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

WASEDA UNIVERSITY

November 19–22, 2019

Contents

早稲田大学
WASEDA University
TOP GLOBAL UNIVERSITY JAPAN
科研費
KAKENHI

Minicourses

COMPUTATIONAL PHASE-FIELD MODELING: APPLICATIONS IN FLUIDS, SOLIDS AND BIOMECHANICS

Hector Gomez[1]

[1] School of Mechanical Engineering, Purdue University

Phase-field modeling is emerging as a promising tool for the treatment of problems with interfaces [1]. The classical description of interface problems requires the numerical solution of partial differential equations on moving domains in which the domain motions are also unknowns. The computational treatment of these problems requires moving meshes and is difficult when the moving domains undergo topological changes. Phase-field modeling may be understood as a methodology to reformulate interface problems as equations posed on fixed domains. In some cases, the phase-field model may be shown to converge to the moving-boundary problem as a regularization parameter tends to zero [1, 2]. However, this is only one interpretation because phase-field models do not need to have a moving-boundary problem associated and can be rigorously derived from classical thermomechanics. In this context, the distinguishing feature is that constitutive models depend on the variational derivative of the free energy. In all, phase-field models open the opportunity for the efficient treatment of outstanding problems in computational mechanics, such as, the interaction of a large number of cracks in three dimensions, cavitation [3], film and nucleate boiling, etc. In addition, phase-field models bring a new set of challenges for numerical discretization that will excite the computational mechanics and computational mathematics communities. These include, for example, higher-order partial-differential spatial operators, stiff semi-discretizations, stable time-stepping algorithms and the treatment of sharp internal layers in the solution. In presentation, I will show how Isogeometric Analysis [4] (a generalization of finite elements that uses functions from computational geometry) presents a unique combination of attributes that can be exploited on phase-field modeling, namely, higher-order accuracy, robustness, two- and three-dimensional geometric flexibility, compact support, and, most importantly, higher-order continuity.

[1] H. Gomez and K. van der Zee, “Computational phase-field modeling”, in *Encyclopedia of Computational Mechincs*, (2017).

[2] H. Gomez, V.M. Calo, Y. Bazilevs, and T.J.R. Hughes, “Isogeometric analysis of the cahn-hilliard phase-field model”, *Computer Methods in Applied Mechanics and Engineering*, **197** (2008) 4333–4352.

[3] H. Gomez, T.J.R. Hughes, X. Nogueira, and V.M. Calo, “Isogeometric analysis of the isothermal navier-stokes-korteweg equations”, *Computer Methods in Applied Mechanics and Engineering*, **199** (2010) 1828–1840.

[4] T.J.R. Hughes, J.A. Cottrell, and Y. Bazilevs, “Isogeometric analysis: CAD, finite elements, NURBS, exact geometry, and mesh refinement”, *Computer Methods in Applied Mechanics and Engineering*, **194** (2005) 4135–4195.

— — — — — — — —

COMPUTATIONAL MODELING OF PROSTATE CANCER

Guillermo Lorenzo[1], Thormas J.R. Hughes[1], Alessandro Reali[2] and Hector Gomez[3]

[1] Oden Institute for Computational Engineering and Sciences, The University of Texas at Austin
[2] Dipartimento de Ingegneria Civile e Architettura, Università degli Studi di Pavia
[3] School of Mechanical Engineering, Purdue University

