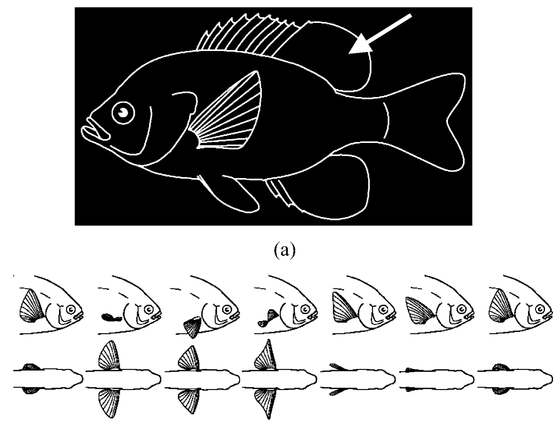
Computational Modeling in Biohydrodynamics: Trends, Challenges, and Recent Advances

Rajat Mittal

***Abstract*—Computational modeling is assuming increased significance in the area of biohydrodynamics. This trend has been enabled primarily by the widespread availability of powerful computers, as well as the induction of novel numerical and modeling approaches. However, despite these recent advances, computational modeling of flows in complex biohydrodynamic configurations remains a challenging proposition. This is due to a multitude of factors, including the need to handle a wide range of flow conditions (laminar, transitional, and turbulent), the ubiquity of two-way coupled interaction between the fluid and moving/deformable structures, and, finally, the requirement of accurately resolving unsteady flow features. Recently, as part of an Office of Naval Research sponsored review, the objective of which was to distill the science related to biology-based hydrodynamics for maneuvering and propulsion, an extensive survey of computational biohydrodynamics was undertaken. The key findings of this survey are reported in this paper.**

***Index Terms*—Biohydrodynamics, computational fluid dynamics, computational modeling, numerical simulation.**



(a)

(b)

Fig. 1. (a) Bluegill sunfish (sketch courtesy of G. V. Lauder, Harvard University, Cambridge, MA) with soft dorsal fin indicated. (b) Lateral and dorsal views of the pectoral fin strokes of Embiotoca lateralis, a benthic maneuverer [4].

I. Introduction

EXPERIMENTAL investigations in biohydrodynamics are limited by their inability to provide full-field, spatially and temporally resolved, velocity and pressure measurements. Some of these limitations are intrinsic to the methods being used, whereas others are associated with the specific conditions associated with biohydrodynamics. Chief among these are the conditions imposed by the need to work with live animals, since it is often difficult to control/predict the motion and location of these animals under test conditions. Methods for controlling the subject that are overly invasive (such as tethering or excessive confinement) run the risk of modifying the natural motion/gait of the subject. In addition to the traditional approach of painstakingly conditioning the subjects to respond somewhat predictably in test conditions; recently, some novel control methods based on visual stimuli have been employed [1], which might hold promise for future studies. However, even if some amount of repeatability and control can be instilled in these tests through some minimally invasive means, it is usually difficult to instrument the test subjects with sensors to the extent needed without disrupting the natural behavior of the subject. For instance, no method currently exists for extracting the surface pressure and shear stress distribution on a structure as delicate as the flapping pectoral fin of a fish. Such measurements are of critical importance, since they allow for direct measurement of the hydrodynamical performance of the locomotor under consideration and would also provide good insight into the flow physics. The final hurdle faced by experiments is the need to make measurements in the vicinity of bodies/boundaries that are undergoing large motions, since this causes problems for invasive (hot wire probes, etc.), as well as noninvasive (laser doppler velocimetry, particle image velocimetry, etc.) measurement methods. Some of the above limitations have motivated the development of articulated mechanical models of these animals [2], which can be controlled and instrumented in a manner adequate for detailed engineering analysis.

Another approach that holds potential for overcoming most of the above limitations is computational modeling. In order to examine the challenge posed by biohydrodynamics to computational modeling and simulation, consider the swimming of the bluegill sunfish [*Lepomis macrochirus*; Fig. 1(a)] that has been the subject of a detailed study by Drucker and Lauder [3]. Depending on the gait, the pectoral and median, as well as tail fins, can all be involved in propulsion and maneuvering. For typical specimens, the Reynolds number based on the fish body length ($L \sim 20$ cm) and velocity (0.5 Ls^{-1}) is about 20 000. At this Reynolds number, the attached flow over the body is most likely laminar, but is expected to transition rapidly to turbulence in regions of flow separation that might occur downstream of appendages. The fins of these and similar fish are also highly flexible, have complex planforms and undergo complicated motions [see Fig. 1(b)]. The flow over the fins can be character-

Manuscript received September 28, 2004; revised September 7, 2004. This work was supported by the Office of Naval Research (ONR) under Grant N00014-03-1-0004, monitored by Dr. P. Bandhyopadhyay.

The author is with the Department of Mechanical and Aerospace Engineering, The George Washington University, Washington, DC 20052 USA (e-mail: mittal@gwu.edu).

Digital Object Identifier 10.1109/JOE.2004.833215

0364-9059/04$20.00 © 2004 IEEE
Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

ized in terms of a Stokes frequency parameter ($S = \omega Al/\nu$ where ω, A and l are the fin angular frequency, amplitude, and length, respectively). Typical fin beat frequency of about 3 Hz and fin amplitude and size of about 2 and 5 cm, respectively, gives $S \approx 18\ 000$, which again is in the range where laminar attached flow would quickly transition to turbulence post separation. It is worthwhile to note that flow separation and transition to turbulence on a fin cannot only have a large impact on the hydrodynamic loading of that fin, but can also drastically alter the flow conditions experienced by any downstream fins/appendages.

Thus, assuming that the above conditions are prototypical of biohydrodynamic flow configurations, especially in the context of low-speed maneuvering, the key factors to be considered in computational modeling of these configurations are as follows.

1) Wide range of flow conditions: Typical Reynolds numbers for swimming fishes/cetaceans can vary from $O(10^2)$ to $O(10^7)$. The flow can be laminar, transitional, or turbulent, or a combination of all three. In addition, the surrounding flow environment can be steady or unsteady.
2) Moving boundaries: Biohydrodynamic flows of interest are often associated with moving boundaries, may they be flapping fins or undulating bodies.
3) Two-way fluid-structure coupling: In many cases, the control surfaces (fins, appendages, etc.) are highly flexible and can undergo large deformations as a result of the hydrodynamic loading. This deformation can in turn have a significant effect on the flow, which can then modify the loading itself. In some situations, the internal structural stress distribution may be of as much interest as the external hydrodynamics.
4) Unsteady flow mechanisms: The presence of moving and flexible control surfaces and/or the unsteady flow environment leads to configurations in which the dominance of unsteady flow mechanisms (added mass effects, dynamic stall, vortex shedding, vortex pairing, and vortex-body and vortex-fin interactions) is a rule rather than an exception.

Until about a decade ago, it was not feasible to simulate such flows with all of their attendant complexities. However, the rapid increase in computing power and the availability of sophisticated simulation approaches has now brought these simulations within the realm of possibility. In the following sections, we summarize the state of the art in computational modeling and simulation as it pertains to biohydrodynamics. This includes a critical evaluation of computational approaches used to date and a survey of other approaches that hold promise in this area. Furthermore, we examine the information that has been gleaned from these simulations *vis-a-vis* its impact on our understanding of biohydrodynamic phenomena.

II. Overview of Computational Approaches: Capabilities and Limitations

A. Flow Modeling

Both inviscid and viscous flow models have been employed in past studies in biohydrodynamics. Inviscid flow simulations are relatively inexpensive, but carry the usual weaknesses inherent in this approach. In addition, their application to biohydrodynamic flows can be further limited by the fact that these flows are often dominated by separated shear layers and associated vortex structures, which are a direct consequence of viscous effects. However, examination of results from inviscid computations serves the purpose of clearly delineating the flow regimes in which viscous flow mechanisms are dominant. Furthermore, these computations are quite inexpensive and allow for the rapid estimation of hydrodynamics forces and other gross flow features over a large parameter space. Examples of application of these methods to biohydrodynamics flows can be found in [5]–[13]. However, inviscid models have a limited range of validity and, given that viscous flow computations of these flows are well within the reach of present day computers, we focus primarily on viscous flow models.

