International Workshop on Multi-Phase Flows: Analysis, Modelling and Numerics

November 27–30, 2018
Waseda University, Tokyo, Japan

Abstracts

Main-Course Lectures:

Invited Lectures (50 minutes):

Short Presentations (20 minutes):

早稲田大学 WASEDA University

TOP GLOBAL UNIVERSITY JAPAN

科研費 KAKENHI

Yuri BAZILEVS

Brown University, Providence, USA

① An Introduction to VMS and IGA for Incompressible Flows

The Variational Multiscale (VMS) concept emerged in the late 90s from the group of T.J.R Hughes as a new paradigm for multiscale modeling. The first connection to turbulence modeling was made shortly thereafter. Isogeometric Analysis came on the scene in the mid 2000's, also from the group of T.J.R. Hughes, and was largely motivated by establishing a better link between FEM and CAD. Shortly thereafter, a residual-based VMS concept was developed and combined with IGA to produce a powerful framework for the simulation of turbulent flows. Space-Time versions of VMS were developed more recently by T.E. Tezduyar and K. Takizawa, and are routinely employed, in conjunction with IGA, to carry out some of the most impressive CFD and FSI simulations. This presentation will introduce VMS and IGA, and show earlier turbulent-flow computations. The presentation will conclude with extensions of the framework to two-fluid flows, and show free-surface flow calculations at small and large spatial scales.

② Compressible Flow and IGA-Meshfree Coupling for Air-Blast FSI

This presentation will focus on stabilized methods for compressible flows showing the underlying theory and recent simulations. The presentation will then shift gears to cover the integration of the developed compressible-flow formulation into a new framework for air-blast FSI using an immersed approach coupling IGA and Meshfree methods.

Date

① NOV 27 (TUE) 12:00–12:50 ② NOV 28 (WED) 12:00–12:50

Raphaël DANCHIN
LAMA, Université Paris-Est Créteil, Paris, France

Fourier analysis, Littlewood-Paley decomposition and compressible Navier-Stokes equations

Since the end of the 80ies, Fourier analysis methods have known a growing importance in the study of linear and nonlinear PDE's, and, in particular, for investigating evolutionary fluid mechanics equations. The present lectures overview some recent results based on Fourier analysis and paradifferential calculus, for the compressible Navier-Stokes equations. Focus is on the well-posedness theory for the Initial Value Problem in the case where the fluid domain is $\mathbb{R}^d$ with $d \geq 2$, and on some asymptotic properties of global small solutions. The tentative plan of the lectures is as follows:

1. First lecture : basic Fourier analysis, Besov spaces and product laws.
2. Second lecture : endpoint maximal regularity estimates for the heat equation and Besov estimates for the transport equation.
3. Third lecture : Solving the compressible Navier-Stokes equations : local-in-time existence and uniqueness (Eulerian coordinates versus Lagrangian coordinates).
4. Fourth lecture : Global-in-time results and time decay estimates.

Date

① NOV 27 (TUE) 10:00–10:50 ② NOV 28 (WED) 10:00–10:50
③ NOV 29 (THU) 10:00–10:50 ④ NOV 30 (FRI) 10:00–10:50

Yasunori MAEKAWA
Kyoto University, Kyoto, Japan

Stability of physically reasonable solutions in two dimensions and related topics

The flow past an obstacle is a fundamental object in fluid mechanics. In 1967 R. Finn and D. R. Smith proved the unique existence of stationary solutions, called the physically reasonable solutions, to the Navier-Stokes equations in a two-dimensional exterior domain modeling this type of flows when the Reynolds number is sufficiently small. The asymptotic behavior of their solution at spatial infinity has been studied in details and well understood by now, while its stability has remained open due to the difficulty specific to the two-dimensionality. In this lecture we prove that the physically reasonable solutions constructed by R. Finn and D. R. Smith are asymptotically stable with respect to small and well-localized initial perturbations. In particular, we discuss the starting problem proposed by R. Finn, which studies the convergence of the Navier-Stokes flows past an obstacle that is started from the rest and was solved in the three dimensional case by G. P. Galdi, J. G. Heywood, and Y. Shibata in 1997. A special focus in the two dimensional case will be given on the condition how to accelerate the obstacle.

