THESIS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY

A Large Eddy Simulation Based Fluid-Structure Interaction Methodology with Application in Hydroelasticity

Andreas Feymark

Department of Shipping and Marine Technology
Chalmers University of Technology
Gothenburg, Sweden
2013

A Large Eddy Simulation Based Fluid-Structure Interaction Methodology with Application in Hydroelasticity
Andreas Feymark
Gothenburg, 2013
ISBN: 978-91-7385-840-3

Doktorsavhandlingar vid Chalmers tekniska högskola
Ny serie Nr 3521
ISSN: 0346-718X

Department of Shipping and Marine Technology
Division of Marine Design
Chalmers University of Technology
SE - 412 96 Gothenburg, Sweden
Telephone: +46 (0)31-7721000

Printed by Chalmers Reproservice
Gothenburg, Sweden, 2013

– FELIX, QUI POTEST RERUM COGNOSCERE CAUSAS –

A Large Eddy Simulation Based Fluid-Structure Interaction Methodology with Application in Hydroelasticity

Andreas Feymark
Department of Shipping and Marine Technology
Division of Marine Design
Chalmers University of Technology

Abstract

The phenomenon of hydroelasticity is a subarea of Fluid-Structure Interaction (FSI) and of major importance in many engineering applications related to hydrodynamics and naval architecture e.g. wave-induced vibrations, such as springing, whipping and slamming, propeller singing, composite propellers or turbines, acoustic signatures from naval vessels, highly loaded thin propeller blades, and cavitation erosion. Some of these phenomena can be assessed with reasonable reliability, but in cases where medium- to small-scale flow features are important the computational models need to be further developed to improve predictive capability and enable new conceptual designs.

The work presented in this thesis has this kind of development as objective and a method capable of providing hydroelasticity predictions based on LES is presented and validated. The problem is particularly challenging as the densities of the fluid and the structure are comparable and an implicit coupling is thus needed to ensure a stable solution procedure. Furthermore, LES is not well established in the FSI context and especially not within the area of hydroelasticity. High resolution of the computation is necessary and the algorithm needs to run efficiently on large parallel computer systems. Reliable results also include predicting the correct separation pattern, in general on smoothly curved geometries. To address this a validation of LES in terms of predicting the correct separation pattern was performed and presented here, including also the development and validation of a LES turbulence trip model.

The results presented can be divided into three parts, firstly the prediction and validation of open separation phenomena around a prolate spheroid, secondly the prediction and validation of the flow around an oscillating cylinder and thirdly the development of a fluid-structure interaction methodology for hydrodynamic applications and corresponding prediction and validation of the deformation of a flexible hydrofoil. The results all show a good agreement with experimental data, thus supporting the validity of the fluid-structure interaction methodology for hydroelastic applications presented within the scope of this thesis. Finally, the parallel performance of the implementation is analyzed through both weak and strong scaling and found to be satisfactory.

Keywords: naval architecture, large eddy simulation, forced oscillation, hydrodynamics, hydroelasticity, numerical simulation, subgrid modeling, fluid-structure interaction, trip model

Preface

This thesis consists of an introduction and five papers. Please note that the author changed his surname from Karlsson to Feymark in January 2009.

Paper I

Karlsson, A., Fureby, C., "LES of the Flow Past an Inclined 6:1 Prolate Spheroid," *47th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, Orlando, Florida, Jan. 5-8, AIAA 2009-1616, 2009.
Contributions: Contributed in the planning and writing of the paper, performed the LES, post-processing and analysis of results.

Paper II

Feymark, A., Chapuis, M., Fureby, C., Liefvendahl, M., "Large Eddy Simulation of High Re Number Partially Separated Flow," *50th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*, Nashville, Tennessee, Jan. 6-9, AIAA 2012-100, 2012.
Contributions: Contributed in the planning and writing of the paper, performed the LES of the prolate spheroid, development of the trip-model, post-processing and analysis of results.

Paper III

Feymark, A., Alin, N., Bensow, R. E., Fureby, C., "Numerical Simulation of an Oscillating Cylinder using Large Eddy Simulation and Implicit Large Eddy Simulation," *Journal of Fluids Engineering*, No. 3, Vol. 1342, pp. 031205, 2012.
Contributions: Contributed in the planning and writing of the paper, performed the LES, post-processing and analysis of results.

Paper IV

Feymark, A., Alin, N., Bensow, R. E., Fureby, C., "Large-Eddy Simulation of an Oscillating Cylinder in a Steady Flow," *AIAA Journal*, No. 4, Vol. 55, April, 2013.
Contributions Contributed in the planning and writing of the paper, performed the LES, post-processing and analysis of results.

Paper V

Feymark, A., Cesur, A., Alin, N., Fureby, C., Revstedt, J., Bensow, R. E., "Fluid-Structure Interaction using Parallel Open Source Software with Application in Hydroelasticity," *In preparation for submission*.
Contributions: Contributed in the planning and writing of the paper, performed the majority of the code development, performed the LES, post-processing and analysis of results.

Acknowledgement

I would like to express my sincere thankfulness to my supervisors and mentors Rickard Bensow, for always taking his time and being a good listener, Christer Fureby, for his commitment and inspiring visions and Niklas Alin for always bringing joy and providing new perspectives. I would, furthermore, like to thank my colleagues at the department, especially Leslie who has, since my first day at Shipping and Marine Technology, been one of my best friends, and my colleagues at Lund University, Alper Cesur and Johan Revstedt. I would also like to thank my family, Kristin, Roger, Catrin, Jonas, Annelie, Malin, Evelina, Albin, Lova, Colin and Dylan for always being there for me, you are the best.

Financial support is provided by VINNOVA through Lighthouse maritime competence centre.

Table of Contents

1. Background

Since the first ships saw the light of the day naval architects have been forced to take the impact of the strains and stresses induced by the sea into consideration. Although a complete model of the dynamic behavior of a ship would be of great interest and importance the evolution of naval architecture has, due to a strong dependency on the development of computational methods and resources, resulted in a division of the field into distinct subject areas.

The term hydroelasticity was first introduced by Heller and Abramson, [1], as the naval counterpart to aeroelasticity. As such, hydroelasticity generally falls under the subject of Fluid-Structure Interaction (FSI), describing the interaction between interconnected structural deflection and fluid flow. Moreover, there exist two distinct coupling approaches. The first one being the monolithic approach where the fluid and the solid systems of equations are solved simultaneously, including the boundary conditions, [2–4]. In the second one, the partitioned approach, a staggered approach is utilized, [5–7]. The monolithic approach is considered to be more stable and accurate however expensive in computational time while the partitioned approach gives more flexibility with the advantage of possible usage of already existing (feature rich) solvers. The possibility to use already existing also reduces the programming required compared with a monolithic approach. The partitioned schemes can also be subdivided into explicit and implicit coupling schemes. Explicit coupling schemes solve the fluid and solid equations without any sub-iteration, leading to inexact matching of the coupling conditions at the fluid-solid interface. Explicit coupling is generally favored in aeroelastic applications, [8], where the difference in density between solid and fluid is large. In contrast, implicit coupling schemes are often used when solid and fluid densities are comparable, such as in hydroelastic and biomedical applications.

Hydroelasticity is a broad area and of great importance in the analysis and prediction of, e.g. wave-induced vibrations, such as, springing, whipping and slamming, propeller singing, flexible propeller blades, signature levels from submarines, structural fatigue, wave induced movements and loads of marine structures, properties and sea loads on rapidly moving vessels and human comfort and fatigue. Springing as a nautical term refers to stationary vertical vibrations of the hull girder due to oscillating wave loads whereas whipping refers to transient hull girder vibrations due to impact, not necessarily only by waves. Both these phenomena may dominate the contribution to fatigue for some vessels but are today, due to their intrinsic complexity, disregarded in the International Association of Classification Societies (IACS) rules, [9]. Slamming which is the impact of the hull bottom on the water surface, resulting in very high loads on the ship structures, is nevertheless considered in the IACS rules.