The inclusion of viscous effects immediately raises the possibility of transition and turbulence, both of which then have to be accounted for in the simulations. A number of different options are available for modeling flow turbulence in numerical simulations and these are discussed here in the context on biohydrodynamic flows.

1) Reynolds-Averaged Navier Stokes (RANS) Modeling: In this approach, the spatial and temporal resolution is such that almost none of the turbulence scales can be resolved and the effect of these scales has to be accounted through turbulence models [14]–[16]. For flows that are steady in the mean, RANS simulations typically compute only the mean flow quantities. However, for flows with a large-scale externally imposed unsteadiness, such as that encountered in biohydrodynamics, unsteady RANS (URANS) is carried out with time-accurate schemes in which an attempt is made to compute directly and resolve the large scales associated with this unsteadiness.

RANS has found widespread use in the engineering community as a relatively inexpensive tool for flow analysis. Its obvious advantage over inviscid models is that it can account for viscous effects. At a basic level, this eliminates the D'Alembert's Paradox, which accounts for erroneous prediction of inviscid models for flapping foils at small amplitudes. In addition, viscous models no longer require imposition of the trailing-edge flow tangency condition and, therefore, allow for flow separation at any location on the foil/body. This is of critical importance for flapping foils, since the leading-edge stall is considered a key mechanism in these flows. As will be discussed later, the big advantage that RANS has over other viscous modeling approaches [such as direct numerical simulation (DNS) and large-eddy simulation (LES)] is that it is relatively inexpensive even for the complex configurations that are usually encountered in biohydrodynamics.

The main limitations of RANS approach are as follows: RANS provides only limited information about the flow. Even for time-accurate simulations of unsteady flows, such as those encountered in biohydrodynamics, the RANS simulations are designed to include only the largest flow structures (those that scale with the dominant flow-length scale) in the flow and the smaller scales are not included. The extent to which the absence of the smaller flow structures affects the prediction of the larger vortex structures is quite problem dependent and usually cannot be judged *a priori*.

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

The second issue with RANS is with regard to its ability to predict separation and separated flows. Since most turbulence models used in conjunction with RANS assume that the flow at the wall follows the behavior of a canonical attached boundary layer, prediction of flow separation has always been accepted as a weakness of this method. This is especially the case when the separation occurs over gently curving surfaces due to an adverse pressure gradient. In cases in which the separation is "massive" (e.g., leading edge stall over a pitching airfoil), this might not be a significant issue. However, even if separation is predicted correctly, conventional RANS approaches do not usually provide a reliable prediction of the flow in the separated region.

The final limitation of RANS models is their applicability to relatively low-Reynolds number flows. Conventional turbulence models are designed for classical high Reynolds number turbulence flows. Their application to flows at lower Reynolds numbers when the flow is transitional is more problematic. Typical Reynolds numbers in low-speed maneuvering situations are $O(10^4)$, which falls very much in the transitional range. For these flows, turbulence models specially designed for low-Reynolds number flows [12] might be more appropriate. However, even with these models, issues such as *ad hoc* specification of transition could diminish the predictive capability of RANS for such flows.

Examples of the application of URANS to biohydrodynamic flows can be found in the work of Jones and coworkers [6]–[8], Ramamurti and coworkers [10], [17], [18], Tuncer and Platzer [19], and Isogai *et al.* [20]. Some key results from these simulations will be discussed in later sections.

2) DNS: On the other end of the spectrum of flow modeling approaches from RANS is DNS, where the grid and temporal resolution is such as to resolve all the turbulence scales down to the dissipation range. Thus, in DNS there is no need to include a turbulence model and this type of modeling procedure should be expected to give results with incomparable accuracy. Furthermore, these simulations provide temporally and spatially accurate information about all the scales in the flow. The turbulence statistics and frequency spectra as well as vortex dynamics are also accurately predicted, thereby allowing a comprehensive analysis of flow. However, this detailed information comes at a very high cost and it has been estimated that the grid requirement of a DNS scales as $\mathrm{Re}^{9/4}$ [21]. Consequently, DNS is limited to relatively low Reynolds numbers and is difficult to find in the literature adequately validated DNSs of relatively complex flows for Reynolds numbers greater than about a few thousand [22], [23]. This range of Reynolds number would seem to allow for investigation of low-speed maneuvering biohydrodynamics, except that DNS is limited to relatively simple flows such as bluff-body wakes [22]–[24]. Although there have been some DNS studies of relatively complex flows [25], DNS has yet to be applied to flows as complex as a flexible flapping pectoral fin. All of the necessary ingredients for performing such simulations already exist and, with the widespread availability of large-scale high-performance parallel computers, it is expected that such simulations will become feasible in the next few years.

3) LES: This approach lies between the RANS and DNS approaches. In LES, a time-accurate simulation is carried out with resolution sufficient to resolve all of the energy-containing scales down to the inertial subrange. Only the scales not resolvable on the mesh are modeled through a subgrid-scale (SGS) turbulence model. Furthermore, since all the energy-containing scales are resolved directly, these simulations provide data over a wide range of dynamically significant scales in the flow and are, therefore, capable of predicting higher order statistics.

Simplified estimates indicate that the grid requirement for LES scales as Re^2 [26], which is a slower increase than that for DNS. Thus, LES of flows in relatively complex geometries with Reynolds numbers up to about 10 000 have been carried out where extensive validation against experimental data have confirmed the accuracy of the computed results [27]–[29].

It should be noted that, in LES, since only the subgrid scales are modeled, there is relatively less sensitivity to the model parameters. Furthermore, the introduction of the dynamic SGS model [30] has removed most of the *ad hoc* parameter adjustments that are prevalent in RANS computations. In the dynamic modeling procedure, the eddy viscosity is computed during the computation by estimating the energy being transferred from the resolved to the SGS modes. Furthermore, the dynamic model automatically detects laminar regions and turns off. It is also capable of predicting transition and automatically produces the correct wall behavior. Thus, LES with dynamic model is quite well suited for prediction of complex flows [21]. Based on our previous experience, it seems that LES of flows encountered in low-speed maneuvering hydrodynamics is feasible with current computer hardware. Given that the LES methodology provides data sufficient for examining the key flow mechanisms in low-speed maneuvering biohydrodynamics, it should emerge as the methodology of choice in the coming years.

4) Detached-Eddy Simulation (DES): As discussed above, LES clearly is a useful tool for detailed analysis of the physics of biohydrodynamic flows. However, given the significant computational resources required, LES would not be well suited for covering a large parameters space. It would also not be appropriate, for instance, for rapid engineering analysis of a biorobotic pectoral fin. For these tasks, one would typically turn to the RANS approach. However, as discussed above, the predictive capability of RANS can be seriously compromised in separated flows, such as those found for flapping foils. The RANS method is designed to model the entire spectrum of turbulent motions. While this might be adequate for simple attached flows, RANS turbulence models are unable to accurately predict phenomena occurring in flows that exhibit large separation. Unsteady massively separated flows are characterized by geometry-dependent eddies, the affect of which cannot be represented well in RANS turbulence models. In order to overcome some of these deficiencies, a new approach, DES, has been proposed [16], which combines elements of RANS and LES. This approach can be considered a modification of the RANS approach where the geometry-dependent unsteady turbulent motions in the separated flow region are represented with an LES-type approach. Separated flows computed using this approach have shown marked improvement in accuracy over corresponding RANS computations [31]. The computational expense of this approach is higher than that of a corresponding RANS, but significantly less than a corresponding LES. Based

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

on the above discussion, it would seem that DES would be preferable over RANS for rapid analysis of biohydrodynamic flow configuration.