Date

① NOV 29 (THU) 11:00–11:50 ② NOV 29 (THU) 14:30–15:20
③ NOV 30 (FRI) 11:00–11:50 ④ NOV 30 (FRI) 14:30–15:20

Kenji TAKIZAWA[1] and Tayfun E. TEZDUYAR[2,1]
[1] Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
[2] Rice University, Houston, Texas, USA

① Introduction to Space–Time Computational Flow Analysis and Mesh Update Methods

The Deforming-Spatial-Domain/Stabilized ST (DSD/SST) method [1] was introduced for computation of flows with moving boundaries and interfaces (MBI), including fluid–structure interactions (FSI). In MBI computations the DSD/SST functions as a moving-mesh method. Moving the fluid mechanics mesh to track an interface enables mesh-resolution control near the interface and, consequently, high-resolution boundary-layer representation near fluid–solid interfaces. The DSD/SST method is an alternative to the Arbitrary Lagrangian–Eulerian (ALE) method, which is an older and more commonly used moving-mesh method. Because of its stabilization components "SUPG" and "PSPG," the DSD/SST method is now also called "ST-SUPS." The ST Variational Multiscale (ST-VMS) method [2] is the VMS version of the DSD/SST method. The VMS components of the ST-VMS are from the residual-based VMS (RBVMS) method [3]. Moving-mesh methods require mesh update methods. Mesh update typically consists of moving the mesh for as long as possible and remeshing as needed. With the key objectives being to maintain the element quality near solid surfaces and to minimize the frequency of remeshing, a number of advanced mesh update methods [4] were developed to be used with the ST-SUPS method, including those that minimize the deformation of the layers of small elements placed near solid surfaces. Some of these methods have also been used with the ALE-VMS method. The advanced mesh update methods developed more recently [5, 6, 7] have been used mostly with the ST-VMS method, and some of the methods are unique to the ST framework.

References

[1] T. E. Tezduyar, "Stabilized Finite Element Formulations for Incompressible Flow Computations", *Advances in Applied Mechanics*, **28** (1992) 1–44.

[2] K. Takizawa and T. E. Tezduyar, "Multiscale Space–Time Fluid–Structure Interaction Techniques", *Computational Mechanics*, **48** (2011) 247–267.

[3] Y. Bazilevs, V. M. Calo, J. A. Cottrell, T. J. R. Hughes, A. Reali, and G. Scovazzi, "Variational multiscale residual-based turbulence modeling for large eddy simulation of incompressible flows", *Computer Methods in Applied Mechanics and Engineering*, **197** (2007) 173–201.

[4] T. Tezduyar, S. Aliabadi, M. Behr, A. Johnson, and S. Mittal, "Parallel Finite-Element Computation of 3D Flows", *Computer*, **26** (10) (1993) 27–36.

[5] K. Takizawa, B. Henicke, A. Puntel, N. Kostov, and T. E. Tezduyar, "Space–Time Techniques for Computational Aerodynamics Modeling of Flapping Wings of an Actual Locust", *Computational Mechanics*, **50** (2012) 743–760.

[6] K. Takizawa, T. E. Tezduyar, J. Boben, N. Kostov, C. Boswell, and A. Buscher, "Fluid–structure interaction modeling of clusters of spacecraft parachutes with modified geometric porosity", *Computational Mechanics*, **52** (2013) 1351–1364.

[7] K. Takizawa, T. E. Tezduyar, A. Buscher, and S. Asada, "Space–Time Interface-Tracking with Topology Change (ST-TC)", *Computational Mechanics*, **54** (2014) 955–971.

Date

① NOV 27 (TUE) 14:30–15:20

② Space–Time Computational Flow Analysis with Isogeometric Discretization and Special Space–Time Methods

We present a space–time (ST) computational method that brings together three ST methods in the framework of the ST-VMS [1] method: the ST Slip Interface (ST-SI) [2] and ST Topology Change (ST-TC) [3] methods and ST Isogeometric Analysis (ST-IGA) [4]. The integration of these methods enables computational analysis in the presence of multiple challenges. The challenges include accurate representation of boundary layers near moving solid surfaces even when they come into contact, and handling a high level of geometric complexity. The ST-VMS, as a moving-mesh method, maintains high-resolution boundary layer representation near solid surfaces. The ST-TC method enables moving-mesh computation of flow problems with contact between moving solid surfaces or other TC, maintaining high-resolution representation near the solid surfaces. The ST-SI method was introduced for high-resolution representation of the boundary layers near spinning solid surfaces. The mesh covering a spinning surface spins with it, and the SI between the spinning mesh and the rest of the mesh accurately connects the two sides. In some cases, the SI connects the mesh sectors containing different moving parts, enabling a more effective mesh moving. Integrating the ST-SI and ST-TC methods [5] enables high-resolution representation even when the contact is between solid surfaces covered by meshes with SI. It also enables dealing with contact location change and contact sliding. Integrating the ST-IGA with the ST-SI and ST-TC methods increases flow solution accuracy while keeping the element density in narrow spaces near contact areas at a reasonable level. We give several examples of challenging computations carried out with this integrated ST-SI-TC-IGA method [6], including the computations for a realistic aortic-valve model with prescribed leaflet motion.