The ongoing development of computational software and resources suggests that it would, in the long term, be possible to analyze hydroelasticity phenomena using three-dimensional non-linear theories based on traditional Computational Fluid Dynamics (CFD) solvers and corresponding structural mechanicals solvers within the field of

Computation Solid Dynamics (CSD). That would mean access to detailed predictions that could, for example, facilitate the development of the simplified potential flow based methods frequently used today. For a comprehensive summary of the computational methods related to springing, whipping and slamming, see Chen *et al*, [10]. Although being important, predictions of these kinds of effects are not the aim of the FSI methodology presented within this thesis.

The phenomenon of propeller singing, ranging from a deep grunting sound trough to a high-pitched noise, may on a particular design be more or less unpredictable within the bounds of the present analysis capabilities, [11]. Singing is believed to be the cause of vortex shedding mechanism in the turbulent and separated part of the boundary layer on the blade surface, [11]. The sensitivity to small design changes and the complexity of the problem might be understood realizing that two propellers of the same design can be manufactured so that one propeller will sing whilst the other will not. The practical solution of the problem is to introduce a chamfer to the trailing edge of the blade, with the purpose to disrupt the boundary layer growth in the trailing edge region in order to prevent the vortex shedding mechanism. To better understand the singing phenomenon and to be able to avoid certain design regions more refined methods providing detailed information is needed. The present work may contribute in this area.

In the research and development of flexible propeller blades the occurrence of hydroelasticity is obvious. The use of flexible propellers, in terms of composite propellers, was according to Mouritz *et al*, [12], first used on Soviet fishing boats in the 1960s. The possible benefits of using composite propellers are weight reduction, smoother take-up of power, reduced noise, reduced blade induced vibration, better cavitation erosion resistance and better fatigue performance. Since the 1980s, performance tests of composite propellers have been conducted on a range of naval vessels, including landing crafts, minesweepers, torpedoes, small boats, and trimarans; much of the scientific information is however not available in the open literature, Mouritz *et al*, [12]. However, the composite flexible propeller problem has been investigated using a low-order potential-based three-dimensional Boundary Element Method (BEM) and a Finite Element Method (FEM), Young, [13]. In this study Young concluded that it is important to include the effect of fluid-structure interaction in the analysis of flexible composite propellers, the reason being that the blade deformation changes the local flow field and thus the fluid pressure distributions, cavitation patterns, and resulting propeller efficiency. The sum of all these unsteady effects consequently makes the problem complex and thus accurate methods are needed to enable the predictions of the correct propeller noise and the hull, shaft and blade vibrations.

Related to the flexible propeller blade problem are acoustic signatures from submerged hulls. These acoustic emissions are associated with e.g. low-frequency hull vibrations excited from the transmission of fluctuating force through the propeller-shafting system, cavitation and vibrations in the machinery or any other onboard equipment. In naval vessels, where not being detected may be of vital importance, the radiated noise may cause problems. The problem is often addressed e.g. by tuning the machinery design to produce a minimum of noise, designing propellers to reduce cavitation and by increasing the hydrodynamic efficiency to minimize perturbations in the water. In recent work by Caresta & Kessissoglou, [14], the structural and acoustic responses of a submarine hull under harmonic force excitation are studied using a combined BEM and FEM approach. However, to really understand and be able to analyze these problems the use of a model taking into account the unsteady viscous effects is required.

FSI related processes are not just found in shipping but everywhere around us in our daily life, for example sound vibrations induced by headphone membranes, the interaction between the atmosphere and the earth surface that governs the weather, the beautiful sound of a well tuned acoustic guitar and trees bending in the wind. Common for all these examples, apart from the FSI, is the occurrence of turbulence, a state of fluid motion that is characterized by apparently random and chaotic three-dimensional vorticity. The forces asserted on the object are to a large extent governed by the medium and larger scales of the flow. The development of the medium and larger scales is in addition dependent on the interaction with the smallest and dissipative scales of the flow. The most common way of mathematically modeling the movement of fluids is the Navier-Stokes Equations (NSE). The approach involves treating the fluid as a continuum and introducing macroscopic fluid properties, like viscosity and density, that have to be experimentally, theoretically or empirically estimated. There is presently for almost all cases no feasible way of fully solving the NSE and the development of new models and the validation of computational methods to approximate the equations play a central role in CFD, [15,16]. The major problem in solving the equations is the large range of scales that needs to be resolved, ranging from the Kolmogorov length scale, under the assumption that the smallest turbulent scales are universal, up to largest integral scale, representing the boundary conditions, [17].

The computational cost for a simulation covering the complete range of scales in the flow, also known as a Direct Numerical Simulation (DNS), [18], is therefore very high even for the simplest flows. It has, however, been confirmed that for certain types of flow and depending on the parameter of interest, simplifications of the NSE may be done. For the shipping industry, the most important simplified methods are potential flow methods, [19], Reynolds Averaged Navier-Stokes (RANS) approaches, [20], Large Eddy Simulations (LES), [21], and Detached Eddy Simulation (DES), [22]. The potential flow method, where the flow is assumed to be inviscid and irrotational, is often used when very fast results are required. The more sophisticated method, RANS, based on a separation of the flow into an averaged and a fluctuating part, is often used when more detailed information about the flow field is required. Despite the extensive use, the reliability in predicting flows influenced by massive separation, large-scale unsteadiness, or by physical flow processes beyond those covered by the incompressible NSE is uncertain with RANS, [23]. These flows are more likely to at least require the use of DES or LES, methods in which the large scales of the flow, responsible for the major part of the momentum transport, are resolved on the grid, whereas only the small (subgrid) scale momentum transport needs to be modeled using a subgrid model. The drawback with DES and LES is a high computational cost that in most cases requires access to a computer cluster and a generous amount of disk space. However, as recently stated by Breuer *et al*, [7], LES is not well established in the FSI context. This is especially true within the area of hydroelasticity, [10].

In this thesis work OpenFOAM, [23–35], is used as the CFD software. OpenFOAM is a free, open source CFD software package developed by OpenCFD Ltd at ESI Group and distributed by the OpenFOAM Foundation, [24]. The first development of OpenFOAM started in the late 1980s at Imperial College, London. The aim was to develop a more powerful and flexible general simulation platform than the standard at the time. This led to the choice of C++ as programming language, due to its highest modularity and object-oriented features. OpenFOAM was originally not freely available but was released as open source in 2004.

2. Thesis Objective and Scope

The main objective of this thesis work is to develop and validate an FSI methodology for hydroelastic applications, in order to facilitate studies of e.g. propeller singing, new propeller concepts, possibly involving composite materials, acoustic signatures from naval vessels and cavitation erosion. In other words the method needs to be able to capture transient, rapid and small-scale phenomena correlated with for instance a cavitation collapse.

In turn, to perform the hydroelastic and hydroacoustic studies aimed for, detailed information about the associated transient flow field and its interaction with the corresponding structure is required. To address these requirements Large Eddy Simulation is employed within this work. Moreover, the flow and structural solvers should rely on open source codes in order to facilitate changes in, and development of, the code and enable easy access to all classes and functions available within the different software.

The structural deformation of the, in FSI, considered structural object is highly dependent on the surface forces induced by the flow. In order to validate the prediction of these forces a non-deforming oscillating cylinder in a steady flow was chosen as a first FSI benchmark case. The computational case assumes no structural response and the surface forces are given only by the flow physics, a phenomenon referred to as one-way FSI. In addition, the LES needs to handle moving grids.

The surface forces, in addition, are highly dependent on the correct separation pattern, especially when considering moving objects. Thus, a thorough validation of the separation pattern predicted by LES is needed. Subsequently, an FSI methodology for hydroelastic applications is to be developed and validated against experimental data of a deforming hydrofoil.

The objectives can thus be summarized as follows:

- Increase understanding of three-dimensional open flow separation and validate the prediction of separation.
- Increase understanding of and validate LES on moving grids.
- Develop and validate a LES based FSI methodology for hydroelastic applications.

Studies assessing the first bullet are covered by appended Paper I & II, the second bullet by Paper III & IV and the last bullet by Paper V. Main results and experiences are briefly summarized in the following sections of the introduction, for details the reader is referred to the appended papers.