B. Numerical Methodology

The type of grid employed quite often distinguishes one numerical simulation methodology from another. The following describes the various types of grid-based methodologies that are available along with comments on the suitability of each type of grid for biohydrodynamic flows.

1) Body-Conformal Grid Methods: The conventional approach to simulate flows with complex boundaries is to generate a body-conformal grid, i.e., a grid that conforms to the shape of the boundaries. Within this approach, two different types of grids are employed; the structured grid and the unstructured grid. The key advantage of structured grid methods over the unstructured grid approach is that discretization of conservation laws on these grid lead to systems of equations that are amenable to a powerful line/block-successive overrelaxation scheme (SOR) iterative technique. These methods work well for relatively simple geometries. As the geometry gets more complex, single-block grids might not suffice and one then has to turn to multiblock meshes. However, in configurations where complex moving boundaries are present, such as those common in biohydrodynamics, even conventional multiblock structured grid methods would be difficult to work with. For such flows, overset structured grids [32] are a good alternative.

Unstructured grid based methods are better suited than structured grid-based methods for simulation of biohydrodynamic flows due to their ability to handle complex geometries. Unstructured grid methods can be used in conjunction with finite volume, finite element, or spectral element-type discretization methods. For example, Ramamurti and Sandberg [17] have used a conventional finite-element method in their computations, whereas other groups [33] use spectral-element discretization techniques. An arbitrary Lagrangian–Eulerian (ALE) methodology is usually employed to handle moving boundaries wherein moving/deformable grids are employed. These methods offer great flexibility for the modeling of flow with complex moving boundaries. However, remeshing algorithms can significantly increase the complexity of the solution procedure and can compromise the robustness and accuracy of the solution procedure.

Recent developments [34] in unstructured grid methods for LES are promising, since these provide an appropriate framework for coupling the power of unstructured grid with that of the LES methodology. Use of such methods in biohydodynamic flows involving complex moving boundaries have yet to be undertaken.

2) Cartesian Grid/Immersed Boundary Methods: In recent years, a new approach has come to fore, which is well suited for simulating flows with complex moving boundaries. In the 1970s, Peskin introduced his "immersed boundary method," which is used to simulate flow in a modeled human heart [35], [36]. The key feature of this method was that simulations with complex moving boundaries were carried out on stationary Cartesian grids, which eliminated the need for the complicated remeshing algorithms that are usually employed with conventional body-conformal methods. Since then, a number of different variations of this methods have been developed [37]–[43]. These methods have been used to a large variety of flows with complex moving boundaries including cardiovascular flows [36], multiphase flows [37], [43], fluid-structure interaction [44], [45], fluid machinery [46], microfluidic devices [47], biological locomotion [48], and flapping foils [49].

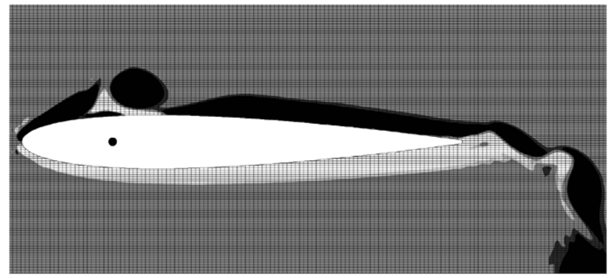


Fig. 2. Example of a nonconformal Cartesian grid for flow over an airfoil undergoing pitch oscillations.

TABLE I
PARAMETERS COVERED IN RANS STUDIES

	Re	h_1^*	k	α_1	ϕ
Tuncer & Platzer [19]	10^3 - 10^5	0.0125-1.0	0.5-7.85	0^o - 20^o	30^o,75^o,90^o
Isogai et al. [20]	10^5	1.0, 2.0	0.0-1.0	10^o, 20^o	30^o-150^o
Ramamurti et al. [18]	10^4, 10^3	0, 1	0-14	2,4, 15	30^o-140^o

These methods provide a unique capability for simulating flows with complex moving boundaries and, as such, are ideally suited for simulation of biohydrodynamic flows. In Sections III-A and III-B, we will show computed results for a configuration involving two flapping foils where the computations have been performed using a Cartesian grid method. Fig. 2 shows an example of a Cartesian grid that is being used to simulate flow past a pitching foil at $\mathrm{Re} = 12\,000$.

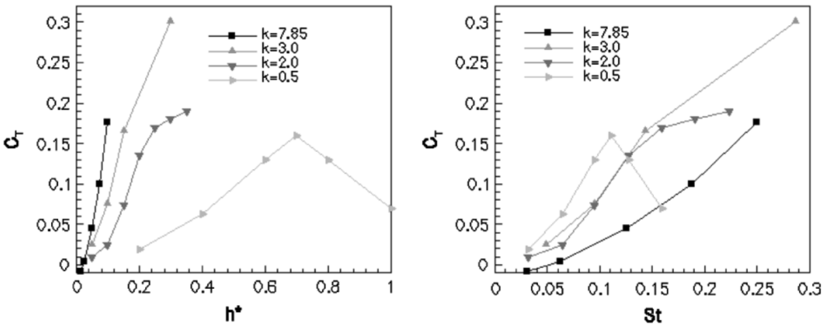
III. APPLICATION TO BIOHYDRODYNAMICS

A. Two-Dimensional (2-D) Flapping Foils

A number of groups have employed numerical simulations to investigate the hydrodynamics of flapping foils. Chief among these are the groups at the U.S. Naval Postgraduate School Monterey, CA, [6]–[8], [15] and Naval Research Laboratory, Washington, DC, (Ramamurti and coworkers [10], [17], [18]). Both groups have employed Euler as well as RANS codes for the analysis of flapping foils, but here we focus primarily on the RANS calculations. Another RANS study that will be discussed in detail here is that of Isogai *et al.* [20]. For the following discussion, consider a simple pitch-and-plunge (or heave) motion of a foil prescribed as

$$h = h_1 \sin(2\pi f t)$$
$$\alpha = \alpha_o + \alpha_1 \sin(2\pi f t + \phi)$$

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.



(a) (b)

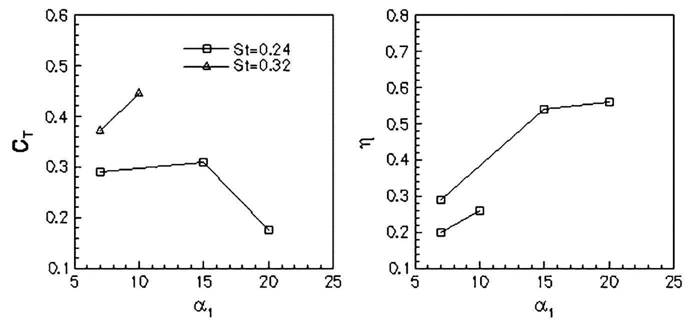
Fig. 3. (a) Variation of thrust with amplitude h_1^* and frequency k for a foil undergoing plunge oscillations, as computed by Tuncer and Plazer [19]. (b) Data replotted using Strouhal number (St) shows better collapse.

where h_1 is the plunge (or heave) amplitude, α_o is the mean pitch angle, and α_1 is the amplitude of the sinusoidal pitch angle variation. Furthermore, f is the flapping frequency and ϕ is the phase difference between the pitch and plunge motions. In addition to α_o, α_1, and ϕ, the nondimensional parameters that govern the fluid dynamics of this configuration are normalized plunge amplitude $h_1^* = h_1/c$, the Reynolds number $\mathrm{Re}_\infty = U_\infty c/\nu$ (where U_∞ is the freestream velocity, c is the foil chord, and ν is the kinematic viscosity of the fluid), and the Strouhal number $\mathrm{St} = 2h_1 f/U_\infty$ based on the wake width [50]. An alternative frequency parameter based on the foil chord $k = 2\pi fc/U_\infty$ has also been used in a number of studies [19], [51]. Note that $kh_1^* = \pi\mathrm{St}$ and that this product is also equal to the peak plunge velocity normalized by the freestream velocity.

