Mathematisches Forschungsinstitut Oberwolfach

Report No. 53/2007

Mini-Workshop: Theory and Numerics of Fluid-Solid Interaction

Organised by
Giovanni P. Galdi, Pittsburgh
Rolf Rannacher, Heidelberg
Stefan Turek, Dortmund

November 25th – December 1st, 2007

Abstract. This volume contains the abstracts of a series of talks given at a mini-workshop in Oberwolfach on the theory and numerics of continuum mechanical fluid-solid/structure interaction. The characteristics of these coupled multi-field problems are that the displacement of the solid/structure has a direct influence on the surrounding flow and vice versa. This interaction is generally nonlinear making the modeling complicated. The mathematical analysis concentrates on the well-posedness of the models in order to provide a rigorous explanation of fundamental experiments. Various competing numerical approaches are discussed based on different variational formulations and mainly using finite element methods.

Mathematics Subject Classification (2000): 35Q30, 65M60, 76A05, 76D05, 74B20, 74F10, 76M10.

Introduction by the Organisers

The theme of the mini-workshop was mathematical and computational aspects of modeling the interaction of viscous fluids with solids or elastic structures. The characteristics of this fluid-solid/structure interaction (FSI) setting is that the displacement of the solid/structure has a direct influence on the surrounding flow area. One dilemma in modeling the coupled dynamics of flow and solid/structure is that the fluid model is normally based on an Eulerian perspective in contrast to the usual Lagrangian approach for the solid model. This makes the setup of a common variational description difficult and complicates the mathematical analysis of the coupled problem as well as its numerical simulation. The whole area is rich of challenging theoretical, computational as well as application-related problems.

Over the last forty years, the interaction of fluid flow with rigid or elastic bodies or structures has represented one of the major focal points in several branches of engineering research. It is required for many industry applications such as biomedical, aerofoil flutter and civil engineering. A simple example is the motion of single particles in a viscous liquid, which has many practical applications. Another type of FSI problems occurs in the interaction of "exterior" fluid flow with elastic structures, e.g., air flow around an aircraft wing during flight, the cabin acoustics, or the vibration of a civil engineering structure under wind loading. Interesting biomedical applications occur in hemodynamics, e.g., blood flow in elastic vessels for modeling stent design. "Interior" FSI problems occur, for example, when liquids are transported in closed containers, in the mixing of polymers, and in vibration in pipeline systems.

Despite the numerous applications of processes involving FSI, the physical mechanism of even some of the most elementary phenomena is far from being understood. The orientation of symmetric particles in a uniform steady flow of a viscoelastic liquid is a typical example that presents several underlying unanswered questions. For instance, what is the physical characteristic of the liquid that determines the orientation of the particle, and why very simple-shaped symmetric particles, like cylinders, present a different orientation in the *same* liquid according to whether they have flat or round ends ("shape-tilting"). Further, subtile problems occur in the context of modeling the interaction of flow and *elastic* structures. Questions are, e.g., the modeling of the collision of rigid particles or the wall touching of elastic structures, boundary conditions for the truncation of large channel systems to smaller portions, and instabilities in the coupling mechanism related to the occurrence of aneurysm in blood vessel walls.

In the last decade another powerful tool has been employed to investigate FSI phenomena including particulate flow, namely, direct numerical simulation (DNS). The results, in some cases, are quite impressive, like those, for instance, simulating the three-dimensional motion of thousands of spheres in a fluidized bed or the 3D simulation of oscillatory blood flow through a piece of an elastic vessel. However, so far, these three different tools (experiments, mathematics, numerics) have been more or less independently used to investigate FSI problems.

The mini-workshop concentrated on the following key questions:

- The first set of questions was related to the flow of a single rigid particle in a viscous fluid, e.g., explanation of the shape-tilting phenomenon, stability of quasi-steady motion depending on particle shape and fluid characteristics, "effective" continuum fluid model for the real fluid and many free particles.
- The second set of questions was related to the interaction of a viscous fluid in a channel with an elastic wall, e.g., conditions for the well-posedness of the mathematical model, non-reflecting artificial boundary conditions, stability of flow through elastic pipe systems.
- The third set of questions was related to special FSI problems occurring in hemodynamics, e.g., the flow of many deformable bodies in a viscous

Newtonian or non-Newtonian liquid and their attachment at the vessel wall.

- The fourth set of questions was related to the efficient numerical simulation of processes involving FSI, e.g., strongly coupled versus partitioned solution approach, fixed-grid versus moving-grid methods, ALE versus Eulerian method, treatment of large deformation and collision, sensitivity-driven mesh adaptivity.

The mini-workshop provided the platform for intensive discussion of the problems described above and, by the combined effort of theoretical and numerical analysts as well as experimental physicists, stimulated new ways for attacking the many open questions.

Workshop: Mini-Workshop: Theory and Numerics of Fluid-Solid Interaction

Table of Contents

Abstracts

Fictitious boundary method and grid deformation for particulate flows

Dan Anca, Stefan Turek
(joint work with Wan Decheng)

Several numerical simulation techniques for particulate flows have been developed over the past decades. In these methods, the fluid flow is governed by the continuity and momentum equations, while the particles are governed by the equation of motion for a rigid body. The flow field around each individual particle is resolved such that the hydrodynamical forces between the particle and the fluid is obtained from the fluid solutions. Hu, Joseph and coworkers [1, 2], Galdi [3] and Maury [4] developed a finite element method based on unstructured grids to simulate the motion of large numbers of rigid particles in Newtonian and viscoelastic fluids. This approach is based on an Arbitrary Lagrangian-Eulerian (ALE) technique. Both the fluid and solid equations of motion are incorporated into a single coupled variational equation. The method developed by them excludes the explicit formulation of the torques and hydrodynamical forces. At each time step, a new mesh is generated when the old one becomes too distorted, and the flow field is projected onto the new mesh. In this scheme, the positions of the particles and grid nodes are updated explicitly, while the velocities of the fluid and the solid particles are determined implicitly.

Glowinski, Joseph, Patankar and coauthors [5, 6, 7, 8] proposed a distributed Lagrange multiplier (DLM)/fictitious domain method for the direct numerical simulation of large number of rigid particles in fluids. In the DLM method, the entire fluid-particle domain is assumed to be a fluid and then to constrain the particle domain to move with a rigid motion. The fluid-particle motion is treated implicitly using a combined weak formulation in which the mutual forces cancel. This formulation permits the use of a fixed structured grid thus eliminating the need for remeshing the domain. Our group [9, 10, 11, 12] presented another multigrid fictitious boundary method (FBM) for the detailed simulation of particulate flows. This method is based on a fixed unstructured FEM background grid. The motion of the solid particles is modelled by the Newton-Euler equations. Based on the boundary conditions applied at the interface between the particles and the fluid which can be seen as an additional constraint to the governing Navier-Stokes equations, the fluid domain can be extended into the whole domain which covers both fluid and particle domains.

For obtaining improvements in accuracy and efficiency, the adaptive mesh method seems to be a very powerfool tool. There are many existing adaptive mesh methods to achieve this purpose. Over the past decade, several adaptive techniques have been developed, namely the so-called h-, p- and r- methods. The first two do static regridding with fixed time, where the h-method does automatic refinement or coarsening of the spatial mesh based on a posteriori error estimates or error indicators and the p-method takes higher or lower order approximations

locally as needed. In contrast, the r-method (also known as moving mesh method) relocates grid points in a mesh having a fixed number of nodes in such a way that the nodes remain concentrated in regions of rapid variation of the solution or corresponding moving interfaces. The r-method is a dynamic method which means that it uses time stepping or pseudo-time stepping approaches to construct the desired transformation. The r-method or moving mesh method differs from the h-, and p-methods in that the former keeps the same number of mesh points throughout the entire solution process, while the later have to treat the tedious hanging node problems. Thus, the size of computation and data structure is fixed, which enables the r-method much easier to incorporate into most CFD codes without the need for the changing of system matrix structures and special interpolation procedures. The r-method has received more and more attention due to some new developments which clearly demonstrate its potential for problems such as those having moving interfaces [13, 14, 15, 16, 17].

The primary objective is to combine our multigrid fictitious boundary method (FBM) [9, 11] with the moving mesh method described in [17] for the simulation of the particulate flows to check the accuracy of the proposed combining method comparing its results with the previous pure multigrid fictitious boundary method (FBM). As we have shown in [11], the use of the multigrid FBM does not require to change the mesh during the simulations, although the rigid particles vary their positions. The advantage is that no expensive remeshing has to be performed while a fixed mesh can be used such that in combination with appropriate data structures and fast CFD solvers very high efficiency rates can be reached. However, the accuracy for capturing the surfaces of solid particles is only of first order which might lead to accuracy problems. For a better approximation of the particle surfaces, we adopt a deformed grid, created from an equidistant cartesian mesh, in which the topology is preserved, only the grid spacing is changed such that the grid points are concentrated near the surfaces of the rigid particles. Only the solution of additional linear Poisson problems in every time step is required for generating the deformation grid, which means that the additional work is significantly less than the main fluid-particle part.

In our numerical studies on particulate flow, the motion of the fluid is modelled by the well-known Navier-Stokes equations with the constraint of an incompressible fluid. For the particles, we use the model of Newton-Euler equations which treat both translation and rotational moves of the particles. As a method belonging to the DLM group of methods, in our FBM method the hydrodinamical forces and torques acting on the particles are computed. The trick we use in order not to use the surface integrals that define these forces, is to use a function that can decide if a point is inside or outside the particles. This function permits us to define the hydrodynamical forces and torques by a volume based integral representation and that is an important advantage since there is no need to reconstruct the shape of the interfaces of the particles. The integral over each element covering the whole domain can be exactly calculated with a standard Gaussian quadrature of sufficiently high order and the volume integrals need to be computed only in one

layer of mesh cells around the particle, which leads to a very efficient treatment. Also, besides these interactions between fluid and particles, there is a no slip condition imposed at the interfaces of the particles. In the numerical simulation of many particles in a fluid, normally interactions between particles or particles-wall occur. For treating this situation, we use a modified repulsive range collision model presented by Glowinski which allows particles to come close together, even overlapping and then separate from each other.

The algorithm of the multigrid FEM fictitious boundary method for solving the coupled system of fluid and particles can be summarized as follows:

(1) Given the positions and velocities of the particles, solve the fluid equations in the corresponding fluid domain involving the position of the particles for the fictitious boundary conditions.
(2) Calculate the corresponding hydrodynamic forces and the torque acting on the particles and compute the collision forces.
(3) Solve the kinematic equations to get the translational and angular velocities of the particles, and then obtain the new positions and velocities of the particles.
(4) Set the new fluid domain and fictitious boundary conditions, and then advance to solve for the new velocity and pressure of the fluid phase as described in step 1.
(5) Optionally, the adapting mesh method can be an additional step in this algorithm.

We present the combination of the multigrid fictitious boundary method and the moving mesh method for the simulations of particulate flow with many moving rigid particles. The new approach directly improves accuracy upon the previous pure multigrid FBM for particulate flows. It is also computationally cheap and simple to implement. Since the size of computation and data structure of the moving deformation meshes are fixed, this enables the proposed method much easier to be incorporated into most CFD codes without the need for changing of the matrix structures and for special interpolation procedures. It is suitable to accurately and efficiently perform the direct numerical simulation of particulate flows with large number of moving particles. Some numerical examples as a single moving particle in a fluid, or the drafting, kissing and tumbling of two disks in a channel were performed to see that the presented method can significantly improve the accuracy for dealing with the interaction between the fluid and the particles, and can be easily applied to real particulate flows with many moving particles.

References

[1] Hu, H.H., Joseph, D.D., Crochet, M.J.: Direct Simulation of Fluid Particle Motions. Theor. Comp. Fluid Dyn., **3**, 285 – 306 (1992)
[2] Hu, H.H., Patankar, N.A., Zhu, M.Y.: Direct Numerical Simulations of Fluid-Solid Systems Using the Arbitrary Lagrangian-Eulerian Techniques. J. Comput. Phys., **169**, 427 – 462 (2001)

[3] Galdi, G¿P., Heuveline, V.: Lift and Sedimentation of particles on the Flow of a Viscoelastic Liquid in a Channel. CRC Press, Free and Moving Boundaries: Analysis, Simulation and Control (2007)
[4] Maury, B.: Direct Simulations of 2D Fluid-Particle Flows in Biperiodic Domains. J. Comput. Phy., **156**, 325 – 351 (1999)
[5] Glowinski, R., Pan, T.W., Hesla, T.I., Joseph, D.D.: A Distributed Lagrange Multiplier/Fictitious Domain Method for Particulate Flows. Int. J. Multiphase Flow, **25**, 755 – 794 (1999)
[6] Patankar, N.A., Singh, P., Joseph, D.D., Glowinski, R., Pan, T.W.: A New Formulation of the Distributed Lagrange Multiplier/Fictitious Domain Method for Particulate Flows. Int. J. Multiphase Flow, **26**, 1509 – 1524 (2000)
[7] Glowinski, R., Pan, T.W., Hesla, T.I., Joseph, D.D., Periaux, J.: A Fictitious Domain Approach to the Direct Numerical Simulation of Incompressible Viscous Flow Past Moving Rigid Bodies: Application to Particulate Flow. J. Comput. Phy., **169**, 363 – 426 (2001)
[8] Glowinski, R.: Finite Element Methods for Incompressible Viscous Flow. In *Handbook of numerical analysis*, **Vol. IX**, Ciarlet, P.G and Lions, J.L., Editors, North-Holland, Amsterdam, 701 – 769 (2003)
[9] Turek, S., Wan, D.C., Rivkind, L.S.: The Fictitious Boundary Method for the Implicit Treatment of Dirichlet Boundary Conditions with Applications to Incompressible Flow Simulations. Challenges in Scientific Computing, Lecture Notes in Computational Science and Engineering, Vol. **35**, Springer, 37 – 68 (2003)
[10] Wan, D.C., Turek, S., Rivkind, L.S.: An Efficient Multigrid FEM Solution Technique for Incompressible Flow with Moving Rigid Bodies. Numerical Mathematics and Advanced Applications, ENUMATH 2003, Springer, 844 – 853 (2004)
[11] Wan, D.C., Turek, S.: Direct Numerical Simulation of Particulate Flow via Multigrid FEM Techniques and the Fictitious Boundary Method. Int. J. Numer. Method in Fluids, Vol. **5**, Nr. 51, 531-566 (2006)
[12] Wan, D.C., Turek, S.: An Efficient Multigrid-FEM Method for the Simulation of Liquid-Solid Two Phase Flows. J. for Comput. and Appl. Math., Vol. **203**, Nr. 2, 561-580 (2007)
[13] Bochev, P.B., Liao, G., de la Pena, G.C.: Analysis and Computation of Adaptive Moving Grids by Deformation. Numerical Methods for Partial Differential Equations, **12**, 489 (1996)
[14] Liao, G., Semper, B.: A Moving Grid Finite–Element Method Using Grid Deformation. Numerical Methods for Partial Differential Equations, **11**, 603 – 615 (1995)
[15] Liu, F., Ji, S., Liao, G.: An Adaptive Grid Method and its Application to Steady Euler Flow Calculations. SIAM Journal on Scientific Computing, **20**(3), 811 – 825 (1998)
[16] Cai, X.X., Fleitas, D., Jiang, B., Liao, G.: Adaptive Grid Generation Based on Least–Squares Finite–Element Method. Computers and Mathematics with Applications, **48**(7-8), 1077 – 1086 (2004)
[17] Grajewski, M., Köster, M., Kilian, S., Turek, S.: Numerical Analysis and Practical Aspects of a Robust and Efficient Grid Deformation Method in the Finite Element Context. submitted to SISC (2006)
[18] Dacorogna, B., Moser, J.: On a Partial Differential Equation Involving the Jacobian Determinant. Annales de le Institut Henri Poincare, **7**, 1 – 26 (1990)
[19] Turek, S.: Efficient Solvers for Incompressible Flow Problems. Springer Verlag, Berlin-Heidelberg-New York (1999)
[20] Turek, S.: **FEATFLOW**–Finite element software for the incompressible Navier–Stokes equations: User Manual, Release 1.2, University of Dortmund (1999)
[21] Turek, S.: A comparative study of time stepping techniques for the incompressible Navier–Stokes equations: From fully implicit nonlinear schemes to semi–implicit projection methods, Int. J. Numer. Meth. Fluids, 22, 987 – 1011 (1996)
[22] Turek, S.: On discrete projection methods for the incompressible Navier–Stokes equations: An algorithmical approach, Comput. Methods Appl. Mech. Engrg., 143, 271 – 288 (1997)

[23] Schäfer, M., Turek, S.: Benchmark computations of laminar flow around cylinder. E.H. Hirschel (editor) Flow Simulation with High-Performance Computers II. Volume 52 of Notes on Numerical Fluid Mechanics, Vieweg, 547 – 566 (1996)
[24] Singh, P., Hesla, T.I., Joseph, D.D.: Distributed Lagrange multiplier method for particulate flows with collisions. Int. J. Multiphase Flow, **29**, 495 – 509 (2003)
[25] Diaz-Goano, C., Minev, P., Nandakumar, K.: A Lagrange multiplier/fictitious domain approach to particulate flows, in: Margenov, W., Yalamov (Eds), Lecture Notes in Computer Science, Vol. 2179, Springer, 409 – 422 (2001)

Experimental observations of sedimentation in simple and complex fluids

Andrew Belmonte

(joint work with Torben Grumstrup, Anandhan Jayaraman, Joseph B. Keller, Michael Sostarecz)

To say that mathematics has had a long and fruitful relationship with experimentation in the physical world would be a vast understatement - the two are fundamentally related in their origins and development. Yet we are often compartimentalized away from the laboratories where observations are made, where mathematical ideas may be both tested and born. I present an overview of three different experimental studies related to the topic of Fluid-Structure Interactions, all involving freely falling objects. These experiments were performed in the Pritchard Laboratories in the Department of Mathematics at the Pennsylvania State University, one of a growing number of mathematics departments in the United States which have experimental laboratories.