3. The CFD-CSD Coupling Approach

The FSI methodology employed in this thesis is based on a one-to-one mapping of the fluid and structural meshes connecting the two domains. This approach, applied together with an averaged CFD method, has previously been employed by Campbell *et al*, [36]. This approach has also been proven to be stable in a FEM context, [37].

In Table 1, the proposed general FSI prediction procedure is described. The first step involves the generation of the fluid and structural mesh using suitable software, e.g. Gmsh, [38,39]. The mesh generation could be done either by producing the meshes simultaneously, or by creating either one of them and then extracting the interface boundary, from which the second mesh could be created. In step two the meshes are converted into the OpenFOAM mesh format. The choice of first converting both meshes into OpenFOAM format is mainly due to the extensive number of conversion and domain decomposition routines available within the OpenFOAM toolbox. In step three the meshes are independently decomposed; this is the most flexible option. However, one might suggest decomposing the domains in such a way that the communication over the fluid-structure interface is minimized. Results, nevertheless, show that the communication time is insignificant compared with the solution of the structural and fluid equations. In step four parallel communication maps connecting the domains are set up. The reason that it is possible to do this prior to the actual simulation is that the interface meshes on both the fluid and solid side are forced to have the same topological description. One limitation, however, is that either the structural or fluid domain could be, depending on the requirements on accuracy, too well resolved, resulting in an unnecessarily long computational time. The actual transfer of data between the solvers is here implemented using the parallel vectors of the Portable, Extensible Toolkit for Scientific Computation (PETSc), [40,41]. In step five the user runs the simulation, generally on a computer cluster. It is in most cases reasonable to start from a somewhat developed flow field before the deformation is activated. This is mainly due to instability issues and is the procedure used to produce the predictions presented within this work. In step six the simulation result is post-processed using suitable software. OOFEM currently supports output in the open-source VTK XML format, [42], which is supported by post-processers MayaVi, [43], and ParaView, [44]. For the OpenFOAM data there are however several formats available, allowing for post-processing using both open source, [43,44], and closed source software e.g. FieldView, [45], and TecPlot, [46].

Table 1 General user procedure Fluid-Structure Interaction simulation

1. Generate fluid and structural mesh.
2. Convert meshes into OpenFOAM mesh format
3. Independently decompose the domains
4. Set up parallel communication maps and create OOFEM input files
5. Run the simulation
6. Post processing

In Figure 1, a schematic drawing of the implicit coupling is shown. The procedure could be briefly described as follows: First the fluid velocities and pressure are solved for. Then the pressure and viscous forces asserted on the solid are transferred to the structural solver to act as a boundary condition and the structural displacement is solved for. Subsequently the convergence criterion is evaluated. Assuming that the first sub-iteration is considered, the sub-iteration number is increased by one and the Aitken relaxation step is performed. The new fluid interface displacement has now been estimated and the fluid mesh is to be moved. Next up is solving the flow equations and after that the structural equations. When that is done the convergence criterion is once again evaluated. Assuming that the convergence criterion is fulfilled the iteration now goes to the next time-step and if the total time exceeds the end time the simulation is stopped.

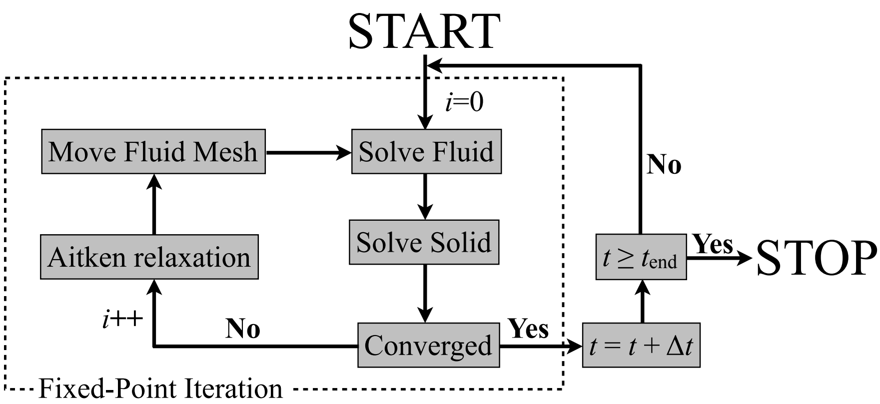

Figure 1 Overview of the implicit coupling algorithm used. Where i is the sub-iteration number, t the time, Δt the time-increment and t_{end} the end-time.

If the computational grid is too coarse to resolve the flow in the wall boundary layer, which is likely to be the case in most engineering flows, a model must be used to account for the presence of the wall. Such models are usually based on statistical arguments together with the mean velocity profiles of the viscous sub-layer and the logarithmic region, [18]. In this research the first cell adjacent to the wall is modified according to the law of the wall, this is described in more detailed in Paper II.

When a moving or deforming computational grid is used, the temporal derivatives introduce a rate of change of the cell volume and a mesh motion flux, due to the mesh convection. The relationship between the temporal derivative and the change in cell volume must satisfy the space conservation law, [47], in order to conserve mass. The change in cell volume is calculated from the sum of the mesh motion fluxes during the current time step rather than from the grid velocity making it consistent with the cell volume calculation. Here the mesh points on the wall are defined explicitly and the resulting mesh deformation is accounted for using a Laplace equation where a diffusion parameter controls the displacement of the internal grid points, [26].

4. Validation of Numerical Predictions

Validation of the predictions against experimental data is a vital part for the main part of the work in this thesis. It is however in many case not obvious how validation could be used and what to expect from it, especially in cases relating to transient phenomena. Due to these reasons, the different aspects of validation of numerical predictions will be discussed in terms of purpose, difficulties and usefulness.

The output from experimental measurements and numerical simulations are intended to be relevant characteristic data of a specific design, engineering problem or phenomenon. The intention of this collecting of data could, for example, be to determine the better out of two designs or to shed light on a previously unexplained phenomenon.

Moreover, the experimental data is something that in most validation is treated as the truth, in other words what the simulation is supposed to look like. One problem with this is that while the boundary conditions of the numerical prediction are well defined the boundary condition of the experimental setup is impossible or very hard to define and control. The reasons for the large confidence in experimental data are most likely related to tradition, and the fact that experiments per definition represent real world physics although the actual configuration includes modeling approximations, related to e.g. boundary locations and definitions

The insight about the difficulty in describing the boundary conditions of the experimental set up naturally leads to a discussion about the whole point of doing validation. The truth is that much understanding could be gained from validation, regardless of whether the numerical prediction and experimental data coincide. In addition, if the data differs a lot it indicates that either some important factor is overlooked in the experimental setup or that the mathematical model does not capture or lacks some relevant phenomenon. It is also very important to determine to what extent the validation could be performed and to ensure that the experimental and numerical results are post-processed in the same fashion, in order to minimize the risk of misinterpretations.

Moreover, if an already existing experiment is to be used for validation it is central to understand the purpose behind the experiment. The reason is that the purpose of the experiment to a large extent will determine the accuracy by which the involved parameters have been measured and the assumptions made. In addition, since far from all relevant information about an experiment could be included in a thesis, report or article contact with the corresponding experimentalists is of vital importance.

The extent of comparison could for example include the general shape of the transient drag and lift force curve or the position and occurrence of vortices and its corresponding separation pattern. It could also involve more specific measures such as the resistance of a ship hull. In some cases certain experimental measurements are not possible or widely restricted. In these cases numerical predictions could aid by providing more details about the process. When such an approach is used the validation against the measurable variables would serve as a support for the conclusions made from the predictions.

In the specific case of FSI the availability of high-quality experimental data that could be used for validating the corresponding models is limited. What makes it problematic is for example that the experimental setup most often includes some kind of elastic suspension of the structure and that the material of the structure may be hard to model. The material could for instance be rubber, which facilitates experiments with large deformations at relatively low flow speeds but nevertheless is hard to model.