It should be noted that a complete numerical investigation of the parameter space of even this simple flapping foil configuration has yet to be undertaken. However, recent RANS simulations of Tuncer and Platzer [19], when put together with those of Isogai *et al.* [20], represent the most comprehensive numerical evaluation of this configuration and we discuss the salient results from these studies. Both Isogai *et al.* [20] and Tuncer and Platzer [19] employ a NACA 0012 foil and solve the Reynolds averaged compressible Navier–Stokes equations on a structured grid. Tuncer's code employs a third-order upwind-biased scheme in space, whereas Isogai's code employs a total variation diminishing (TVD) scheme. Both codes use the Baldwin–Lomax turbulence model. As in most RANS calculations, a fine mesh is employed near the body but away from the body; the mesh is coarsened and the numerical viscosity is allowed to dissipate the vortex structures. Table I summarizes the range of parameters examined in these studies and the following figures show a sampling of the data obtained from these simulations.

Fig. 3(a) shows the effect of flapping amplitude and frequency on the thrust produced by the foil. These are two parameters that are known to have a significant effect on the thrust, which is confirmed by these computations. Replotting of the same data against the Strouhal number in Fig. 3(b) leads to a better collapse, which is in line with the findings of Triantafyllou *et al.* [50].

Fig. 4 shows the variation in thrust coefficient and efficiency (η) for a foil undergoing combined pitch and heave and trends are similar to those observed by Anderson *et al.* [9], [51]. In both



(a) (b)

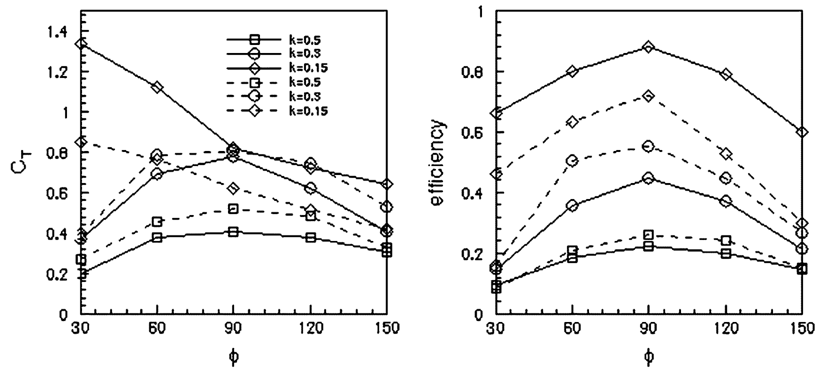
Fig. 4. Effect pitch amplitude and Strouhal number on (a) thrust and (b) efficiency for airfoil undergoing combined pitch and heave. ($h_1^* = 0.75, \phi = 75°$). Results from computations of Tuncer and Platzer [19].

computational studies, larger values of pitch amplitude have not been examined. Tuncer and Platzer [19] indicate that the solver has convergence problems for high pitch amplitudes. This might possibly be due to inadequate resolution of the large-scale dynamic stall vortices that form at high pitch amplitudes.

Fig. 5 shows the effect of varying the phase angle between pitch and plunge on the thrust and efficiency. The trend that emerges here, i.e., maximal thrust and efficiency at a phase angle of about roughly 90°, is also well established in experiments [5], [9]. It is interesting to note that these two studies employ very similar computational methodologies and simulate precisely the same configuration. Furthermore, both indicate that the results are grid independent. However, despite this, the resulting thrust and efficiencies for some cases differ by over 30%. As noted by both groups in their respective papers, the cause for this discrepancy is not clear. In our view, this clearly indicates that simulation fidelity is not guaranteed by grid independence in these RANS computations and that turbulence modeling effects have to be examined.

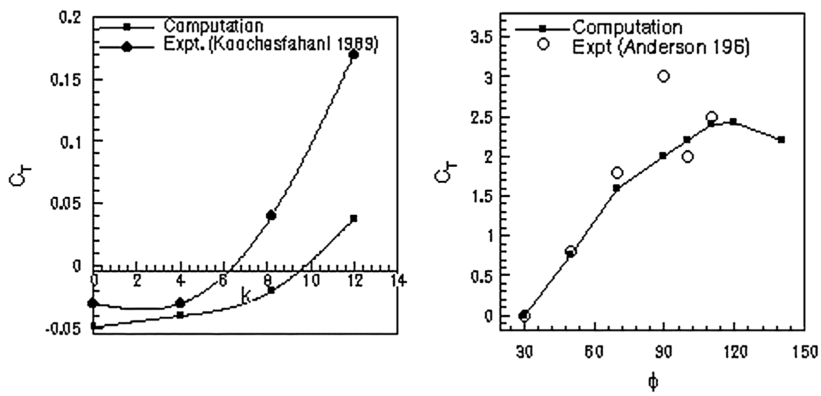
Ramamurti *et al.* [18] have also simulated a similar configuration (NACA 0012 foil undergoing pitch and heave motion in a freestream). However, Ramamurti *et al.* solve the incompressible Navier–Stokes equations on an unstructured grid, which is a significant departure from these other simulations. A relatively complex mesh movement algorithm is employed, which allows for the generation of a body-conformal mesh at every time step. The solver has been used to study the foil in pitch only, as well as combined pitch and heave. Fig. 6(a) shows a

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.



(a) (b)

Fig. 5. Effect of phase between pitch and plunge motions on (a) thrust and (b) efficiency. Comparison of Tuncer and Platzer [19] (solid lines) and Isogai *et al.* [20] (dashed lines) data.



(a) (b)

Fig. 6. Comparison between simulations (Ramamurti *et al.*) and experiments [9], [52]. (a) Effect of frequency parameter for pitch oscillations of 2° at $\mathrm{Re} = 1.2 \times 10^4$. (b) Effect of phasing between the pitch-and-plunge motion for $k = 3.8$ and $\mathrm{Re} = 1.1 \times 10^3$.

comparison of computed data for the 2° pitch oscillation case with experiments of Koochesfehani [52]. As noted by the authors, there is significant mismatch between the experiments and computations at higher frequencies. Ramamurti *et al.* [51] make a convincing case that the discrepancy is due to inaccuracies in measuring the thrust in the experiment. A procedure to improve thrust estimation in experiments has recently been proposed [53]. Also shown in Fig. 6(b) are the computed results of the foil in pitch-and-heave motion, where the configuration is similar to that of the experiments of Anderson [51]. The plot shows the variation of thrust with change in phase angle between the pitch-and-heave motions and the computed results are found to be generally consistent with the experiments.

It would seem that we now have a relatively good understanding of the effect of the frequency (St or k), amplitude (h_1^*), and pitch and heave phase difference (ϕ) on the hydrodynamic performance of 2-D flapping foils. The effect of the Reynolds number on the foil performance is somewhat less well understood. Most simulations and experiments of flapping foils have been carried out in the subcritical $(\mathrm{Re} < 10^6)$ regime [7], [19], [20], [54], since this is the range relevant to the swimming of small fish. Within this subcritical range, the general trend is that thrust increases with Reynolds number [54]. However, from an engineering point of view, it might be useful to understand the operation of flapping foils at postcritical Reynolds numbers $(\mathrm{Re} > 10^6)$, since autonomous underwater vehicles (AUVs) often operate in this regime. The effect of α_o on the foil performance also is not well understood. Since the primary effect of α_o is on the transverse (lift) force and not the thrust, past studies, which have focused mainly on propulsion and not maneuvering, have tended to disregard this parameter. A nonzero value of α_o could produce significant magnitudes of transverse force, which could be used in turning and pitching, as well as rolling maneuvers. There have been some experimental studies that have examined the effect of this parameter [55]–[58]. In a study of hover modes by Freymuth [55], it was shown that the angle of the hover jet (and, therefore, the direction of the thrust vector) could be manipulated quite easily using this parameter. Further understanding of this parameter is needed, however, especially with regard to its effect on the hydrodynamic forces.