Oscillating sedimentation in micellar fluids. The first observation is the nontransient oscillations of a falling sphere at low Reynolds numbers in a peculiar non-Newtonian fluid - a wormlike micellar fluid [1, 2]. This fluid is an aqueous solution of long tubular self-assembling aggregates of surfactants (wormlike micelles), which like polymers lead to a viscoelasticity in the fluid. However only transient oscillations have been observed in polymeric fluids [3]. The mathematical understanding of this oscillating sedimentation, in terms of any appropriate constitutive equation for these fluids, is an open problem. In fact it is only recently that constitutive models have explicitly taken into account the fundamental difference between wormlike micellar and polymeric fluids. The observation of a plateau in the stress-shear rate curve (steady rheology) [4, 5] initially led to the consideration of the Johnson-Segalman equation [6] or its variants as an appropriate model, since it also leads to an apparent plateau in shear stress [7, 8]. The first constitutive modeling of wormlike micelles was due to Cates and coworkers, treating the relaxation of stress by the standard reptation with the addition of breaking events (see e.g. [9]). More recent constitutive models include a modified associative network model, the BSPPM model [10], a modified FENE (finite extensibility nonlinear elastic) model [11, 12], and several multi-species models for the different populations of broken (smaller) or unbroken (larger) micelles [13, 14]. The appropriateness of these constitutive models are presently being tested in comparison

with the rheology of micellar fluids; whether they can reproduce hydrodynamic instabilities such as the oscillating sedimentation discussed here is not yet known, and may involve serious challenges for numerical simulation.

Stable sedimenting toroidal drops. The second observation involves the sedimentation of an aqueous polymeric fluid drop which forms a stable torus in an immiscible Newtonian oil [15]. While other examples of toroidal drops (and bubbles) were known previously, these were restricted to Newtonian fluids, and in each case the torus was unstable. The observations closest to our system involve two miscible fluids [16, 17] or fluids with a very low surface tension [18]. In the latter case, the torus was observed to expand rapidly due to viscous dissipation effects until it broke into smaller drops. It is not yet understood how the elasticity of the torus we observe counteracts the viscous dissipation in the vortex ring. However, there may be a parallel with a soliton-like toroidal vortex which has been observed numerically in the Ginzburg-Landau equation with a cubic-quintic nonlinearity [19, 20]. The soliton is a classic example of nonlinearity balancing dissipation, resulting in the existence of an undamped structure (the soliton) in a dissipative system. We postulate here that these viscoelastic toroidal drops exist for the same reasons as the 'stable vortex solitons' [20].

Acoustic ripples on long entrained bubbles. Finally I discuss our recent observation of acoustic ripples on long entrained air bubbles behind rapidly falling spheres in water (also ethanol) [21]. The bubble is produced during the impact of a solid sphere onto the free surface, as a result of the pinch-off of the long cavity formed by the sphere [22]. We have found that the acoustic emission of this pinch-off excites coherent ripples on the bubble surface, which remain fixed in the lab frame [21]. Based on this we have formulated a potential flow model for a slightly compressible fluid, which explains these ripples as the spatial rectification of the acoustic bubble oscillations by the moving object [21].

The earliest mathematical model for the dynamics of a bubble in a fluid was written by Rayleigh [23], coupling an irrotational flow to a spherical bubble with a changing radius $R(t)$ and internal pressure $p(R)$:

$$\rho(R\ddot{R} + \frac{3}{2}\dot{R}^2) = p(R) - p_\infty(t)$$

where ρ is the density of the fluid and p_∞ is the far-field pressure. Extending this approach to our approximately cylindrical bubble leads to the well-known log term for 2D harmonic functions. To avoid this difficulty we treat the fluid velocity outside of the cylindrical bubble as irrotational but not completely incompressible, so that the velocity potential satisfies a wave equation outside of the bubble [24, 25] instead of being harmonic. This was solved by Epstein & Keller for the case of purely temporal (spatially homogeneous) oscillations, for which they obtained an analytic formula for damped oscillations, which we will denote as $a_0(t)$ [26].

By imposing a boundary condition that these oscillations vanish at the position of the sedimenting sphere $z(t) = z_0 - U_0 t$, where U_0 is the speed of the sphere, and by further postulating that the local displacement of the surface due to the

rippling η is written

$$\eta(z,t) = a_0(t) + \eta_R(z),$$

where η_R is a reflected standing wave due to the boundary condition, we obtain a formula for the ripples directly from Epstein & Keller's solution, which reproduces many of our observations [21]. Future research in this area will center on the pattern-forming properties of this system, including aspects such as the threshold for the ripples, which disappear for slower or shorter bubbles, and nonlinear spectral issues such as the interactions of multiple high frequency modes.

Open mathematical problems. At the end of this talk four mathematical questions were posed, related to sedimentation and the experimental observations we have made:

• *S1 (oscillating sphere)*: Is there a Hopf bifurcation to steady oscillations of a sphere or other object sedimenting in a non-Newtonian fluid? This should be constitutive equation dependent, and not occur for polymeric models.

• *S2 (toroidal drop)*: Can the addition of elastic or other effects into the constitutive equation stabilize a sinking vortex ring?

• *S3 (entrained bubble)*: What is the drag on a sphere with an attached bubble? Existence of solution with mixed boundary conditions?

• *S4 (ripples)*: Surface waves with nonlocal (oscillating pressure) coupling on free boundary of irrotational fluid?

References

[1] A. Jayaraman & A. Belmonte, *Oscillations of a solid sphere falling through a wormlike micellar fluid*, Phys. Rev. E **67** (2003), 65301–4.

[2] S. Chen & J. P. Rothstein, *Flow of a wormlike micelle solution past a falling sphere*, J. Non-Newtonian Fluid Mech. **116** (2004), 205–234.

[3] G. H. McKinley, in *Transport Processes in Bubbles, Drops, and Particles*, edited by R. P. Chhabra & D. De Kee, (Taylor & Francis, London, 2001), 2nd ed.

[4] G. Porte, J.-F. Berret, & J. L. Harden, *Inhomogeneous Flows of Complex Fluids: Mechanical Instability Versus Non-Equilibrium Phase Transition*, J. Physique II **7** (1997), 459.

[5] E. Cappelaere, J.-F. Berret, J. Decruppe, P. Cressely, & R. Lindner, *Rheology, birefringence, and small-angle neutron scattering in a charged micellar system: Evidence of a shear-induced phase transition*, Phys. Rev. E **56** (1997), 1869–1878.

[6] M. Johnson, D. Segalman, *A model for viscoelastic fluid behaviour which allows non-affine deformation*, J. Non-Newtonian Fluid Mech. **2** (1977), 255–270.

[7] D. S. Malkus, J. S. Nohel, & B. J. Plohr, *Analysis of new phenomena in shear flow of non-Newtonian fluids*, SIAM J. Appl. Math. **51** (1991), 899–929.

[8] Y. Y. Renardy, *Spurt and instability in a two-layer Johnson-Segalman liquid*, Theoret. Comput. Fluid Dynamics **7** (1995), 463–475.

[9] R. Granek & M. E. Cates, *Stress relaxation in living polymers: results from a Poisson renewal model*, J. Chem. Phys. **96** (1992), 4758.

[10] Bautista F, Soltero JFA, Pérez-López JH, Puig JE, Manero O, *On the shear banding flow of elongated micellar solutions*, J. Non-Newtonian Fluid Mech. **94** (2000), 57–66.

[11] M. Kröger & R. Makhloufi, *Wormlike micelles under shear flow: a microscopic model studied by nonequilibrium-molecular-dynamics computer simulation*, Phys. Rev. E **53** (1996), 2531–2536.

[12] M. Kröger, *Simple models for complex nonequilibrium fluids*, Phys. Reports **390** (2004), 453–551.
[13] J. Goveas & P. Olmsted, *A minimal model for vorticity and gradient banding in complex fluids*, Eur. Phys. J. E **6** (2001), 79–89.
[14] P. Vasquez, L. P. Cook, & G. McKinley, *A network scission model for wormlike micellar solutions I: Model formulation and homogeneous flow predictions*, J. Non-Newtonian Fluid Mech. **144** (2007), 122–139.
[15] M. C. Sostarecz & A. Belmonte, *Motion and shape of a viscoelastic drop falling through a viscous fluid*, J. Fluid Mechanics **497** (2003), 235–252.
[16] M. Kojima, E. J. Hinch, & A. Acrivos, *The formation and expansion of a toroidal drop moving in a viscous fluid*, Phys. Fluids **27** (1984), 19–32.
[17] G. Machu, W. Meile, L. C. Nitsche, & U. Schaflinger, *Coalescence, torus formation, and breakup of sedimenting drops: experiments and computer simulations*, J. Fluid Mech. **447** (2001), 299–336.
[18] N. Baumann, D. D. Joseph, P. Mohr, & Y. Renardy, *Vortex rings of one fluid in another in free fall*, Phys. Fluids A **4** (1992), 567–580.
[19] D. Mihalache, D. Mazilu, F. Lederer, Y. Kartashov, L.-C. Crasovan, L. Torner, & B. A. Malomed, *Stable Vortex Tori in the Three-Dimensional Cubic-Quintic Ginzburg-Landau Equation*, Phys. Rev. Lett. **97** (2006), 073904.
[20] D. Mihalache, D. Mazilu, F. Lederer, H. Leblond, & B. A. Malomed, *Stability limits for three-dimensional vortex solitons in the Ginzburg-Landau equation with the cubic-quintic nonlinearity*, Phys. Rev. A **76** (2007), 045803.
[21] T. Grumstrup, J. B. Keller, & A. Belmonte, *Cavity ripples observed during the impact of solid objects into liquids*, Phys. Rev. Lett. **99** (2007), 114502.
[22] G. Birkhoff & E. H. Zarantonello, *Jets, Wakes, and Cavities* (Academic, 1957).
[23] Lord Rayleigh, *On the pressure developed in a liquid during the collapse of a spherical cavity*, Philos. Mag. **34** (1917), 94–98.
[24] Lord Rayleigh, *Theory of Sound* (MacMillian, 1896).
[25] J. B. Keller & I. I. Kolodner, *Damping of Underwater Explosion Bubble Oscillations*, J. Appl. Phys. **27** (1956), 1152.
[26] D. Epstein & J. B. Keller, *Expansion and Contraction of Planar, Cylindrical, and Spherical Underwater Gas Bubbles*, J. Acoustical Soc. Am. **52** (1972), 975–980.

Eulerian vs. Lagrangian methods for fluid-structure interaction

Thomas Dunne

Computational fluid dynamics and computational structure mechanics are two major areas of numerical simulation of physical systems. With the introduction of high performance computing it has become possible to tackle systems with a coupling of fluid and structure dynamics. General examples of such fluid-structure interaction (FSI) problems are flow transporting elastic particles (particulate flow), flow around elastic structures (airplanes, submarines) and flow in elastic structures (haemodynamics, transport of fluids in closed containers). In all these settings the dilemma in modeling the coupled dynamics is that the fluid model is normally based on an Eulerian perspective in contrast to the usual Lagrangian approach for the solid model. This makes the setup of a common variational description difficult. However, such a variational formulation of FSI is needed as the basis of a consistent approach to residual-based a posteriori error estimation and mesh

adaptation as well as to the solution of optimal control problems by the Euler-Lagrange method.

Combining the Eulerian and the Lagrangian setting for describing FSI involves conceptional difficulties. On the one hand the fluid domain itself is time-dependent and depends on the deformation of the structure domain. On the other hand, for the structure the fluid boundary values (velocity and the normal stress) are needed. In both cases values from the one problem are used for the other, which is costly and can lead to a drastic loss of accuracy. A common approach to dealing with this problem is to separate the two models, solve each one after the other, and so converge iteratively to a solution, which satisfies both together with the interface conditions Solving the separated problems serially multiple times is referred to as a 'partitioned approach'. For advanced examples of this approach see [4].

A basic partitioned approach does not contain a variational equation for the fluid-structure interface. To achieve this, usually an auxiliary unknown coordinate transformation function ζ_f is introduced for the fluid domain. With its help the fluid problem is rewritten as one on the transformed domain, which is fixed in time. Then, all computations are done on the fixed reference domain and as part of the computation the auxiliary transformation function ζ_f has to be determined at each time step.

Such, so-called 'arbitrary Lagrangian-Eulerian' (ALE) methods are also treated in the authors doctoral dissertation [7]. Multiple good examples and quantitative results can be found in [4], e.g. [11, 16].

Both, the partitioned and the transformation approach overcome the Euler-Lagrange discrepancy by explicitly tracking the fluid-structure interface. This is done by mesh adjustment or aligning the mesh to match the interface and is generally referred to as 'interface tracking'. Both methods leave the structure problem in its natural Lagrangian setting.

We follow the alternative (and to our knowledge new) way of posing the fluid as well as the structure problem in a *fully Eulerian framework*. In the Eulerian setting a phase variable is employed on the fixed mesh to distinguish between the different phases, liquid and solid. This approach to identifying the fluid-structure interface is generally referred to as 'interface capturing', a method commonly used in the simulation of multiphase flows, [12, 13]. Examples for the use of such a phase variable are the Volume of Fluid (VoF) method [10] and the Level Set (LS) method [14, 15]. In the classical LS approach the distance function has to continually be reinitialized, due to the smearing effect by the convection velocity in the fluid domain. This makes the use of the LS method delicate for modeling FSI problems particularly in the presence of cornered structures. To cope with this difficulty, we introduce a variant of the LS method that makes reinitialization unnecessary and which can easily cope with cornered structures.

The method we describe does not depend on the specific structure model. The key variable in structure dynamics is the deformation, and since this depends on the deflection, it is understandable why structure dynamics is preferably described in the Lagrangian frame. To be able to describe the deformations in the Eulerian

frame, we introduce the 'Initial Positions set' (IP set) of all structure points. This set is then transported with the structure velocity in each time step. Based on the IP set points and their Eulerian coordinates the displacement is now available in an Eulerian sense. Also its gradient has to be rewritten appropriately. Since the fluid-structure interface will be crossing through cells, we will have to also transport the IP set in the fluid domain.

If we were to use the fluid velocity for the convection of the IP set, this would lead to entanglement of the respective displacements, which would 'wreak havoc' on the interface cells. This is a known problem with LS approaches. A common way for fixing this problem has been to occasionally fix the LS field between the time steps. The problem with this approach is that the variational formulation is no longer consistent. As an alternative, we harmonically continue the structure velocity into the fluid domain. In the fluid domain we then use this velocity for the convection of the IP set. Since an IP set is available in both domains, we can always at each point determine if it belongs to the fluid or solid part of the model.

Again, this approach is similar to the LS approach. But when developing a complete variational formulation the two key characteristics of the LS approach also become the main cause of concern: reinitialization and the signed distance function. Although the problem of reinitialization here can also be avoided by using an harmonically extended velocity, the trouble concerning corner approximation persists. In contrast to this, by using an initial position set, we are deforming a virtual mesh of the structure, which is extended into the whole domain.

Based on the Eulerian variational formulation of the FSI system, we use the 'dual weighted residual' (DWR) method, described in [2, 3], to derive 'goal-oriented' a posteriori error estimates. The evaluation of these error estimates requires the approximate solution of a linear dual variational problem. The resulting a posteriori error indicators are then used for automatic local mesh adaption. The full application of the DWR method to FSI problems requires a Galerkin discretization in space as well as in time. Due to the use of a difference scheme in time, we are limited to 'goal-oriented' mesh adaptation in computing steady states or (somewhat heuristically) to quasi-steady states within the time stepping process.

The method for computing FSI described is validated at a stationary model problem that is a lid-driven cavity involving the interaction of an incompressible Stokes fluid with an incompressible neo-Hookean solid. Then, as a more challenging test the self-induced oscillation of a thin elastic bar immersed in an incompressible fluid is treated (FLUSTRUK-A benchmark described in [16] and [11]). For this test problem, our fully Eulerian method is also compared against a standard 'arbitrary Lagrangian-Eulerian' (ALE) approach. The possible potential of the fully Eulerian formulation of the FSI problem is indicated by its good behavior for large structure deformations.