Prostate cancer is a major health burden among aging men worldwide. Benign prostatic hyperplasia (BPH) is another urogenital condition of the prostate in ageing men, which often coexists with prostate cancer. BPH causes the prostate to gradually enlarge over time, which may induce bothersome lower urinary tract symptoms. Recent pathological studies have shown that prostatic tumors growing in larger prostates tend to present more favorable clinical features. This suggests that large prostates may exert a protective effect against prostate cancer, but the underlying mechanisms are largely unknown. We propose a mechanical explanation for this phenomenon. The mechanical stress fields created by growing solid tumors are known to exert an inhibitory effect on their dynamics. Prostate enlargement due to BPH and the confinement of the organ in the pelvic region contribute to these mechanical stress fields, hence further restraining prostate cancer growth. To explore this hypothesis, we run a simulation study using a mechanically-coupled organ-scale computational model of prostate cancer growth over the actual anatomy of a patient's prostate with coexisting tumor and BPH. We leverage isogeometric analysis to handle the nonlinearities in the model, the complex anatomy of the prostate, and the intricate tumor morphologies. Our simulations show that a history of benign prostatic enlargement creates mechanical stress fields that impede prostate cancer growth. These results suggest major changes in the clinical management of BPH and prostate cancer to account for this mechanical interaction. The computational technology developed in this study may also assist physicians to improve diagnosis and predict pathological outcomes of both diseases on an organ-scale, patient-specific basis.

[1] G. Lorenzo, M.A. Scott, K. Tew, T.J.R. Hughes, Y.J. Zhang, L. Liu, G. Vilanova, and H. Gomez, "Tissue-scale, personalized modeling and simulation of prostate cancer growth, proceedings of the national academy of sciences", in *Proceedings of the National Academy of Sciences*, (2016).

[2] G. Lorenzo, T.J.R. Hughes, P. Dominguez-Frojan, A. Reali, and H. Gomez, "Computer simulations suggest that prostate enlargement due to benign prostatic hyperplasia mechanically impedes prostate cancer growth", in *Proceedings of the National Academy of Sciences*, (2019).

[3] G. Lorenzo, M.A. Scott, K. Tew, T.J.R. Hughes, and H. Gomez, "Hierarchically refined and coarsened splines for moving interface problems, with particular application to phase-field models of prostate tumor growth", *Computer Methods in Applied Mechanics and Engineering*, **319** (2017) 515–548.

Date

① NOV 22 (FRI) 14:30–15:20

② NOV 22 (FRI) 16:50–17:40

— — — — — — — —

Decay estimates of gradient of a generalized Oseen evolution operator arising from time-dependent rigid motions in exterior domains

TOSHIAKI HISHIDA

NAGOYA UNIVERSITY, JAPAN

Let us consider the motion of a viscous incompressible fluid past a rotating rigid body in 3D, where the translational and angular velocities of the body are prescribed but time-dependent. In a reference frame attached to the body, we have the Navier-Stokes system with the drift and (one half of the) Coriolis terms in a fixed exterior domain. The existence of the evolution operator $T(t,s)$ in the space L^q generated by the linearized non-autonomous system was proved by Hansel and Rhandi [1] and the large time behavior of $T(t,s)f$ in L^r for $(t-s)\to\infty$ was then developed by the present author [2] when f is taken from L^q with $q\le r$. The contribution of the present lecture concerns such L^q-L^r decay estimates of $\nabla T(t,s)$ with optimal rates, which must be useful for the study of stability/attainability of the Navier-Stokes flow in several physically relevant situations. Our main theorem (arXiv:1908.04080) completely recovers the L^q-L^r estimates for the autonomous case (Stokes and Oseen semigroups, those semigroups with rotating effect) in 3D exterior domains, which were established by [4], [6], [5], [3] and [7].

[1] T. Hansel and A. Rhandi, The Oseen-Navier-Stokes flow in the exterior of a rotating obstacle: the non-autonomous case, *J. Reine Angew. Math.* **694** (2014), 1–26.

[2] T. Hishida, Large time behavior of a generalized Oseen evolution operator, with applications to the Navier-Stokes flow past a rotating obstacle, *Math. Ann.* **372** (2018), 915–949.

[3] T. Hishida and Y. Shibata, L_p-L_q estimate of the Stokes operator and Navier-Stokes flows in the exterior of a rotating obstacle, *Arch. Rational Mech. Anal.* **193** (2009), 339–421.

