This is a postprint version of the following published document:

Moriche, M., Raiola, M., Disceti, S., Ianiro, A., Flores, Ó., García-Villalba, M. (2020) Assessing aerodynamic force estimation with experiments and simulations of flapping-airfoil flows on the verge of three-dimensionality. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*. 234(2), pp.: 428-444.

DOI: <https://doi.org/10.1177/0954410019867570>

© 2020 by Institution of Mechanical Engineers

Assessing aerodynamic force estimation with experiments and simulations of flapping-airfoil flows on the verge of three-dimensionality

M. Moriche¹, M. Raiola², S. Discetti²,
A. Ianiro², O. Flores², M. García-Villalba²

¹ Inst. for Hydromechanics, Karlsruhe Institute of Technology,
Karlsruhe, Germany

² Bioingeniería e Ing. Aeroespacial, Universidad Carlos III de Madrid,
Leganés, Spain

July 12, 2019

Abstract

This paper reports a combined experimental and numerical study of the flow over a rigid airfoil in flapping motion. The setup consists of a heaving and pitching airfoil at a moderate Reynolds number ($Re = 500 - 3600$), at a Strouhal number $St = 0.1$. The aim is to assess the accuracy of two-dimensional (2D) direct numerical simulations in predicting aerodynamic forces in a flow configuration which is nominally 2D but is at the verge of three-dimensionality (3D). The assessment is carried out with experiments, including flow field and aerodynamic force measurements with particle image velocimetry and a load cell. The comparative study shows a good qualitative agreement between the experiments and the simulations at comparable Reynolds numbers both in terms of forces and flow fields, but with some quantitative differences. The quantitative discrepancies between experiments and simulation are analyzed and reduced to inherent differences between experimental and computational setups. It is observed that the significant differences are apparent almost exclusively in the wake evolution. Nonetheless, this is shown to have a minor effect on the aerodynamic force estimation. Overall, the trends observed when varying the mean pitch angle and the pitching amplitude are the same in both experiments and simulations. This suggests that 2D/3D effects do not alter significantly the relationship between the unsteady flow mechanisms (i.e. leading edge vortex) and the aerodynamic forces in the parametric range considered here.

Notation

c	Airfoil chord
c_t, c_l	Aerodynamic thrust and lift coefficients
c_f	Total aerodynamic force coefficient $c_f = \sqrt{c_t^2 + c_l^2}$
f	Frequency of oscillation
h	Vertical displacement of pivoting point
h_0	Amplitude of heaving motion
k	Reduced frequency $k = 2\pi f/U_\infty$
u, w	Streamwise and vertical velocity components
LEV	Leading Edge Vortex
TEV	Trailing Edge Vortex
Re	Reynolds number $Re = cU_\infty/\nu$
St	Strouhal number $St = f h_0/U_\infty$
T	Period of oscillation $T = 1/f$
U_∞	Free stream velocity
α_e	Effective angle of attack $\alpha_e = \theta - \text{atan}(\dot{h}/U_\infty)$
Δt	Time step
Δx	Mesh width
θ	Pitching angle
θ_0	Amplitude of pitching motion
θ_m	Mean pitch angle
ν	Kinematic viscosity
σ	Angle between the aerodynamic force and the free stream
ω	Spanwise vorticity

1 Introduction

The development of bio-inspired devices for swimming, flying or power generation has led to an increased interest in the understanding of the unsteady fluid dynamics of flapping foils. There is a broad literature on this topic as reviewed by several authors [1, 2, 3, 4, 5, 6]. Both in nature and in engineering applications, the geometry of the wings and their motion kinematics are three-dimensional (3D), and complicated by additional effects like flexibility [5, 6]. Nonetheless, the investigation of the aerodynamic force generation in the simplified scenario of 2D rigid airfoil in heaving and/or plunging motion has for long offered interesting research avenues [7]. It has been shown that the vortical structures that are formed around the airfoil during the oscillation cycle are major actors in the process of force generation [8]. The most important structures are the leading edge vortex (LEV) [9] and the trailing edge vortex (TEV). It has been found that a constructive interaction between LEV and TEV can lead to high propulsive efficiencies [10]. Although the main physical mechanisms of force generation in this configuration are fairly well understood, it is difficult to systematize the knowledge, since the flow is governed by several non-dimensional parameters: the Reynolds number (Re), the Strouhal number (St), the reduced frequency (k), the airfoil shape, etc. This has led to a large number of numerical [10, 11, 12, 13, 14] and experimental [15, 16, 17, 18, 19, 20, 21] studies. Only few studies have combined numerical simulations and experiments [22, 23, 24].

Both approaches have their own limitations, which easily become apparent when they are used as testing/predictive tools for flapping wing performance. The use of simulations is often hindered by their computational cost. Indeed, most numerical studies conducted using direct numerical simulations (DNS) are restricted to relatively low Reynolds numbers (Re often below 3000, with Re defined using the free stream velocity and the airfoil chord). This partly explains the lack of combined numerical and experimental studies, since the latter usually cover a somewhat higher range of Re , typically above 5000. For example, the combined study of Brunton et al. [23] reported numerical simulations at $Re = 100$ and experiments at $Re = 65000$. It should be noted, however, that there are a few heroic simulations at higher Re , for stationary wings for $Re = 400000$ [25] and for moving wings for $Re = 20000$ [26]. Also using Large Eddy Simulation techniques it is possible to perform more affordable simulations at high Reynolds numbers [12].

Some limitations of the experiments concern the setup. For instance, in order to reproduce a nominally 2D geometry, the foils employed in the experiments need to have a large aspect ratio,

Figure 1: Outline of the airfoil motion

which is sometimes difficult to achieve. Even when it is achieved, there are uncertainties associated with the presence of end walls, gaps between the airfoil and the end walls (needed to enable some clearance for the motion), or even interference with the mechanical setup employed to move the foil. Sometimes, it is possible to avoid these limitations. For instance, Andersen et al. [24] used a soap film in a nearly two-dimensional water tunnel to perform experiments of a 2D flapping airfoil.

Additionally, experiments are affected by measurement uncertainty. In case of direct force measurements, this uncertainty is bound by the sensitivity of the measurement device, as well as by precise compensation of spurious contributions (e.g. inertia forces). Such direct force measurements have been reported by several authors [16, 18, 19, 21]. Water tunnels have been a widely preferred choice because, for a given airfoil and Re , the aerodynamic forces in water are one order of magnitude larger than in air, while inertia forces are one order of magnitude smaller, resulting in a substantial reduction of the measurement relative uncertainty. Owing to the two-dimensionality of the problem, the flow field organization is often quantitatively measured in experiments with planar Particle Image Velocimetry (PIV) [15, 16, 18, 20], although there are some studies that report also stereoscopic and tomographic PIV [17].