References

[1] K. Takizawa and T. E. Tezduyar, "Multiscale Space–Time Fluid–Structure Interaction Techniques", *Computational Mechanics*, **48** (2011) 247–267.

[2] K. Takizawa, T. E. Tezduyar, H. Mochizuki, H. Hattori, S. Mei, L. Pan, and K. Montel, "Space–time VMS method for flow computations with slip interfaces (ST-SI)", *Mathematical Models and Methods in Applied Sciences*, **25** (2015) 2377–2406.

[3] K. Takizawa, T. E. Tezduyar, A. Buscher, and S. Asada, "Space–Time Interface-Tracking with Topology Change (ST-TC)", *Computational Mechanics*, **54** (2014) 955–971.

[4] K. Takizawa, T. E. Tezduyar, Y. Otoguro, T. Terahara, T. Kuraishi, and H. Hattori, "Turbocharger Flow Computations with the Space–Time Isogeometric Analysis (ST-IGA)", *Computers & Fluids*, **142** (2017) 15–20.

[5] K. Takizawa, T. E. Tezduyar, S. Asada, and T. Kuraishi, "Space–Time Method for Flow Computations with Slip Interfaces and Topology Changes (ST-SI-TC)", *Computers & Fluids*, **141** (2016) 124–134.

[6] K. Takizawa, T. E. Tezduyar, T. Terahara, and T. Sasaki, "Heart valve flow computation with the integrated Space–Time VMS, Slip Interface, Topology Change and Isogeometric Discretization methods", *Computers & Fluids*, **158** (2017) 176–188.

Date

② NOV 28 (WED) 14:30–15:20

TAYFUN E. TEZDUYAR[1,2] and KENJI TAKIZAWA[2]
[1] RICE UNIVERSITY, HOUSTON, TEXAS, USA
[2] WASEDA UNIVERSITY, 3-4-1 OOKUBO, SHINJUKU-KU, TOKYO 169-8555, JAPAN

① Introduction to Stabilized Methods for Computational Flow Analysis

Stabilized methods now play an indispensable role flow in analysis (see [1] for some examples from fluid–structure interaction analysis). The main components of the early stabilized methods were the Streamline-Upwind/Petrov-Galerkin (SUPG) [2, 3] and Pressure-Stabilizing/Petrov-Galerkin (PSPG) [4] stabilizations, which are still used very widely. The SUPG method stabilizes the computations against numerical oscillations caused by dominant advection terms, and the PSPG method enables using equal-order basis functions for velocity and pressure in incompressible flow. They are both residual-based methods, where the stabilization term added to the Galerkin formulation has, as a factor, some residual of the governing equations. This consistency of these stabilized methods brings the stabilization without trading off the accuracy. We provide an introduction to the stabilized methods in the context of the advection–diffusion equation and Navier–Stokes equations of incompressible flows. In stabilized methods, an embedded stabilization parameter, known as "τ," plays a significant role. This parameter involves a measure of the local length scale (also known as "element length") and other parameters such as the element Reynolds and Courant numbers. We describe some of the introductory concepts and early definitions [5, 6] of the stabilization parameters.

References

[1] Y. Bazilevs, K. Takizawa, and T. E. Tezduyar, *Computational Fluid–Structure Interaction: Methods and Applications*, Wiley, February 2013.

[2] A. N. Brooks and T. J. R. Hughes, "Streamline Upwind/Petrov-Galerkin Formulations for Convection Dominated Flows with Particular Emphasis on the Incompressible Navier-Stokes Equations", *Computer Methods in Applied Mechanics and Engineering*, **32** (1982) 199–259.

[3] T. J. R. Hughes and T. E. Tezduyar, "Finite Element Methods for First-order Hyperbolic Systems with Particular Emphasis on the Compressible Euler Equations", *Computer Methods in Applied Mechanics and Engineering*, **45**, (1984) 217–284.

[4] T. E. Tezduyar, "Stabilized Finite Element Formulations for Incompressible Flow Computations", *Advances in Applied Mechanics*, **28** (1992) 1–44.

[5] T. E. Tezduyar and Y. J. Park, "Discontinuity Capturing Finite Element Formulations for Nonlinear Convection-Diffusion-Reaction Equations", *Computer Methods in Applied Mechanics and Engineering*, **59** (1986) 307–325.

[6] T. E. Tezduyar, "Computation of Moving Boundaries and Interfaces and Stabilization Parameters", *International Journal for Numerical Methods in Fluids*, **43** (2003) 555–575.