As stressed in previous paragraphs, the comparison has to be done carefully taking into account the difficulty of exactly defining the problem in experiments and the numerical accuracy of the simulation. For instance, mounting a tripping device on the experimental model will in many cases clearly change the flow field. Tripping devices are in most cases very small and either hard to numerically resolve or to geometrically define. This makes the influence of the trip extremely challenging to model. In addition, if one actually manages to model the effect of a certain type of trip it is unlikely to work for other trip types.

In summary, validation should be performed with great care. Nevertheless, when done correctly validation could result in greater understanding of the corresponding physical phenomena and provide increased confidence in the computational models.

5. Summary

In this section the path taken towards a feasible and reliable LES based FSI methodology for hydroelastic applications will be presented. In the first subsection a summary of the research done on unsteady separation at curved smooth surfaces will be presented, then follows the research on transient forces on an oscillating cylinder and in the end a summary of the FSI solver developed within this research.

A. Large Eddy Simulation of Separated Flow (PAPER I & II)

Motivation

Flows around maneuvering ships, submarines and underwater vehicles are complicated, and often experience three-dimensional open flow separation resulting in the unsteady forces and moments that constitute one of the fundamental phenomena of hydroelastic and hydroacoustic predictions. In addition, separation prediction is to be considered as something fundamental in computational fluid dynamics. However, separation on smooth curved surfaces is difficult predict and there is more to be done in terms of developing models and increasing understanding.

In Paper I & II the main study is based on the inclined 6:1 prolate spheroid, investigated experimentally by Wetzel *et al*, [48,49]. What makes this case interesting is the exhibited open flow separation, the availability of high fidelity experimental data and the effect of the tripping device mounted at the nose of the model. For the LES computations near wall modeling is applied, different sub-grid models utilized and since the experimental study is carried out using a tripped model, one relatively simple and one more sophisticated trip model are developed and tested together with the LES models. A more detailed description of the different trip models is available in Paper I & II. The validation focuses on the mean velocity distribution in the region of separation, as well as on the skin friction and the surface streamlines.

Results

Figure 2 shows a perspective view of the wall modeled LES predictions, of the flow around the inclined prolate spheroid, for two different subgrid models, namely the Mixed Model (MM), [31], and the Localized Dynamic k-Equation Model (LDKM), [50]. In the absence of a trip model two longitudinal vortices roll up on the back of the prolate spheroid, one on each side of the symmetry plane. In the figures that follow, the x corresponds to the distance measured in the axial direction, starting at the nose, and L is the length of the model.

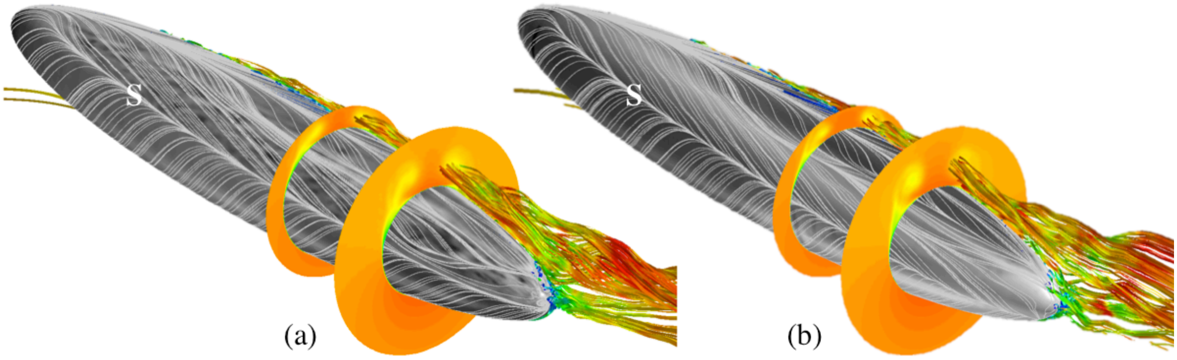

Figure 2 Perspective views of the flow past the 6:1 prolate spheroid in terms of surface streamlines (in white), streamlines released from the hull colored by the axial velocity, contours of the axial velocity at x/L=0.600 and x/L=0.772, respectively. Body-surface is colored by the time-averaged friction velocity, (a) LES-MM and (b) LES-LDKM.

In Figure 3 the perspective view of the wall modeled tripped flow around the prolate spheroid is shown. It could be observed that the separation pattern on the surface of the prolate spheroid has changed substantially due to the trip model. In Figure 2 one distinct separation line is observed, one on each side of the symmetry plane and here indicated by the letter S, while in Figure 3 two separation lines are observed.

Corresponding to the two separation lines are two longitudinal vortices, one large and the other small. The separation patterns in Figure 3 are observed to coincide well with experimental observation, [48].

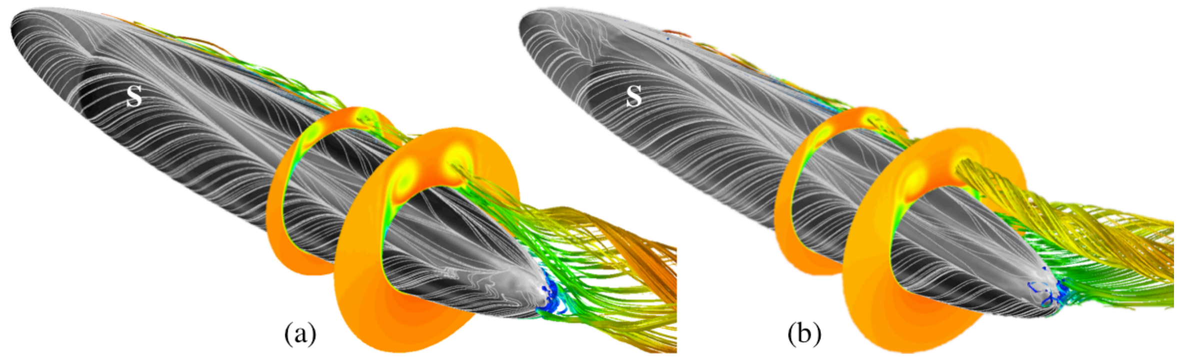

Figure 3 Perspective views of the flow past the 6:1 prolate spheroid in terms of surface streamlines (in white), streamlines released from the hull colored by the axial velocity, contours of the axial velocity at x/L=0.600 and x/L=0.772, respectively. Body-surface is colored by the time-averaged friction velocity, (a) LES-MM +trip model and (b) LES-LDKM+trip model.

The trip also introduces a more distinct low velocity region on the leeward-side of the spheroid, observed as the blue region in Figure 4 where the velocity, U, tangential to the body surface is shown at x/L=0.600 and x/L=0.772. It also becomes evident, when comparing experiments and LES that the presence of the trip is not something that could be neglected.

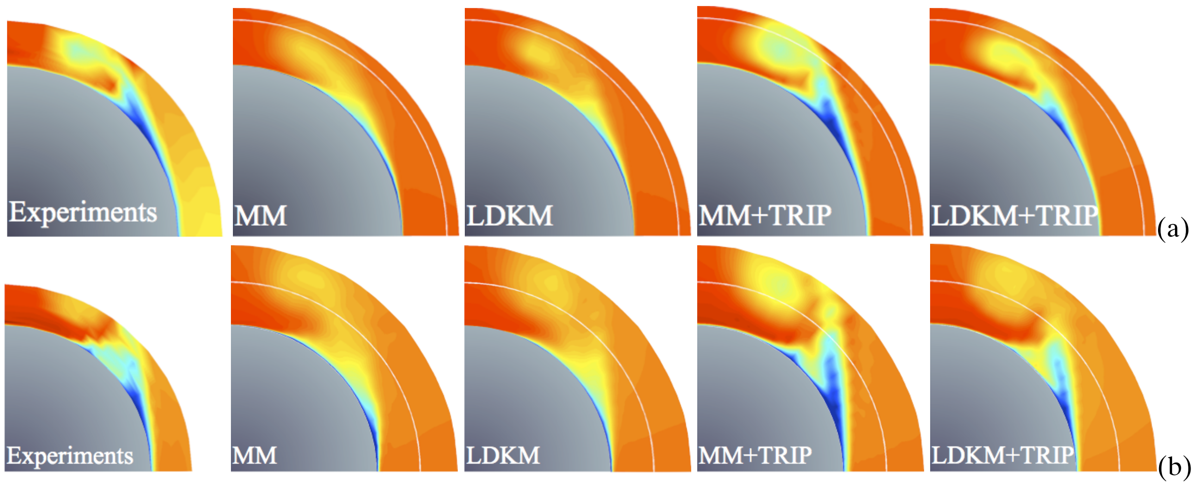

Figure 4 Cross-sectional contours of the velocity, U, tangent to the body surface in the axial direction and pointing towards the tail of the body at (a) x/L=0.600 and (b) x/L=0.772, respectively, for the experimental data and for all four LES models investigated.