Finally, it should be noted that the flow around a flapping airfoil is truly two-dimensional only at low Re . Increasing the Reynolds number beyond about $Re \approx 1000$, and depending on the geometry and kinematics of the airfoil, the flow starts to develop 3D features [27, 28, 29, 30], even when the geometry and kinematics of the airfoil are nominally two-dimensional. A complete 3D-DNS results, however, in a larger computational cost and large parametric studies become unaffordable. On the other hand, a 2D-DNS probably does not provide an accurate representation of the flow since the vortex stretching term, which plays an important role in the 3D dynamics of the flow, vanishes identically in 2D. Recent literature includes works using 2D DNS simulations at Re way beyond 1000; for instance Ref. [31] reports the use of 2D DNS to study the performance augmentation of in-line tandem foils at $Re = 7000$. Whether this affects the estimation of the aerodynamic forces in flapping airfoils, it is an open question.

The present work aims to compare the aerodynamic forces predicted using experiments and 2D-DNS, for a configuration at the verge of three-dimensionality. We report a combined experimental/numerical study on a free-stream flow over a rigid airfoil in heaving and pitching motion. The study is carried out for a set of 6 different wing kinematics using both water-tunnel experiments and 2D-DNS. The force and flow field measurements are performed at $Re = 3600$ while the simulations are conducted in the range $Re = 500 - 3600$, providing matching to allow for a direct comparison and at the same time to explore Reynolds number effects using the simulations results. The problem description and the experimental/computational details are reported in Section 2. The comparative analysis of experiments and simulations (Section 3) enables the evaluation of the ability of 2D simulations to provide the unsteady forces measured in the experiments, for a flow which is on the verge of becoming fully three-dimensional. In addition, when discrepancies in the forces are found, the differences are assessed through flow fields and force components comparison.

Table 1: Summary of kinematics analyzed.

Case	$\theta_m(^{\circ})$	$\theta_0(^{\circ})$	$\alpha_{e,max}(^{\circ})$	$\alpha_{e,min}(^{\circ})$
A00	0	0	32	-32
A10	0	10	22	-22
A20	0	20	12	-12
B00	10	0	42	-22
B10	10	10	32	-12
B20	10	20	22	-2

2 Computational and experimental details

2.1 Problem description

The present study considers a uniform flow with a free-stream flow of velocity U_{∞} around a NACA 0012 airfoil of chord c in heaving, $h(t)$, and pitching, $\theta(t)$, motion. Both heaving and pitching motions are prescribed by sinusoidal laws

$$h(t) = h_0 \cos(2\pi f t), \quad (1)$$

$$\theta(t) = \theta_m + \theta_0 \cos\left(2\pi f t + \frac{\pi}{2}\right), \quad (2)$$

where f is the frequency of oscillation, h_0 and θ_0 are the heaving and pitching amplitude, respectively, and θ_m is the mean pitch angle. The heaving is considered positive for upward displacement; the pitching is considered positive for nose-up motion, as sketched in Fig.1. The pivoting point x_p for the pitching motion is located at $c/4$ from the leading edge. The amplitude of the heaving motion is fixed to $h_0 = c$ and the reduced frequency is set to $k = 2\pi f c/U_{\infty} = 0.2\pi$, resulting in a period of oscillation of $T = 10c/U_{\infty}$ and a Strouhal number $St = fh_0/U_{\infty} = 0.1$. The phase shift between the heaving and pitching motion is set to $\pi/2$ because it is shown both experimentally [8] and numerically [32] that this value enhances thrust. A set of six cases with different kinematics is obtained by setting the mean pitch angle to $\theta_m = 0^{\circ}$ and 10° , and the pitching amplitude to $\theta_0 = 0^{\circ}, 10^{\circ}$ and 20° (see Eqs. 1 and 2). To simplify the discussion of the results, each case is labelled with a letter representing the mean pitch angle, A for $\theta_m = 0^{\circ}$ and B for $\theta_m = 10^{\circ}$, and a number indicating the value of the pitching amplitude θ_0 . The entire set of kinematics employed in this study is reported in Table 1. The table also reports the maxima and minima of the effective angle of attack $\alpha_e = \theta - \tan^{-1}(\dot{h}/U_{\infty})$, defined as the angle between the chord of the airfoil and the relative wind at the pivoting point. The Reynolds number based on the chord and on the free stream velocity is set to $Re = 3600$ in the experiments and to $Re = 500, 1000$ and 3600 in the simulations.

2.2 Experimental setup

The experiments are carried out in the water tunnel of the Aerospace Engineering Group at the *Universidad Carlos III de Madrid*. The tunnel has a rectangular test chamber ($500 \times 550 \text{ mm}^2$). The tested model is a wing with rectangular planform (span equal to 490 mm and chord equal to 30 mm), milled from aluminum, providing enough stiffness to avoid bending at the prescribed flapping frequency. The blockage ratio provided by the wing is equal to 0.0072. Since the wing spans side-to-side the test chamber, with a small clearance ($\sim 5 \text{ mm}$) to avoid contact with the lateral walls, three-dimensional effects at the wing tip are of secondary importance. Additionally the large aspect ratio (16.3) ensures with good approximation two-dimensionality of the flow conditions throughout a large extent of the wing span. All the reported experimental tests have been carried out at a free-stream velocity of 0.12 m/s, resulting in a chord-based Reynolds number equal to 3600. The turbulence intensity of the water tunnel for this working condition has been estimated to be about 1% by means of PIV measurements. The wing has been actuated at 0.4 Hz in order to match the Strouhal number and reduced frequency of the simulations. The wing actuation is achieved with a four-bar linkage driven by two synchronized linear actuators (positioning precision 0.01 mm) and connected to the wing through a load cell at $L/3$ from the tip. The four-bar linkage is sketched in Fig. 2. The center of rotation is not located along the wing chord, being at a distance of

Figure 2: Sketch of the experimental setup.

$d = 12$ mm to avoid direct contact between the wing and the linkage. Consequently, in addition to the prescribed pitching/heaving motion, the wing is subjected to a streamwise translation motion $s(t)$ defined as $s(t) = d \sin(\theta(t))$. This leads to a maximum peak-to-peak amplitude for the streamwise motion of $0.14c$ in cases A10 and B10, and $0.27c$ in cases A20 and B20. The maximum streamwise velocities induced by this motion are relatively small ($0.04U_\infty$ for cases A10 and B10, and $0.09U_\infty$ for cases A20 and B20); this is expected to have a negligible effect on the aerodynamic force generation mechanism.