Date

① NOV 27 (TUE) 11:00–11:50

Kenji TAKIZAWA[1] and Tayfun E. TEZDUYAR[2,1]
[1] Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
[2] Rice University, Houston, Texas, USA

② Space–Time Computational Analysis: From Inception to New Generations

Space–Time (ST) Variational Multiscale (ST-VMS) method [1] and its predecessor ST-SUPS [2] have a good track record in computational analysis of complex fluid–structure interactions (FSI) and flows with moving boundaries and interfaces (MBI). The classes of challenging problems with successful analysis range from spacecraft parachute FSI to wind-turbine aerodynamics, from flapping-wing aerodynamics of an actual locust to fluid mechanics of heart valves. When an FSI or MBI problem requires high-resolution representation of boundary layers near solid surfaces, ALE and ST methods, where the mesh moves to follow the fluid–solid interface, meet that requirement. Moving-mesh methods have been practical in more classes of complex FSI and MBI problems than commonly thought of. With a number of complementary methods [3, 4, 5, 6] introduced recently, the ST methods can now do even more. We provide a brief description of how the ST methods started, how they evolved over the years, and the classes of challenging problems that can be solved.

References

[1] K. Takizawa and T. E. Tezduyar, "Multiscale Space–Time Fluid–Structure Interaction Techniques", *Computational Mechanics*, **48** (2011) 247–267.

[2] T. E. Tezduyar, "Computation of Moving Boundaries and Interfaces and Stabilization Parameters", *International Journal for Numerical Methods in Fluids*, **43** (2003) 555–575.

[3] K. Takizawa, T. E. Tezduyar, A. Buscher, and S. Asada, "Space–Time Interface-Tracking with Topology Change (ST-TC)", *Computational Mechanics*, **54** (2014) 955–971.

[4] K. Takizawa, T. E. Tezduyar, H. Mochizuki, H. Hattori, S. Mei, L. Pan, and K. Montel, "Space–time VMS method for flow computations with slip interfaces (ST-SI)", *Mathematical Models and Methods in Applied Sciences*, **25** (2015) 2377–2406.

[5] K. Takizawa, T. E. Tezduyar, S. Asada, and T. Kuraishi, "Space–Time Method for Flow Computations with Slip Interfaces and Topology Changes (ST-SI-TC)", *Computers & Fluids*, **141** (2016) 124–134.

[6] K. Takizawa, T. E. Tezduyar, Y. Otoguro, T. Terahara, T. Kuraishi, and H. Hattori, "Turbocharger Flow Computations with the Space–Time Isogeometric Analysis (ST-IGA)", *Computers & Fluids*, **142** (2017) 15–20.

Date

② NOV 28 (WED) 11:00–11:50

Harald van BRUMMELEN
Department of Mechanical Engineering,
Eindhoven University of Technology,
Eindhoven, The Netherlands

Phase-field models for binary-fluid flows and elasto-capillary fluid–solid interaction

Phase-field models [1] have emerged as a comprehensive and versatile paradigm for modeling binary-fluid flows, by virtue of their inherent treatment of topological changes and their rigorous thermodynamic substructure. Such topological changes may pertain to fusion or fissioning of interfaces or to the motion of the contact line (triple point) corresponding to the intersection of the fluid–fluid meniscus with an adjacent solid substrate. Hence, equipped with suitable boundary conditions, phase-field models also provide an inherent description of wetting phenomena [2,3]. In conjunction with a standard Arbitrary-Eulerian-Lagrangian formulation, phase-field models of binary fluids provide a suitable model for so-called elasto-capillary fluid–solid interaction [4,5], i.e. the deformation of a soft solid substrate due to capillary forces in the fluid–fluid meniscus.

In these lectures, we will consider phase-field models for binary-fluid flows and their use in elasto-capillary fluid–solid interaction simulations. We will focus mostly on the Navier–Stokes–Cahn–Hilliard (NSCH) equations, but some digression to the Navier–Stokes–Korteweg (NSK) equations will be made to elucidate the main differences between these two phase-field models. The first lecture introduces basic aspects of the Cahn–Hilliard phase-field model [6] and its conjunction with the Navier–Stokes equations. In the second lecture, we will consider the application to elasto-capillary FSI.

References

[1] H. Gomez and K.G. van der Zee, *Computational phase-field modeling*, Encyclopedia of Computational Mechanics (E. Stein, R. de Borst, and T.J.R. Hughes, eds.), John Wiley & Sons, Ltd., 2nd ed., 2017.

[2] D. Jacqmin, *Contact-line dynamics of a diffuse fluid interface*, J. Fluid Mech. 402 (2000), 57–88.

[3] T. Qian and P. Wang, X.-P.and Sheng, *Molecular scale contactline hydrodynamics of immiscible flows*, Phys. Rev. E 68 (2003), 016306.