B. Large Eddy Simulation of an Oscillating Cylinder (PAPER III & IV)

Motivation

As a first step towards studying hydroelasticity, the flow around a circular cylinder undergoing streamwise sinusoidal oscillations was studied in Paper III & IV using Large Eddy Simulation (LES). The LES predictions were validated against experimental data of Cetiner, [51]. It should be stressed that validation is performed in a transient sense, which most often is not the case. Due to the transient dependency the difficulty and complexity is of course higher compared with time-averaged comparisons, for which it is easier to determine specific quantities with corresponding error bounds.

The oscillating cylinder case includes access to high fidelity experimental data and was chosen as a validation case in order to assess the reliability of LES on moving geometries and corresponding deforming meshes. The flow exhibits many of the important flow phenomena, such as a von Kármán vortex street and Kelvin-Helmholtz instabilities and was investigated in terms of correlation of the time varying loading of the cylinder with the flow development. Moreover, the constantly changing relative velocity, because of the oscillation, results in a rich blend of vortices of different sizes. The sensitivity of the LES to subgrid model and grid resolution was investigated in terms of difference in lift and drag force predictions.

Results

It became quite obvious that the foundation on which the validation of the oscillating cylinder predictions stands clearly relies on the post-processing of the experimental data. This becomes evident in Figure 5 where two different Lissajous curves representing the same data set are presented.

The reason for the discrepancy is that, in the experiment, the frequency by which the cylinder oscillates rather than the position of the cylinder is specified. In other words, to create the Lissajous curves the displacement of the cylinder needs to be reconstructed using the oscillation frequency. How big the error will become then depends on the length of the signal and the accuracy by which the frequency is given.

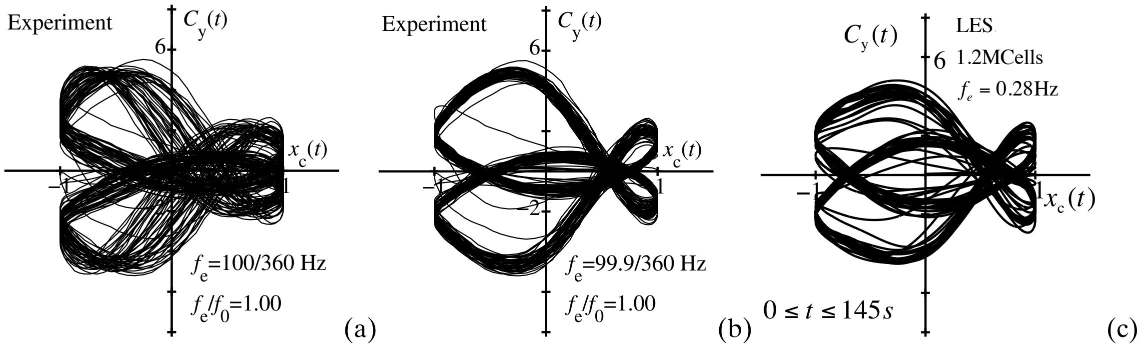

Figure 5 Example of Lissajous curves emphasizing the sensitivity of these graphs to the frequency by which the cylinder oscillates, f_e. (a) f_e=100/360 Hz and (b) f_e=99.9/360 Hz of 100 cylinder oscillations and in (c) LES prediction of 40 cylinder oscillations. Experimental data provided by Cetiner, [51]. Here f_0 is the von Kármán shedding frequency.

In Figure 6 the fascinating flow pattern corresponding to a single cylinder oscillation at the fundamental frequency is shown. Although, the sequence has been reduced to include only five snapshots, the evolution of the flow field and the corresponding drag and lift force curves could quite easily be followed. For a detailed description, the reader is referred to Paper IV.

For the predictions of the oscillating cylinder case a good agreement with experimental drag and lift forces was found, considering both the general shape of the Lissajous curves, used to describe the force variation, as well as the magnitude of these forces, implying that all grids and models used within this work were able to make sufficiently accurate predictions of the lift and drag force, then including the overall separation behavior.

In the comparison between ILES and explicit LES, it was observed that the subgrid model has an impact on how the fluid responds to the movement of the cylinder, possibly a result of different separation points. Some minor differences were found compared with the experimental results, mainly explained by differences in the computational and experimental setup. In addition, a small monotonic increase in force magnitude with increasing grid resolution and an even smaller dependency on the subgrid model was found.