Hover modes (for which $U_\infty = 0$), which are relevant to low-speed maneuvering hydrodynamics also need further examination. In addition to the experimental work of Freymuth [55], [59], hover modes for 2-D foils have been examined by Gustafson and Leben [60], Wang [61], and Mittal *et al.* [49]. One key finding from the study of Wang [61] was the rapid increase in the hydrodynamic force with heave amplitude h_1^* up to about $h_1^* = 2$, beyond which the force was relatively insensitive to this parameter. This result is in line with the findings of Freymuth [55]. Mittal *et al.* [49] simulated the flow associated with an elliptic airfoil in hover. The primary variable in their study was α_1, which was varied from 30° to 75°. The simulations indicated that variation in the pitch amplitude produced a range of wake topologies (see Fig. 7). For $\alpha_1 = 45^\circ$, an inverse Karman

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

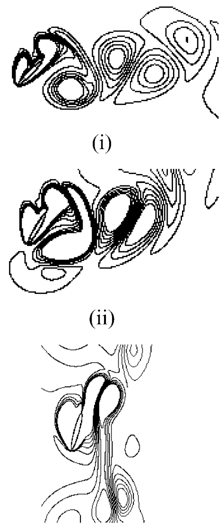


Fig. 7. Wake topologies for a flapping foil in hover computed in Mittal *et al.* [49]. (i) Vortex street for $\alpha_1 = 45°$, (ii) vortex dipoles for $\alpha_1 = 60°$, and (iii) pair of vortex dipoles at $\alpha_1 = 75°$.

vortex street was generated, which was associated with optimal thrust production, as estimated by the ratio of the mean thrust and the root-mean-square (rms) transverse force

Experiments clearly show that vortex dynamics in the wake often hold the key to the performance of flapping foils [3], [49], [51], [52], [55]. However, as noted earlier, the RANS approach does not naturally lend itself to simulation and analysis of wake topologies. Flow separation, which usually precedes the formation of the wake, is not easily predicted using RANS. Most RANS computations coarsen the grid rapidly away from the wall and in the wake, and rely on numerical dissipation to damp out the vortex structures. Tuncer and Platzer [19] have attempted to use Lagrangian tracers in conjunction with their RANS computations to visualize the wake topology and provide some qualitative validation of their computational methodology. However, direct visualization of the wake topology through an adequately resolved vorticity field is highly desirable and DNS, LES, and DES type techniques are well suited for this purpose.

Most past studies of flapping foils, due to their focus on propulsion, have limited themselves primarily to understanding the dependence of mean thrust and efficiency on the foil parameters. However, the magnitude of oscillatory components of both the thrust and transverse (lift) force on the foil can often be significantly larger than the mean values. Due to their large magnitudes, these oscillatory components could significantly affect the dynamics and control of an AUV powered by flapping foils [62]. It is, therefore, crucial to parameterize the oscillatory components of force in addition to the mean components.

Finally, the focus of past studies has primarily been on foils undergoing flapping motion. However, as discussed by Walker and Westneat [63], flapping motion is relevant for energy-efficient operation, such as is required during cruising, whereas rowing is more relevant to slow speed, maneuvering (starting, stopping, yawing, etc.) motion. Recent blade-element computations of Walker and Westneat [64] indicate that even though flapping motion is more efficient at all flow speeds, higher thrust can be generated at low speed through a rowing motion. However, little is known about the wake topologies and other flow details for fins undergoing a rowing motion. This aspect could be easily investigated through viscous flow simulations.

B. Three-Dimensional (3-D) Flapping Foils

We focus now on computations of 3-D flapping foil, which have more relevance in the context of low-speed maneuvering biohydrodynamics. For such foils, the aspect ratio (AR), defined as $(\text{span})^2/(\text{area})$, also has to be considered. It should be noted that many highly maneuverable fish have relatively low AR fins. For instance, mean ARs of the pectoral, dorsal, and anal fins for a boxfish *Ostracion meleagris* have been measured to be 1.9, 1.5, and 1.7, respectively [65], and those of a striped surfperch were found to be in the 1.8–2.5 range [4]. Furthermore, fish fins usually have highly complex 3-D planforms. There are relatively few computational studies of 3-D flapping foils, primarily because such computations tend to be highly complex and expensive. Liu and Kawachi [67] were one of the first to perform viscous flow computations of 3-D flapping foils. The configuration chosen in this study attempted to match the experiments of Van den Berg and Ellington [68]. In this experiment, a mechanical flapper was used to study the fluid dynamics of a hawkmoth wing, which had a relatively complex 3-D planform. The Reynolds number (Re) and frequency parameter (k) were 4000 and 0.74, respectively, and a relatively coarse mesh with 72 000 points was used. The computations were validated qualitatively by comparing with the smoke visualizations of Van den Berg and Ellington [68]. The computational study was used to examine the role of leading edge vortices on the force production. In a similar vein, Ramamurti *et al.* [10] have simulated the flow generated by a 3-D foil in hover where the flow configuration matches the *Drosophila melanogaster* wing experiments of Dickinson *et al.* [69]. These computations are primarily inviscid although, interestingly, comparisons with viscous simulations seem to indicate that viscous effects are not significant [10]. Computations are validated quantitatively by comparing computed forces against experimental measurements. This study examines the effect of phasing between translational and rotational motions and confirms the findings of Dickinson *et al.* [69] that rotational mechanisms play an important role in the production of hydrodynamic forces. Mittal *et al.* [70] have examined the vortex structures in the wake of low-aspect ratio heaving foils and have found that the wakes of such foils are dominated by vortex loops.

Perhaps the most ambitious simulations of 3-D flapping foils to date are those of the bird wrasse pectoral fin by Ramamurti *et al.* [71], which attempt to match the experiments of Walker and Westneat [66]. The pectoral fin kinematic data extracted from the experiments has been used to develop a realistic computational model of the 3-D flexible pectoral fin. Steady, quasisteady, and unsteady viscous flow computations have been carried out and the computed hydrodynamic forces have been correlated with the formation of flow structures in the vicinity of the pectoral fin. Since no detailed fluid flow measurements were made for the swimming fish in the experiment of Walker and Westneat [66], strong validation of the computations could not be provided. Simulation of four cycles of the pectoral fin motion

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

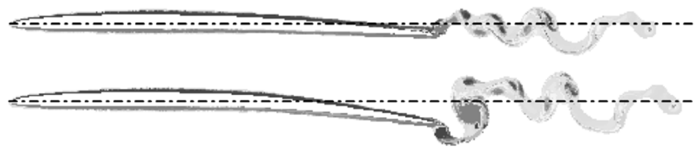


Fig. 8. Sequence of spanwise vorticity contour plots for flow past a flexible foil computed using a Lagrangian vortex method [76], [77].

required about 15 h on eight processors of a 400-MHz R12K SGI Origin 2000 [72].

Simulations of realistic and highly complex configurations. such as the bird wrasse by Ramamurti *et al.* [71]. certainly point to the growing role that computational modeling will play in biohydrodynamics. However, due to the many complex features that are simultaneously included in these models, it generally is quite difficult to extract clear insight into the underlying flow mechanisms and to delineate flow features/mechanisms that are universal from those that are specific to a particular configuration. Recent digital particle image velocimitry (DPIV) experiments of live swimming fishes by Drucker and Lauder [3] indicate that vortex rings are a dominant feature in the pectoral fin wake. It would, therefore, be of interest to examine wake topologies of 3-D flapping foils and connect these wake topologies to optimal thrust conditions, as has been done for 2-D flapping foils [9]. The fin planform and aspect-ratio are also important factors for 3-D flapping foils and recent experiments [63], [73], [74] have examined the correlation of fin shape with performance. The experiments of Combes and Daniel [73] with a spotted ratfish (*Hydrolagus colliei*) suggest that the AR as well as the proportion of area in the outer one-fifth of the wing can characterize the hydrodynamic performance of the flapping fin. Computational modeling is ideally suited for examining this issue further, since confounding variables can be effectively eliminated in these models.