The force measurements are performed with a 6 degrees-of-freedom load cell (ATI Industrial Automations Nano-17 IP68). The resonant frequency of the load cell is 3000Hz. Phase-averaging over 500 flapping cycles is performed in order to reduce the noise content of the measurements. The aerodynamic force coefficients are obtained after removing the inertia forces of the wing and the height-dependent bias induced by the load-cell waterproofing. The inertia forces have been measured in a *dry experiment*, i.e. with empty tunnel. The height-dependent bias has been determined with a static *wet experiment*, i.e. with filled test section, no free-stream flow and very slow wing motion (actuation frequency of 0.04Hz). The resolution of the force coefficients measurements at the testing condition is about 0.03 according to the manufacturer specifications. In order to remove high frequency electromagnetic noise induced by the water tunnel pumps running at 13Hz, the load-cell signal has been low-pass filtered with a cutoff at 10Hz. The sensor response is also affected by a long-period thermal drift, which is corrected through detrending. However, the sensor drift also produces a bias error on the average force which can not be removed through detrending. The uncertainty on the force coefficients after the phase-averaging is mostly determined by the bias error on the mean force, which has been experimentally estimated to be lower than 0.1 according to the typical load-cell thermal drift and the typical duration of the measurements. The uncertainty on the phase-averaged fluctuating part of the force is negligible (~ 0.001).

Flow field measurements are performed with 2D PIV. The PIV setup is sketched in Fig. 2. The light sheet (1 mm thickness), intersecting the wing at $L/3$ from both the tip and the load cell, is provided by a dual cavity Nd:Yag Quantel Evergreen laser (200 mJ/pulse at 15 Hz). A 5.5 Mpixels Andor sCMOS camera is used to capture frames with a resolution of about 8.5 pix/mm. The flow is seeded with neutrally-buoyant polyamide particles (56 μ m diameter). The PIV system is synchronized with the movement system in order to provide phase-averaged measurements over 55 flapping cycles. The images are pre-processed to remove the background intensity via an eigenbackground subtraction [33]. Velocity fields are obtained through image digital correlation [34], using a multi-pass [35] image deformation algorithm [36] with a B-spline interpolation [37]. The final interrogation window is equal to 32×32 pixels and 75% overlap. The spatial resolution of the PIV measurement corresponds to about 32 independent vectors per chord. The typical displacement uncertainty is equal to 0.1 pixels [38], thus corresponding to an uncertainty of 0.8% of the free stream velocity at the present image resolution.

Figure 3: Outline of the computational domain and boundary conditions used in the simulations. The outlet boundary condition is an advective type boundary condition where u_c is the advective velocity. u and w represent the streamwise and vertical velocities, respectively.

2.3 Computational setup

Direct Numerical Simulations (DNS) are performed with TUCAN [29, 39], an in-house code that solves the Navier-Stokes equations for an incompressible flow. In TUCAN, the domain is discretized using a uniform, Cartesian, staggered grid in which derivatives are approximated by finite differences of second order. The presence of the body is modelled with the immersed boundary method proposed by Uhlmann [40]. Time integration of the Navier-Stokes equations is performed with a pressure correction formulation and a 3-stages, semi-implicit Runge-Kutta method. More information on the flow solver can be found in Moriche [39] including the details of the validation test cases, both for 2D flows (Poiseuille flow, Taylor-Green vortices, flow over stationary and moving cylinders) and 3D flows (fully-developed turbulent channel flow, laminar flow around a sphere).

The 2D computational domain used in this work is sketched in Fig. 3. This domain has $[12c \times 8c]$ in the streamwise and vertical directions, respectively. The airfoil is located in middle of the domain in the vertical direction and $3c$ downstream of the inlet in the streamwise direction, leaving a distance of eight chords between the trailing edge of the airfoil and the outlet. At the inflow boundary a constant free-stream velocity is imposed while at the outflow a convective boundary condition is imposed. At the top and bottom boundaries free-slip boundary conditions are imposed. (see Fig. 3).

The grid resolution for the cases with $Re = 500$ and $Re = 1000$ was selected based on grid refinement studies as reported in previous works [32]. The grid resolution for the case with $Re = 3600$ was selected based on a grid refinement study for the case A10. We performed three simulations using 144, 216 and 324 grid points per chord. Using the case with 324 points per chord as reference, both resolutions of 144 and 216 points per chord show differences in the mean and standard deviation values of the lift coefficient smaller than 1%. The differences for the values of the standard deviation of the thrust coefficient are slightly larger, about 5% for the case with 216 points per chord, but smaller than 1% for the mean values of the thrust. In any case, the instantaneous lift and thrust coefficient values for these three resolutions (see figure 4) show very good agreement between the cases with 216 and 324 points per chord. Thus, the spatial resolution was set to $c/\Delta x = 72, 128, 216$ for the simulations with $Re = 500, 1000, 3600$, respectively. These values for the resolution lead, for the simulation with $Re = 500$, to a grid of $[864 \times 576]$ points in the streamwise and vertical directions, respectively. The number of points increases to $[1536 \times 1024]$ for the simulation with $Re = 1000$ and to $[2592 \times 1728]$ for the simulation with $Re = 3600$. The time step is determined in such a way that the Courant number is always lower than 0.1. This results in $\Delta t = 4.17 \cdot 10^{-4} c/U_\infty$ for the simulations with $Re = 500$, $\Delta t = 2.34 \cdot 10^{-4} c/U_\infty$ for the simulations with $Re = 1000$ and $\Delta t = 1.56 \cdot 10^{-4} c/U_\infty$ for the simulations with $Re = 3600$.

In all the simulations, except for the case A00 at $Re = 3600$, the flow reaches a periodic state. For the periodic cases the L_2 norm of the difference in thrust and lift coefficients from one cycle

Figure 4: a) Thrust and b) lift coefficient for one cycle of the case A10 ($Re = 3600$) for different resolutions: $c/\Delta x = 144$ (black), $c/\Delta x = 216$ (blue) and $c/\Delta x = 324$ (red).

to the next one is smaller than 3% of the root-mean-square (rms) values, whereas for case A00 at $Re = 3600$ this value is approximately 12%. In order to compute the time evolution of the force coefficients along a cycle for the simulation of case A00 at $Re = 3600$, the last 10 cycles are used to perform a phase average. The convergence error for the time averaged force coefficients of this case is estimated using the standard deviation of the mean values in each cycle, resulting in an absolute error of 0.0053 for the thrust coefficient and 0.0143 for the lift coefficient. For the rest of the simulations no phase average is required.

3 Results

The discussion of the results is divided in two parts. First, a comparison of the estimation/measurement of the aerodynamic forces in terms of mean values and evolution along the flapping cycle is presented (Sec.3.1). In the remainder of the section, the differences are explained in terms of flow features organization and Reynolds number effects. Finally, a quantitative comparison of the discrepancies and similarities of experiments and simulations in phase-averaged forces is carried out.

3.1 Aerodynamic forces

The aerodynamic force is decomposed into its lift and thrust components, oriented respectively in the direction orthogonal and opposite to the free stream velocity. These force components are reported in non-dimensional form as lift and thrust coefficients c_l, c_t , obtained dividing the force components per unit span by the airfoil chord and the free-stream dynamic pressure $0.5\rho U_\infty^2$ (being ρ the density of the fluid). Note that, for the experiments, this is equivalent to non-dimensionalize the total force with respect to the wing surface and the dynamic pressure.

Table 2: Summary of cases analyzed by simulations and experiments. Mean and rms values of the aerodynamic force coefficients are calculated from phase averaged values, except for the simulations that reached a periodic state ($t_{av}/T = 1$).