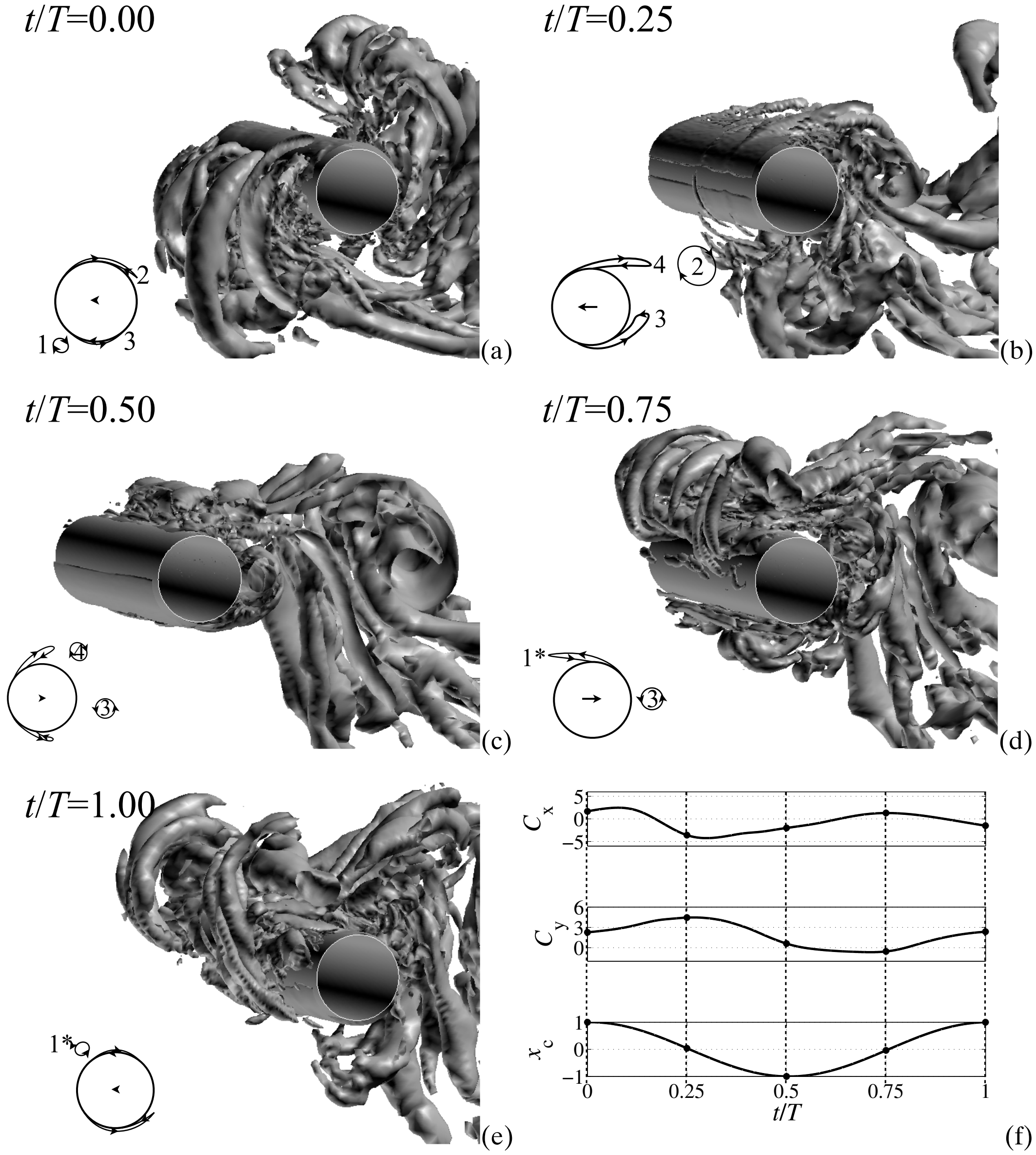

Figure 6 Iso-surface of the second invariant of the velocity gradient, λ_2, together with schematic pictures at five instants during one characteristic cylinder cycle, (a) to (e), and in (f) a time series of the normalized displacement of the cylinder, x_c, the drag and lift forces, C_x and C_y. In all panels is the oscillation period $T=1/f_e$.

C. A Large Eddy Simulation Based Fluid-Structure Interaction Methodology with Application in Hydroelasticity (PAPER V)

Motivation

One important difference between hydroelasticity and its aeronautic counterpart aeroelasticity is the difference in density ratio between the corresponding fluid and structure. The effect of more comparable densities, which is the case in hydroelasticity, is that a small disturbance in the displacement of the fluid-structure interface yields a large imbalance in the equilibrium at the interface. The phenomenon is referred to as the added mass effect.

FSI codes could be divided into two distinct groups, namely monolithic and partioned (segregated) solvers. In the monolithic approach the whole system of equations is solved for simultaneously. Moreover, in the partitioned approach the equilibrium is solved for in an iterative manner. In this research the choice fell on the partitioned approach, hence facilitating the use of already developed CFD and CSD solvers.

In addition, if a fluid and structure, with comparable densities, is coupled without taking the added mass effect into consideration the simulation most likely will diverge. In order to resolve this issue a conventional relaxation technique is included in the proposed methodology and is explained in detail in Paper V.

The experiences gained from solely LES could not be directly transferred to FSI in order to get a good measure of how computationally expensive an FSI simulation actually would be. To get detailed information about the capability and performance of the solver the case of a cantilever beam in a steady flow was studied, involving also a grid convergence study. In addition, many of the hydrodynamic phenomena require high resolution both in time and space, thus the need of a parallel solver is evident. Since parallelization does not, per default, guarantee good performance an additional scalability study was performed on the cantilever beam.

The software used in this research was chosen to be open source in order to avoid licensing costs and to have full control of the solvers on both the fluid and structural side. The choice of solvers fell on OpenFOAM, [24,25], for the CFD and OOFEM, [52,53], for the CSD.

Results

The results from the performance study exhibit overall good scaling properties as seen in Figure 7. Nevertheless, the mesh solver exhibits one notable limitation as when the number of cells per core decreases the communication increases and hence also the wall-time. In addition, the results from the grid study indicate the need for resolving both the structure and the fluid.

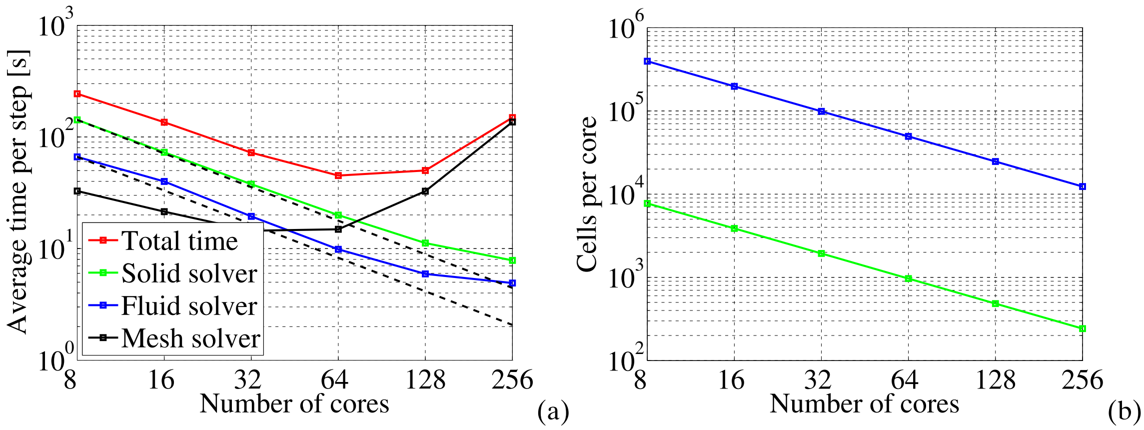

Figure 7 Scaling properties for a fixed mesh size of 5.2 Mcells (a) Average time per time-step versus number of cores. (b) Number of cells per core in fluid and structural domain.

The characteristic of the deforming cantilever beam studied in Paper V exhibits a von Kármán vortex street, seen in Figure 8b. The von Kármán vortex street is characterized by an alternating vortex separation with a corresponding alternating surface force. In the FSI case the alternation results in a locked-in vibration of the beam.

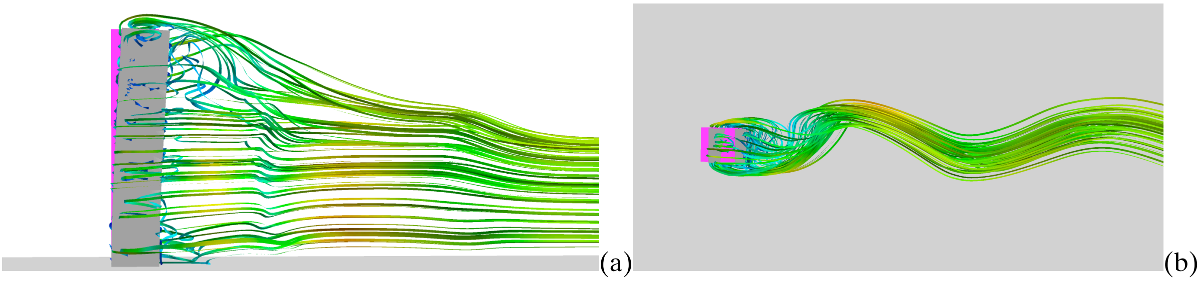

Figure 8 Instantaneous streamlines, colored by |v|, around the deformed beam. Reference position shown in pink.

In Figure 9 the pressure field on the deforming hydrofoil is shown. The hydrofoil was chosen due to its geometrical similarities with the profile of a propeller blade and because of easy access to experimental data and an initiated communication with the experimentalist behind the study. The average deformation in the y-direction is approximately 3-4% of the span length and agrees well with the experimental data, Figure 10. Thus supporting the validity of the proposed FSI methodology. The strengths of the presented FSI algorithm was shown to rely on the effective and accurate coupling between the fluid and solid solvers and the use of open source software.

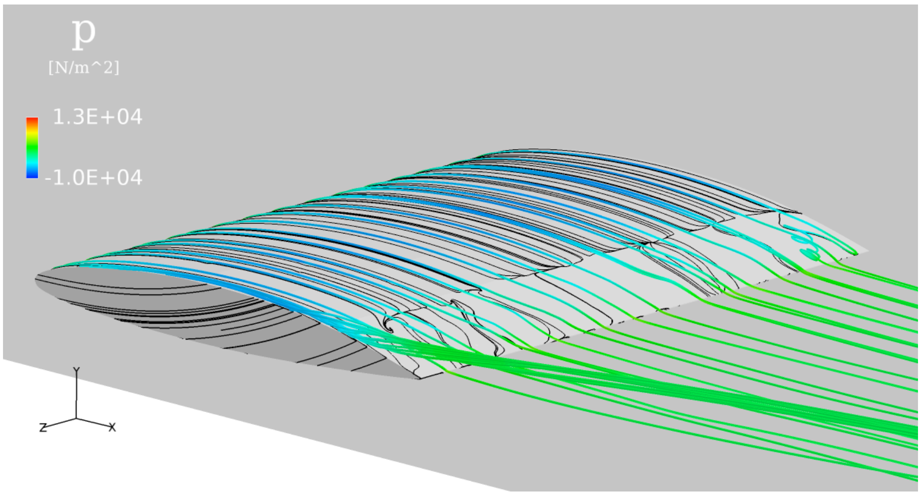

Figure 9 Velocity streamlines, colored by pressure, seeded on the surface of the deformable hydrofoil together with surface streamlines colored in black.