C. Effect of Fin Flexion

Fins of most small highly maneuverable fish tend to be highly flexible. Fin flexion, when controlled actively, allows for on-demand morphing of the fin and can provide precise control over a large operational envelope. The recent study of Combes and Daniel [73] indicates that flexion in the pectoral fin of a ratfish reduces the thrust and increases the efficiency. Apart from this, little work has been done to understand the penalties/benefits of fin flexion. This issue is another that is well suited for investigation through computational modeling. In computational models, flexion can be varied in a systematic manner in order to examine its effect on fin performance. Inclusion of flexion would require a two-way coupled solution procedure and would have to incorporate an appropriate constitutive model for the fin material. Examples of flexible foil simulations include [75] and [76]. In the work of Grant [76], simulations of flow past flexible foils are being carried out using a Lagrangian vortex method [77]. In this study, the simulations are being used to examine the effect of forced oscillatory flexing in a foil and Fig. 8 shows some qualitative results from these ongoing simulations.

D. Full-Body Hydrodynamics

The presence of the body can potentially have a significant effect on locomotor performance, especially during maneuvering motions. However, relatively little work [54], [71] has been done in using numerical simulations to study full-body hydrodynamics. The issue of body–fin interaction, therefore, remains open and needs to be investigated through simulations and experiments. Due to the computational costs associated with simulating full body hydrodynamics, RANS and DES seem best suited for this purpose. These modeling techniques can be used to identify critical regions in the parameter space, which can then be subjected to detailed study through LES and DNS.

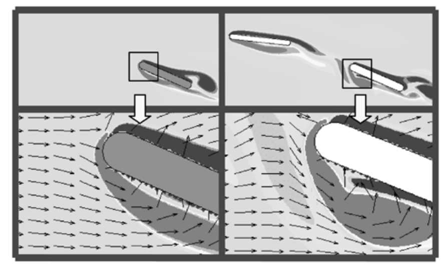


Fig. 9. Comparison of computed flow past tail fin (a) without and (b) with upstream dorsal fin.

E. Biologically Inspired Active Control

This is another area that is of importance in biomimetic hydromechanical systems. Consider, for example, the bluegill sunfish (Fig. 1), which has been the subject of investigation by Drucker and Lauder [3]. Based on DPIV data, Drucker and Lauder have hypothesized that vortex structures shed by the soft dorsal fin (indicated by an arrow in Fig. 1) could enhance the thrust of the tail fin. This hypothesis has been examined through 2-D numerical simulations by Akhtar [78]. Kinematic data from the experiments [3] has been used to set up a simulation of flow past two flapping foils so as to mimic the interaction of the soft-dorsal and tail fins. These simulations employ a Cartesian grid method [39], [40] wherein the simulations are carried out on a stationary Cartesian grid.

Thrust and efficiency of the tail fin with and without the upstream soft-dorsal fin has been computed and compared. In the 2-D simulations, the thrust and efficiency of the tail fin is found to increase by about 100% and 50%, respectively, thereby providing strong support for the hypothesis of Drucker and Lauder [3]. The simulations indicate that the vortex structures from the dorsal fin increase the apparent pitch angle of the tail fin at mid-stroke (see Fig. 9). This leads to the formation of a large leading edge vortex that stays attached to the fin and increases the thrust on the foil. The 2-D simulations do not account for all the effects that are expected to be present in this flow and 3-D simulations with more realistic geometries would be needed to further verify these findings. In general, however, such thrust augmentation strategies could potentially have large payoffs in

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

engineered biomimetic systems where even small increases in efficiency could translate to larger payloads and/or extended operational envelope.

IV. Conclusion

Computational modeling and simulation of biohydrodynamic flows is still in its infancy. Computational modeling gives us the ability to eliminate confounding variables and isolate effects and mechanisms associated with chosen parameters. Computations also provide detailed full-field information about the flow. Due to this, it is expected that computational modeling will play an increasing role in the analysis of these flow configurations in the future. In addition to inviscid and RANS/URANS modeling that has been the norm in this area, it is expected that more accurate modeling techniques, such as DES, LES, and DNS, will also be brought to bear on these problems. These techniques provide significantly more information about the flow, which is essential for understanding the flow physics. Due to the complex geometries involved, unstructured grid method will be more widely employed than body-conformal structured grid methods. In addition, Cartesian grid/immersed boundary methods are also expected to find extensive use in computational modeling of biohydrodynamic configurations. Systematic and comprehensive validation remains a crucial component of these computational investigations and requires that quantitative fluid flow data, such as that extracted from DPIV, be available from experiments.

The following key areas/issues in biohydrodynamics have been identified in which progress could be made using computational modeling:

- further parameter survey of 2-D flapping foils;
- parameterization of oscillatory components of forces on flapping foils;
- performance and wake topologies of 3-D foils, including systematic investigation of AR and planform effects;
- examination of hovering and rowing motions, which are relevant to low-speed maneuvering biohydrodynamics;
- penalties/benefits of fin flexion;
- biologically inspired active control strategies.