Case	θ_m	θ_0	Data (Re)	t_{av}/T	$\bar{c}_t$	c_t^{rms}	$\bar{c}_l$	c_l^{rms}
A00	0°	0°	DNS (500)	1	-0.1437	0.2199	0.0000	1.9757
			DNS (1000)	1	-0.0883	0.2097	-0.0000	1.9831
			DNS (3600)	10	-0.0682 ± 0.0053	0.2254	0.0021 ± 0.0143	2.0684
			Exp. (3600)	500	-0.1934 ± 0.1	0.1318	0.0713 ± 0.1	1.7980
A10	0°	10°	DNS (500)	1	-0.0095	0.2753	0.0000	1.4295
			DNS (1000)	1	0.0866	0.2927	0.0000	1.5506
			DNS (3600)	1	0.1575	0.2477	0.0000	1.5681
			Exp. (3600)	500	0.1213 ± 0.1	0.2108	0.0706 ± 0.1	1.3075
A20	0°	20°	DNS (500)	1	0.0061	0.1467	-0.0000	0.7251
			DNS (1000)	1	0.0834	0.2033	0.0000	0.8367
			DNS (3600)	1	0.1743	0.2130	0.0001	0.9677
			Exp. (3600)	500	0.0038 ± 0.1	0.2478	-0.0734 ± 0.1	0.6892
B00	10°	0°	DNS (500)	1	-0.2004	0.3353	0.3376	1.9409
			DNS (1000)	1	-0.1187	0.3517	0.2542	2.0119
			DNS (3600)	1	-0.1066	0.3775	0.3590	2.0955
			Exp. (3600)	500	-0.1714 ± 0.1	0.3860	0.5924 ± 0.1	2.2596
B10	10°	10°	DNS (500)	1	-0.1206	0.2306	0.5251	1.3099
			DNS (1000)	1	-0.0780	0.2575	0.6984	1.3889
			DNS (3600)	1	-0.0213	0.2447	0.8154	1.4425
			Exp. (3600)	500	-0.1091 ± 0.1	0.2416	0.7274 ± 0.1	1.2719
B20	10°	20°	DNS (500)	1	-0.1294	0.1650	0.5009	0.7341
			DNS (1000)	1	-0.0649	0.2169	0.6512	0.8611
			DNS (3600)	1	-0.0113	0.2380	0.8734	0.9241
			Exp. (3600)	500	-0.1632 ± 0.1	0.3407	0.7608 ± 0.1	0.7420

Table 2 presents the time-averaged force coefficients ($\bar{c}_l, \bar{c}_t$) and the rms of the fluctuation of the force coefficient ($c_l^{\text{rms}}, c_t^{\text{rms}}$) for the entire set of simulations and experiments. For A00, B00, B10 and B20, both simulations and experiments predict the generation of a mean drag ($\bar{c}_t < 0$). For cases A10 and A20, both simulations and experiments predict mean thrust, except the simulation at $Re = 500$ of case A10, which shows marginal drag. The discrepancies found in $\bar{c}_t$ between the experiments and simulations at $Re = 3600$ are of the same order, or smaller, than among previous works [8, 41]. It should be noted also that these discrepancies lay within the uncertainty of the measurements for most of the cases.

The rms values of thrust seem to vary with Re for the cases with maximum pitching amplitude, i.e. A20 and B20 ($\theta_0 = 20^\circ$). For these cases, the peak value of $|\alpha_e|$ is the lowest among the cases under study (see Table 1). This results in a lower intensity of the LEV and, in turn, in a higher relative importance of the force generated when the boundary layer is attached (and thus of the friction forces). The high sensitivity of friction forces to the Reynolds number may therefore explain this discrepancy. For most of the cases, the agreement between the rms values of the fluctuating part of the thrust force (c_t^{rms}) obtained with experiments and simulations seems to improve as the Re of the simulation increases.

The mean lift coefficients obtained from experiments and simulations are generally in good agreement. For the cases with $\theta_m = 0^\circ$ no mean lift is generated, as expected due to the vertical symmetry of the motion. Data agreement is within the measurement and convergence uncertainties of experiments and simulations, respectively. For the cases with $\theta_m = 10^\circ$, the mean lift in both simulations and experiments is positive. The discrepancy between the values obtained from the simulations at $Re = 3600$ and from the experiments are again within the measurement and convergence uncertainty, except for case B00 for which the simulations predict a significantly lower mean lift than the experimental measurements.

The rms values of the fluctuating part of the lift (c_l^{rms}) obtained from experiments and simulations show reasonable agreement. The simulations at $Re = 3600$ result in a larger value of c_l^{rms}

Figure 5: Thrust coefficient as a function of time. a) A00, b) A10, c) A20, d) B00, e) B10 and f) B20. The results from DNS with $Re = 500$ (—), $Re = 1000$ (—) and $Re = 3600$ (—) are represented together with the results from the experiments (—). The yellow shaded area represents the uncertainty of the experimental measurement. The light (dark) grey region represents the downstroke (upstroke). The effective angle of attack α_e of each case is included in each panel (—).

when compared to the experimental measurements, except for case B00, for which the measured value is less than 10% larger than the value predicted by simulations at $Re = 3600$.

Fig. 5 shows the time evolution of the thrust coefficient during one cycle together with the effective angle of attack α_e . Experiments and simulations present qualitative agreement, with two local maxima (one for the downstroke, one for the upstroke). The only exception is case B00, in which no local maximum during the downstroke is observed for both experiment and simulation at $Re = 3600$. Similarly, Fig. 6 shows the time evolution of the lift coefficient. It can be observed that the lift evolution is characterized by a single maximum during the downstroke and a single minimum during the upstroke, following closely the evolution of the effective angle of attack (α_e , shown in Fig. 5) both in terms of the amplitude of the maxima/minima and in terms of the sign of the lift.

Although it is customary to analyze the aerodynamic force by decomposing it into lift and thrust, for the comparison in the present case it is revealing to analyze the magnitude and orientation of the force. We start by analyzing the orientation in Fig. 7, which shows, as a function of time, the angle, σ , between the total aerodynamic force and the perpendicular to the free stream direction, positive clockwise (see Fig. 8 for a sketch that illustrates the definition of σ for both positive and negative effective angle of attack). When the force is small, the orientation of the force is not very relevant, and therefore the lines are faded when the total force coefficient $c_f < 0.5$ ($c_f = \sqrt{c_l^2 + c_t^2}$). The results are presented alongside the angle of the chord-normal direction with respect to the free stream.

For all the cases, the orientation of the force predicted from simulations and experiments is in good agreement in a large portion of the cycle. The agreement is especially satisfactory for the simulation with $Re = 3600$. The best agreement in force orientation roughly corresponds to the phases in which maximum force magnitude is observed (see Fig. 9 for reference). In these phases the force orientation coincides with the direction normal to the chord, thus confirming that the most relevant component of the force is the chord-normal one, as already assessed by Moriche et al. [32]. It is worth remarking that the experimental uncertainty of the force angle is less than 5% when $c_f > 1$, calculated according to [42].