Further details and results concerning Eulerian and ALE-Formulations of FSI problems as well as adaptivity for FSI problems can be found in the authors doctoral dissertation [7] as well as in [8, 4]. All computations and visualizations

were done using the flow-solver package GASCOIGNE [9] and the graphics package VISUSIMPLE [1, 17].

References

[1] R. Becker, Th. Dunne, *VisuSimple: An interactive open source visualization utility for scientific computing*, in "Reactive Flows, Diffusion and Transport: From Experiments via Mathematical Modeling to Numerical Simulation and Optimization, Final Report of SFB (Collaborative Research Center) 359" Jäger, Willi; Rannacher, Rolf; Warnatz, Jürgen (Eds.) Springer; (October 31, 2006) 676 p., ISBN-10: 3-540-28379-X ISBN-13: 978-3-540-28379-9 . See [17].

[2] R. Becker and R. Rannacher, Weighted a-posteriori error estimates in FE methods, Lecture ENUMATH-95, Paris, Sept. 18-22, 1995, in: *Proc. ENUMATH-97*, (H.G. Bock, et al., eds), pp. 621-637, World Scientific Publ., Singapore, 1998.

[3] R. Becker and R. Rannacher, An optimal control approach to error estimation and mesh adaption in finite element methods, *Acta Numerica 2000* (A. Iserles, ed.), pp. 1-101, Cambridge University Press, 2001.

[4] H.-J. Bungartz, M. Schäfer (Eds.), *Fluid-Structure Interaction Modelling, Simulation, Optimization*, Springer Series: Lecture Notes in Computational Science and Engineering, Vol. 53 2006, VIII, 394 p., 251 illus., Softcover ISBN-10: 3-540-34595-7, ISBN-13: 978-3-540-34595-4

[5] Th. Dunne *An Eulerian approach to fluid-structure interaction and goal-oriented mesh refinement*, Proc. "Finite Elements for Flow Problems (FEF05)", IACM Special Interest Conference supported by ECCOMAS, April 4-6, 2005, Swansea, Wales, UK, appeared in Int. J. Numer. Meth. Fluids 2006; 51: 1017-1039.

[6] Th. Dunne *Adaptive Finite Element Approximation of Fluid-Structure Interaction Based on an Eulerian Variational Formulation*, Proc. "European Conference on Computational Fluid Dynamics (ECCOMAS CFD 2006)", September 5-8, 2006, Egmond aan Zee, The Netherlands.

[7] Th. Dunne. *Adaptive Finite Element Approximation of Fluid-Structure Interaction Based on Eulerian and Arbitrary Lagrangian-Eulerian Variational Formulations* Institute of Applied Mathematics, University of Heidelberg, doctoral dissertation, 2007.

[8] Th. Dunne., R. Rannacher, *Adaptive finite element simulation of fluid structure interaction based on an Eulerian variational formulation*, p. 371-386, in H.-J. Bungartz, M. Schäfer (Eds.) *Fluid-Structure Interaction Modelling, Simulation, Optimization*, Springer Verlag, (2006). [4]

[9] GASCOIGNE, A C++ numerics library for scientific computing, `http://gascoigne.uni-hd.de/` .

[10] C. W. Hirt and B. D. Nichols, Volume of Fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics* 39, 201-225 (1981).

[11] J. Hron, S. Turek, *A Monolithic FEM/Multigrid Solver for an ALE Formulation of Fluid-Structure Interaction with Applications in Biomechanics*, p. 146-170, in H.-J. Bungartz, M. Schäfer (Eds.) *Fluid-Structure Interaction Modelling, Simulation, Optimization*, Springer Verlag, (2006). [4]

[12] D. D. Joseph and Y. Y. Renardy, *Fundamentals of two-fluid dynamics. Part I*, Springer, New York, 1993. Math. Theory and Applications.

[13] D. D. Joseph and Y. Y. Renardy, *Fundamentals of two-fluid dynamics. Part II*, Springer, New York, 1993.

[14] S. Osher and J. A. Sethian, Propagation of fronts with curvature based speed: algorithms based on Hamilton-Jacobi formulations, *Journal of Computational Physics* 79, 12 (1988).

[15] J.A. Sethian, *Level set methods and fast marching methods.* Cambridge University Press, 1999.

[16] S. Turek, J. Hron, *Proposal for Numerical Benchmarking of Fluid-Structure Interaction between an Elastic Object and Laminar Incompressible Flow* p. 371-386, in H.-J. Bungartz, M. Schäfer (Eds.) *Fluid-Structure Interaction Modelling, Simulation, Optimization*, Springer Verlag, (2006). [4]

[17] VISUSIMPLE, An open source interactive visualization utility for scientific computing, `http://visusimple.uni-hd.de/` . See [1].

Segregated vs. monolithic solvers for fluid-structure interaction problems

Matthias Heil

(joint work with Andrew Hazel, Richard Muddle and Jonathan Boyle)

We compare the relative performance of monolithic and segregated (partitioned) solvers for large-displacement fluid-structure interaction (FSI) problems within the framework of oomph-lib, the object-oriented multi-physics finite-element library, available as open-source software at `http://www.oomph-lib.org`. Monolithic solvers are widely acknowledged to be more robust than their segregated counterparts, but are believed to be too expensive for use in large-scale problems. We demonstrate that monolithic solvers are competitive even for problems in which the fluid-solid coupling is weak and, hence, the segregated solvers converge within a moderate number of iterations. The efficient monolithic solution of large-scale FSI problems requires the development of preconditioners for the iterative solution of the linear systems that arise during the solution of the monolithically-coupled fluid and solid equations by Newton's method. We demonstrate that recent improvements to oomph-lib's FSI preconditioner, based on [3], result in mesh-independent convergence rates under uniform and non-uniform (adaptive) mesh refinement, and explore its performance in a number of two- and three-dimensional test problems involving the interaction of finite-Reynolds-number flows with shell and beam structures, as well as finite-thickness solids.

References

[1] Heil M , Hazel AL (2006) oomph-lib – An Object-Oriented Multi-Physics Finite-Element Library. In: *Fluid-Structure Interaction*, Editors: M. Schafer and H.-J. Bungartz. Springer (Lecture Notes on Computational Science and Engineering), pp. 19–49.

[2] Heil M (2004) An efficient solver for the fully-coupled solution of large-displacement fluid-structure interaction problems. Computer Methods in Applied Mechanics and Engineering, 193:1-23.

[3] Elman HC, Silvester DJ, Wathen AJ, Finite Elements and Fast Iterative Solvers with Applications in Incompressible Fluid Dynamics. (Oxford University Press, 2006).

On the numerical simulation of the free fall of a solid in a fluid: the Newtonian and viscoelasic case

Vincent Heuveline

1. Introduction

The ability to predict the motion of a solid falling in a fluid has a far reaching impact in areas ranging from meteorology, sedimentology, aerospace engineering, DNA sequencing, biology (*e.g.* models for animal flights) and electrophoresis (see e.g. [2] for a detailed bibliography). Despite the seemingly simple problem formulation, even elementary phenomenological aspects related to this problem are, if at all, not well understood and the object of intensive research activities. These efforts concentrate not only on the mathematical analysis but also on the development of adequate experimental devices toward experimental investigation. One main difficulty is due to the fact that the considered fluid-structure interaction may result in highly intricate motion patterns (see [2] and referenced therein).

Further the falling behaviour greatly depends on the properties of the considered fluid. It is well known from experiments that the orientation of long cylinders or ellipsoids, when dropped from rest, dramatically depends on whether the fluid is purely Newtonian or not. In Newtonian fluids, such a body always reach a stable equilibrium orientation with its broadside parallel to the horizontal, while in non-Newtonian fluids the so-called *tilt angle* which describes the angle formed by the long axis of symmetry of the particle with the horizontal may vary from 0^0 to 90^0, depending on weight, length and shape of the particle as well as on the material properties of the liquid (see e.g. [6, 7]). While the mechanism of orientation is quite well understood for the case of a Newtonian fluid at least for small Reynolds number [8, 11], there is until now no satisfactory theory to explain the tilt angle phenomenon for non-Newtonian liquids.

The lecture presented new methods which allows to tackle such problems by means of numerical simulation. From a numerical point of view, this problem is difficult since it requires the accurate simulation of the fluid-structure coupling under the additional constraints of large computational domains as well as large integration times. Besides the study of fundamental issues related to the free fall problem in viscous flows, the purpose of this development is also to be able to calibrate or validate simplified models for the free fall problem. In that context, a strong emphasis is given to the accurate determination of values such as the drag, the lift and the torque acting on the body under very weak assumptions on its geometry.

2. Problem formulation and solution process

Our approach relies on the description of the free fall problem by means of an exterior flow problem where the liquid fills the whole space. This well established formulation [13, 12, 5] relies on the assumption that the wall effects related to the enclosure are negligible and do not influence the motion of the falling body.

In accordance with experiments (see [5, p.7] and quoted references), one may reasonably assume that this property holds with respect to the final steady state orientation of the falling body. The crucial advantage of this approach is that the reformulation of the resulting governing equations in the body frame leads to a problem where the region occupied by the liquid, i.e. the computational domain, is not time-dependent any more.

We consider the free fall of a solid body $\mathcal{S} \subset \mathbb{R}^d$ $(d = 2, 3)$ in an incompressible liquid $\mathcal{L}$ filling the whole space $\mathcal{D} := \mathbb{R}^d \backslash \mathcal{S}$. The solid body $\mathcal{S}$ is assumed to be a bounded domain and the velocity of its mass center C (resp. its angular velocity) in the inertial frame $\mathcal{F}$ are denoted by $\mathcal{V}_C$ (resp. Ω). The region occupied by $\mathcal{S}$ at time t is described by $S(t)$. In the inertial frame $\mathcal{F}$ the equations of conservation of momentum and mass of $\mathcal{L}$ in their non-conservative form are given by

$$\left.\begin{array}{c} \rho \frac{\partial \mathrm{v}}{\partial t} + \rho(\mathrm{v} \cdot \nabla)\mathrm{v} = \rho g + \nabla \cdot \mathcal{T}(\mathrm{v}, \mathrm{p}) \\ \nabla \cdot \mathrm{v} = 0 \end{array}\right\} \text{ for } (x,t) \in \bigcup_{t>0} [\mathbb{R}^d \backslash S(t)] \times \{t\}, \tag{1}$$

where ρ is the constant density of $\mathcal{L}$, v and p are the Eulerian velocity field and pressure associated with $\mathcal{L}$, $\mathcal{T}$ is the Cauchy stress tensor and ρg is the force of gravity which is assumed to be the only external force.

We assume a Newtonian fluid or a *second-order* liquid model for which the Cauchy stress tensor is given by

$$\mathcal{T}(\mathrm{v}, \mathrm{p}) := \mathcal{T}_{NS}(\mathrm{v}, \mathrm{p}) + \mathcal{S}(\mathrm{v}). \tag{2}$$

Here, $\mathcal{T}_{NS}$ denotes the standard Cauchy stress tensor for Newtonian liquids,

$$\mathcal{T}_{NS}(v, p) := -\mathrm{p}\mathbf{1} + \mu(\nabla \mathrm{v} + (\nabla \mathrm{v})^T), \tag{3}$$

where μ is the shear viscosity parameter. The tensor $\mathcal{S}(v)$ of viscoelastic extra-stresses is given by

$$\mathcal{S}(\mathrm{v}) := \alpha_1 \mathcal{A}_2(\mathrm{v}) + \alpha_2 \mathcal{A}_1^2(\mathrm{v}), \tag{4}$$

where $\mathcal{A}_1$ and $\mathcal{A}_2$ denote the first- and second-order Rivlin-Ericksen tensors,

$$\mathcal{A}_1(\mathrm{v}) = \nabla \mathrm{v} + (\nabla \mathrm{v})^T, \tag{5}$$

and

$$\mathcal{A}_2(\mathrm{v}) = \frac{\partial \mathcal{A}_1(\mathrm{v})}{\partial t} + (\mathrm{v} \cdot \nabla)\mathcal{A}_1(\mathrm{v}) + \mathcal{A}_1(\mathrm{v}) \cdot (\nabla \mathrm{v})^T + \nabla \mathrm{v} \cdot \mathcal{A}_1(\mathrm{v}). \tag{6}$$

α_1 and α_2 are the so-called *quadratic constants* which are related to the non-Newtonian properties of the model. Let

$$\varepsilon := \alpha_2/\alpha_1, \tag{7}$$

we impose $\varepsilon = -1.8$ under the usual constraints $\alpha_1 < 0$ and $\alpha_2 > 0$. The initial boundary conditions are given by

$$\mathrm{v}(x, 0) = 0, \ \lim_{|x| \to \infty} \mathrm{v}(x, t) = 0 \text{ for } x \in \mathbb{R}^d \backslash S(t), \tag{8}$$

$$\mathrm{v}(x, t) = \mathcal{V}_C(t) + \Omega(t) \times (x - x_C(t)) \text{ for } x \in \partial S(t). \tag{9}$$

The fluid/body coupling occurs through the Dirichlet boundary condition (9).

The proposed approach relies on an explicit treatment of the fluid-body coupling. An intrinsic difficulty considering this two-step approach is related to the fact that the data occurring in the flow equations are generally non-smooth with regard to the time variable since explicitly updated. In order to cope with this difficulty we consider for instationary motions a time stepping strategy which is based on the implicit fractional-step-θ scheme. For the viscoelastic case, where the steady motion are of interest, the solver is based on a modified quasi-Newton scheme which allows to avoid iterations trough the unneeded transient states. The spatial discretization relies on the finite element method. The overall numerical scheme is supplemented by a numerical stability analysis based on the principle of linearized stability with respect to the body equations in order to filter numerically unstable steady state solutions. We refer to [10, 4, 2, 3] for a detailed description of the overall solution process.

3. Some numerical results

We consider the two dimensional sedimentation problem of a rectangle falling in a second-order fluid. The density (resp. the size) of the rectangular body is denoted by ρ_S (resp. $l_1 \times l_2$). In that context the setup of the considered free fall problem is described by means of the following non-dimensional numbers:

$$Re = \frac{\rho |V_C(\infty)| D}{\mu}, \qquad \textit{Reynolds number} \tag{10}$$

$$We = \frac{|\alpha_1||V_C(\infty)|}{D\mu}, \qquad \textit{Weissenberg number}, \tag{11}$$

where D corresponds to the diameter of the solid body. Based on these definitions we then can define the so-called *elasticity number* in the following way:

$$E = \frac{We}{Re} = \frac{|\alpha_1|}{D^2 \rho}. \tag{12}$$

We keep the following parameters constant: $l_1 = 1$, $l_2 = 2$, $\mu = 30$, $\rho = 0.773$. The tilt-angle dependency with respect to ρ_S for the corresponding values of α_1 is depicted in Figure 1. This result clearly show the complex and highly non-linear character related to the orientation of the solid body.

References

[1] S. Bönisch, V. Heuveline and P. Wittwer Adaptive Boundary Conditions for Exterior Flow Problems. *Journal of Mathematical Fluid Mechanics*, Vol. 7, No. 1, pp. 85-107, 2005.

[2] S. Bönisch and V. Heuveline On the Numerical Simulation of the Instationary Free Fall of a Solid in a Fluid. I. The Newtonian Case. *Computers & Fluids*, to appear.

[3] S. Bönisch and V. Heuveline On the Numerical Simulation of the free fall of a solid in a fluid. II. The viscoelastic case. *Technical Report, IWR Heidelberg, 2004-32*, 2004.

[4] S. Bönisch, V. Heuveline and P. Wittwer Second Order Adaptive Boundary Conditions for Exterior Flow Problems: Non-symmetric Stationary Flows in two Dimensions. *Journal of Mathematical Fluid Mechanics*, Vol. 8, pp. 1-26 , 2006.

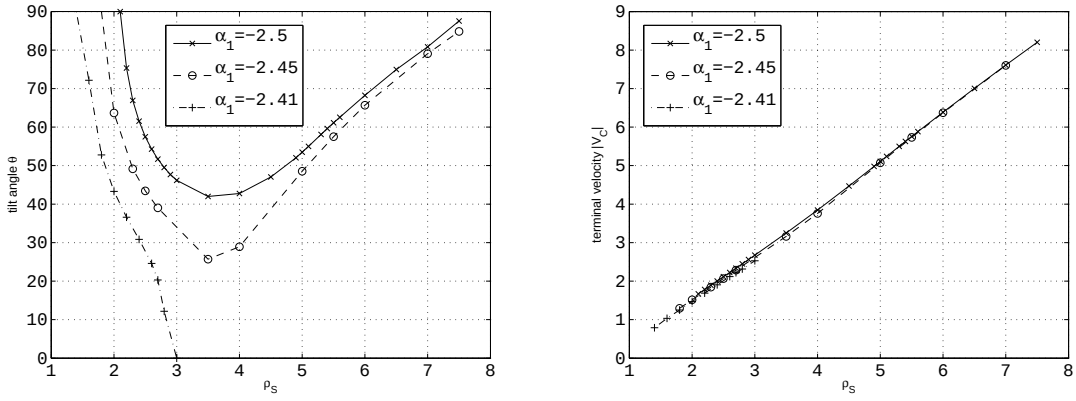

FIGURE 1. Plot of the tilt-angle (left) and norm of the falling velocity $V_C(\infty)$ assuming $\alpha_1 \in \{-2.5; -2.45; -2.41\}$ and $\rho_S \in [1.4; 7.5]$.