References

[1] L. Tammero and M. H. Dickinson, "The influence of visual landscape on the free-flight behavior of the fruit fly Drosophila melanogaster," *J. Exp. Biol.*, vol. 205, pp. 327–343, 2002.
[2] M. S. Triantafyllou and S. Triantafyllou, "An efficient swimming machine," *Sci. Amer.*, vol. 272, pp. 64–70, 1995.
[3] E. Drucker and V. Lauder, "Experimental hydrodynamics of fish locomotion: Functional insights from wake visualization," *Integ. Comp. Biol.*, vol. 42, pp. 243–257, 2002.
[4] E. G. Drucker and J. S. Jensen, "Kinematic and electromyographic analysis of steady pectoral fin swimming in the surfperches," *J. Exp. Biol.*, vol. 200, pp. 1709–1723, 1997.
[5] J. Katz and D. Weihs, "Hydrodynamic propulsion by large amplitude oscillations of an airfoil with chordwise flexibility," *J. Fluid Mech.*, vol. 88, pp. 485–497, 1978.
[6] K. D. Jones, J. C. S. Lai, I. H. Tuncer, and M. F. Platzer, "Computational and experimental investigation of flapping-foil propulsion," presented at the 1st Int. Symp. Aqua Bio-Mechanisms/Int. Seminar on Aqua Bio-Mechanisms, Tokai Univ. Pacific Center, Honolulu, HI, Aug. 2000.
[7] K. D. Jones, B. M. Castro, O. Mahmoud, and M. F. Platzer, "Numerical and experimental investigation of flapping-wing propulsion in ground effect AIAA 2002-0866," presented at the AIAA 40th Aerospace Sciences Meeting, Reno, NV, Jan. 2002.
[8] K. D. Jones and M. F. Platzer, "On the design of efficient micro air vehicles," in *Design and Nature Comparing Design in Nature with Science and Engineering*, C. A. Brebbia, L. J. Sucharov, and P. Pascolo, Eds, Southampton, U.K.: WIT, 2002, pp. 67–76.
[9] J. M. Anderson, K. Streitlien, D. S. Barrett, and M. S. Triantafyllou, "Oscillating foils of high propulsive efficiency," *J. Fluid Mech.*, vol. 360, pp. 41–72, 1998.
[10] R. Ramamurti and W. C. Sandberg, "A three-dimensional computational study of the aerodynamic mechanisms of insect flight," *J. Exp. Biol.*, vol. 205, no. 10, pp. 1507–1518, 2002.
[11] P. Watts and F. E. Fish, "The influence of passive, leading edge tubercles on wing performance," presented at the 12th Int. Symp. Unmanned Untethered Submesible Technolody, Autonomous Undersea Systems Inst., Durham, NH, 2001.
[12] C. E. Lan, "The unsteady quasivortex-lattice method with applications to animal propulsion," *J. Fluid Mech.*, vol. 93, no. 4, pp. 747–765, 1979.
[13] M. J. C. Smith, "Simulating moth wing aerodynamics: Toward the development of flapping-wing technology," in *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 34, 1996, pp. 1348–1355.
[14] D. C. Wilcox, Turbulence Modeling for CFD, DCW Ind., La Cañada, CA, 1998.
[15] B. S. Baldwin and H. Lomax, "Thin layer approximation and algebraic model for separated turbulent flow," *Amer. Inst. Aeronaut. Astronaut. J.*, pp. 78–757, 1978.
[16] P. R. Spalart, W. H. Jou, M. Strelets, and S. R. Allmaras, "Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach," in *Proceedings First AFOSR International Conference DNS/LES, Advances in DNS/LES*, C. Liu and Z. Liu, Eds. Columbus, OH: Greyden, Aug. 1997.
[17] R. Ramamurti and W. C. Sandberg, "A three-dimensional computational study of the aerodynamic mechanisms of insect flight," *J. Exp. Biol.*, vol. 205, no. 10, pp. 1507–1518, 2002.
[18] R. Ramamurti, W. C. Sandberg, and R. Löhner, "Simulation of flow about flapping airfoils using a finite element incompressible flow solver," in *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 39, 2001, pp. 253–260.
[19] I. H. Tuncer and M. F. Platzer, "Computational study of flapping airfoil aerodyanmics," *Amer. Inst. Aeronaut. Astronaut. J. Aircraft*, vol. 37, no. 3, pp. 514–520, 2000.
[20] K. Isogai, Y. Shinmoto, and Y. Watanabe, "Effects of dynamic stall on propulsive efficiency and thrust of flapping airfoil," *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 37, no. 10, pp. 1145–1151, 1999.
[21] U. Piomelli, "Large-eddy simulation: Achievements and challenges," *Progr. Aerosp. Sci.*, vol. 35, pp. 335–362, 1999.
[22] E. Karniadakis and S. Triantafyllou, "Three-dimensional dynamics and transition to turbulence in the wake of bluff objects," *J. Fluid Mech.*, vol. 238, p. 1, 1992.
[23] R. Mittal, J. J. Wilson, and F. M. Najjar, "Symmetry properties of the transitional sphere wake," in *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 40, 2002.
[24] R. Mittal and S. Balachandar, "Effect of three-dimensionality on the lift and drag of nominally two-dimensional cylinders," *Phys. Fluids*, vol. 7, no. 8, pp. 1841–1865, 1995.
[25] J. Newman and E. Karniadakis, "Simulations of flow past a freely vibrating cable," *J. Fluid Mech.*, vol. 344, no. 95, pp. 95–121, 1997.
[26] J. S. Baggett, J. Jimenez, and A. Kravchenko, Resolution requirements in large-eddy simulations of shear flows, in Annu. Res. Briefs, NASA Ames/Stanford Univ., pp. 51–66, 1997. Center Turbulence Res..
[27] H.-J. Kaltenbach, M. Fatica, R. Mittal, T. S. Lund, and P. Moin, "Study of flow in a planar asymmetric diffuser using large-eddy simulation," *J. Fluid Mech.*, vol. 390, pp. 151–186, 1999.
[28] D. You, R. Mittal, M. Wang, and P. Moin, "Progress in large-eddy simulation of a rotor tip-clearance flow," presented at the 12th Annu. DoD HPCMP User Group Conf., Austin, TX, 2002.
[29] C. Muthanna, K. S. Wittmer, and W. J. Devenport, "Turbulence structure of the flow downstream of a compressor cascade with tip leakage," in *Proc. American Institute Aeronautics and Astronautics 36th Aerospace Sciences Meeting*, Reno, NV, Jan. 1998, pp. 98–420.
[30] M. Germano, U. Piomelli, P. Moin, and W. H. Cabot, "A dynamic sub-grid scale eddy viscosity model," *Phys. Fluids*, vol. A, no. 3, pp. 1760–1765, 1991.
[31] K. D. Squires, J. R. Forsythe, S. A. Morton, W. Z. Strang, K. E. Wurtzler, R. F. Tomaro, M. J. Grismer, and P. R. Spalart, "Progress on detached-eddy simulation of massively separated flows," *Amer. Inst. Aeronaut. Astronaut.*, no. 2002–1021, 2002.
[32] W. M. Chan and R. L. Meakin, "Advances toward automatic surface domain decomposition and grid generation for overset grids," presented at the Amer. Inst. Aeronaut. Astronaut. 13th Computational Fluid Dynamics Conf., AIAA Paper 97-1979, Snowmass, CO, 1997.

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.

[33] E. Karniadakis and S. J. Sherwin, *Spectral/hp Element Methods for CFD*. Oxford, U.K.: Oxford Univ. Press, Mar. 1999.

[34] F. Ham, S. Apte, G. Iaccarino, X. Wu, H. Herrmann, G. Constantinescu, K. Mahesh, and P. Moin, "Unstructured LES of realistic multiphase flows in gas turbine combustors," in *CTR Annu. Res. Briefs*, 2003, pp. 139–160.

[35] C. S. Peskin, "Numerical analysis of blood flow in the heart," *J. Comput. Phys.*, vol. 25, pp. 229–252, 1977.

[36] C. S. Peskin and D. M. McQueen, "A three-dimensional computational model of blood flow in the heart I. Immersed elastic fibers in a viscous incompressible fluid," *J. Comput. Phys.*, vol. 81, pp. 372–405, 1989.

[37] G. Tryggvason, B. Bunner, O. Ebrat, and W. Tauber, "29th Computational Fluid Dynamics," Von Karman Institute for Fluid Dynamics, ser. Lecture Ser. 1998-03.

[38] R. J. Leveque and Z. Li, "The immersed interface method for elliptic equations and discontinuous coefficients and singular sources," in *SIAM J. Numer. Anal.*, vol. 21, 1994, pp. 101–9.

[39] T. Ye, R. Mittal, H. S. Udaykumar, and W. Shyy, "An accurate Cartesian grid method for simulation of viscous incompressible flows with complex immersed boundaries," *J. Comput. Phys.*, vol. 156, pp. 209–240, 1999.

[40] H. S. Udaykumar, R. Mittal, P. Rampunggoon, and A. Khanna, "A sharp interface Cartesian grid method for simulating flows with complex moving boundaries," *J. Comput. Phys.*, vol. 174, pp. 345–380, 2001.

[41] H. S. Udaykumar, R. Mittal, and P. Rampunggoon, "Interface tracking finite volume method for complex solid-fluid interactions on fixed meshes," *Commun. Numer. Methods Eng.*, vol. 18, pp. 89–97, 2002.

[42] R. Glowinski, T. W. Pan, and J. Periaux, "Fictitious domain methods for the simulation of Stokes flow past a moving disk," in *Computational Fluid Dynamics*, J. A. Desideri, C. Hirsh, P. Le Tallec, M. Pandolfi, and J. Periaux, Eds, Chichester, U.K.: Wiley, 1996, pp. 64–70.

[43] R. Fedkiw and X.-D. Liu, "The ghost fluid method for viscous flows," in *Innovative Methods for Numerical Solutions of Partial Differential Equations*, M. Hafez and J.-J. Chattot, Eds, Singapore: World Scientific, 2002, pp. 111–143.

[44] R. Mittal, V. Seshadri, S. Sarma, and H. S. Udaykumar, "Computational modeling of fluidic micro-handling processes," presented at the 5th Int. Conf. Modeling and Simulation of Microsystems, San Juan, PR, 2002.

[45] R. Mittal, C. Bonilla, and H. S. Udaykumar, "Cartesian grid methods for simulating flows with moving boundaries," in *Computational Methods and Experimental Measurements XI*, C. A. Brebbia, G. M. Carlomagno, and P. Anagnostopoulous, Eds. Halkidiki, Greece: WIT, 2003.

[46] E. A. Fadlun, R. Verzicco, P. Orlandi, and J. Mohd-Yusof, "Combined immersed-boundary/finite-difference methods for three-dimensional complex flow simulations," *J. Comput. Phys.*, vol. 161, pp. 35–60, 2000.