[4] H. Iwashita, L_q-L_r estimates for solutions of the nonstationary Stokes equations in an exterior domain and the Navier-Stokes initial value problems in L_q spaces, *Math. Ann.* **285** (1989), 265–288.

[5] T. Kobayashi and Y. Shibata, On the Oseen equation in the three dimensional exterior domains, *Math. Ann.* **310** (1998), 1–45.

[6] P. Maremonti and V.A. Solonnikov, On nonstationary Stokes problems in exterior domains, *Ann. Sc. Norm. Sup. Pisa* **24** (1997), 395–449.

[7] Y. Shibata, On the Oseen semigroup with rotating effect, *Functional Analysis and Evolution Equations, The Günter Lumer Volume*, 595–611, Birkhäuser, Basel, 2008.

Date

① NOV 19 (TUE) 11:00–11:50

② NOV 20 (WED) 11:00–11:50

— — — — — — — —

Introduction to Immersed Boundary Method

MING-CHEN HSU

IOWA STATE UNIVERSITY, USA

Immersogeometric Analysis is a geometrically flexible framework that has been recently proposed for the modeling and simulation of CFD, fluid–structure interaction (FSI), and biomedical applications. This novel method makes direct use of the CAD boundary representation (B-rep) of a complex design structure by immersing it into a non-body-fitted discretization of the surrounding fluid domain, thereby eliminating the challenges associated with time-consuming and labor-intensive geometry cleanup and mesh generation/manipulation. This approach also effectively deals with FSI problems involving structures with complex motion that leads to large deformations of the fluid domain, including changes of topology. The key ingredients to achieving high simulation accuracy, including imposing the Dirichlet boundary conditions weakly using Nitsche's method and faithfully capturing the geometry in intersected elements, will be discussed. The variational formulation for immersogeometric FSI analysis is derived using an augmented Lagrangian approach to weakly enforce kinematic constraints. A hybrid arbitrary Lagrangian–Eulerian/immersogeometric methodology, in which a single computation combines both a body-fitted, deforming-mesh treatment of some fluid–structure interfaces and a non-body-fitted treatment of others, is also developed under the same framework. Finally, the capabilities of immersogeometric methods that can be effectively integrated with optimization methods to improve engineering designs using high-fidelity FSI analysis will be demonstrated.

— — — — — — — —

Fluid–Structure Interaction Modeling and Analysis of Heart Valves

MING-CHEN HSU

IOWA STATE UNIVERSITY, USA

Bioprosthetic heart valves (BHVs) are prostheses fabricated from xenograft biomaterials for treating valvular disease. While these devices have mechanical and blood flow characteristics similar to the native valves, the durability remains limited to 10–15 years with device failure continues to result from leaflet structural deterioration mediated by fatigue and tissue mineralization. Improving BHV design remains an important clinical goal and represents a unique cardiovascular engineering challenge. We believe there is a profound need to develop a general understanding of heart valve mechanism through novel simulation technologies that take advantage of fluid–structure interactions (FSI). In this work, we present a framework for modeling BHVs using recently proposed isogeometric analysis based parametric design platform and immersogeometric FSI analysis. We simulate the coupling of the deforming aortic root, the parametrically designed prosthetic valves, and the surrounding blood flow under physiological conditions. The results demonstrate the effectiveness of the proposed framework in practical computations with greater levels of physical realism. A parametric study is carried out to investigate the influence of the geometry on heart valve performance, indicated by the effective orifice area the coaptation area. The simulation result of the best performed prosthetic design is compared with the phase-contrast MRI data to demonstrate the qualitative similarity of the flow patterns in the ascending aorta. Recent developments in FSI modeling and simulation of transcatheter heart valves will also be discussed.

Date

① NOV 22 (FRI) 12:00–12:50

② NOV 22 (FRI) 15:30–16:20

— — — — — — —

On Leray Problems for Steady Navier–Stokes System in 2D Bounded and Exterior Domains

Mikhail V. Korobkov

School of Mathematical Sciences, Fudan University, Shanghai, China
Voronezh State University, Voronezh, Russia

These talks are based on some results obtained in our joint papers with K.Pileckas and R.Russo (see, e.g., [4]–[6]). In the first lectures we discuss the existence theorem for plane and axially symmetric spatial flows for boundary value problem of stationary Navier-Stokes system in bounded domains under necessary and sufficient condition of zero total flux.