[5] G. P. Galdi. On the motion of a rigid body in a viscous liquid: A mathematical analysis with applications. In *Friedlander, S. (ed.) et al., Handbook of mathematical fluid dynamics. Vol. 1.*, pages 653–791. Amsterdam: Elsevier., 2002.

[6] G.P. Galdi. Slow steady fall of a rigid body in a second-order fluid. *J. Non-Newtonian Fluid Mech.*, 93:169–177, 2000.

[7] G.P. Galdi, M. Pokorny, A. Vaidya, D.D. Joseph, and J. Feng. Orientation of symmetric bodies falling in a second-order liquid at non-zero Reynolds number. *Math. Models Methods Appl. Sci.*, 12(11):1653–1690, 2002.

[8] G.P. Galdi and A. Vaidya. Translational steady fall of symmetric bodies in a Navier-Stokes liquid, with application to particle sedimentation. *J. Math. Fluid Mech.*, 3(2):183–211, 2001.

[9] G. Galdi and V. Heuveline Lift and Sedimentation of Particles in the Flow of a Viscoelastic Liquid in a Channel. *CRC Press*, 2007.

[10] V. Heuveline Adaptive Finite Elements for the Steady Free Fall of a Body in Newtonian Fluid. *Comptes Rendus de l'Académie des Sciences, Mécanique*, 2005, 333/12, 896-909.

[11] T.W. Pan, R. Glowinski, and G. P. Galdi. Direct simulation of the motion of a settling ellipsoid in Newtonian fluid. *J. Comput. Appl. Math.*, 149(1):71–82, 2002.

[12] D. Serre. Chute libre d'un solide dans un fluide visqueux incompressible. Existense. *Jap. J. Appl. Math.*, 4(1):99–110, 1987.

[13] H.F. Weinberger. Variational properties of steady fall in Stokes flow. *J. Fluid Mech.*, 52:321–344, 1972.

A monolithic FEM solver for fluid structure interaction in ALE formulation

JAROSLAV HRON

(joint work with M. Mádlík, Stefan Turek, M. Razzaq)

Both problems of viscous fluid flow and of elastic body deformation have been studied separately for many years in great detail. The Eulerian (or spatial) description is well suited for a problem of fluid flowing through some spatially fixed region. In such a case the material particles can enter and leave the region of interest. The fundamental quantity describing the motion is the velocity vector.

On the other hand the Lagrangian (or referential) description is well suited for a problem of deforming a given body consisting of a fixed set of material particles. In this case the actual boundary of the body can change its shape. The fundamental quantity describing the motion in this case is the vector of displacement from the referential state. In the case of fluid-structure interaction problem we can still use the Lagrangian description for the deformation of the solid part. The fluid flow now takes place in a domain with boundary given by the deformation of the structure which can change in time and is influenced back by the fluid flow. The mixed ALE description of the fluid has to be used in this case. The fundamental quantity describing the motion of the fluid is still the velocity vector but the description is accompanied by a certain displacement field which describes the change of the fluid domain. This displacement field has no connection to the fluid velocity field and the sole purpose of its introduction is to provide a transformation of the current fluid domain and corresponding governing equations to some fixed reference domain.

1. Monolithic fluid structure interaction problem formulation

We will use the superscripts ^{s} and ^{f} to denote the quantities connected with the solid and fluid. Let us assume that the both materials are incompressible and all the processes are isothermal and let us denote the constant densities of each material by ϱ^f, ϱ^s. We denote by Ω_t^f the domain occupied by the fluid and Ω_t^s by the solid at time $t \in [0,T]$. Let $\Gamma_t^0 = \bar{\Omega}_t^f \cap \bar{\Omega}_t^s$ be the part of the boundary where the solid interacts with the fluid and $\Gamma_t^i, i = 1,2,3$ be the remaining external boundaries of the solid and the fluid. Let the deformation of the solid part be described by the displacement $\boldsymbol{u}^s$ and the velocity $\boldsymbol{v}^s$. The fluid flow is described by the velocity field $\boldsymbol{v}^f$ defined on the fluid domain Ω_t^f. Further we define the auxiliary mapping, denoted by the corresponding displacement $\boldsymbol{u}^f$. We require that the mapping to be sufficiently smooth, one to one and has to satisfy a suitable boundary value problem. In the context of the finite element method this will describe the artificial mesh deformation inside the fluid region and it will be constructed as a solution to a suitable boundary value problem.
The standard momentum and mass balance of the fluid in the time dependent fluid and solid are considered (see [2, 1] for details).
The interaction is due to the exchange of momentum through the common part of the boundary Γ_t^0. On this part we require that the forces are in balance and simultaneously the no slip boundary condition for the fluid, i.e.

$$\boldsymbol{\sigma}^f \boldsymbol{n} = \boldsymbol{\sigma}^s \boldsymbol{n} \quad \text{on } \Gamma_t^0, \qquad \boldsymbol{v}^f = \boldsymbol{v}^s \quad \text{on } \Gamma_t^0. \tag{1}$$

The remaining external boundary conditions can be of the any standard kind of boundary conditions.
In order to obtain the monolithic description we introduce the domain $\Omega = \Omega^f \cup \Omega^s$, where Ω^f, Ω^s are the domains occupied by the fluid and solid in the initial

undeformed state, and two fields defined on this domain as

$$\boldsymbol{u} : \Omega \times [0,T] \to \mathcal{R}^3, \qquad \boldsymbol{v} : \Omega \times [0,T] \to \mathcal{R}^3,$$

such that the field $\boldsymbol{v}$ represents the velocity at the given point and $\boldsymbol{u}$ the displacement on the solid part and the artificial displacement in the fluid part, taking care of the fact that the fluid domain is changing with time,

$$\boldsymbol{v} = \begin{cases} \boldsymbol{v}^s & \text{on } \Omega^s, \\ \boldsymbol{v}^f & \text{on } \Omega^f, \end{cases} \qquad \boldsymbol{u} = \begin{cases} \boldsymbol{u}^s & \text{on } \Omega^s, \\ \boldsymbol{u}^f & \text{on } \Omega^f. \end{cases} \tag{2}$$

Due to the condition (1) both fields are continuous across the interface Γ_t^0.
The complete set of the equations can be written as

$$\frac{\partial \boldsymbol{u}}{\partial t} = \begin{cases} \boldsymbol{v} & \text{in } \Omega^s, \\ \Delta \boldsymbol{u} & \text{in } \Omega^f, \end{cases} \tag{3}$$

$$\frac{\partial \boldsymbol{v}}{\partial t} = \begin{cases} \frac{1}{J\varrho^s} \operatorname{Div} \mathbf{P}^s & \text{in } \Omega^s, \\ -(\operatorname{Grad} \boldsymbol{v})\mathbf{F}^{-1}(\boldsymbol{v} - \frac{\partial \boldsymbol{u}}{\partial t}) + \frac{1}{J\varrho^f} \operatorname{Div}(J\boldsymbol{\sigma}^f \mathbf{F}^{-T}) & \text{in } \Omega^f, \end{cases} \tag{4}$$

$$0 = \begin{cases} J - 1 & \text{in } \Omega^s, \\ \operatorname{Div}(J\boldsymbol{v}\mathbf{F}^{-T}) & \text{in } \Omega^f, \end{cases} \tag{5}$$

with the boundary conditions

$$\boldsymbol{u} = \mathbf{0}, \quad \boldsymbol{v} = \boldsymbol{v}_B \quad \text{on } \Gamma^1, \qquad \boldsymbol{u} = \mathbf{0} \quad \text{on } \Gamma^2, \qquad \boldsymbol{\sigma}^s \boldsymbol{n} = \mathbf{0} \quad \text{on } \Gamma^3. \tag{6}$$

In order solve the balance equations we need to specify the constitutive relations for the stress tensors. For the fluid we use the incompressible Newtonian relation and the neo-Hookean material constitutive equation is used for the structure which gives the Cauchy stress tensor as

$$\boldsymbol{\sigma}^f = -p^f \mathbf{I} + \mu(\nabla \boldsymbol{v}^f + (\nabla \boldsymbol{v}^f)^T), \qquad \boldsymbol{\sigma}^s = -p^s \mathbf{I} + \varrho^s \alpha(\mathbf{F}\mathbf{F}^{\mathrm{T}} - \mathbf{I}), \tag{7}$$

where μ represents the viscosity of the fluid and p^f is the Lagrange multiplier corresponding to the incompressibility constraint.

2. Discretization and solution algorithm

Here we restrict ourselves to two dimensions. The three dimensional extension can be found in [1]. The time discretization is done by the Crank-Nicholson scheme which is only conditionally stable but it has better conservation property than for example the implicit Euler scheme. The discretization in space is done by the finite element method. Our treatment of the problem as a one system suggests to use the same finite elements on both, the solid part and the fluid region. Since both materials are incompressible we have to choose a pair of finite element spaces know to be stable for the problems with incompressibility constraint. One possible choice is the conforming biquadratic, discontinuous bilinear Q_2, P_1 pair. The discretized

system of nonlinear algebraic equations is solved using Newton method as the basic iteration. One step of the Newton iteration can be written as

$$\boldsymbol{X}^{n+1} = \boldsymbol{X}^n - \omega \left[\frac{\partial \boldsymbol{\mathcal{R}}}{\partial \boldsymbol{X}}(\boldsymbol{X}^n)\right]^{-1} \boldsymbol{\mathcal{R}}(\boldsymbol{X}^n) \tag{8}$$

The dumped Newton method with line search improves the chance of convergence by adaptively changing the length of the correction vector ω. An adaptive time-step selection was found to help in the nonlinear convergence. A heuristic algorithm was used to correct the time-step length according to the convergence of the nonlinear iterations in the previous time-step. If the convergence was close to quadratic, i.e. only up to three Newton steps were needed to obtain required precision, the time step could be slightly increased, otherwise the time-step length was reduced.

The structure of the Jacobian matrix $\frac{\partial \boldsymbol{\mathcal{R}}}{\partial \boldsymbol{X}}$ is

$$\frac{\partial \boldsymbol{\mathcal{R}}}{\partial \boldsymbol{X}}(\boldsymbol{X}) = \begin{pmatrix} A_{vv} & A_{vu} & B_v \\ A_{uv} & A_{uu} & B_u \\ B_v^T & B_u^T & 0 \end{pmatrix}, \tag{9}$$

and it can be computed by finite differences from the residual vector $\boldsymbol{\mathcal{R}}(\boldsymbol{X})$

$$\left[\frac{\partial \boldsymbol{\mathcal{R}}}{\partial \boldsymbol{X}}\right]_{ij} (\boldsymbol{X}^n) \approx \frac{[\boldsymbol{\mathcal{R}}]_i(\boldsymbol{X}^n + \alpha_j \boldsymbol{e}_j) - [\boldsymbol{\mathcal{R}}]_i(\boldsymbol{X}^n - \alpha_j \boldsymbol{e}_j)}{2\alpha_j}, \tag{10}$$

where $\boldsymbol{e}_j$ are the unit basis vectors in $\mathcal{R}^n$ and coefficients α_j are adaptively taken according to the change in the solution in the previous time step. Since we know the sparsity pattern of the Jacobian matrix in advance, it is given by the used finite element method, this computation can be done in an efficient way so that the linear solver remains the dominant part in terms of the cpu time.

References

[1] J. Hron, M. Mádlík, *Fluid-structure interaction with applications in biomechanics*, Nonlinear Analysis: Real World Applications **8** (2007), 1431–1458.

[2] J. Hron, S. Turek, *A Monolithic FEM Solver for ALE Formulation of Fluid Structure Interaction with Configurations for Numerical Benchmarking*, Papadrakakis, M.; Onate, E.; Schrefler Eds., Computational Methods for Coupled Problems in Science and Engineering (2005)

Simulations of particulate flows

Howard H. Hu

Direct numerical simulation of particulate flows, which resolves the flow field around each particle, remains a computationally challenging problem. Finite element methods based on a moving unstructured mesh, such as the Particle-Mover with arbitrary Lagrangian-Eulerian (ALE) method of Hu [1], use meshes that conform to the shape of the particles, offer local mesh refinement around the particle surface, and are shown to be efficacious for small numbers of particles at moderate

Reynolds numbers. However, they become computationally expensive for systems of dense particle suspensions due to frequent remeshing and projection.
In this presentation, we will cover several topics related to direct numerical simulations of particulate flows. The first is on direct numerical simulations of rigid particles in viscous fluid flows using an explicit finite-difference scheme. The scheme relaxes the condition of incompressibility and uses explicit update for the governing equations. In particulate flows, the time steps for the simulations are limited by the movement of the particles, and are generally small. The condition on the time step for stable explicit schemes is not too restrictive. In [2] a finite difference scheme based on MacCormack method is formulated for particulate flows. The MacCormack method is a two step predictor-corrector explicit scheme. To enforce boundary conditions on the particle surfaces, a local Taylor series expansion was used to enforce no-slip approximately by assigning velocities to the grid points nearest the surface such that velocity on the surface would be zero. The stresses were numerically integrated on the particle surfaces to evaluate the hydrodynamic forces and moments acting on the particles, and the particles were moved accordingly. It was found that a relative fine mesh, of at least 20 grid spacings across the particle diameter at Reynolds number around 20, was needed to obtain smooth pressures on the particle surfaces. The dense mesh also requires a smaller time step because of the stability condition for the explicit scheme.
The effort to relieve this bottleneck led to a different implementation of boundary condition on the particle surfaces. In a coordinate system moving with the particle, there is a region next to the particle surface where inertial effects are small, so the equations for Stokes flow provide a reasonable approximation. A spectral solution for the Stokes equations next to the particle surface could then be matched with the finite difference solution of the full Navier-Stokes equations away from the particle surface [3, 4]. The spectral solution exactly satisfies the no-slip condition on the particle surface, and explicitly gives the forces and moments acting on the particle. Additional advantages of this technique include superior accuracy of the scheme on a relatively coarse grid for intermediate particle Reynolds numbers, ease of implementation, and the elimination of the need to track the particle surface.
In this presentation, we will show results calculated for both 2D and 3D particulate flow problems with an explicit Lax-Wendroff method using this special treatment of the boundary conditions at the particle surface. The Lax-Wendroff scheme is formally identical to the MacCormack scheme in [2] for convection-diffusion equations, but it is a one-step method rather than a predictor-corrector scheme. Consequently, the boundary conditions need only be applied once per time step. Besides this simplification, it was found that the one-sided finite differences used in the MacCormack scheme introduced an asymmetry in the forces, leading to an artificial lift even at low Reynolds numbers when the grid was relatively coarse. The Lax-Wendroff scheme uses only central differences, which eliminates the asymmetry.

The second topic is on a numerical technique to simulate dynamics of elastic deformable objects in viscous fluids. In solving problems with fluid-structure interaction, it is common to have one solver for the Navier-Stokes equations for the fluid phase, another solver for the elasticity equations for the solid phase, and to iterate between them. The variables in the fluid are the velocity and pressure, and in the solid are the components of the displacement field. For finite elasticity problems, the Lagrangian Cauchy-Green stress tensor is often used. For particulate flows, however, the large number and large deformation of the particles make the traditional formulation difficult to implement.
In this presentation, we will present a numerical scheme that solves the large deformation of incompressible elastic solid particles with a velocity-stress formulation, which models the elastic solid as a Òneo-HookeanÓ material. In this formulation the displacement field is eliminated. It is known that the Eulerian Almansi strain tensor is related to the strain rate tensor through an evolution equation with a lower-convected time derivative. It is then proposed that in a Òneo-HookeanÓ material, the stress tensor is proportional to this Almansi strain tensor. The result is that the stress tensor within the solid satisfies a constitutive equation that is similar to those commonly used to describe viscoelastic fluids. Therefore, mixed finite element techniques developed for viscoelastic fluids are adapted here to simulate dynamics of elastic objects in viscous fluids.
In our implementation, the movement and the deformation of the particles are handled with an Arbitrary Lagrangian Eulerian (ALE) scheme to track the exact fluid-solid interface. The coupling between the solid and fluid phase is enforced by assuming that the material velocity and the traction force are continuous across the interface between the solid and the liquid. However, the pressure and stress components are allowed to have jumps across the fluid-solid interface. The coupled fluid-solid equations are discretized by a Galerkin finite element method with mixed elements for velocity, pressure and stress components. The combined algebraic equations for the fluid-solid system are solved using iterative solvers. This monolithic solution scheme is demonstrated to be stable and is capable to resolve large deformations of the solid particles. Results for the deformations of particles in sedimentation, shear flow and Poiseuille flow will be presented.
The third topic is on numerical models that simulate motion of dielectric particles in an electrolyte. Dielectrophoresis (or DEP) is a phenomenon in which a force is exerted on a dielectric particle when it is suspended in a non-uniform electric field. Most suspensions in biological applications involve electrolyte where an ionic double layer may be formed next to particle surface due to the induced-charge on its surface when the frequency of the applied AC fields is not too high. This double layer affects the dielectrophoretic motion of the particle. It modifies the net dipole moment of the particle and at the same time produces slip velocity on the particle surface. The effects of the double layer on the dielectrophoretic motion of particles will be discussed.