Fig. 9 shows the time evolution of c_f . The main difference between the values obtained from the simulations and the experiments is a scale factor of about 0.8 for almost all the cases. For example, approximating the first half of the downstroke with a linear law, for case A00 the slope $dc_f/d(t/T)$ corresponding to the simulations is about 10 while that for the experiment is about 8. This might be ascribed to the presence of the force balance and of the linkage which might result in a decrease of the wing area effectively responsible of the aerodynamic performance, and/or to

Figure 6: Lift coefficient as a function of time. a) A00, b) A10, c) A20, d) B00, e) B10 and f) B20. The results from DNS with $Re = 500$ (—), $Re = 1000$ (—) and $Re = 3600$ (—) are represented together with the results from the experiments (—). The light (dark) grey region represents the downstroke (upstroke). The effective angle of attack α_e of each case is included in each panel (—).

the end-wall effects mentioned in the introduction.

Besides this difference, the overall agreement between simulation at $Re = 3600$ and experiments is acceptable.

The discrepancies appear mostly in the second half of both the downstroke and upstroke, where the experimentally measured force seems to grow during a slightly longer time interval than in the simulation. This yields a difference in the time to peak force in experiments and simulations. The difference in the time to peak forces of c_f is also present among the DNS data for different Re , specially for the cases with smaller α_e . This suggests that the difference might be ascribed to different skin friction between simulations and experiments (which should be more important at low effective angles of attack).

Finally, note that for the present cases, the force coefficient c_f and the lift coefficient c_l are of the same order, which is typically between 2 and 10 times larger than the thrust coefficient c_t . Hence, the differences observed in Fig. 5 are comparable to the differences found in Figs. 6 and 9.

3.2 Flow visualization

We now turn our attention to the flow structure by analyzing flow visualizations from both experiments and simulations. The complete set of visualizations is provided in the Appendix A and animations of each case can be found in the supplementary material online.

The visualizations are presented in terms of the spanwise component of the vorticity field $\omega = (\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x})$, where positive (negative) vorticity corresponds to clockwise (counter-clockwise) rotating vortices.

Fig. 10 shows the vorticity field of case B10 ($\theta_m = 10^\circ$, $\theta_m = 10^\circ$) obtained from the simulations at $Re = 3600$ and from the experiment. Four different phases are plotted, which correspond to $t/T = 0, 0.25, 0.5$ and 0.75 . Animations are also provided in the supplementary material online. This case has been selected to illustrate the different vortical structures appearing in the flow, since it has a high effective angle of attack during the downstroke ($\alpha_e = 32^\circ$ at $t/T = 0.25$), and a moderately low effective angle of attack during the upstroke ($\alpha_e = -12^\circ$ at $t/T = 0.75$). As a consequence, strong vortical structures develop in the downstroke, while only weak vortical structures appear in the upstroke.

At the end of the upstroke (Figs. 10a and b), the boundary layer in the upper surface of the airfoil is attached, while the boundary layer in the lower surface is partially detached, with a small vortex (labeled A) being shed to the wake at the trailing edge of the airfoil. At mid-downstroke, Figs. 10c and d show a detached boundary layer in the upper surface of the airfoil, with a clear LEV (labeled B1) and a secondary vortex (labelled B2) closer to the trailing edge. The secondary

Figure 7: Angle (in degrees) between the aerodynamic force and the vertical direction (i.e., perpendicular to the free-stream, see sketch in Fig. 8). a) A00, b) A10, c) A20, d) B00, e) B10 and f) B20. The results from DNS with $Re = 500$ (—), $Re = 1000$ (—) and $Re = 3600$ (—) are represented together with the results from the experiments (—) and the angle of the chord-normal direction with respect to the free-stream (—). The light (dark) grey region represents the downstroke (upstroke) and the lines are faded when the aerodynamic force coefficient is small ($c_f < 0.5$).

Figure 8: Definition of the angle σ (figure 7) between the aerodynamic force ($\vec{F}$) and the vertical direction (i.e., perpendicular to the free stream, $\vec{U}_\infty^\perp$). The definition of the direction perpendicular to the chord ($c^\perp$) and the direction along the effective incoming velocity seen by the airfoil ($\vec{U}$) are also included in the figure. Note that $c^\perp$ points a) towards the upper surface of the airfoil when $\alpha_e > 0$ and b) towards the lower surface when $\alpha_e < 0$.

Figure 9: Time history of the aerodynamic force coefficient $c_f = \sqrt{c_t^2 + c_l^2}$ for cases a) A00, b) A10, c) A20, d) B00, e) B10 and f) B20. The results from DNS with $Re = 500$ (—), $Re = 1000$ (—) and $Re = 3600$ (—) are represented together with the results from the experiments (—). The light (dark) grey region represents the downstroke (upstroke). Also, dashed lines in a) represent the slope of the force coefficient of the experiment (black) and DNS at $Re = 3600$ (blue) during the first half of the downstroke to support the discussion.

vortex B2 in the simulation is more concentrated than in the experiment, therefore results more easily identifiable. As B1 separates and moves downstream, it merges with B2 and forms a dipole with the TEV at the end of the downstroke (Fig. 10e and f, see structure labeled C). This dipole is then shed into the wake, where it is advected downstream by the free stream (see structure E in Figs. 10g and h), but at different velocities. For that reason, in Fig. 10a the dipole has left the domain, while it is still visible in Fig. 10b (labelled structure D). Note that, during the first half of the upstroke the effective angle of attack is small enough for the boundary layers in the upper and lower surfaces of the airfoil to remain attached (see Figs. 10g and h). The boundary layer at the lower surface of the airfoil separates and rolls up into a vortex close to the end of the upstroke, resulting in vortex A in Figs. 10a and b. This vortex is weaker than the one shed at the end of the downstroke (structure C in Figs. 10e and f).

The direct comparison between the experiment and the DNS shows that the evolution of the vortical structures is qualitatively very similar. Nearby the airfoil, the same vortical structures are observed in the DNS and the experiment, with similar intensities and location (vortices A, B1, B2, C). This observation is consistent with most of the cases of the database (the reader is referred to the flow visualizations of the simulations at $Re = 3600$ and the experiments in the Appendix A and the animations in the Supplementary Material). This resemblance of the flow nearby the airfoil justifies the agreement between the forces estimated with computations and measurements. Indeed, previous works by Moriche et al. [32] and by Martín-Alcántara et al. [14] have shown that only the vortical structures close to the airfoil contribute to the aerodynamic force.

In spite of that, some differences are observable. For instance, in the cases B00, B10 and B20, the flow visualizations (figures 16, 17 and 18 and corresponding animations) show differences in the position and size of the LEV and of the TEV, which might be correlated to the differences in the time to peak total force discussed in Fig. 9. The present dataset suggests that the differences in the LEV are more effective in affecting the c_l history (see case B00, flow visualization and c_l at $0.25 < t/T < 0.5$). On the other hand, differences in the TEV seem to mainly influence c_t (see case B20, flow visualization and c_t at $t/T = 0.5$).

