close to a cylinder boundary to account for the thickness of the boundary layer

Figure 2. Study 1. Time evolution of the dimensionless fluid forces, for $Sk = \{10^1, 10^2, 10^3, 10^4\}$, $\eta = 2$ and $KC = 10^{-2}$. The red and blue solid lines correspond to the theoretical estimations of [50], see (16). The dashed lines (indistinguishable from the solid lines for $Sk \geq 10^2$) correspond to the theoretical asymptotic expansion $Sk \to \infty$, see (18). The symbols correspond to the numerical predictions.

and obtain an accurate estimation of the damping terms. In this work, a compromise between the time of calculation and the precision (throughout of this work, we indicate the relative deviations of our numerical results compared to other theoretical or experimental references) is made to choose adequate values for lc and lc_{ne}.

3.3. Results and discussion

The inner cylinder is imposed a sinusoidal displacement in the x-direction for five time periods, i.e. $u^* = \sin(t^*) e_x$ with $t^* \in \{0, 10\pi\}$. The numerical simulations are performed for $\eta \in \{1.25, 1.5, 2\}$, $Sk \in \{10^1, 10^2, 10^3, 10^4\}$ and $KC = 10^{-2}$.

The time evolution of the dimensionless fluid forces is represented in Figures 2 and 3. A perfect agreement is obtained between theoretical estimations and numerical predictions. The fluid forces are sinusoidal functions in phase opposition. In Figures 4 and 5, we show that the amplitudes h_j (resp. phases φ_j) decrease (resp. increase) with both Sk and η, recovering the asymptotic limits as $Sk \to \infty$ (inviscid limit) and as $\eta \to \infty$ (isolated cylinder limit).

Figure 3. Study 1. Time evolution of the dimensionless fluid forces, for $\varepsilon = \{1.25, 1.5, 2\}$, $Sk = 10^4$ and $KC = 10^{-2}$. The red and blue solid lines correspond to the theoretical estimations of [50], see (16). The symbols correspond to the numerical predictions.

The evolution of the added coefficients is depicted in Figures 6 and 7. The numerical predictions are in very good agreement with the theoretical estimations, even if a tiny difference is observed in the range of low Stokes numbers in which the theoretical estimations reach their limit of validity. Still, we confirm that the self (resp. cross) added coefficients decrease (resp. increase) with both Sk and ε, recovering the asymptotic limits as $Sk \to \infty$ (inviscid limit) and as $\varepsilon \to \infty$ (isolated cylinder limit).

We introduce the quantity δ defined as the relative deviation between the numerical and the exact theoretical predictions [50] of some quantity Q: $\delta = |Q_{th} - Q_{num}|/|Q_{th}|$. Figure 8 and Tables A1, A2 in Appendix A show that δ is very low for the added mass coefficients and it becomes greater for the added damping coefficients as Sk increases but seems to be weakly influenced by ε. In fact, for large values of Sk, the damping coefficients become very small and even small differences from the theoretical value produce significant deviations. Physically, this is related to the thickness of the boundary layer, which tends to zero: to reduce δ for the damping coefficients a finer mesh at the cylinder boundaries would be required. We shall note that the theoretical approach [49, 50] is fully linear since the convective term $2\pi KC(v^\star \cdot \nabla)v^\star$ of the Navier–Stokes equation (10b) is neglected. In the numerical simulations, the nonlinear convective term is retained through a small but nonzero Keulegan–Carpenter number $KC = 10^{-2}$. However, this

Figure 4. Study 1. Evolution of the magnitudes η_j and phases φ_j with the Stokes number Sk, for $\varepsilon = 2$ and $KC = 10^{-2}$. The red and blue solid lines correspond to the theoretical estimations of [50], see (16). The dashed lines correspond to the theoretical asymptotic expansion $Sk \to \infty$, see (18). The horizontal dashed lines correspond to the theoretical inviscid limits as $Sk \to \infty$. The symbols correspond to the numerical predictions.

Figure 5. Study 1. Evolution of the magnitudes η_j and phases φ_j with the dimensionless separation distance ε, for $Sk = 10^4$ and $KC = 10^{-2}$. The red and blue solid lines correspond to the theoretical estimations of [50], see (16). The dashed lines correspond to the theoretical asymptotic expansion $Sk \to \infty$, see (18). The horizontal dashed lines correspond to the theoretical isolated cylinder limits as $\varepsilon \to \infty$. The symbols correspond to the numerical predictions.

difference might slightly affect the deviation between the theoretical and numerical results. In any case, the relative deviation for $m^{(1)}_{self}$ (resp. m_{cross}) is smaller than $\simeq 0.6\%$ (resp. $\simeq 0.2\%$) while the relative deviation for $c^{(1)}_{self}$ (resp. c_{cross}) is more pronounced, with $\simeq 5.5\%$ (resp. $\simeq 6.7\%$).

A further analysis is carried out to check the scale invariance of the numerical results obtained for $\varepsilon = 2$, $Sk = 10^4$ and $KC = 10^{-2}$ (see Table A3 in Appendix A). In the numerical simulation, the dimensional quantities were: $D_1 = 1$, $D_2 = 2$, $U = 10^{-2}$ unit of length and $\Omega = 0.0633$ unit of frequency. In this new case study, we set $D_1 = 0.0316$, $D_2 = 0.0632$, $U = 0.0316 \times 10^{-2}$ unit of length and $\Omega = 63.3$ unit of frequency, such that they are self similar configurations in terms of dimensionless numbers: $\varepsilon = 2$, $Sk = 10^4$ and $KC = 10^{-2}$. As expected, the new numerical predictions