References

[1] H.H. Hu, N.A. Patankar, and M.Y. Zhu, *Direct numerical simulations of fluid- solid systems using the arbitrary lagrangian-eulerian technique*, J. Comput. Phys. **169** (2001), 42–462.

[2] A. Perrin and H. H. Hu, *An explicit finite-difference scheme for simulation of moving particles*, J. Comput. Phys. **212** (2006), 166–187.

[3] Z. Zhang and A. Prosperetti, *A method for particle simulation*, Transactions of the ASME, **70** (2003), 64–74.

[4] Z. Zhang and A. Prosperetti, *A second-order method for three-dimensional particle simulation*, J. Comput. Phys., **210** (2005), 292–324.

Numerical simulation of free-surface flows with application to ball-bearings

Stefan Knauf

(joint work with Rolf Rannacher)

The dynamics of ball-bearings is essentially affected by fluid-structure interaction. In particular, the contact between the rolling ball and the curved surface is established through a thin film of lubricant. This thin film has a free surface and the contact area between ball and "outer" surface is very small. Therefore forces acting on the ball lead to high pressures in the contact region. In such situations the common Navier-Stokes equations are no longer an adequate model, in fact one has to assume a pressure-dependent viscosity.
From the computational point of view, the problem is tackled with an "Arbitrary-Lagrangian-Eulerian" (ALE) method. The extreme geometry with an aspect ratio of the contact region of ~ 1000 is so far not efficiently treatable with common multigrid methods.

References

[1] C. T. Wilson,*Stirring and transport enhancement in a continuously modulated free-surface flow*, Journal of Fluid Mechanics **565**, 319–351.

[2] J. Hron *Simple Flows of Fluids with Pressure-Dependent Viscosities*, Proceedings: Mathematical, Physical and Engineering Sciences **457**, 1603–1622.

[3] A. J. Moore *The behaviour of lubricants in elastohydrodynamic contacts*, Proceedings of the I MECH E Part J Journal of Engineering Tribology **211**, 91–106.

[4] T. Dunne *Adaptive Finite Element Approximation of Fluid-Structure Interaction Based on Eulerian and Arbitrary Lagrangian-Eulerian Variational Formulations*, PhD Thesis.

[5] S. Bair *Rheological challenges and opportunities for EHL*,IUTAM Symposium on Elastohydrodynamics and Micro-elastohydrodynamics, Springer 2006.

Fictitious domain methods for particulate flows

Peter Minev
(joint work with Antoine Dechaume, Shi Jin, Chidambaranathan Veeramani)

Modelling of particulate flows is one of the most difficult and often misunderstood ares of Computational Fluid Dynamics. There is a variety of proposed models, however, none of them is based on a solid theoretical foundation, thoroughly verified in experiments. Therefore, it is very important to approach the problem by solving directly the incompressible Navier-Stokes equations in a domain filled with rigid particles and try to extract some useful average characteristics of such flows. This is an extremely difficult computational task and there are very few available methods that could be applied. In this talk we describe the variant of the fictitious domain method (FDM) which was proposed by our group in [1] and [2]. Several validation examples of flows with rigid particles are also presented.
The FDM gained a wide popularity in the last decade, after the sequence of papers by Glowinski and his collaborators (see [3] and the references therein), mostly due to its relative simplicity when applied to flows containing many rigid particles or fluid-structure interaction problems with complicated geometries. Essentially, it is an Eulerian method which does not require re-meshing and therefore it can be relatively easily applied to large scale problems requiring parallelization. In this talk we discuss the method recently proposed in [2] which is applied for the discretization of the incompressible Navier-Stokes equations

$$\frac{D\hat{\mathbf{u}}_1}{Dt} = -\nabla \hat{p}_1 + \frac{1}{Re}\nabla^2 \hat{\mathbf{u}}_1, \quad \nabla \cdot \hat{\mathbf{u}}_1 = 0 \text{ in } \Omega_1 \tag{1}$$

containing n particles whose equations of motion are given by

$$\rho_{2,i}\frac{d\mathbf{U}_i}{dt} = (\rho_{2,i} - \rho_1)\frac{1}{Fr}\mathbf{e}_g + \frac{\rho_1}{V_i}\mathbf{F}_i \tag{2}$$

$$\mathbf{I}_i\frac{d\boldsymbol{\omega}_i}{dt} + \boldsymbol{\omega}_i \times \mathbf{I}_i\boldsymbol{\omega}_i = \mathbf{T}_i. \tag{3}$$

Here $i = 1, \ldots, n$, Fr is the Froude number, $\mathbf{e}_g$ is the unit vector in the direction of gravity, V_i is the volume of the particle, $\mathbf{F}_i = \int\limits_{\partial\Omega_{2,i}} \hat{\boldsymbol{\sigma}}_1 \cdot \mathbf{n} ds$ is the total hydrodynamic force acting on the particle, $\hat{\boldsymbol{\sigma}}_1 = -\hat{p}_1\boldsymbol{\delta} + 1/Re(\nabla\hat{\mathbf{u}}_1 + (\nabla\hat{\mathbf{u}}_1)^T)$ is the stress tensor of the fluid, $\mathbf{n}$ is the unit normal pointing out of the particle, $\mathbf{I}_i$ is its tensor of inertia, and $\mathbf{T}_i$ is the hydrodynamic torque about its center of mass. This set is supplemented by the kinematic equations for the centroids $\mathbf{X}_i$

$$\frac{\partial \mathbf{X}_i}{\partial t} = \mathbf{U}_i, \quad i = 1, \ldots, n.$$

1. Discretization

The discretization proceeds with the following steps (see [2] for the details of the derivation of the algorithm)

- **Advection-diffusion substep.**

The center of mass of the $i-$th particle is predicted explicitly by

$$\mathbf{X}_i^{p,n+1} = \mathbf{X}_i^{n-1} + 2\delta t \mathbf{U}_i^n \tag{4}$$

where δt is the time step. Then we solve for $\mathbf{u}_1^*$ from

$$\begin{aligned} &\tau_0 \mathbf{u}_1^* - \frac{1}{Re}\nabla^2 \mathbf{u}_1^* = -\tau_1 \tilde{\mathbf{u}}_1^n - \tau_2 \tilde{\mathbf{u}}_1^{n-1} - \nabla p_1^n + \frac{\rho_{2,i}-\rho_1}{\rho_1}\mathbf{G}, \text{ in } \Omega \\ &\mathbf{u}_1^* = 0 \text{ on } \partial\Omega \end{aligned} \tag{5}$$

where $\tau_0 = 3/(2\delta t), \tau_1 = -2/\delta t, \tau_2 = 1/(2\delta t)$, and $\tilde{\mathbf{u}}_1^n$, $\tilde{\mathbf{u}}_1^{n-1}$ are the velocities from time levels $n, n-1$ which are advected alongside an approximation of the characteristics (see [4] for details). The particle velocities are predicted by

$$\tau_0 \mathbf{U}_i^* + \tau_1 \mathbf{U}_i^n + \tau_2 \mathbf{U}_i^{n-1} = \frac{1}{V_i} \int\limits_{\Omega_{2,i}(t^{n+1})} \left(\tau_0 \mathbf{u}_1^* + \tau_1 \tilde{\mathbf{u}}_1^n + \tau_2 \tilde{\mathbf{u}}_1^{n-1}\right) d\Omega. \tag{6}$$

- **Projection substep.**

On the next substep we impose the incompressibility constraint by solving

$$\begin{aligned} &\tau_0(\mathbf{u}_1^{**} - \mathbf{u}_1^*) = -\nabla(p_1^{n+1} - p_1^n) \text{ in } \Omega \\ &\nabla \cdot \mathbf{u}_1^{**} = 0 \text{ in } \Omega \\ &\mathbf{u}_1^{**} \cdot \mathbf{n} = 0 \text{ on } \partial\Omega, \end{aligned} \tag{7}$$

$\mathbf{n}$ being the outward normal to $\partial\Omega$.

- **Rigid body constraint.**

$$\begin{aligned} \mathbf{u}_1^{n+1} = \mathbf{u}_1^{**} + \Bigg[(\mathbf{U}_i^* - \mathbf{u}_1^{**}) + \frac{1}{2V_i}\left(\int\limits_{\Omega_{2,i}(t^{n+1})} \nabla \times \mathbf{u}_1^{n+1} d\Omega \right) \times (\mathbf{x} - \mathbf{X}_i^{p,n+1}) \\ + \frac{1}{V_i}\frac{\rho_1}{\rho_{2,i}} \int\limits_{\Omega_{2,i}(t^{n+1})} (\mathbf{u}_1^{**} - \mathbf{U}_i^*)\, d\Omega \Bigg] 1_{\Omega_2} \text{ in } \Omega, \end{aligned} \tag{8}$$

$$\mathbf{U}_i^{n+1} = \frac{1}{V_i}\frac{\rho_1}{\rho_{2,i}} \int\limits_{\Omega_{2,i}(t^{n+1})} \mathbf{u}_1^{**} d\Omega + \left(1 - \frac{\rho_1}{\rho_{2,i}}\right)\mathbf{U}_i^*,$$

where

$$1_{\Omega_2} = \begin{cases} 1, & \text{in } \Omega_2 \\ 0, & \text{in } \Omega_1. \end{cases} \tag{9}$$

is the characteristic function of Ω_2. One way to solve this set is to approximate $\nabla \times \mathbf{u}_1^{n+1}$ by $\nabla \times \mathbf{u}_1^{**}$ and so to impose the rigid body constraint fully explicitly. The other possibility is to solve the equations iteratively.

Finally, the angular velocity is computed from

$$\mathbf{U}_i + \boldsymbol{\omega}_i \times (\mathbf{x} - \mathbf{X}_i) = \mathbf{u}_1, \text{ in } \Omega_{2,i}, \quad i = 1, \ldots, n, \tag{10}$$

and the centroids positions are corrected according to

$$\mathbf{X}_i^{n+1} = \mathbf{X}_i^n + 0.5\delta t(\mathbf{U}_i^{n+1} + \mathbf{U}_i^n). \tag{11}$$

1.1. **Numerical results.** The method has been validated on a 3D sedimentation of a single and multiple spherical particles, a spherical particle in a Poiseuille flow (see [2]) and on a 3D ellipsoid in a Poiseuille flow (see [5]).

References

[1] C. Diaz-Goano, P.D. Minev and K. Nandakumar, *A fictitious domain/finite element method for particulate flows*, J. Comp. Phys. **192** (2003), 105–123.

[2] C. Veeramani, P.D. Minev and K. Nandakumar, *A Fictitious Domain Formulation for Flows with Rigid Particles: a non-Lagrange Multiplier Version*, J. Comp. Phys. **224** (2007), 867–879.

[3] R. Glowinski, T. Pan, T. Hesla, D. Joseph and J. Periaux, *A fictitious domain approach to the direct numerical simulation of incompressible viscous flow past moving rigid bodies: Application to particulate flow*, J. Comp. Phys. **169** (2001), 363–389.

[4] P.D. Minev and C.R. Ethier, *A semi-implicit projection algorithm for the Navier-Stokes equations with application to flows in complex geometries*, Notes on Numerical Fluid Mechanics **73** (1999), 223–231.

[5] A. Dechaume, W. Finlay and P.D. Minev, *Migration of an ellipsoidal particle in Hagen-Poiseuille flow*,submitted to Phys. Fluids.

On the steady free fall of an elastic body in a Navier-Stokes liquid

Mads Kyed

(joint work with Giovanni P. Galdi)

A ridgid body is said to perform a steady free fall in a Navier-Stokes liquid if, in a frame attached to the body, the motion of the liquid, as prescribed by the Navier-Stokes equations, is time independent. In [1] the existence of such steady free falls for ridgid bodies was shown for the first time. It is the purpose of this work to extend the notion of a steady free fall to an elastic (deformable) body and investigate the circumstances under which it can be performed.

When a body is falling freely in a liquid, under the influence of gravity, it may, in addition to translation, perform a rotation. If the body is elastic, it may furthermore deform due to the forces exerted on it by the fluid flowing past it. In our mathematical analysis of the problem, we consider the translation, rotation, deformation, and the motion of the liquid to be the unknowns. The gravitational force and the (stress free) shape of the body are known. We will assume the motion of the liquid is governed by the Navier-Stokes equations and that the elastic body is a St.Venant-Kirchoff material. Our main result is the existence of a steady free fall of an elastic body provided the effective mass is sufficiently small and the shape of the body satisfies a certain condtition.

The condition we need to impose on the (stress free) shape of the body is that of an isolated direction of fall. This condition was also imposed by Weinberger in [1]. In the ridgid body case, Serre proved in [2] that this condition is not necessary and that, in fact, any ridgid body can perform a steady free fall in a Navier-Stokes liquid. The proof of Serre exploits the possibility of formulating the free fall problem in a weak sense. Unfortunately, such a weak formulation is not compatible with the nonlinear elasticity equations of a St.Venant-Kirchoff material, which is the main reason we are not able to reproduce the result of Serre in the elastic body case. For further results on the ridgid body case we refer the reader to [3].

The mathematical study of the interaction between a Navier-Stokes liquid and elastic structures is relative new. For results in the steady-state case we refer the reader to [5, 4, 6]. All of these works are focused on a setting where the liquid is contained in a (bounded) container with elastic walls. Recently, the (exterior) flow of a Navier-Stokes liquid past an elastic body, fixed in space, has been studied in [7]. For results on similar unsteady problems we refer to [10, 11, 8, 9]. For applications of the mathematical study we refer to [12].

We denote by $\Omega \subset \mathbb{R}^3$ the domain occupied by an elastic body in a stress free configuration and assume that Ω is falling freely, under the influence of gravity, in a liquid occupying the exterior domain $\mathcal{E} := \mathbb{R}^3 \setminus \Omega$. We shall say that the elastic body can perform a steady free fall if there exists a deformation, $\chi : \Omega \to \mathbb{R}^3$, and a rotating frame of reference, $\mathcal{F}$, centered in the center of mass of the body, such that the equations of motion in $\mathcal{F}$ posses a steady solution. The equations of motion are constituted by the Navier-Stokes equations in the liquid domain and the non-linear elasticity equations in the domain occupied by the deformed body. Written in the frame $\mathcal{F}$, we arrive at the following coupled systems for the time independent equations:

$$(1) \qquad \begin{cases} \rho_E(\omega \wedge \omega \wedge y + \omega \wedge \xi) = \mathrm{div} T_E^\chi + \rho_E\, G & \text{in } \chi(\Omega) \\ T_E^\chi \cdot n^\chi = T_F \cdot n^\chi & \text{on } \partial\big(\chi(\Omega)\big) \end{cases}$$

$$(2) \qquad \begin{cases} \rho_F(\nabla v(v-U) + \omega \wedge v) = \mathrm{div} T_F & \text{in } \mathbb{R}^3 \setminus \big(\chi(\Omega)\big) \\ \mathrm{div}(v) = 0 & \text{in } \mathbb{R}^3 \setminus \big(\chi(\Omega)\big) \\ v = U := \xi + \omega \wedge y & \text{on } \partial\big(\chi(\Omega)\big) \\ \lim\limits_{|y| \to \infty} v = 0 & \end{cases}$$

$$(3) \qquad \big\{\, G \wedge \omega = 0 \quad , \quad |G| = 1 \,.$$

Here ρ_E and ρ_F denote the densities of the elastic body and fluid, respectively. T_E^χ denotes the Cauchy stress tensor of the elastic material and T_F the classical Cauchy stress tensor of a Navier-Stokes fluid. Furthermore, n^χ is the normal on $\partial\big(\chi(\Omega)\big)$, ω is the axis of rotation of $\mathcal{F}$, ξ the velocity of the center of mass of the body, and G denotes the gravitational force, the magnitude of which we have normalized to 1.