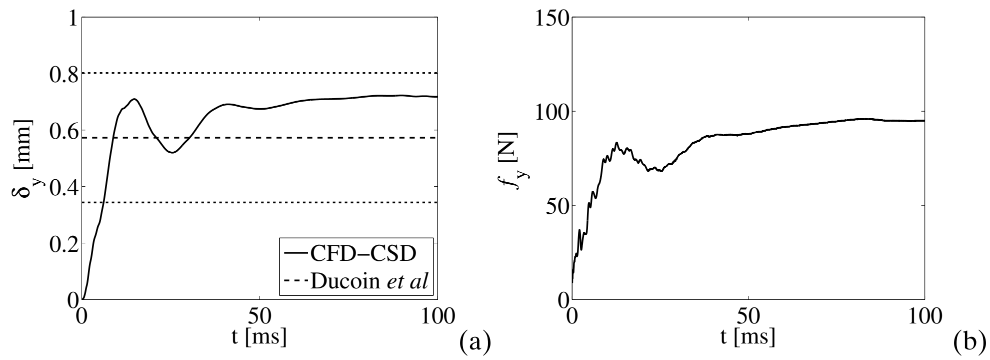

Figure 10 (a) Maximum displacement in y-direction at the tip of the hydrofoil. With FSI prediction shown as a solid line and experimental data of Ducoin, [54], dashed line, experimental uncertainty represented by dotted lines. (b) Force in the y-direction integrated over the wetted surface of the hydrofoil.

6. Conclusions

The principal advantages of the LES are its ability to capture the large-scale structures, separation pattern and the unsteady effects of the considered flow. This is apparent in all studies included in this thesis, that is in the validation based on the experimental study of the oscillating cylinder by Cetiner *et al*, [51,55,56], the 6:1 prolate spheroid by Wetzel *et al*, [57–59], and in the deforming hydrofoil by Ducoin *et al*, [54,60,61]. In addition, the predictive capability of LES for flow with three-dimensional open flow separation has been shown to be satisfactory in terms of capturing the correct separation pattern. In the choice of subgrid model there is no obvious choice. However, looking at all predictions included in this thesis LDKM and ILES+WM generally exhibits results that agree well with the experiments. The trip model developed has shown to improve the predictions both with respect to the near-wall flow and the separation pattern. Although this is true both for the prolate spheroid and the ellipsoid case, also included in Paper II, the model still needs to be tested on trips of different types of studs.

It has been found that in a validation process it is evidently important to understand how the different data has been produced and post-processed. In addition for the deforming hydrofoil, the oscillating cylinder, and 6:1 prolate spheroid case there has been an open dialog with the experimentalists behind the studies. This approach increased the confidence both in the predictions and the validations and a lot of misunderstandings were avoided due to a higher level of understanding of the differences between the experimental and mathematical set-up.

A LES based FSI methodology for hydroelastic applications has been developed and an initial validation against the experimental data of Ducoin, [54], has been performed. The results from the deforming hydrofoil validation show that the solver is capable of producing prediction of hydroelastic model tests, which is very promising and provides great confidence in the future predictive capabilities of the solver.

7. Outlook

The FSI solver development performed within the scope of this thesis is to be considered as basic research. Basic research in this context refers to the fact that the actual research is focused more on the solver methodology then the applications aimed for. It should be noted that before the solver had actually produced it first results the capability of the method was relatively unknown.

To actually see what needs to be done in terms of improving different parts of the methodology more hand on experience is needed. The reason for the uncertainty is the complexity of the problem that is the computational time is highly dependent on mesh resolution, time-step, geometrical properties and density ratio. Nevertheless, as mentioned in Section 5 the mesh solver seems to be one bottleneck that possibly could be improved. The first step towards an enhancement would be to investigate the capabilities of the different mesh motion solvers available within OpenFOAM.

There are numerous applications where the proposed FSI solver could provide interesting predictions and increase the understanding. One such application is in biometrics where for instance the dynamics of fish could be of interest. Possible aims could be to either come up with a completely new propulsion design inspired by the hydroelastic properties of the fish or to study the fish scales with the aim of reducing ship resistance. Moreover, there are boats actually already today replicating the efficient swimming action of a fish.

There is also the possibility to model tidal water turbines, where the blades are long and relatively flexible. It is in this case very likely that the bending of the blade will change the flow field properties and is of importance. The possible gain from increased understanding would then be increase efficiency.

Within a couple of years predictions including both cavitation and FSI should be possible. One purpose of performing these predictions would be to investigate the usage of flexible propellers in order to reduce cavitation, or remove it, on a submarine or to investigate the sensitivity of the cavitation development to vibration.

References

[1] Heller SR, Abramson HN. Hydroelasticity: A New Naval Science. Journal of the American Society for Naval Engineers 1959;71:205–9.

[2] Hübner B, Walhorn E, Dinkler D. A Monolithic Approach to Fluid–Structure Interaction Using Space–Time Finite Elements. Computer Methods in Applied Mechanics and Engineering 2004;193:2087–104.

[3] Michler C, Hulshoff SJ, Van Brummelen EH, De Borst R. A Monolithic Approach to Fluid–Structure Interaction. Computers & Fluids 2004;33:839–48.

[4] Barker AT, Cai X-C. Scalable Parallel Methods for Monolithic Coupling in Fluid–Structure Interaction with Application to Blood Flow Modeling. Journal of Computational Physics 2010;229:642–59.

[5] Matthies HG, Steindorf J. Partitioned Strong Coupling Algorithms for Fluid–Structure Interaction. Computers & Structures 2003;81:805–12.

[6] Lee J, Lee S. Fluid–Structure Interaction Analysis on a Flexible Plate Normal to a Free Stream at Low Reynolds Numbers. Journal of Fluids and Structures 2012;29:18–34.

[7] Breuer M, De Nayer G, Münsch M, Gallinger T, Wüchner R. Fluid–Structure Interaction Using a Partitioned Semi-Implicit Predictor–Corrector Coupling Scheme for the Application of Large-Eddy Simulation. Journal of Fluids and Structures 2012;29:107–30.

[8] Farhat C, Van der Zee KG, Geuzaine P. Provably Second-Order Time-Accurate Loosely-Coupled Solution Algorithms for Transient Nonlinear Computational Aeroelasticity. Computer Methods in Applied Mechanics and Engineering 2006;195:1973–2001.

[9] Storhaug G. Experimental Investigation of Wave Induced Vibrations and Their Effect on the Fatigue Loading of Ships. Trondheim: Norwegian University of Science and Technology, Faculty of Engineering Science and Technology, Department of Marine Technology; 2007.

[10] Chen X, Wu Y, Cui W, Jensen JJ. Review of Hydroelasticity Theories for Global Response of Marine Structures. Ocean Engineering 2006;33:439–57.

[11] Carlton J. Marine Propellers and Propulsion. 2nd ed. Amsterdam Oxford: Elsevier Butterworth-Heinemann; 2007.

[12] Mouritz A., Gellert E, Burchill P, Challis K. Review of Advanced Composite Structures for Naval Ships and Submarines. Composite Structures 2001;53:21–42.

[13] Young YL. Fluid–Structure Interaction Analysis of Flexible Composite Marine Propellers. Journal of Fluids and Structures 2008;24:799–818.