Also, the simulations present some vortical structures in the wake which are either absent or substantially weaker in intensity in the experiments (see vortices labeled F, I and L in Fig. 10). This might suggest that 3D vortex dynamics becomes important in the wake dynamics of the experiments, allowing for additional instabilities. In the simulations, the 2D character of the solution allows for these vortices to persist, independently of their intensity and size. In the experiments, the large aspect ratio of the wing in the experiment and the measures taken to reduce

Figure 10: Vorticity contours of case B10 ($\theta_m = 10^\circ$, $\theta_0 = 10^\circ$) obtained from (a,c,e,g) DNS at $Re = 3600$ and from (b,d,f,h) experiment at $Re = 3600$. The dashed line is an outline used as support for the discussion. The times plotted are (a,b) $t/T = 0$, (c,d) $t/T = 0.25$, (e,f) $t/T = 0.5$, and (g,h) $t/T = 0.75$. The insets show the time evolution of c_l and c_d around the time plotted in each panel, in black for the experiment and blue for the DNS at $Re = 3600$.

Figure 11: Vorticity contours of case B20 ($\theta_m = 10^\circ$, $\theta_0 = 20^\circ$) obtained from simulations at $Re = 500, 1000, 3600$ and from experiments at $Re = 3600$.

3D effects from the tips are effective to generate a quasi-2D flow around the airfoil. The large LEV and TEV, consequently, are advected keeping its quasi-2D character, but the smaller unstable vortices break down and tend to disappear from the 2D representation.

The idea that the quasi-2D character of the flow is maintained is supported by previous works. For instance, Koochesfahani [43] showed that a small axial flow appears because of the presence of lateral wall in the experimental setup, but the 2D character of the flow was maintained. In a different work, Visbal et al. [12] analyzed the effect of the lateral walls by means of simulations. They found that, whenever the LEV and TEV are strong, the variations of the structures along the span are not important, but when these structures are shed into the wake, they tend to break losing their quasi-2D character. Nonetheless, the impact on force generation (and thus in the accuracy of force prediction with 2D-DNS) is minimal, as observed in Section 3.1. This is consistent with the scenario depicted by Moriche et al. [29], where 3D instabilities are advected downstream and the flow nearby the wing is 2D, resulting in very small differences in the forces between 2D and 3D simulations of a similar configuration.

In most of the cases presented here, the interaction of the LEV and TEV leads to the formation of a dipole that is advected downstream in the wake (see labels C, E and D in Fig. 10). The trajectory and the intensity of these dipoles are different in the experiments and the simulations. Note that these differences tend to occur relatively far from the airfoil, when the dipoles are in the wake, which again suggests that this has a relatively small impact on the forces. The rationale is the same as for the breakup of small vortices: as long as the flow near the airfoil is 2D, the 3D effects of advected flow structures have little impact on the aerodynamic forces. Comparing results from 2D and 3D simulations, Moriche et al. [29] documented differences of approximately 5% in the rms of the fluctuation of the force coefficients, and of approximately 15% in the mean lift coefficient. These values are consistent with the differences observed in the present cases. Besides that, the position of the wake with respect to the airfoil is essentially the same both in the simulation and the experiments (see black dashed lines in Fig. 10).

Figure 12: Cross-correlation coefficient ρ_ω (near field — ; far field —) and ρ_f (—). of simulation and experiment. To support the discussion the aerodynamic force magnitude c_f obtained in the experiment is included (—).

3.3 Reynolds number effect on the flow organization

The rich database collected in this study allows also for a deeper analysis on the effects of the Reynolds number on the flow field. Fig. 11 shows vorticity contours at four different time instants for case B20. The figure presents the results obtained from the simulations at different Re and from the experiments. The large pitching amplitude of this case results in a smaller effective angle of attack ($-2^\circ \leq \alpha_e \leq 10^\circ$, as shown in Fig. 5b) during the upstroke ($0.5 \leq t/T \leq 1$) than in B10, resulting in an attached boundary layer during the complete upstroke. During the downstroke, the effective angle of attack is moderately high ($10^\circ \leq \alpha_e \leq 22^\circ$). This results in the generation of a strong dipole formed by a TEV (blue) and a LEV (red) at the end of the downstroke ($t/T = 0.5$). The effect of increasing Re is to decrease the thickness of the boundary layers and the size of the vortices formed during the downstroke. As a consequence, the orientation, size and position of the dipole shed at the end of the downstroke varies with Re , becoming more compact and traveling further upwards as Re increases. It is worth noting that this dipole does not travel upwards in the experiments, where it is only advected downstream. Finally, the Re also affects the growth of the Kelvin-Helmholtz (K-H) instabilities in the shear layer (wake) generated during the upstroke. This can be observed at $t/T = 0.25$ (Fig. 11), where the simulation at $Re = 3600$ presents clear K-H vortices in the wake that are absent at $Re = 500$ and 1000 . As expected, the wake in the DNS at $Re = 3600$ is qualitatively more similar to the experimental results than the wakes at $Re = 500$ and 1000 . Besides that, it is worth noting that in the experiments the development of the K-H instabilities of the wake is faster than in the simulations, even if the DNS at $Re = 3600$ is considered (see Fig. 11 at $t/T = 0$). Since the development of K-H vortices is a 2D process, the most likely explanation for this difference is the free-stream turbulence of the experiment, which might trigger earlier the development of the instability.

3.4 Quantitative comparison

In this section a quantitative assessment of the similarities and discrepancies observed between the experiments and the simulations at $Re = 3600$ is presented. The evaluation is performed using the cross-correlation coefficients between both the phase-averaged vorticity fields (ρ_ω) and

aerodynamic forces (ρ_f),

$$\rho_\omega = \frac{\iint (\omega_{exp} - \langle \omega_{exp} \rangle) (\omega_{sim} - \langle \omega_{sim} \rangle) dx dz}{\sqrt{\iint (\omega_{exp} - \langle \omega_{exp} \rangle)^2 dx dz} \sqrt{\iint (\omega_{sim} - \langle \omega_{sim} \rangle)^2 dx dz}}, \quad (3)$$

$$\rho_f = \frac{\|\mathbf{F}_{exp} \cdot \mathbf{F}_{sim}\|}{\|\mathbf{F}_{exp}\| \|\mathbf{F}_{sim}\|}, \quad (4)$$

where ω and $\mathbf{F}$ are the spanwise vorticity field and the aerodynamic force, respectively, the data obtained from the experiment (simulation) is identified by the subscript *exp* (*sim*), and the operators $\|\cdot\|$ and $\langle \cdot \rangle$ represent the L_2 norm and the spatial average operator, respectively. Fig. 12 reports, for each case, the cross-correlation coefficient of the vorticity fields ρ_ω , calculated both in the near field ($-0.5 < x/c < 1.5$) and in the far field ($1.5 < x/c < 6.5$), together with the cross-correlation coefficient of the aerodynamic force ρ_f . Also, the force coefficient from the experiment $c_{f,exp}$ is included in the figure to support the discussion. The uncertainty of the force measurements has a small effect on the force cross-correlation coefficient ρ_f , resulting in a relative uncertainty, calculated according to [42], of less than 3% when $c_f > 1$.