Note that we have neglected gravity as a body force in the liquid equations. We thereby also discard the hydrostatic pressure in $(1)_2$, which would be a time dependent force in any frame attached to the body and therefore have rendered the notion of a steady free fall meaningless for compressible bodies. Neglecting gravity in the liquid equations is physically reasonable when $\rho_F << \rho_E$. We note that in this setting the effective mass of the elastic body is equal to the absolute mass.
When initially dropped in a liquid, the elastic body may undergo some ridgid motion before attaining its terminal state. Thus the direction of the body is inherently unknown in the steady state free fall problem. Equivalent to letting the direction of the body be unknown, however, we treat the direction of the gravity as an unknown instead. Thus the unknowns in (1)-(3) are ω, ξ, G, χ, and (v, p). Our main result is indeed the existence of a solution $(\omega, \xi, G, \chi, v, p)$ of (1)-(3).
In order to solve (1)-(3) we first pull back the equations to the reference domain. For this purpose we extend χ to $\mathbb{R}^3$ such that, when restricted to $\mathcal{E}$, χ maps $\mathcal{E}$ onto $\mathbb{R}^3 \setminus \big(\chi(\Omega)\big)$. This enables us, by putting $w := v \circ \chi$ and $q := p \circ \chi$, to reformulate (1)-(3) as:

$$\text{(4)} \qquad \begin{cases} \rho_E^R(\omega \wedge \omega \wedge \chi_u + \omega \wedge \xi) = \mathrm{div} T_E + \rho_E^R\, G & \text{in } \Omega \\ T_E \cdot n = T_F^u \cdot n & \text{on } \partial\Omega \end{cases}$$

$$\text{(5)} \qquad \begin{cases} \rho_F(\nabla w\, A_u\, (w - U) + J_u\, \omega \wedge w) = \mathrm{div} T_F^u & \text{in } \mathcal{E} \\ \mathrm{div}(A_u w) = 0 & \text{in } \mathcal{E} \\ w = U := \xi + \omega \wedge \chi_u & \text{on } \partial\Omega \\ \lim\limits_{|x| \to \infty} w = 0 \end{cases}$$

$$\text{(6)} \qquad \big\{\, G \wedge \omega = 0 \quad , \quad |G| = 1 \,.$$

Here u denotes the displacement vector field corresponding to the deformation χ_u. T_E denotes the first Piola-Kirchoff stress tensor, the constitutive equations of which are given in terms of u by the assumption that the elastic material is of St.Venant-Kirchoff type. Moreover, $A_u := (\mathrm{cof} \nabla \chi_u)^\top$, $J_u := \det \nabla \chi_u$, $\rho_E^R := J_u \rho_E$, and $T_F^u := \big(\mu(\nabla w \nabla \chi_u^{-1} + \nabla \chi_u^{-\top} \nabla w^\top) - qI\big) A_u^\top$.
Problem (4) is at pure Neumann problem, which is only solvable when the corresponding compatibility conditions are satisfied. Denoting by $\mathcal{N}_E$ the nonlinear part of T_E, the compatibility conditions can be expressed as:

$$\text{(7)} \qquad \int_\Omega \rho_E^R(\omega \wedge \omega \wedge \chi_u + \omega \wedge \xi) - \rho_E^R G - \mathcal{N}_E \,\mathrm{d}x = \int_{\partial\Omega} \big(T_F^u - \mathcal{N}_E\big) \cdot n \,\mathrm{d}S$$

$$\text{(8)} \qquad \int_\Omega x \wedge \big(\rho_E^R(\omega \wedge \omega \wedge \chi_u + \omega \wedge \xi) - \rho_E^R G - \mathcal{N}_E\big) \,\mathrm{d}x = \int_{\partial\Omega} x \wedge \big(\big(T_F^u - \mathcal{N}_E\big) \cdot n\big) \,\mathrm{d}S \,.$$

Compared to the ridgid body case, the equations above replace the conservation of linear and angular momentum.

In our main proof of existence, we linearize the equations (4)-(8) around an isolated direction of fall. We are then able to prove existence of a solution, under a suitable smallness condition, using a fixed point approach.

References

[1] H.F. Weinberger, *On the steady fall of a body in a Navier-Stokes fluid*, Proc. Symp Pure Math. **23** (1973), 421–440.

[2] D. Serre, *Chute libre d'un solide dans un fluide visqueux incompressible. Existence*, Jap. J. Appl. Math. **4(1)** (1987), 99–110.

[3] G.P. Galdi, *On the Motion of a Rigid Body in a Viscous Liquid: A Mathematical Analysis with Applications*, Friedlander, S. (ed.) et al., Handbook of mathematical fluid dynamics. Vol. 1. Amsterdam: Elsevier, (2002), 653-791.

[4] C. Grandmont, *Existence for a three-dimensional steady-state fluid-structure interaction problem*, J. Math. Fluid Mech. **4(1)** (2002), 76-94.

[5] M. Rumpf, *On equilibria in the interaction of fluids and elastic solids*, Heywood, J. G. (ed.) et al., Theory of the Navier-Stokes equations. Proceedings of the third international conference on the Navier-Stokes equations: theory and numerical methods, Oberwolfach, Germany, June 5–11, 1994. Singapore: World Scientific. Ser. Adv. Math. Appl. Sci. 47, (1998), 136-158.

[6] C. Surulescu, *On the stationary interaction of a Navier-Stokes fluid with an elastic tube wall*, Appl. Anal. **86(2)** (2007), 149-165.

[7] G.P. Galdi and M. Kyed, *Steady Flow of a Navier-Stokes Liquid Past an Elastic Body*, (to appear) (2007).

[8] D. Coutand and S. Shkoller, *Motion of an elastic solid inside an incompressible viscous fluid*, Arch. Ration. Mech. Anal., **176(1)** (2005), 25-102.

[9] D. Coutand and S. Shkoller, *The interaction between quasilinear elastodynamics and the Navier-Stokes equations*, Arch. Ration. Mech. Anal., **179(3)** (2006), 303-352.

[10] C. Grandmont and Y. Maday, *Fluid-structure interaction: a theoretical point of view*, Fluid-structure interaction, Innov. Tech. Ser., Kogan Page Sci., London (2003), 1-22.

[11] A. Chambolle, B. Desjardins, M.J. Esteban and C. Grandmont, *Existence of weak solutions for the unsteady interaction of a viscous fluid with an elastic plate* J. Math. Fluid Mech., **7(3)** (2005), 368-404.

[12] G.P. Galdi, A.M. Robertson, R. Rannacher and S. Turek, *Hemodynamical Flows: Modeling, Analysis and Simulation*, Oberwolfach Seminar Series Vol. 35, Birkhäuser-Verlag, 2008.

Adaptive finite element simulation of fluid structure interaction

Rolf Rannacher

(joint work with Thomas Dunne)

The main issue of this talk is automatic mesh adaptation in the finite-element discretization of fluid-structure interaction problems. Four different types of mesh refinement/adaptation are considered:

- *uniform mesh refinement* using several steps of uniform (edge) bisection of a coarse initial mesh,
- *zonal mesh refinement* using a purely geometry-based criterion by marking all cells for refinement which have certain prescribed distances from the fluid-structure interface,

- *local mesh refinement* guided by qualitative smoothness properties (ZZ post-processing technique, energy-type error indicators, ...) where all cells are marked for refinement which have error indicators above a certain threshold.
- *local mesh refinement* driven by sensitivity-controled residual-based criteria (Dual Weighted Residual - DWR - method).

The DWR method (see Becker/Rannacher [4], [5], Bangerth/Rannacher [1]) provides a general framework for the derivation of "goal-oriented" a posteriori error estimates together with criteria of mesh adaptation for the Galerkin discretization of general linear and nonlinear variational problems, including optimization problems. The DWR method is based on a *variational* formulation of the problem. Time discretization can be treated by using a fully space-time Galerkin method. Starting point is an abstract semi-linear variational problem: *find* $U \in U^D + W^0$, *such that*

$$A(U)(\Psi) = F(\Psi) \quad \forall \Psi \in W^0,$$

and its (stabilized) Galerkin finite element approximation: *find* $U_h \in U_h^D + W_h$, *such that*

$$A(U_h)(\Psi_h) + STAB = F(\Psi) \quad \forall \Psi_h \in W_h^0.$$

For concrete discretizations of the Navier-Stokes equations fitting into this framework, we refer to Rannacher [11], Becker/Braack/Rannacher/Richter [2], and Braack/Richter [8]. Let the value $J(U)$ for some (linear) functional $J(\cdot)$ are to be computed. Then, the approximation is to be controlled in terms of the error

$$|J(U) - J(U_h)| = |J(U - U_h)| \leq TOL.$$

The existence of the directional derivative

$$A'(U)(\Phi, \Psi) := \lim_{\tau \to 0} \frac{1}{\tau} \big\{ A(U + \tau\Phi)(\Psi) - A(U)(\Psi) \big\}, \quad \Phi, \Psi \in W^0,$$

is assumed. We introduce the bilinear form

$$L(U, U_h)(\Phi, \Psi) := \int_0^1 A'(U_h + s(U - U_h))(\Phi, \Psi)\, ds,$$

and formulate the "dual problem"

$$L(U, U_h)(\Phi, Z) = J(\Phi) \quad \forall \Phi \in W^0.$$

The existence of a solution $Z \in W^0$ of the dual problem is assumed. Taking $\Phi = U - U_h \in W^0$ in the dual problem yields the error representation

$$\begin{aligned} J(U - U_h) &= L(U, U_h)(U - U_h, Z) \\ &= \int_0^1 A'(U_h + s(U - U_h))(U - U_h, Z)\, ds = A(U)(Z) - A(U_h)(Z). \end{aligned}$$

Further, using the perturbed Galerkin orthogonality property

$$A(U)(\Psi_h) - A(U_h)(\Psi_h) = \text{“}STAB\text{''}, \quad \Psi_h \in W_h^0,$$

we obtain

$$\begin{aligned} J(U-U_h) &= A(U)(Z-\Psi_h) - A(U_h)(Z-\Psi_h) - STAB \\ &= F(Z-\Psi_h) - A(U_h)(Z-\Psi_h) - STAB \\ &=: \rho(U_h)(Z-\Psi_h) - STAB = \sum_{K\in\mathbb{T}_h} \rho_K(U_h)(Z-\Psi_h) - `STAB', \end{aligned}$$

where $\Psi_h \in W_h^0$ is an arbitrary element. This leads us to the "theoretical" a posteriori error estimate

$$|J(U-U_h)| \approx \sum_{K\in\mathbb{T}_h} \rho_K(U_h)\,\omega_K(Z),$$

which involves the unknown dual solution Z. In order to obtain a "practical" error estimate the following steps are used:

- *Linearization of dual problem:*

$$L(U,U_h)(\Phi,\Psi) \approx L(U_h,U_h)(\Phi,\Psi) = A'(U_h)(\Phi,\Psi)$$

- *Discretization of dual problem:* discrete dual solution $Z_h \in W_h^0$

$$A'(U_h)(\Phi,Z_h) = J(\Phi_h) \quad \forall \Phi_h \in W_h^0.$$

- *Approximation of dual solution:* From Z_h, we generate improved approximations to Z in a post-processing step by patch-wise higher-order interpolation. On 2×2-patches of cells in $\mathbb{T}_h$ the 9 nodal values of the piecewise bilinear Z_h are used to construct a patch-wise biquadratic function $\tilde{Z}_h$.

This results in the "practical" a posteriori error estimate

$$|J(U-U_h)| \approx \eta := \sum_{K\in\mathbb{T}_h} \rho_K(U_h)\,\omega_K(\tilde{Z}_h).$$

On the basis of this error estimator the following algorithm of mesh adaptation is used: Let an error tolerance TOL be give.

(1) Compute the primal solution U_h on the current mesh, starting from some initial state, e.g., that with zero deformation.
(2) Compute the solution $\tilde{Z}_h$ of the approximate discrete dual problem.
(3) Evaluate the cell-error indicators $\eta_K := \rho_K(U_h)\,\omega_K(\tilde{Z}_h)$.
(4) If $\eta < TOL$ (the given tolerance) then accept U_h and evaluate $J(U_h)$, otherwise proceed to the next step.
(5) Determine the 30% cells with largest and the 10% cells with smallest values of η_K. The cells of the first group are refined and those of the second group coarsened. Then, continue with Step 1.

The application of the DWR method to the discretization of fluid-(rigid) structure interaction problems is described in Bönisch [6], Boenisch/Dunne/Rannacher [7] and for fluid-(elastic) structure interaction in Dunne/Rannacher [10], Dunne [9].

References

[1] W. Bangerth, R. Rannacher: *Adaptive Finite Element Methods for Differential Equations*, Lectures in Mathematics, ETH Zürich, Birkhäuser, Basel 2003.

[2] R. Becker, M. Braack, R. Rannacher, and Th. Richter: *Mesh and model adativity for flow problems*, Final Report of SFB 359 'Reactive Flow, Diffusion and Transport', University of Heidelberg (W. Jäger, R. Rannacher, J. Warnatz, eds), Springer, Berlin-Heidelberg, 2006.

[3] R. Becker, H. Kapp, R. Rannacher: *Adaptive finite element methods for optimal control of partial differential equations: basic concepts*, SIAM J. Optimization Control 39, 113-132 (2000).

[4] R. Becker, R. Rannacher: *A feed-back approach to error control in finite element methods: basic analysis and examples*, East-West J. Numer. Math. 4, 237-264 (1996).

[5] R. Becker, R. Rannacher: *An optimal control approach to error estimation and mesh adaptation in finite element methods*, Acta Numerica 2000 (A. Iserles, ed.), pp. 1-102, Cambridge University Press, 2001.

[6] S. Bönisch: *Adaptive Finite Element Methods for Rigid Particulate Flow Problems*, Dissertation, Institute of Applied Mathematics, University of Heidelberg, 2006.

[7] S. Bönisch, Th. Dunne, and R. Rannacher: *Lecture on numerical simulation of liquid-structure interaction*, in Lecture Notes of Oberwolfach Seminar "Hemodynamical Flows: Aspects of Modeling, Analysis and Simulation", Nov. 20-26, 2005, Mathematical Research Institute Oberwolfach (G.P" Galdi, R. Rannacher, A.M. Robertson, S. Turek, eds), Birkhäuser, Zürich, to appear 2007.

[8] M. Braack, Tgh. Richter: *Solving multidimensional reactive flow problems with adaptive finite elements*, Final Report of SFB 359 'Reactive Flow, Diffusion and Transport', University of Heidelberg (W. Jäger, R. Rannacher, J. Warnatz, eds), Springer, Berlin-Heidelberg, 2006.

[9] Th. Dunne: *Adaptive Finite Element Approximation of Fluid-Structure Interaction Based on Eulerian and Arbitrary Lagrangian-Eulerian Variational Formulations*, Dissertation, Institute of Applied Mathematics, University of Heidelberg, 2007.

[10] Th. Dunne, R. Rannacher: *Adaptive finite element approximation of fluid-structure interaction based on an Eulerian variational formulation*, in Fluid-Structure Interaction: Modelling, Simulation, Optimisation (H.-J. Bungartz and M. Schäfer, eds), Springer's LNCSE-Series, Vol. 53, Springer, Berlin-Heidelberg, 2006.

[11] R. Rannacher: *Finite element methods for the incompressible Navier-Stokes equations*, in Fundamental Directions in Mathematical Fluid Mechanics (P. Galdi, J. Heywood, and R. Rannacher, eds.), pp. 191-293, Birkhäuser, Basel, 2000.