[4] E.H. van Brummelen, H. Shokrpour Roudbari, and G.J. van Zwieten, *Elasto-capillarity simulations based on the Navier-Stokes-Cahn-Hilliard equations*, Advances in Computational Fluid-Structure Interaction and Flow Simulation, Modeling and Simulation in Science, Engineering and Technology, Birkhäuser, 2016, pp. 451–462.

[5] E.H. van Brummelen, M. Shokrpour Roudbari, G. Simsek, and K.G. van der Zee, *Binary-fluid–solid interaction based on the Navier–Stokes–Cahn–Hilliard equations*, Fluid Structure Interaction (S. Frei, B. Holm, T. Richter, T. Wick, and H. Yang, eds.), Radon Series on Computational and Applied Mathematics, vol. 20, De Gruyter, 2017, pp. 283–328.

[6] A. Novick–Cohen, *The Cahn–Hilliard equation*, Handbook of Differential Equations: Evolutionary Equations, vol. 4, Elsevier, 2008.

Date

① NOV 27 (TUE) 15:30–16:20 ② NOV 28 (WED) 15:30–16:20

Thomas EITER
Technical University of Darmstadt, Darmstadt, Germany

Motion of a liquid drop under time-periodic forcing

We consider a liquid drop under the influence of a time-periodic external force in two different settings: in a vacuum and in a container filled with another fluid. After certain manipulations of the corresponding equations of motion, the systems describing these free boundary value problems have well-posed linearizations. However, in the one-phase case the existence of time-periodic solutions requires additional compatibility conditions. For this reason, the examination of the two corresponding linear systems has to be carried out in different ways.

Date NOV 29 (THU) 17:50–18:40

— — —

Giovanni P. GALDI (joint work with Jan Dušek)
Department of Mechanical Engineering and Materials Science
Department of Mathematics
University of Pittsburgh, USA

Hopf Bifurcation in Navier-Stokes Flow past a Rotating Obstacle

We provide sufficient conditions for the existence of a time-periodic family of solutions branching out from steady-state solutions to the Navier-Stokes equations in the exterior of a body that is allowed to translate with non-zero speed, and rotate with constant angular velocity $\omega_\star \neq 0$. The main analytic difficulty associated with this problem is due to the fact that the relevant linearized operator possesses a fairly complicated essential spectrum that touches the imaginary axis at a countable number of points, including 0. This implies that all "classical" approaches fail. However, under an appropriate correlation in the neighborhood of the bifurcation point between frequency of the periodic branch and $\omega_\star$, we can show the existence of such a branch. The method consists in reducing the original problem to the study of a coupled nonlinear system of the elliptic-parabolic type. Moreover, the assumed correlation is supported by several numerical tests.

Date NOV 29 (THU) 12:00–12:50

Matthias HIEBER

Technical University of Darmstadt, Darmstadt, Germany

Dynamics of the Ericksen-Leslie Model for Nematic Liquid Crystal Flows

In this talk we consider the Ericksen-Leslie model for nematic liquid crystal flows with general Leslie and general Ericksen tensor. We prove local as well as global well-posedness results for strong solutions and investigate furthermore equilibria sets and the longtime behaviour of solutions. This is joint work with Jan Pruess.

Date NOV 30 (FRI) 12:00–12:50

— — —

Toshiaki HISHIDA

Nagoya University, Nagoya, Japan

Attainability of steady flows as limits of unsteady Navier-Stokes flows around a rigid body rotating from rest

Let us consider a viscous incompressible flow around a rigid body in 3D. Suppose both the fluid and the body are initially at rest. If the body starts to move from the rest state until the terminal velocity after some finite time, then the large time behavior of the flow would be related to the associated steady problem for the Navier-Stokes system in exterior domains. In 1965, Finn raised the question whether the unsteady Navier-Stokes flow converges to the steady flow, what is called the physically reasonable solution, as $t \to \infty$ in a sense as long as the terminal velocity is small enough (Finn's starting problem). If that is the case, the steady flow is said to be attainable by following the terminology of Heywood. Since the steady flow does not have the finite energy, we do need the L^q-theory to study the starting problem. For the case when the motion of the rigid body is translation, the problem was successfully solved by Galdi, Heywood and Shibata (Arch. Rational Mech. Anal. 1997). It would be worth while finding more possibilities of attainability of the steady flows as limits of unsteady Navier-Stokes flows starting from large motions being in L^3. This was actually studied by Hishida and Maremonti (J. Math. Fluid Mech. 2018) though the terminal translational velocity is still assumed to be sufficiently small. In this presentation I would like to address the problem in which the body starts to rotate from the rest state instead of the translation and show the attainability of steady flows as limits of evolution of the Navier-Stokes fluid motions starting from the rest state or even from large motions provided that the terminal angular velocity after some finite time is small enough. Unlike the translational case, decay properties of the Stokes semigroup with rotational effect is not enough to analyze the starting problem because of the growth of the rotational velocity at spatial infinity. To overcome this difficulty, large time behavior of the evolution operator generated by a non-autonomous linearized operator is useful. For the latter situation mentioned above in which the fluid starts from large motions being in L^3, the evolution operator plays an important role as an auxiliary flow that attains the initial velocity, and then the remaining part is constructed as a Leray-Hopf weak solution (with the strong energy inequality) which can be identified with a decaying solution after a large instant. Finding such an instant is a crucial step.