[14] Caresta M, Kessissoglou NJ. Acoustic Signature of a Submarine Hull under Harmonic Excitation. Applied Acoustics 2010;71:17–31.

[15] Zou L, Larsson L, Orych M. Verification and Validation of CFD Predictions for a Manoeuvring Tanker. Journal of Hydrodynamics, Ser. B 2010;22:438–45.

[16] Vuorinen V, Larmi M, Schlatter P, Fuchs L, Boersma BJ. A Low-Dissipative, Scale-Selective Discretization Scheme for the Navier–Stokes Equations. Computers & Fluids 2012;70:195–205.

[17] Orszag SA. Analytical Theories of Turbulence. Journal of Fluid Mechanics 2006;41:363.

[18] Pope S. Turbulent Flows. Cambridge New York: Cambridge University Press; 2000.

[19] Hess JL. Panel Methods in Computational Fluid Dynamics. Annual Review of Fluid Mechanics 1990;22:255–74.

[20] Wilcox D. Turbulence Modeling for CFD. La Cãnada, California: DCW Industries; 2006.

[21] Sagaut P. Large Eddy Simulation for Incompressible Flows: An Introduction. Third Edit. Berlin New York: Springer-Verlag; 2006.

[22] Spalart PR. Detached-Eddy Simulation. Annual Review of Fluid Mechanics 2009;41:181–202.

[23] Karlsson A, Fureby C. LES of the Flow Past a 6:1 Prolate Spheroid. 47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition, American Institute of Aeronautics and Astronautics; 2009.

[24] OpenFOAM® - The Open Source Computational Fluid Dynamics (CFD) Toolbox n.d.

[25] Weller HG, Tabor G, Jasak H, Fureby C. A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques. Computers in Physics 1998;12:620.

[26] Jasak H. Dynamic Mesh Handling in OpenFOAM. 47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition, American Institute of Aeronautics and Astronautics; 2009, pp. 1–10.

[27] Feymark A, Chapuis M, Fureby C, Liefvendahl M. Large Eddy Simulation of High Re Number Partially Separated Flow. 50th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, American Institute of Aeronautics and Astronautics; 2012.

[28] Feymark A, Alin N, Bensow R, Fureby C. Numerical Simulation of an Oscillating Cylinder Using Large Eddy Simulation and Implicit Large Eddy Simulation. Journal of Fluids Engineering 2012;134:031205.

[29] Feymark A, Alin N, Bensow RE, Fureby C. Large-Eddy Simulation of an Oscillating Cylinder in a Steady Flow. AIAA Journal 2013;51:1–11.

[30] Persson T, Liefvendahl M, Bensow RE, Fureby C. Numerical Investigation of the Flow Over an Axisymmetric Hill Using LES, DES, and RANS. Journal of Turbulence 2006;7:N4.

[31] Bensow RE, Fureby C. On the Justification and Extension of Mixed Models in LES. Journal of Turbulence 2007;8:N54.

[32] Feymark A, Alin N, Bensow R, Fureby C. LES of an Oscillating Cylinder in a Steady Flow. 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, American Institute of Aeronautics and Astronautics; 2010.

[33] Karlsson J, Feymark A, Eskilsson C. Implementation of Anisotropic Mesh Refinement in OpenFOAM. 15th Numerical Towing Tank Symposium, Cortona, Tuscany: 2012, pp. 117–22.

[34] Cesur A, Feymark A. Fluid-Structure Interaction of the Transient Evolution of a Cantilever Beam. Proceedings of the 25th Nordic Seminar on Computational Mechanics, Lund: 2012, pp. 49–52.

[35] Feymark A, Cesur A. Fluid-Structure Interaction Using Open Source Software. 14th Numerical Towing Tank Symposium, Poole, Dorset: 2011.

[36] Campbell RL, Paterson EG. Fluid–Structure Interaction Analysis of Flexible Turbomachinery. Journal of Fluids and Structures 2011;27:1376–91.

[37] Fick PW, Van Brummelen EH, Van der Zee KG. On the Adjoint-Consistent Formulation of Interface Conditions in Goal-Oriented Error Estimation and Adaptivity for Fluid–Structure Interaction. Computer Methods in Applied Mechanics and Engineering 2010;199:3369–85.

[38] Geuzaine C, Remacle J-F. Gmsh: A 3-D Finite Element Mesh Generator with Built-in Pre- and Post-Processing Facilities. International Journal for Numerical Methods in Engineering 2009;79:1309–31.

[39] Gmsh: A Three-Dimensional Finite Elment Mesh Generator with Built-In Pre- and Post-Processing Facilities n.d.

[40] PETSc: Home Page n.d.

[41] Balay S, Gropp WD, McInnes LC, Smith BF. Efficient Management of Parallelism in Object Oriented Numerical Software Libraries. In: Arge E, Bruaset AM, Langtangen HP, editors. Modern Software Tools in Scientific Computing, Birkhäuser Press; 1997, pp. 163–202.

[42] VTK - The Visualization Toolkit n.d.

[43] MayaVi - 3D Scientific Data Visualization and Plotting n.d.

[44] ParaView - Open Source Scientific Visualization n.d.

[45] FieldView 13 - Intelligent Light and FieldView n.d.

[46] Tecplot 360 - Visualization Software n.d.

[47] Demirdžić I, Perić M. Space Conservation Law in Finite Volume Calculations of Fluid Flow. International Journal for Numerical Methods in Fluids 1988;8:1037–50.

[48] Wetzel TG. Unsteady Flow Over a 6:1 Prolate Spheroid. Virginia Polytechnic Institute and State University, 1996.

[49] Goody M, Simpson R, Engel M. Mean Velocity and Pressure and Velocity Spectral Measurements Within a Separated Flow Around a Prolate Spheroid at Incidence. Archive Set 664, American Institute of Aeronautics and Astronautics; 1998.

[50] Menon S, Kim W-W. High Reynolds Number Flow Simulations Using the Localized Dynamic Subgrid-Scale Model. 34th AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada: American Institute of Aeronautics and Astronautics; 1996.

[51] Cetiner O. Flow Structure and Loading Due to an Oscillating Cylinder in Steady Current. Lehigh University; 1998.

[52] OOFEM - Free Object Oriented Finite Element Code n.d.

[53] Patzák B, Bittnar Z. Design of Object Oriented Finite Element Code. Advances in Engineering Software 2001;32:759–67.

[54] Ducoin A. Etude Experintale et Numerique du Chargement Hydrodynamique des Corps Portants en Regime Transitoire Avec Prise en Compte du Couplage Fluide Structure. École Centrale de Nantes, 2008.

[55] Cetiner O, Rockwell D. Streamwise Oscillations of a Cylinder in a Steady Current. Part 1. Locked-On States of Vortex Formation and Loading. Journal of Fluid Mechanics 2001;427:1–28.

[56] Cetiner O, Rockwell D. Streamwise Oscillations of a Cylinder in Steady Current. Part 2. Free-Surface Effects on Vortex Formation and Loading. Journal of Fluid Mechanics 2001;427:29–59.

[57] Chesnakas CJ, Simpson RL. Full Three-Dimensional Measurements of the Cross-Flow Separation Region of a 6:1 Prolate Spheroid. Experiments in Fluids 1994;17:68–74.

[58] Chesnakas CJ, Simpson RL. Measurements of the Turbulence Structure in the Vicinity of a 3-D Separation. Journal of Fluids Engineering 1996;118:268.

[59] Chesnakas CJ, Taylor D, Simpson RL. Detailed Investigation of the Three-Dimensional Separation About a 6:1 Prolate Spheroid. AIAA Journal 1997;35:990–9.

[60] Ducoin A, Astolfi JA, Deniset F, Sigrist J-F. An Experimental and Numerical Study of the Hydroelastic Behavior of an Hydrofoil in Transient Pitching Motion. First International Symposium on Marine Propulsors, Trondheim, Norway: 2009.

[61] Ducoin A, André Astolfi J, Gobert M-L. An Experimental Study of Boundary-Layer Transition Induced Vibrations on a Hydrofoil. Journal of Fluids and Structures 2012;32:37–51.