On the existence of time-periodic motions of a rigid body in a Navier-Stokes liquid

Ana Leonor Silvestre

(joint work with Giovanni P. Galdi)

Consider a mechanical system $\{\mathcal{S}, \mathcal{L}\}$ constituted by a rigid body $\mathcal{S}$ moving in a Navier-Stokes liquid $\mathcal{L}$ that fills the three dimensional region exterior to $\mathcal{S}$.
The question that we would like to address is the following. Assume that the forces and torques acting on $\{\mathcal{S}, \mathcal{L}\}$ are periodic, of period T. Then, does the system $\{\mathcal{S}, \mathcal{L}\}$ execute a time-periodic motion of period T?
In order to make the fluid domain time-independent, the motion of $\{\mathcal{S}, \mathcal{L}\}$ will be described in a reference frame attached to the body. We will assume that the external forces and torques acting on $\mathcal{S}$ and $\mathcal{L}$ are given in an inertial frame, so

they will be unknown in the frame in which we study the motion of the system $\{\mathcal{S}, \mathcal{L}\}$. More precisely, we will assume that the force and torque acting on $\mathcal{S}$ are of the form

$$F_{\mathcal{S}}(t) = \mathbb{F}(t)\hat{a},$$
$$M_{\mathcal{S}}(t) = \mathbb{M}(t)\hat{b},$$

with $\hat{a},\ \hat{b} \in S^2$ and time-independent, and $\mathbb{F},\ \mathbb{M}$ $T-$periodic scalar functions. We will also assume that the external force acting on $\mathcal{L}$ is conservative. In this case, the motion of $\{\mathcal{S}, \mathcal{L}\}$ is described by the following equations (see [1])

$$\left.\begin{array}{l} \partial_t v = \nabla \cdot T(v,p) + (V - v) \cdot \nabla v - \omega \times v \\ \nabla \cdot v = 0 \end{array}\right\} \text{ in } \Omega \times \mathbb{R}$$
$$v = V \quad \text{at } \Sigma \times \mathbb{R}$$
$$\lim_{|x| \to \infty} v(x,t) = 0,\ t \in \mathbb{R}$$
$$m\frac{d\xi}{dt} + m\omega \times \xi = \mathbb{F}\hat{A} - \int_{\Sigma} T(v,p) \cdot n,\ t \in \mathbb{R} \tag{1}$$
$$I \cdot \frac{d\omega}{dt} + \omega \times (I \cdot \omega) = \mathbb{M}\hat{B} - \int_{\Sigma} x \times T(v,p) \cdot n,\ t \in \mathbb{R}$$
$$\frac{d\hat{A}}{dt} = \omega \times \hat{A},\ \frac{d\hat{B}}{dt} = \omega \times \hat{B},\ t \in \mathbb{R}$$

where (v, p) represents the velocity and pressure of $\mathcal{L}$, (ξ, ω) represents the velocity of the rigidy body, with $V(x,t) = \xi(t) + \omega(t) \times x$, and

$$\hat{A}(t),\ \hat{B}(t) \in S^2\,(t \in \mathbb{R}) \tag{2}$$

are the unknown directions of the force and the torque acting on $\mathcal{S}$, respectively. Before stating our results ([2]) on the existence of time-periodic solutions for the fluid-structure interaction problem (1), it is convenient to introduce appropriate spaces of test functions. We set

- $C^{\infty}_{T,per} := \{\varphi \in C^{\infty}([0,T]) : \varphi(0) = \varphi(T)\}$
- $\mathcal{C}_{T,per}(\Omega)$ the space formed by functions $\Phi \in C^{\infty}(\Omega \times [0,T])$, satisfying:
  (1) $\nabla \cdot \Phi = 0$, in $\Omega \times [0,T]$,
  (2) there exists $\Phi_1, \Phi_2 \in C^{\infty}([0,T])$ such that $\Phi(x,t) = \Phi_1(t) + \Phi_2(t) \times x$, for x in a neighborhood of Σ and $t \in [0,T]$,
  (3) for each Φ, there exists $R > \delta(\mathcal{S})$ (diameter of $\mathcal{S}$), such that $\Phi(x,t) = 0$, for $|x| \geq R$ and $t \in [0,T]$,
  (4) $\Phi(x,0) = \Phi(x,T)$, for all $x \in \Omega$.

Definition. We shall say that $(v, \xi, \omega, \hat{A}, \hat{B})$ is a $T-$periodic weak solution to system (1) if

(1) $D(v) \in L^2(0,T;L^2(\Omega))$ and $\nabla \cdot v = 0$,
(2) $\xi, \omega, \in L^2(0,T;\mathbb{R}^3)$, $\hat{A}, \hat{B} \in L^2(0,T;S^2)$,
(3) $(v, \xi, \omega, \hat{A}, \hat{B})$ satisfy

$$\int_0^T \left(\int_\Omega v \cdot \partial_t \Phi + m\xi \cdot \frac{d\Phi_1}{dt} + \omega \cdot I \cdot \frac{d\Phi_2}{dt} + 2\mu \int_\Omega D(v) : D(\Phi) + \right.$$
$$\left. + \int_0^T \left(\int_\Omega (v - V) \cdot \nabla v \cdot \Phi + \int_\Omega \omega \times v \cdot \Phi + m\xi \times \omega \cdot \Phi_1 + (I \cdot \omega) \times \omega \cdot \Phi_2 \right) \right.$$
$$= \int_0^T \left(\mathbb{F}\hat{A} \cdot \Phi_1 + \mathbb{M}\hat{B} \cdot \Phi_2 \right), \forall \Phi \in \mathcal{C}_{T,per}(\Omega),$$

and

$$\int_0^T \hat{A} \cdot \frac{d\varphi}{dt} + \int_0^T \omega \times \hat{A} \cdot \varphi = \int_0^T \hat{B} \cdot \frac{d\varphi}{dt} + \int_0^T \omega \times \hat{B} \cdot \varphi = 0, \forall \varphi \in C^\infty_{T,per}.$$

The main results presented in the talk can be summarized in the following theorems.

Theorem 1. Let $\Omega \subset \mathbb{R}^3$ be the domain exterior to a rigid body $\mathcal{S}$. Let $\mathbb{F}, \mathbb{M} \in C([0,T];\mathbb{R})$ with $\mathbb{F}(0) = \mathbb{F}(T)$ and $\mathbb{M}(0) = \mathbb{M}(T)$. Then there exists at least one $T-$periodic weak solution to the system $\{\mathcal{S}, \mathcal{L}\}$.

Theorem 2. Let $\Omega \subset \mathbb{R}^3$ be the domain exterior to a rigid body $\mathcal{S}$ with boundary of class C^2. Let $\mathbb{F}, \mathbb{M} \in C([0,T];\mathbb{R})$ with $\mathbb{F}(0) = \mathbb{F}(T)$ and $\mathbb{M}(0) = \mathbb{M}(T)$. There exists a positive constant $C_0 = C_0(\mathcal{S}, T)$ such that if

$$\|\mathbb{F}\|_\infty + \|\mathbb{M}\|_\infty \leq C_0$$

then there exists $(v, p, \xi, \omega, \hat{A}, \hat{B})$ such that

$$v \in C([0,T];L^2(\Omega_R)),\ \partial_t v \in L^2(0,T;L^2(\Omega_R))$$
$$\xi, \omega \in H^1(0,T;\mathbb{R}^3),\ \hat{A} \in C([0,T];S^2) \cap H^1(0,T;\mathbb{R}^3)$$
$$\nabla v \in L^2(0,T;H^1(\Omega)) \cap L^\infty(0,T;L^2(\Omega))$$
$$\nabla p \in L^2(0,T;L^2(\Omega))$$

and satisfying the system (1) a.e. with $v(0) = v(T)$, $\xi(0) = \xi(T)$, $\omega(0) = \omega(T)$, $\hat{A}(0) = \hat{A}(T)$, $\hat{B}(0) = \hat{B}(T)$.

The proofs, as in [1], are based on the invading domains technique. In each bounded domain the existence of a solution is established by Galerkin method. The main difficulty in this process is the fact that we have to find approximating solutions satisfying (2). An article describing the details of our approach will be submitted soon [2].

References

[1] G. P. Galdi and A. L. Silvestre, *Strong solutions to the problem of motion of a rigid body in a Navier-Stokes liquid under the action of prescribed forces and torques*, in Nonlinear Problems in Mathematical Physics and Related Topics I , Int. Math. Ser. (N. Y.), 1, Kluwer/Plenum, New York, 121-144 (2002).

[2] G. P. Galdi and A. L. Silvestre, *On the existence of time-periodic motions of a rigid body in a Navier-Stokes liquid*, in preparation.

Analysis and simulation of the motion of solids in a viscous fluid

Marius Tucsnak

(joint work with Jorge San Martin, Jean-François Scheid, Takéo Takahashi)

Understanding the motion of solids immersed in a fluid is a problem is a question which intrigued researchers for a long time. The main mathematical difficulty raised by this issue is that the partial differential equations of the fluid are coupled with the equations of the solid and that the domain field by the fluid is one of the unknowns of the problem. This means that we have to deal with a free boundary value problem.

In the case of rigid bodies immersed in a viscous incompressible fluid filling a bounded domain, several recent papers contributed to the mathematical analysis of the above mentioned difficulties (see, for instance, Desjardins and Esteban [1], San Martín, Starovoitov and Tucsnak [6], Feireisl [2] or Gunzburger, Lee and Seregin [5]). Let us also mention that in the case of a fluid-rigid system filling the whole space the existence of self-propelled motions has been investigated in Galdi [3] and [4].

In this talk we investigate the related problem of self-propelling of solids in a viscous incompressible fluid. The model consists in a solid (called *the creature*) undergoing an undulatory deformation, which is immersed in a viscous incompressible fluid. The displacement of the creature is decomposed into a rigid part and a deformation (undulatory) part. The rigid part of the displacement results from the interaction of the fluid and the solid, whereas the deformation part is given. Since our aim is to possibly consider several creatures and to tackle the case of a bounded fluid-body system, the domain filled by the fluid is one of the unknowns.

Our main theoretical result asserts that the initial and boundary value problem obtained by coupling the Navier-Stokes equations for the fluid to Newton's law for the creature is well-posed in Sobolev type spaces. Our method is based on a change of variables introduced in Takahashi [9] combined to an appropriate lifting of the non rigid part of the velocity field and to some a priori estimates. Our method for proving the existence result can be extended to the case of several immersed bodies.

The second contribution brought in by this work consists in giving a weak formulation (of mixed type) and an approximation scheme for the governing equations.

This scheme is finally tested on some undulatory motions observed by the zoologists in order to get straight-line-swimming or turning. Moreover, we take advantage of the fact that our method still works if we have several immersed bodies, in order to simulate the simultaneous swimming of two fish-like creatures. The results described in this talk have been proved in [7] and in [8].

References

[1] B. Desjardins and M. J. Esteban, *Existence of weak solutions for the motion of rigid bodies in a viscous fluid*, Arch.Rational Mech.Anal. **146** (1999), 59–71.

[2] E. Feireisl, *On the motion of rigid bodies in a viscous incompressible fluid.*, J. Evol. Equ. **3** (2003), 419–441.

[3] G. Galdi, *On the steady self-propelled motion of a body in a viscous incompressible fluid*, Arch. Ration. Mech. Anal. **148** (1999), 53–88.

[4] G. P. Galdi, *On the motion of a rigid body in a viscous liquid: a mathematical analysis with applications*, in Handbook of mathematical fluid dynamics, Vol. I, North-Holland, Amsterdam, 2002, 653–791.

[5] M. Gunzburger, H.-C. Lee, and G. Seregin, *Global existence of weak solutions for viscous incompressible flows around a moving rigid body in three dimensions*, J. Math. Fluid Mech. **2** (2000), 219–266.

[6] J. San Martín, V. Starovoitov, and M. Tucsnak, *Global weak solutions for the two-dimensional motion of several rigid bodies in an incompressible viscous fluid*, Arch. Ration. Mech. Anal. **161** (2002), 113–147.

[7] J. San Martín, J.-F Scjeid, T. Takahashi, and M. Tucsnak, *An Initial and Boundary Value Problem Modeling*
Fish-like Swimming, Arch. Ration. Mech. Anal., to appear.

[8] J. San Martín, J.-F. Scheid, T. Takahashi, and M. Tucsnak, *Convergence of the Lagrange-Galerkin method for the equations modelling the motion of a fluid-rigid system*, SIAM J. Numer. Anal. **43** (2005), 1536–1571.

[9] T. Takahashi, *Analysis of strong solutions for the equations modeling the motion of a rigid-fluid system in a bounded domain*, Adv. Differential Equations **8** (2003), 1499–1532.

Benchmarking for fluid-structure interaction with elastic bodies

Stefan Turek

(joint work with Jaroslav Hron)

The main purpose of this benchmark presentation is to describe specific configurations which shall help in future to test and to compare different numerical methods and code implementations for the fluid-structure interaction (FSI) problem. In particular, the various coupling mechanisms, ranging from partitioned, weakly coupled approaches to fully coupled, monolithic schemes are of high interest. Moreover, it shall be possible to examine the quality of different discretization schemes (FEM, FV, FD, LBM, resp., beam, shell, volume elements), and the robustness and numerical efficiency of the integrated solver components shall be a further aspect. This new benchmark is based on the older successful *flow around cylinder* setting developed in [2] for incompressible laminar flow. Similar to these older configurations we consider the fluid to be incompressible and in the laminar regime. The structure is allowed to be compressible, and the deformations of the

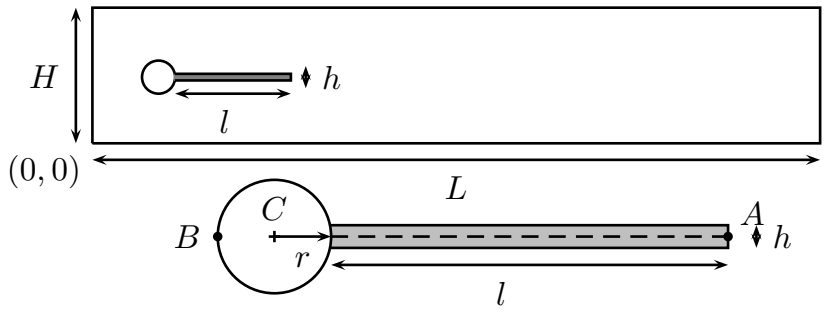

FIGURE 1. Computational domain with the detail of the structure part.

structure should be significant. The overall setup of the interaction problem is such that the solid object with elastic part is submerged in a channel flow. Then, self induced oscillations in the fluid and the deformable part of the structure are obtained so that characteristic physical quantities and plots for the time-dependent results can be provided.

We consider the flow of an **incompressible Newtonian fluid** interacting with an **elastic solid**. We denote by Ω_t^f the domain occupied by the fluid and Ω_t^s by the solid at the time $t \in [0, T]$. Let $\Gamma_t^0 = \bar{\Omega}_t^f \cap \bar{\Omega}_t^s$ be the part of the boundary where the elastic solid interacts with the fluid. The fluid is considered to be **Newtonian**, **incompressible** and its state is described by the velocity and pressure fields $\mathbf{v}^f, p^f$. The constant density of the fluid is ϱ^f and the viscosity is denoted by ν^f. The Reynolds number is defined by $\Re = \frac{2r\bar{V}}{\nu^f}$, with the mean velocity $\bar{V} = \frac{2}{3}v(0, \frac{H}{2}, t)$, r radius of the cylinder and H height of the channel (see Fig. 1).
The structure is assumed to be **elastic** and **compressible**. Its configuration is described by the displacement $\mathbf{u}^s$, with velocity field $\mathbf{v}^s = \frac{\partial \mathbf{u}^s}{\partial t}$. The material is specified by giving the Cauchy stress tensor $\boldsymbol{\sigma}^s$ (the 2nd Piola-Kirchhoff stress tensor is then given by $\boldsymbol{S}^s = J\boldsymbol{F}^{-1}\boldsymbol{\sigma}^s\boldsymbol{F}^{-T}$) by the following constitutive law for the **St. Venant-Kirchhoff** material ($\boldsymbol{E} = \frac{1}{2}(\boldsymbol{F}^{\mathrm{T}}\boldsymbol{F} - \boldsymbol{I})$)

$$\boldsymbol{\sigma}^s = \frac{1}{J}\boldsymbol{F}\left(\lambda^s(\operatorname{tr}\boldsymbol{E})\boldsymbol{I} + 2\mu^s\boldsymbol{E}\right)\boldsymbol{F}^{\mathrm{T}}, \qquad \boldsymbol{S}^s = \lambda^s(\operatorname{tr}\boldsymbol{E})\boldsymbol{I} + 2\mu^s\boldsymbol{E} \tag{1}$$

The boundary conditions on the fluid solid interface are assumed to be

$$\boldsymbol{\sigma}^f\mathbf{n} = \boldsymbol{\sigma}^s\mathbf{n}, \quad \mathbf{v}^f = \mathbf{v}^s \qquad \text{on } \Gamma_t^0, \tag{2}$$

where $\mathbf{n}$ is a unit normal vector to the interface Γ_t^0. This implies the no-slip condition for the flow, and that the forces on the interface are in balance.
The domain is based on the 2D version of the well-known CFD benchmark in [2] and shown here in figure 1. By omitting the elastic bar behind the cylinder one can exactly recover the setup of the *flow around cylinder* configuration which allows for validation of the flow part by comparing the results with the older flow benchmark. The setting is intentionally non-symmetric [2] to prevent the dependence

		value [m]
channel length	L	2.5
channel width	H	0.41
cylinder center	C	$(0.2, 0.2)$
cylinder radius	r	0.05

		value [m]
elastic structure length	l	0.35
elastic structure thickness	h	0.02
reference point (at $t = 0$)	A	$(0.6, 0.2)$
reference point	B	$(0.2, 0.2)$

TABLE 1. Overview of the geometry parameters

parameter	FSI1	FSI2	FSI3
ϱ^s $[10^3 \frac{\text{kg}}{\text{m}^3}]$	1	10	1
ν^s	0.4	0.4	0.4
μ^s $[10^6 \frac{\text{kg}}{\text{ms}^2}]$	0.5	0.5	2.0
ϱ^f $[10^3 \frac{\text{kg}}{\text{m}^3}]$	1	1	1
ν^f $[10^{-3} \frac{\text{m}^2}{\text{s}}]$	1	1	1
U $[\frac{\text{m}}{\text{s}}]$	0.2	1	2

parameter	FSI1	FSI2	FSI3
$\beta = \frac{\varrho^s}{\varrho^f}$	1	10	1
ν^s	0.4	0.4	0.4
$\text{Ae} = \frac{E^s}{\varrho^f \bar{U}^2}$	3.5×10^4	1.4×10^3	1.4×10^3
$\Re = \frac{Ud}{\nu^f}$	20	100	200
$\bar{U}$	0.2	1	2

TABLE 2. Parameter settings for the full FSI benchmarks.

of the onset of any possible oscillation on the precision of the computation.