Recall that according to the mass conservation law the total flux (i.e. the amount of fluid flows through all the boundary components of the domain) should be zero, it is a necessary condition of solvability. However, in his famous paper of 1933 [3] J. Leray proved the existence of a solution under the stronger assumption that the flux of fluid through each boundary component is zero (this condition means the lack of sources and sinks). The case when the boundary value satisfies only the necessary condition of zero total flux (i.e. when the sources and sinks are allowed) was left open and the problem of existence (or nonexistence) of a solution for such case is known in the scientific community as *Leray's problem.*

The main tool in our approach here is a new analogue of the classical Morse-Sard theorem on critical values for Sobolev functions under minimal smoothness assumptions obtained in the joint papers with J.Bourgain and J.Kristensen [2]. Surprisingly, almost all level sets of these functions turns out to be classically smooth manifolds despite the “fact that functions itself are not smooth — in general they are continuous only.

A similar method was used to solve the Leray problem in an exterior (unbounded) axisymmetric three-dimensional domain without any restrictions on fluxes [5].

The last lectures address to solutions of stationary Navier–Stokes system in two dimensional exterior domains, namely, existence of these solutions and their asymptotic behavior. The talks are based on our recent joint papers (see, e.g., [6]), where the uniform boundedness and uniform convergence at infinity for arbitrary solution with finite Dirichlet integral were established. Here no restrictions on smallness of fluxes are assumed, etc. In the proofs we develop the ideas of the classical paper of Amick [1].

[1] C.J. Amick, On Leray's problem of steady Navier–Stokes flow past a body in the plane, *Acta Math.* **161** (1988), 71–130.

[2] Bourgain J., Korobkov M. V., Kristensen J., On the Morse–Sard property and level sets of $W^{n,1}$ Sobolev functions on $\mathbb{R}^n$, *Journal fur die reine und angewandte Mathematik (Crelles Journal),* **2015**, no. 700 (2015), 93–112.

[3] J. Leray, Étude de diverses équations intégrales non linéaire et de quelques problèmes que pose l'hydrodynamique, *J. Math. Pures Appl.* **12** (1933), 1–82.

[4] Korobkov M. V., Pileckas K. and Russo R., Solution of Leray's problem for stationary Navier-Stokes equations in plane and axially symmetric spatial domains, *Ann. of Math.* **181**, no. 2 (2015), 769–807.

[5] M.V. Korobkov, K. Pileckas and R. Russo, The existence theorem for the steady Navier–Stokes problem in exterior axially symmetric 3D domains, *Math. Ann.* **181**, no. 2 (2015), 769–807.

[6] M.V. Korobkov, K. Pileckas and R. Russo, On convergence of arbitrary D-solution of steady Navier–Stokes system in 2D exterior domains, *Arch. Rational Mech. Anal.* **233**, no. 1 (2019), 385–407.

Date

① NOV 19 (TUE) 10:00–10:50

② NOV 19 (TUE) 12:00–12:50

③ NOV 20 (WED) 10:00–10:50

④ NOV 20 (WED) 12:00–12:50

— — — — — — —

INTRODUCTION TO SPACE–TIME COMPUTATIONAL FLOW ANALYSIS AND MESH UPDATE METHODS

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

The Deforming-Spatial-Domain/Stabilized ST (DSD/SST) method [4] 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 a fluid–solid interface enables high-resolution boundary-layer representation near the interface. The DSD/SST is an alternative to ALE method. Because of its stabilization components "SUPG" and "PSPG," the original DSD/SST is now also called "ST-SUPS." The ST-VMS method [1] is the VMS version of the DSD/SST. 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. 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, 3, 8] have been used mostly with the ST-VMS method, and some of the methods are unique to the ST framework.