For all the cases, the cross-correlation coefficient ρ_ω in the near field is higher than in the far field during the entire cycle. This result is in agreement with the discussion of Fig. 10, where discrepancies between simulations and experiments are mainly observed in the organization of the structures in the wake. Regarding the cross-correlation coefficient of the force, ρ_f is close to one most of the time, except when the modulus of the force is relatively small, $c_f \lesssim 1$. This implies that the orientation of the total force coefficient in experiments and simulations is very similar most of the time, except when the forces are small (see also Fig. 7).

The correlation in the near field is more intense when the flow is dominated by an intense LEV. For instance, cases A00, A10 and B00 (Fig. 12a,b and d), which are characterized by a strong LEV in both the downstroke and the upstroke, present high values of the near-field ρ_ω during both strokes. At the stroke reversal, when the LEV detaches from the airfoil (past the peak of c_f), these cases show a clear decrease in the near-field value of ρ_ω . This could be related to the high sensitivity of the detachment of the LEV, which could also explain the differences in the time to peak aerodynamic forces between experiments and simulations discussed earlier. In fact, it should be noted that Visbal [12] also observed differences in the detachment of the LEV when comparing two experimental setups with the same geometry and kinematics.

Cases B10 and B20 (see Figs. 12e and f), in which a strong LEV is generated during the downstroke only, show the same behaviour described above for $0 \lesssim t/T \lesssim 0.5$. However, ρ_ω in the near field shows some oscillations during the upstroke, when the total aerodynamic force is small, the airfoil has a very small effective angle of attack, and all vorticity in the near field is concentrated in attached boundary layers (see Figs. 17 and 18). Indeed, the oscillating character of ρ_ω seems to be related to the presence of shadows in the PIV setup in regions close to the airfoil surface. Regarding the forces, the value of ρ_f for B10 and B20 is clearly lower during the upstroke (when the total aerodynamic force is small) than during the downstroke (when the total aerodynamic force is large). Furthermore, strong oscillations appear in ρ_f when the value of c_f approaches zero ($0.5 \lesssim t/T \lesssim 0.7$ and $t/T \approx 0.98$ for case B10, and $0.8 \lesssim t/T \lesssim 0.95$ for B20). These oscillations seem to be related to the change of sign of the force when α_e changes sign.

Finally, case A20, which has small effective angle of attack α_e during both strokes, presents a moderately high value of the near-field ρ_ω during the whole cycle, with the aforementioned oscillating character. Also, because of the relatively small value of c_f during both strokes, the value of ρ_f is smaller than in the rest of the cases, especially compared to cases where a strong LEV is formed.

4 Conclusions

A combined experimental/numerical study of the flow over a nominally 2D flapping airfoil at moderate Reynolds numbers ($Re = 500 - 3600$) is reported. Several flapping kinematics have been analyzed by modifying both the pitching amplitude and the mean geometrical angle of attack, with the final aim of assessing discrepancies between experiments and 2D-DNS in a range of Reynolds

number where the flow should be developing 3D features. The study shows a good agreement of the force history both in terms of direction and magnitude in the parametric range considered. The direct comparison of the flow fields from experiments and 2D-DNS highlights that differences in the vortical features arise mostly in the wake. As a consequence, these differences have a low impact on the force. Differences in the wake are mostly related to the presence of strong and localized vortices other than LEV and TEV in the 2D simulations, which may be the result of the absence of three-dimensional vortex stretching and tilting. Differences in the flow over the wing are localized to specific flapping phases, mostly during the LEV detachment and the TEV formation, suggesting that these phases are more sensitive to three-dimensional effects, differences in Reynolds number and free stream turbulence. This suggests that 2D-DNS is still a valid and efficient tool to estimate aerodynamic forces also in configurations where 3D effects are already present, provided that the flow instabilities develop sufficiently far from the wing.

Acknowledgments

This work was supported by grants TRA2013-41103-P (MINECO/Feder,UE) and DPI2016-76151-C2-2-R (AEI/FEDER, UE).

References

- [1] K. V. Rozhdestvensky and V. A. Ryzhov. Aerohydrodynamics of flapping-wing propulsors. *Prog. Aerospace Sci.*, 39:585–633, 2003.
- [2] M. F. Platzer, K. D. Jones, J. Young, and J. C. S. Lai. Flapping-wing aerodynamics: Progress and challenges. *AIAA J.*, 46(9):2136–2149, 2008.
- [3] K. D. von Ellenrieder, K. Parker, and J. Soria. Fluid mechanics of flapping wings. *Exp. Thermal Fluid Sci.*, 32:1578–1589, 2008.
- [4] W. Shyy, H. Aono, Chimakurthi S. K., P. Trizila, C.-K. Kang, C. E. S. Cesnik, and H. Liu. Recent progress in flapping wing aerodynamics and aeroelasticity. *Prog. Aerospace Sci.*, 46:284–327, 2010.
- [5] W. Shyy, H. Aono, C.-K. Kang, and H. Liu. *An introduction to flapping wing aerodynamics*. Cambridge Univ. Press, 2013.
- [6] I. Gursul and D. Cleaver. Plunging oscillations of airfoils and wings: Progress, opportunities, and challenges. *AIAA J.*, page in press, 2018.
- [7] J. Zhang. Footprints of a flapping wing. *J. Fluid Mech.*, 818:1–4, 2017.
- [8] J. M. Anderson, K. Streitlien, D. S. Barret, and M. S. Triantafyllou. Oscillating foils of high propulsive efficiency. *J. Fluid Mech.*, 360:41–72, 1998.
- [9] C. P. Ellington, C. Van Den Berg, A. P. Willmott, and A. L. R. Thomas. Leading-edge vortices in insect flight. *Nature*, 384:626–630, 1996.
- [10] G. C. Lewin and H. Haj-Hariri. Modelling thrust generation of a two-dimensional heaving airfoil in a viscous flow. *J. Fluid Mech.*, 492:339–362, 2003.
- [11] Z. J. Wang. Vortex shedding and frequency selection in flapping flight. *J. Fluid Mech.*, 410:323–341, 2000.
- [12] M. R. Visbal. Numerical investigation of deep dynamic stall of a plunging airfoil. *AIAA J.*, 49(10):2152–2170, 2011.
- [13] J. Choi, T. Colonius, and D. R. Williams. Surging and plunging oscillations of an airfoil at low Reynolds number. *J. Fluid Mech.*, 763:237–253, 2015.
- [14] A. Martín-Alcántara, R. Fernández Fera, and E. Sanmiguel-Rojas. Vortex flow structures and interactions for the optimum thrust efficiency of a heaving airfoil at different mean angles of attack. *Phys. Fluids*, 27:073602, 2015.