Date NOV 29 (THU) 15:30–16:20

Keiichi ITATANI

Department of Cardiovascular Surgery,
Cardiovascular Imaging Research Labo. Adult Congenital Heart Center,
Kyoto Prefectural University of Medicine, Kyoto, Japan

Blood Flow Visualization in Cardiovascular System for Surgical Planning

Heart and vascular diseases have always abnormality in blood flow, and all cardiac surgery aims toward improvement or correction of the flow pattern irrespective of type of diseases or procedures. Therefore, blood flow visualization is an essential guide to inspect the diseases and to plan the surgery.

Classically, there are two types of blood flow visualization methods: the one is based on the flow measurement equipment such as cardiac MRI, and the other one is numerical calculation method such as CFD, or flow simulation. However, actual visualization process is data assimilation because they combine calculation and the measured data to some extent.

The lecture, at first, explains how to realize biological or physiological blood flow in CFD models: how to define boundary conditions in inlets and outlets to realize for example reflections from the peripheral capillaries or autonomic nerve regulations, and how to define fluid viscosity models in diseased turbulent flow. Then CFD model application samples to coronary arterial disease and aortic disease and surgical planning will be discussed.

Second, the lecture explains flow measurement modalities. The representative one is 4D flow MRI based on the phase contrast images. In addition to the validation with CFD models, the application of 4D flow MRI in heart valve disease will be explained. Other novel flow visualization technologies are those using ultrasonography. Because color Doppler information provides only unidirectional flow, flow visualization requires some kind of calculation using measured data, again data assimilation process.

In clinical practice, parameters to estimate mechanical stress on cardiovascular structures would be necessary, and blood flow visualization can provide several parameters based on the theorem of fluid mechanics. Finally we will discuss the derivation process and application of physiological parameters.

Date NOV 28 (WED) 16:50–17:40

— — —

Mads KYED

Technical University of Darmstadt, Darmstadt, Germany

$\mathcal{R}$-boundedness and time-periodic maximal regularity

In recent years, vector-valued multiplier theorems have been widely used to establish maximal regularity for parabolic initial-value problems (IVP). Due to the pioneering work Bourgain and Weis, the task of establishing maximal L_p regularity can be reduced to verifying $\mathcal{R}$-boundedness of a family of resolvent operators. In this talk, I will discuss the possibility of deriving similar L_p estimates for corresponding time-periodic problems based solely on these $\mathcal{R}$-bounds. I will demonstrate how existing IVP $\mathcal{R}$-bounds can be directly applied to flow problems with periodic forcing.

Date NOV 29 (THU) 16:50–17:40

Yuto OTOGURO[1], Kenji TAKIZAWA[1]
and Tayfun E. TEZDUYAR[2,1]
[1] Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
[2] Rice University, Houston, Texas, USA

Stabilization and Discontinuity-Capturing Parameters for Space–Time Flow Computations with Isogeometric Discretizations in Complex Geometry

Stabilized methods, which have been very common in flow computations for many years, typically involve stabilization parameters, and discontinuity-capturing (DC) parameters if the method is supplemented with a DC term. Various well-performing stabilization and DC parameters have been introduced for stabilized space–time (ST) computational methods in the context of the advection–diffusion equation and the Navier–Stokes equations of incompressible and compressible flows. These parameters were all originally intended for finite element discretization but quite often used also for isogeometric discretization. The stabilization and DC parameters we present here for ST computations are in the context of the advection–diffusion equation and the Navier–Stokes equations of incompressible flows and target isogeometric discretization. The parameters are based on a direction-dependent element length expression. The expression is outcome of an easy to understand derivation [1]. The key components of the derivation are mapping the direction vector from the physical ST element to the parent ST element, accounting for the discretization spacing along each of the parametric coordinates, and mapping what we have in the parent element back to the physical element.

We extend the element length calculation method to complex geometries [2, 3] and apply the method to flow analysis of a turbocharger turbine [4].

References

[1] K. Takizawa, T. E. Tezduyar, and Y. Otoguro, "Stabilization and discontinuity-capturing parameters for space–time flow computations with finite element and isogeometric discretizations", *Computational Mechanics*, published online, DOI: 10.1007/s00466-018-1557-x, April 2018.