[1] T.E. Tezduyar, "Stabilized finite element formulations for incompressible flow computations", *Advances in Applied Mechanics*, **28** (1992) 1–44, doi:10.1016/S0065-2156(08)70153-4.

[2] K. Takizawa and T.E. Tezduyar, "Multiscale space–time fluid–structure interaction techniques", *Computational Mechanics*, **48** (2011) 247–267, doi:10.1007/s00466-011-0571-z.

[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, doi:10.1109/2.237441.

[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, doi:10.1007/s00466-012-0759-x.

[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, doi:10.1007/s00466-013-0880-5.

[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, doi:10.1007/s00466-013-0935-7.

[8] T. Kuraishi, K. Takizawa, and T.E. Tezduyar, "Space–time computational analysis of tire aerodynamics with actual geometry, road contact, tire deformation, road roughness and fluid film", *Computational Mechanics*, published online, DOI: 10.1007/s00466-019-01746-8, July 2019, doi:10.1007/s00466-019-01746-8.

— — — — — — —

ARTERIAL WALL MODELING IN COMPUTATIONAL CARDIOVASCULAR ANALYSIS

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

Because the medical-image-based geometries used in patient-specific arterial fluid–structure interaction computations do not come from the zero-stress state (ZSS) of the artery, we need to estimate the ZSS required in the computations. This is one of the most important factors in evaluating the stretch in the arterial wall and also in determining the distribution of the stress along the arterial wall, while the material properties could have a secondary effect. In the first ZSS estimation method [1] the framework was an element-based configuration. The method was tested in [2] on time-dependent anatomical models. It was extended in [3] to isogeometric discretization. The framework was upgraded in [4] to an integration-point-based (IPB) configuration, and we call that IPBZSS estimation. The IPBZSS estimation has two main components. 1. An iteration technique, which starts with a calculated ZSS initial guess, is used for computing the IPBZSS such that when a given pressure load is applied, the medical-image-based target shape is matched. 2. A design procedure, which is based on the Kirchhoff–Love shell model of the artery is used for calculating the ZSS initial guess. Very recently the scope and robustness of the method were further increased by introducing a new design procedure for the ZSS initial guess [5]. The new design procedure has two features. (a) An IPB shell-like coordinate system, which increases the scope of the design to general parametrization in the computational space. (b) Analytical solution of the force equilibrium in the normal direction, based on the Kirchhoff–Love shell model [1], which places proper constraints on the design parameters. This increases the estimation accuracy, which in turn increases the robustness of the iterations and the convergence speed.

[1] K. Takizawa, H. Takagi, T.E. Tezduyar, and R. Torii, “Estimation of element-based zero-stress state for arterial FSI computations”, *Computational Mechanics*, **54** (2014) 895–910, doi:10.1007/s00466-013-0919-7.

[2] K. Takizawa, R. Torii, H. Takagi, T.E. Tezduyar, and X.Y. Xu, “Coronary arterial dynamics computation with medical-image-based time-dependent anatomical models and element-based zero-stress state estimates”, *Computational Mechanics*, **54** (2014) 1047–1053, doi:10.1007/s00466-014-1049-6.

[3] K. Takizawa, T.E. Tezduyar, and T. Sasaki, “Aorta modeling with the element-based zero-stress state and isogeometric discretization”, *Computational Mechanics*, **59** (2017) 265–280, doi:10.1007/s00466-016-1344-5.

[4] T. Sasaki, K. Takizawa, and T.E. Tezduyar, “Aorta zero-stress state modeling with T-spline discretization”, *Computational Mechanics*, **63** (2019) 1315–1331, doi:10.1007/s00466-018-1651-0.

[5] T. Sasaki, K. Takizawa, and T.E. Tezduyar, “Medical-image-based aorta modeling with zero-stress-state estimation”, *Computational Mechanics*, **64** (2019) 249–271, doi:10.1007/s00466-019-01669-4.