A parabolic velocity profile is prescribed at the left channel inflow

$$v^f(0, y) = 1.5\bar{U}\frac{y(H-y)}{\left(\frac{H}{2}\right)^2} = 1.5\bar{U}\frac{4.0}{0.1681}y(0.41-y), \tag{3}$$

such that the mean inflow velocity is $\bar{U}$ and the maximum of the inflow velocity profile is $1.5\bar{U}$. The *no-slip* condition is prescribed for the fluid on the other boundary parts. i.e. top and bottom wall, circle and fluid-structure interface Γ_t^0.

The outflow condition can be chosen by the user, for example *stress free* or *do nothing* conditions. The outflow condition effectively prescribes some reference value for the pressure variable p. While this value could be arbitrarily set in the incompressible case, in the case of compressible structure this will have influence on the stress and consequently the deformation of the solid. In this proposal, we

set the reference pressure at the outflow to have *zero mean value*.

Suggested starting procedure for the non-steady tests is to use a smooth increase of the velocity profile in time as

$$v^f(t,0,y) = \begin{cases} v^f(0,y)\frac{1-\cos(\frac{\pi}{2}t)}{2} & \text{if } t < 2.0 \\ v^f(0,y) & \text{otherwise} \end{cases} \tag{4}$$

where $v^f(0,y)$ is the velocity profile given in (3).

References

[1] S. Turek and J. Hron. Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow. In H.-J. Bungartz and M. Schäfer, editors, *Fluid-Structure Interaction: Modelling, Simulation, Optimisation*, LNCSE. Springer, 2006.

[2] S. Turek and M. Schäfer. Benchmark computations of laminar flow around cylinder. In E.H. Hirschel, editor, *Flow Simulation with High-Performance Computers II*, volume 52 of *Notes on Numerical Fluid Mechanics*. Vieweg, 1996. co. F. Durst, E. Krause, R. Rannacher.

High performance computing techniques for the FEM simulation in solid mechanics

Hilmar Wobker

(joint work with Stefan Turek)

For large scale solid mechanical simulations iterative solving methods are mandatory. In contrast to direct methods, the efficiency of iterative schemes is crucially influenced by different factors:

- physical parameters (e. g. material constants)
- algorithmic parameters (e. g. time step size)
- quality of the underlying computational mesh (anisotropies)
- number of processors in a parallel computing system

We distinguish between three aspects of 'efficiency': The ability of the solving algorithm to exploit the processor's computational power is called the *processor efficiency*, the ratio between computation and communication times describes the *parallel efficiency*, and the convergence behaviour, i. e. the number of iterations needed to achieve a prescribed tolerance, determines the *numerical efficiency*. Our aim is to develop by means of the FEM software package Feast (Finite Element Analysis and Solution Tools) a solver mechanism which at the same time gains high efficiency in all three aspects, while trying to minimise the mentioned influences. Based on the insight that in modern computer architectures not *data processing*, but *data moving* is costly [7], a major drawback of many FEM codes has to be seen in the usage of the standard sparse matrix storage technique. Working with highly adaptive and unstructured meshes results in global matrices that lack the possibility of direct data access. Thus the usage of pointer structures becomes necessary, preventing the employment of cache-oriented techniques and, therefore,

the achievement of satisfactory MFLOP/s rates. What follows is that, for instance, matrix vector multiplications – one of the core components of many computational routines – are often executed with less than one percent of the theoretically possible MFLOP/s rate of the processor. Multigrid solvers perform even more slowly, thus often losing the linear dependency of computing time on the problem size (see [9] for more details). One of the main ideas behind FEAST is to use *locally structured meshes*. To combine this with an adaptive meshing concept, a *patch-oriented* approach is applied: The geometry is described by a coarse mesh consisting of patches which we call *'macros'*.

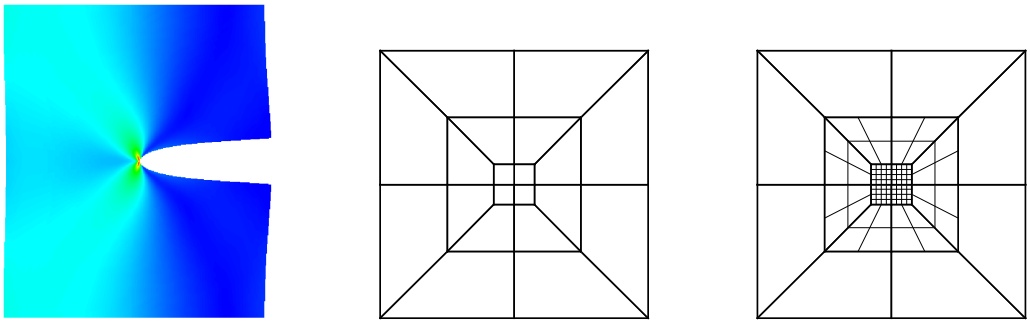

FIGURE 1. Crack opening configuration (von Mises stress, coarse mesh consisting of 20 patches and adaptively refined coarse mesh)

For example, Figure 1 (center) shows the initial coarse mesh for the well known crack opening configuration (left). Driven by automatic or user-guided control mechanisms these macros are then refined in such a way that two conditions are met: every macro describes a *generalized tensor product mesh*, and the refinement levels of two neighbouring macros differ by at most one (see Figure 1, right). Furthermore, in each patch anisotropic refinement and mesh deformation techniques can be applied to obtain more flexible adaptive designs and resolve singularities as in the crack tip (see Figure 2).

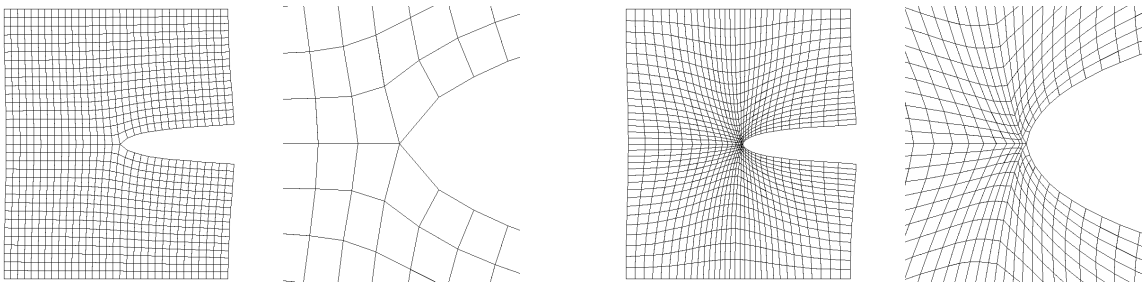

FIGURE 2. Crack opening mesh in non-deformed (left) and deformed state (right)

The crucial benefit of using generalized tensor product patches is the resulting band structure of the local matrices (e. g., exactly nine bands for bilinear elements when

rowwise numbering is applied). Knowing this structure a priori, highly optimized linear algebra routines can be developed, which has been realized by means of the SparseBandedBlas package. Consequently, basic operations like matrix vector multiplications can be performed very efficiently. Furthermore, highly effective, linewise working Gauß-Seidel smoothers (TRIGS, ADITRIGS) can be applied. In the enhanced BlockedBanded version of the SparseBandedBlas package the matrices are further divided into blocks such that the cache of the underlying processor architecture can be optimally exploited. This way much higher MFLOP/s rates than with the standard sparse techniques can be achieved [9].
In connection to this patch-based approach Feast introduces a sophisticated parallelization concept. A main problem of parallelizing a numerical algorithm is that *numerical efficiency* and *parallel efficiency* are often two contrary properties: To achieve good convergence rates *global information* is needed, while *locality* is preferred to minimize communication. Two major concepts for parallelising iterative solvers are *domain decomposition techniques (Schwarz-CG-methods)* and *blockwise smoothed multigrid methods*. Feast uses a generalized MG/DD concept which tries to connect the advantages of the two approaches while, at the same time, avoiding their flaws (e. g. sensitivity to mesh anisotropies, poor ratio between communication and computation time). The basic idea is to apply a *global* multigrid algorithm which is recursively smoothed by *local* multigrids. This concept, called 'ScaRC', is distributed over some hierarchies to optimally exploit the structures of the underlying mesh (see [1] for more details). On the lowest level are the generalized tensor product patches, where mesh-optimized smoothers can be applied. This way mesh irregularities like strong anisotropies can be recursively hidden which helps to minimize the number of global multigrid iterations and, consequently, the amount of global communication. Additionally, by using only tensor product meshes on patch-level the single processors are optimally exploited. Another essential advantage of this modular solving concept is the possibility to schedule the local subtasks to scientific co-processors like graphics cards. By exploiting their much higher memory bandwith one can gain significant speedups compared to pure CPU computations (see [3] for details).
The described Feast concepts are laid out and specialized to efficiently and robustly solve scalar elliptic equations [5]. Problems arising in Computational Solid Mechanics (CSM) are, in general, not scalar, such that Feast is not directly applicable. The remedy is to bring the process of solving vector-valued problems down to the solution of scalar equations. This has been realized by means of the module FeastSolid, a CSM software package based on Feast. The concept shall be shortly illustrated with the prototypical example of 2D linear elasticity. Applying separate displacement ordering to the discretized Lamé equation, leads to a block structured linear equation system

$$\begin{pmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} \\ \mathbf{K}_{21} & \mathbf{K}_{22} \end{pmatrix} \begin{pmatrix} \mathbf{u}_1 \\ \mathbf{u}_2 \end{pmatrix} = \begin{pmatrix} \mathbf{f}_1 \\ \mathbf{f}_2 \end{pmatrix}.$$

This system can be solved by applying a preconditioned Krylov space method (e. g. BiCGstab) using a block Gauss-Seidel preconditioner $\left(\begin{smallmatrix} \mathbf{K}_{11} & \mathbf{0} \\ \mathbf{K}_{21} & \mathbf{K}_{22} \end{smallmatrix}\right)^{-1}$. Since

the blocks on the diagonal correspond to discretizations of scalar elliptic equations, they can be treated by FEAST. Thus the efficiency and robustness of the ScaRC solvers can be transferred to the outer Krylov space iteration. Also the benefit from using GPUs for solving the local scalar subproblems carries over to this more complex solver setting and significantly shortens total runtimes (see [4] for more details).

In many applications incompressibility effects occur, e. g. when working with rubber-like materials or when plastic deformation takes place. It is well known that a FE approximation basing on the standard displacement formulation deteriorates due to a degenerating parameter – a phenomenon widely referred to as *volume locking*. FEASTSOLID overcomes this problem by employing the mixed displacement/pressure formulation resulting in equation systems having the well known saddle point structure

$$\begin{pmatrix} \mathbf{K} & \mathbf{B} \\ \mathbf{B}^T & \mathbf{C} \end{pmatrix} \begin{pmatrix} \mathbf{u} \\ \mathbf{p} \end{pmatrix} = \begin{pmatrix} \mathbf{f} \\ \mathbf{g} \end{pmatrix}.$$

Here, the matrix block $\mathbf{C}$ results from compressibility and stabilization terms. The latter occur when an unstable element combination like Q_1/Q_1 has to be stabilized, e. g., by means of a least-squares technique [2]. Similar saddle point systems arise from the discretization of the incompressible Navier-Stokes equation in the field of Computational Fluid Dynamics (CFD) where much effort has been spent on developing efficient solving procedures. Two prominent solution strategies have been adapted within FEASTSOLID: the pressure Schur complement technique [8] and a block-preconditioning approach [6]. Both of them crucially depend on having an efficient preconditioner for the Schur complement $\mathbf{S} := \mathbf{B}^T\mathbf{K}^{-1}\mathbf{B} - \mathbf{C}$ of the above system. While for the stationary case the pressure mass matrix is known to be an optimal preconditioner [6], the non-stationary case is treated with the technique developed in [8] for the incompressible Navier-Stokes equation. The idea is to distinctly consider the two parts $\mathbf{B}^T\mathbf{K}^{-1}\mathbf{B}$ and $\mathbf{B}^T\mathbf{M}_u^{-1}\mathbf{B}$ of the Schur complement, where $\mathbf{M}_u$ denotes the displacement mass matrix. The first part is treated just as in the stationary case, while the second part can be interpreted as discretized mixed formulation of the Poisson equation and thus can be efficiently preconditioned by the discrete pressure Poisson operator. The final preconditioner is then defined as a suitable linear combination of the two parts, which incorporates the shear modulus and the time step size. The result is a Schur complement preconditioner which is efficient for the whole range of relevant time step sizes. Applying the nonstationary preconditioner involves solving a scalar equation which is, again, efficiently done by FEAST's ScaRC solvers.

References

[1] Ch. Becker, *Strategien und Methoden zur Ausnutzung der High-Performance-Ressourcen moderner Rechnerarchitekturen für Finite-Element-Simulationen und ihre Realisierung in* FEAST *(Finite Element Analysis & Solution Tools)*, PhD thesis, Logos Verlag Berlin (2007)

[2] L.P. Franca, R. Stenberg, *Error analysis of some Galerkin least squares methods for the elasticity equations*, SIAM J. Numer. Anal. **28** (1991)

[3] D. Göddeke, R. Strzodka, J. Mohd-Yusof, P. McCormick, H. Wobker, Ch. Becker and S. Turek, *Using GPUs to Improve Multigrid Solver Performance on a Cluster*, accepted for publication in the International Journal of Computational Science and Engineering (2008)
[4] D. Göddeke, H. Wobker, R. Strzodka, J. Mohd-Yusof, P. McCormick and S. Turek, *Co-processor acceleration of an unmodified parallel structural mechanics code with FEAST-GPU*, Supercomputing Posters (2007)
[5] S. Kilian, *ScaRC als verallgemeinerter Mehrgitter– und Gebietszerlegungsansatz für parallele Rechnerplattformen*, PhD Thesis, Logos Verlag Berlin (2002)
[6] A. Klawonn, *An optimal preconditioner for a class of saddle point problems with a penalty term*, SIAM J. Sci. Comput. **18** (1997)
[7] U. Rüde, *Technological trends and their impact on the future of supercomputers*, High Performance Scientific and Engineering Computing **8** (eds. H.-J. Bungartz, F. Durst and Ch. Zenger), Springer Berlin (1999)
[8] S. Turek, *Efficient Solvers for Incompressible Flow Problems: An Algorithmic and Computational Approach*, Springer Berlin (1999)
[9] S. Turek, A. Runge and Ch. Becker, *The FEAST Indices – Realistic evaluation of modern software components and processor technologies*, Computers and Mathematics with Applications **41** (2001), 1431–1464

Reporter: Rolf Rannacher

Participants

M.Sc. Dan Anca
Fachbereich Mathematik
Technische Universität Dortmund
Vogelpothsweg 87
44227 Dortmund

Prof. Dr. Andrew L. Belmonte
Department of Mathematics
Penn State University
322 McAllister Building
University Park PA 16802
USA

Thomas Dunne
Institut für Angewandte Mathematik
Universität Heidelberg
Im Neuenheimer Feld 294
69120 Heidelberg

Prof. Dr. Matthias Heil
Department of Mathematics
The University of Manchester
Oxford Road
GB-Manchester M13 9PL

Prof. Dr. Vincent Heuveline
Institut f. Angewandte und Numerische
Mathematik - Universität Karlsuhe (TH)
-Rechenzentrum-
Zirkel 2
76128 Karlsruhe

Dr. Jaroslav Hron
Institut für Angewandte Mathematik
Technische Universität Dortmund
Vogelpothsweg 87
44227 Dortmund

Prof. Dr. Howard H. Hu
Dept. of Mechanical Engineering
University of Pennsylvania
220 S. 33rd St.
Philadelphia , PA 19104-6315
USA

Dipl.-Phys. Stefan Knauf
Institut für Angewandte Mathematik
Universität Heidelberg
Im Neuenheimer Feld 294
69120 Heidelberg

Dr. Mads Kyed
Institut
für Mathematik
RWTH Aachen
Templergraben 55
52062 Aachen

Prof. Dr. Peter D. Minev
Department of Mathematical and
Statistical Sciences
University of Alberta
677 Central Academic Building
Edmonton T6G 2G1
CANADA

Prof. Dr. Rolf Rannacher
Institut für Angewandte Mathematik
Universität Heidelberg
Im Neuenheimer Feld 294
69120 Heidelberg

Prof. Dr. Ana Leonor Silvestre
Departamento de Matematica
Instituto Superior Tecnico
Avenida Rovisco Pais, 1
Lisboa 1049-001
PORTUGAL

Prof. Dr. Marius Tucsnak
Institut Elie Cartan
-Mathematiques-
Universite Henri Poincare, Nancy I
Boite Postale 239
F-54506 Vandoeuvre les Nancy Cedex

Dr. Stefan Turek
Institut für Angewandte Mathematik
Technische Universität Dortmund
Vogelpothsweg 87
44227 Dortmund

Hilmar Wobker
Institut für Angewandte Mathematik
Technische Universität Dortmund
Vogelpothsweg 87
44227 Dortmund