[2] Y. Otoguro, K. Takizawa, and T. E. Tezduyar, "Space–time VMS computational flow analysis with isogeometric discretization and a general-purpose NURBS mesh generation method", *Computers & Fluids*, **158** (2017) 189–200.

[3] Y. Otoguro, K. Takizawa, and T. E. Tezduyar, "A General-Purpose NURBS Mesh Generation Method for Complex Geometries", in T. E. Tezduyar, editor, *Frontiers in Computational Fluid–Structure Interaction and Flow Simulation: Research from Lead Investigators under Forty – 2018*, Modeling and Simulation in Science, Engineering and Technology, 399–434, Springer, 2018.

[4] Y. Otoguro, K. Takizawa, T. E. Tezduyar, K. Nagaoka, and S. Mei, "Turbocharger turbine and exhaust manifold flow computation with the Space–Time Variational Multiscale Method and Isogeometric Analysis", *Computers & Fluids*, published online, DOI: 10.1016/j.compfluid.2018.05.019, May 2018.

Date NOV 27 (TUE) 16:50–17:40

Grigory SEREGIN
University of Oxford, Oxford, England

Global weak $L^{3,\infty}$-solutions to Cauchy problem for Navier-Stokes equations

We consider the Cauchy problem for the Navier-Stokes equation in $\mathbb{R}^3\times]0,\infty[$ with the initial datum $u_0 \in L^3_{\text{weak}}$, a critical space containing non-trivial $(-1)-$homogeneous fields. For small $||u_0||_{L^3_{\text{weak}}}$ one can get global well-posedness by perturbation theory. When $||u_0||_{L^3_{\text{weak}}}$ is not small, the perturbation theory no longer applies and, very likely, the local-in-time well-posedness and uniqueness fails. One can still develop a good theory of weak solutions with the following stability property: If $u^{(n)}$ are weak solutions corresponding the the initial datum $u_0^{(n)}$, and $u_0^{(n)}$ converge weakly* in L^3_{weak} to u_0, then a suitable subsequence of $u^{(n)}$ converges to a weak solution u corresponding to the initial condition u_0. This is of interest even in the special case $u_0 \equiv 0$. The talk is based on the joint paper written together with T. Barker and V. Sverak.

Date NOV 30 (FRI) 16:50–17:40

— — —

Zhifei ZHANG
Peking University, Beijing, China

Resolvent estimates and transition threshold for 2-D Couette flow in a finite channel

In this talk, I will talk about the transition threshold problem for the 2-D Navier-Stokes equations around the Couette flow at high Reynolds number Re in a finite channel. We introduce a systematic method to establish the resolvent estimates of the linearized operator and space-time estimates of the linearized Navier-Stokes equations. As an application, we prove that if the initial velocity v_0 satisfies $\|v_0 - (y,0)\|_{H^2} \leq cRe^{-\frac{1}{2}}$ for some small c independent of Re, then the solution of the 2-D Navier-Stokes equations remains within $O(Re^{-\frac{1}{2}})$ of the Couette flow for any time.

Date NOV 30 (FRI) 15:30–16:20

Taro KANAI[1], Kenji TAKIZAWA[1], Tayfun E. TEZDUYAR[2,1], Tatsuya TANAKA[1] and Aaron HARTMANN[2],
[1] Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
[2] Rice University, Houston, Texas, USA

Spacecraft Parachute Compressible-Flow computation with Geometric-Porosity Modeling and Isogeometric Discretization

One of the challenges in computational fluid–structure interaction (FSI) analysis of spacecraft parachutes is the "geometric porosity," a design feature created by the hundreds of gaps and slits that the flow goes through. Accurate geometric-porosity modeling becomes essential for FSI analysis because computations with resolved geometric porosity would be exceedingly time-consuming. The geometric-porosity model introduced earlier with the space–time FSI (STFSI) method enabled successful computational analysis and design studies of the Orion spacecraft parachutes in the incompressible-flow regime [1]. Recently, new porosity models and ST computational methods for compressible-flow aerodynamics were introduced in [2]. These models and methods were tested in finite element computation of a drogue parachute with geometric porosity. The key new component of the ST computational framework was the compressible-flow ST Slip Interface method, introduced in conjunction with the compressible-flow ST SUPG method. Here, we integrate these porosity models and ST computational methods with isogeometric discretization. We use cubic and quadratic NURBS basis functions in structure and fluid mechanics computations, respectively. This gives us a parachute shape that is smoother than what we get from a typical finite element discretization. In the flow analysis, the combination of the ST framework, NURBS basis functions, and the SUPG stabilization assures superior computational accuracy. The computations we present for a drogue parachute show the effectiveness of the porosity models, ST computational methods, and the integration with isogeometric discretization [3].