[47] R. Mittal and P. Rampunggoon, "On the virtual aero shaping effect of synthetic jets," *Phys. Fluids*, vol. 14, pp. 1533–1536, 2002.

[48] L. J. Fauci and C. S. Peskin, "A computational model of aquatic animal locomotion," *J. Comput. Phys.*, vol. 77, pp. 85–108, 1988.

[49] R. Mittal, Y. Utturkar, and H. S. Udaykumar, "Computational modeling and analysis of biomimetic flight mechanisms," presented at the 40th Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 2002.

[50] S. Triantafyllou, M. S. Triantafyllou, and M. A. Grosenbaugh, "Optimal thurst development in oscillating foils with applications to fish propulsion," *J. Fluids Structures*, vol. 7, pp. 205–224, 1992.

[51] J. M. Anderson, "Vortex control for efficient propulsion," Ph.D. dissertation, Massachusetts Inst. Technol., Cambridge, MA, 1996.

[52] M. M. Koochesfahani, "Vortical patterns in the wake of an oscillating airfoil," presented at the Amer. Inst. Aeronaut. Astronaut. 25th Aerospace Sciences Meeting, Reno, NV, Jan. 1987.

[53] D. Bohl and M. Koochesfahani, "MTV measurements of the flow structure downstream of an oscillating airfoil," Amer. Inst. Aeronaut. Astronaut. Paper 2003-4017, 2003.

[54] H. Liu and K. Kawachi, "A numerical study of undulatory swimming," *J. Comput. Phys.*, vol. 155, pp. 223–247, 1999.

[55] P. Freymuth, "Thrust generation by an airfoil in hover modes," *Expts. Fluids*, vol. 9, pp. 17–24, 1990.

[56] K. Ohmi, M. Coutanceau, T. P. Loc, and A. Dulieu, "Vortex formation around an oscillating and translating airfoil at large incidences," *J. Fluid Mech.*, vol. 211, pp. 37–60, 1990.

[57] K. Ohmi, M. Coutanceau, O. Daube, and T. P. Loc, "Further experiments on oscillating and translating airfoil at large incidences," *J. Fluid Mech.*, vol. 225, pp. 607–630, 1991.

[58] M. S. Triantafyllou, F. S. Hover, and S. Licht, "The mechanics of force production in flapping foils under steady-state and transient motion conditions," Dept. Ocean Eng. Testing Tank Facility, Massachusetts Inst. Technol., Cambridge, MA, Rep. 031 903, 2003.

[59] P. Freymuth, "Propulsive vortical signature of plunging and pitching airfoils," in *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 26, 1988, pp. 881–883.

[60] K. E. Gustafson and R. Leben, "Computation of dragonfly aerodynamics," *Comp. Phys. Commun.*, vol. 65, pp. 121–132, 1991.

[61] Z. J. Wang, "Two dimensional mechanism for insect hovering," *Phys. Rev. Lett.*, vol. 85, no. 10, pp. 2216–2219, 2000.

[62] S. N. Singh and A. Simha, "Open-loop and feedback pectoral fin control of biorobotic auv," presented at the 12th Int. Symp. Untethered Submersible Technology, Durham, NH, 2003.

[63] J. A. Walker and M. W. Westneat, "Performance limits of labriform propulsion and correlates with fin shape and motion," *J. Exp. Biol.*, vol. 205, pp. 177–187, 2002.

[64] ——, "Mechanical performance of aquatic rowing and flying," *Proc. R. Soc. London*, ser. B, vol. 267, pp. 1875–1881, 2000.

[65] J. R. Hove, L. M. O'Bryan, M. S. Gordon, P. W. Webb, and D. Weihs, "Boxfishes (*Teleostie: Ostraciidae*) as a model system for fishes swimming with many fins," *I. Kinemat. J. Exp. Biol.*, vol. 204, pp. 1459–1471, 2001.

[66] J. A. Walker and M. W. Westneat, "Labriform propulsion in fishes: Kinematics of flapping aquatic flight in the bird wrasse, Gomphosus varius (Labridae)," *J. Exp. Biol.*, vol. 200, pp. 1549–1569, 1997.

[67] H. Liu and K. Kawachi, "A numerical study of insect flight," *J. Comput. Phys.*, vol. 146, pp. 124–156, 1998.

[68] C. Van den Berg and C. P. Ellington, "The vortex wake of a 'hovering' model hawkmoth," *Phil. Trans. R. Soc. Lond. B*, vol. 352, pp. 317–328, 1997.

[69] M. H. Dickinson, F. O. Lehmann, and S. P. Sane, "Wing rotation and the aerodynamic basics of insect flight," *Sci.*, vol. 284, pp. 1954–1960, 1999.

[70] R. Mittal, I. Akhtar, M. Bozkurttas, and F. M. Najjar, "Toward a conceptual model of a bio-robotic AUV: Pectoral fin hydrodynamics," presented at the 13th Int. Symp. Unmanned Untethered Submersible Technology (UUST), Durham, NH, Aug. 2003.

[71] R. Ramamurti, W. C. Sandberg, R. Löhner, J. A. Walker, and M. W. Westneat, "Fluid dynamics of flapping aquatic flight in the bird wrasse: 3-D unsteady computations with fin deformation," *J. Exp. Biol.*, vol. 205, pp. 2997–3008, 2002.

[72] R. Ramamurti, private communication, 2003.

[73] S. A. Combes and T. L. Daniel, "Shape, flapping and flexion: Wing and fin design for forward flight," *J. Exp. Biol.*, vol. 204, pp. 2073–2085, 2001.

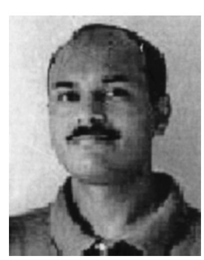
[74] P. C. Wainwright, D. R. Belwood, and M. W. Westneat, "Ecomorphology of locomotion in labrid fishes," Env. Biol. Fish, vol. 65, no. 1, pp. 47–62, 2002, to be published.

[75] W. Shyy, D. A. Jenkins, and R. W. Smith, "Study of adaptive shape airfoils at low Reynolds numbers in oscillatory flow," in *Amer. Inst. Aeronaut. Astronaut. J.*, vol. 35, 1997, pp. 1545–1548.

[76] J. S. Grant, private communication, 2004.

[77] J. S. Marshall, J. R. Grant, A. A. Gossler, and S. A. Huyer, "Vorticity transport on Lagrangian tetrahedral mesh," *J. Comput. Phys.*, vol. 161, pp. 85–113, 2000.

[78] I. Akhtar, "Thrust augmentation through active flow control: Lessons from a bluegill sunfish," M.S. thesis, Dept. Mech. Aerospace Eng., The George Washington Univ., Washington, DC, 2003.



Rajat Mittal was born in New Delhi, India, on April 6, 1967. He received the B.Tech. degree in aeronautical engineering from the Indian Institute of Technology, Kanpur, in 1989, the M.S. degree in aerospace engineering from the University of Florida, Gainesville, in 1991, and the Ph.D. degree in applied mechanics from The University of Illinois, Urbana-Champaign, in 1995.

He joined the Center for Turbulence Research, Stanford University, Stanford, CA, as a Postdoctoral Fellow, where he conducted research in the area of turbulent flows. Subsequently, he joined the Department of Mechanical Engineering, University of Florida, as an Assistant Professor, where he taught from 1996 to 2001. He currently is an Associate Professor in the Department of Mechanical and Aerospace Engineering, The George Washington University, Washington, DC. His research interests include computational fluid dynamics, vortex dominated flows, fluid-structure interaction, flow control, and biofluid dynamics.

Dr. Mittal is a Member of the American Institute of Aeronautics and Astronautics, the American Society of Mechanical Engineers, and the American Physical Society.

Authorized licensed use limited to: b-on: Instituto Politecnico de Tomar. Downloaded on April 19,2024 at 18:43:51 UTC from IEEE Xplore. Restrictions apply.