- [15] D. Rival, T. Prangemeier, and C. Tropea. The influence of airfoil kinematics on the formation of leading-edge vortices in bio-inspired flight. *Exp. Fluids*, 46(5):823–833, 2009.
- [16] Y. S. Baik, L. P. Bernal, K. Granlund, and M. V. Ol. Unsteady force generation and vortex dynamics of pitching and plunging aerofoils. *J. Fluid Mech.*, 709:37–68, 2012.
- [17] A.-J. Buchner, N. Buchmann, K. Kilany, C. Atkinson, and J. Soria. Stereoscopic and tomographic PIV of a pitching plate. *Exp. Fluids*, 52(2):299–314, 2012.
- [18] C. W. Pitt Ford and H. Babinsky. Lift and the leading-edge vortex. *J. Fluid Mech.*, 720:280–313, 2013.
- [19] D. E. Rival, J. Kriegseis, P. Schaub, A. Widmann, and C. Tropea. Characteristic length scales for vortex detachment on plunging profiles with varying leading-edge geometry. *Exp. Fluids*, 55(1):1660, 2014.
- [20] A. Widmann and C. Tropea. Parameters influencing vortex growth and detachment on unsteady aerodynamic profiles. *J. Fluid Mech.*, 773:432–459, 2015.
- [21] D. Floryan, T. Van Buren, C. W. Rowley, and A. J. Smits. Scaling the propulsive performance of heaving and pitching foils. *J. Fluid Mech.*, 822:386–397, 2017.
- [22] K. D. Jones, C. M. Dohring, and M. F. Platzer. Experimental and computational investigation of the Knoller-Betz effect. *AIAA J.*, 36(7):1240–1246, 1998.
- [23] S. L. Brunton, C. W. Rowley, and D. R. Williams. Reduced-order unsteady aerodynamic models at low Reynolds numbers. *J. Fluid Mech.*, 724:203–233, 2013.
- [24] A. Andersen, T. Bohr, T. Schnipper, and J. H. Walther. Wake structure and thrust generation of a flapping foil in two-dimensional flow. *J. Fluid Mech.*, 812, 2017.
- [25] S. M. Hosseini, R. Vinuesa, P. Schlatter, A. Hanifi, and D. S. Henningson. Direct numerical simulation of the flow around a wing section at moderate reynolds number. *Int. J. Heat Fluid Flow*, 61:117–128, 2016.
- [26] M. E. Rosti, M. Omidyeganeh, and A. Pinelli. Direct numerical simulation of the flow around an aerofoil in ramp-up motion. *Phys. Fluids*, 28(2):025106, 2016.
- [27] J. Deng and C. P. Caulfield. Three-dimensional transition after wake deflection behind a flapping foil. *Phys. Rev. E*, 91(4):043017, 2015.
- [28] J. Deng, L. Sun, and X. Shao. Dynamical features of the wake behind a pitching foil. *Phys. Rev. E*, 92(6):063013, 2015.
- [29] M. Moriche, O. Flores, and M. García-Villalba. Three-dimensional instabilities in the wake of a flapping wing at low Reynolds number. *Int. J. Heat Fluid Flow*, 62:44–55, 2016.
- [30] L. Sun, J. Deng, and X. Shao. Three-dimensional instabilities for the flow around a heaving foil. *Phys. Rev. E*, 97(1):013110, 2018.
- [31] L. E. Muscutt, G. D. Weymouth, and B. Ganapathisubramani. Performance augmentation mechanism of in-line tandem flapping foils. *J. Fluid Mech.*, 827:484–505, 2017.
- [32] M. Moriche, O. Flores, and M. García-Villalba. On the aerodynamic forces on heaving and pitching airfoils at low Reynolds number. *J. Fluid Mech.*, 828:395–423, 2017.
- [33] M. A. Mendez, M. Raiola, A. Masullo, S. Discetti, A. Ianiro, R. Theunissen, and J.-M. Buchlin. POD-based background removal for particle image velocimetry. *Exp. Thermal Fluid Sci.*, 80:181–192, 2017.
- [34] C. E. Willert and M. Gharib. Digital particle image velocimetry. *Exp. Fluids*, 10(4):181–193, 1991.
- [35] J. Soria. An investigation of the near wake of a circular cylinder using a video-based digital cross-correlation particle image velocimetry technique. *Exp. Thermal Fluid Sci.*, 12(2):221–233, 1996.

- [36] F. Scarano. Iterative image deformation methods in PIV. *Meas. Sci. Tech.*, 13(1):R1, 2001.
- [37] T. Astarita. Analysis of interpolation schemes for image deformation methods in PIV: effect of noise on the accuracy and spatial resolution. *Exp. Fluids*, 40(6):977–987, 2006.
- [38] J. Westerweel. Fundamentals of digital particle image velocimetry. *Meas. Sci. Tech.*, 8(12):1379, 1997.
- [39] M. Moriche. *A numerical study on the aerodynamic forces and the wake stability of flapping flight at low Reynolds number*. PhD thesis, Universidad Carlos III de Madrid, 2017.
- [40] M. Uhlmann. An immersed boundary method with direct forcing for the simulation of particulate flows. *J. Comp. Phys.*, 209(2):448–476, 2005.
- [41] R. Ramamurti and W. Sandberg. Simulation of flow about flapping airfoils using finite element incompressible flow solver. *AIAA J.*, 39(2):253–260, 2001.
- [42] S. J. Kline and F. A. McClintock. Describing uncertainties in single-sample experiments. *Mech. Eng.*, 75:3–8, 1953.
- [43] M. M. Koochesfahani. Vortical patterns in the wake of an oscillating airfoil. *AIAA J.*, 27(9):1200–1205, 1989.

A Complete set of flow visualizations

Figure 13: Vorticity contours of case A00 ($\theta_m = 0^\circ, \theta_0 = 0^\circ$) obtained from simulations and experiments at $Re = 3600$.

Figure 14: Vorticity contours of case A10 ($\theta_m = 0^\circ, \theta_0 = 10^\circ$) obtained from simulations and experiments at $Re = 3600$.

Figure 15: Vorticity contours of case A20 ($\theta_m = 0^\circ, \theta_0 = 20^\circ$) obtained from simulations and experiments at $Re = 3600$.

Figure 16: Vorticity contours of case B00 ($\theta_m = 10^\circ, \theta_0 = 0^\circ$) obtained from simulations and experiments at $\text{Re} = 3600$.

Figure 17: Vorticity contours of case B10 ($\theta_m = 10^\circ, \theta_0 = 10^\circ$) obtained from simulations and experiments at $\text{Re} = 3600$.

Figure 18: Vorticity contours of case B20 ($\theta_m = 10^\circ, \theta_0 = 20^\circ$) obtained from simulations and experiments at $\text{Re} = 3600$.