References

[1] K. Takizawa, C. Moorman, S. Wright, T. Spielman, and T. E. Tezduyar, "Fluid–Structure Interaction Modeling and Performance Analysis of the Orion Spacecraft Parachutes", *International Journal for Numerical Methods in Fluids*, **65** (2011) 271–285, doi: 10.1002/fld.2348.

[2] K. Takizawa, T. E. Tezduyar, and T. Kanai, "Porosity models and computational methods for compressible-flow aerodynamics of parachutes with geometric porosity", *Mathematical Models and Methods in Applied Sciences*, **27** (2017) 771–806, doi: 10.1142/S0218202517500166.

[3] T. Kanai, K. Takizawa, T. E. Tezduyar, T. Tanaka, and A. Hartmann, "Compressible-Flow Geometric-Porosity Modeling and Spacecraft Parachute Computation with Isogeometric Discretization", *Computational Mechanics*, published online, July 2018, doi: 10.1007/s00466-018-1595-4.

Date NOV 27 (TUE) 17:50–18:10

Takashi KURAISHI[1], Kenji TAKIZAWA[1]
and Tayfun E. TEZDUYAR[2,1]
[1] Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
[2] Rice University, Houston, Texas, USA

ST-SI-TC-IGA Computational Analysis of Flow Around a Tire with Actual Geometry, Road Contact and Tire Deformation

Tire aerodynamics with actual tire geometry, road contact and tire deformation pose tough computational challenges. The challenges include i) the complexity of an actual tire geometry with longitudinal and transverse grooves, ii) the spin of the tire, iii) maintaining accurate representation of the boundary layers near the tire while being able to deal with the flow-domain topology change created by the road contact and tire deformation, and iv) the turbulent nature of the flow. A new space–time (ST) computational method [1], "ST-SI-TC-IGA," is enabling us to address these challenges. The core component of the ST-SI-TC-IGA is the ST Variational Multiscale (ST-VMS) method [2], and the other key components are the ST Slip Interface (ST-SI) [3] and ST Topology Change (ST-TC) [4] methods and the ST Isogeometric Analysis (ST-IGA). The VMS feature of the ST-VMS addresses the challenge created by the turbulent nature of the flow, the moving-mesh feature of the ST framework enables high-resolution flow computation near the moving fluid–solid interfaces, and the higher-order accuracy of the ST framework strengthens both features. The ST-SI enables moving-mesh computation with the tire spinning. The mesh covering the tire spins with it, and the SI between the spinning mesh and the rest of the mesh accurately connects the two sides of the solution. The ST-TC enables moving-mesh computation even with the TC created by the contact between the tire and the road. It deals with the contact while maintaining high-resolution flow representation near the tire. Integration of the ST-SI and ST-TC enables high-resolution representation even though parts of the SI are coinciding with the tire and road surfaces. It also enables dealing with the tire–road contact location change and contact sliding. By integrating the ST-IGA with the ST-SI and ST-TC, in addition to having a more accurate representation of the tire geometry and increased accuracy in the flow solution, the element density in the tire grooves and in the narrow spaces near the contact areas is kept at a reasonable level.

We present computations with the ST-SI-TC-IGA and two models of flow around a rotating tire with road contact and prescribed deformation [5]. One is a simple 2D model for verification purposes, and one is a 3D model with an actual tire geometry and a deformation pattern provided by the tire company. The computations show the effectiveness of the ST-SI-TC-IGA in tire aerodynamics.

References

[1] K. Takizawa, T. E. Tezduyar, T. Terahara, and T. Sasaki, "Heart valve flow computation with the integrated Space–Time VMS, Slip Interface, Topology Change and Isogeometric Discretization methods", *Computers & Fluids*, **158** (2017) 176–188.

[2] K. Takizawa and T. E. Tezduyar, "Multiscale Space–Time Fluid–Structure Interaction Techniques", *Computational Mechanics*, **48** (2011) 247–267.

[3] K. Takizawa, T. E. Tezduyar, H. Mochizuki, H. Hattori, S. Mei, L. Pan and K. Montel, "Space–time VMS method for flow computations with slip interfaces (ST-SI)", *Mathemat-*

ical Models and Methods in Applied Sciences, **25** (2015) 2377–2406.

[4] K. Takizawa, T. E. Tezduyar, A. Buscher, and S. Asada, “Space–Time Interface-Tracking with Topology Change (ST-TC)”, *Computational Mechanics*, **54** (2014) 955–971.

[5] T. Kuraishi, K. Takizawa, and T. E. Tezduyar, “Tire Aerodynamics with Actual Tire Geometry, Road Contact and Tire Deformation”, *Computational Mechanics*, published online, DOI: 10.1007/s00466-018-1642-1, October 2018.

Date NOV 28 (WED) 17:50–18:10