[6] K. Takizawa, T.E. Tezduyar, and T. Sasaki, “Isogeometric hyperelastic shell analysis with out-of-plane deformation mapping”, *Computational Mechanics*, **63** (2019) 681–700, doi:10.1007/s00466-018-1616-3.

Date

① NOV 21 (THU) 15:30–16:20

② NOV 22 (FRI) 11:00–11:50

— — — — — — — —

INTRODUCTION TO STABILIZED METHODS FOR COMPUTATIONAL FLOW ANALYSIS

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

Stabilized methods now play an indispensable role flow analysis (see [1] for some examples from fluid–structure 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 SUPG 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 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, and we mention some newer definitions [2, 8], including those designed for isogeometric analysis.

[1] Y. Bazilevs, K. Takizawa, and T.E. Tezduyar, *Computational Fluid–Structure Interaction: Methods and Applications*. Wiley, February 2013, ISBN 978-0470978771.

[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, doi:10.1016/0045-7825(84)90157-9.

[4] T.E. Tezduyar, "Stabilized finite element formulations for incompressible flow computations", *Advances in Applied Mechanics*, **28** (1992) 1–44, doi:10.1016/S0065-2156(08)70153-4.

[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, doi:10.1016/0045-7825(86)90003-4.

[6] T.E. Tezduyar, "Computation of moving boundaries and interfaces and stabilization parameters", *International Journal for Numerical Methods in Fluids*, **43** (2003) 555–575, doi:10.1002/fld.505.

[7] 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*, **62** (2018) 1169–1186, doi:10.1007/s00466-018-1557-x.

[8] K. Takizawa, Y. Ueda, and T.E. Tezduyar, "A node-numbering-invariant directional length scale for simplex elements", *Mathematical Models and Methods in Applied Sciences*, published online, DOI: 10.1142/S0218202519500581, 2019, doi:10.1142/S0218202519500581.

— — — — — — —

FSI AND FLUID MECHANICS IN CARDIOVASCULAR ANALYSIS

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

Many agree that computational cardiovascular flow analysis can provide surgeons and medical doctors valuable information in a wide range of patient-specific cases, including cerebral aneurysms (see, e.g., [1]), treatment of cerebral aneurysms (see, e.g., [2]), aortas (see, e.g., [9]) and heart valves (see, e.g., [7]). The computational challenges faced in this class of flow analyses also have a wide range, many quite formidable. They include highly-unsteady flows and complex cardiovascular geometries. They also include moving boundaries and interfaces, such as the motion of the heart valve leaflets, contact between moving solid surfaces, such as the contact between the leaflets, and the fluid–structure interaction between the blood and the cardiovascular structure. Many of these challenges have been or are being addressed by the Space–Time VMS (ST-VMS) method [1] and the special methods used in combination with it. The special methods include the ST Slip Interface (ST-SI) method [2, 7], ST Topology Change (ST-TC) [3, 9] method, ST Isogeometric Analysis (ST-IGA) [1, 10, 4], integration of these methods, and a general-purpose NURBS mesh generation method for complex geometries [1, 3]. We will provide an overview of the core and special methods and present examples of challenging computations carried out with these methods.

[1] Y. Bazilevs, K. Takizawa, and T.E. Tezduyar, *Computational Fluid–Structure Interaction: Methods and Applications*. Wiley, February 2013, ISBN 978-0470978771.

[2] K. Takizawa, K. Schjodt, A. Puntel, N. Kostov, and T.E. Tezduyar, "Patient-specific computational analysis of the influence of a stent on the unsteady flow in cerebral aneurysms", *Computational Mechanics*, **51** (2013) 1061–1073, doi:10.1007/s00466-012-0790-y.

[3] K. Takizawa, T.E. Tezduyar, H. Uchikawa, T. Terahara, T. Sasaki, and A. Yoshida, "Mesh refinement influence and cardiac-cycle flow periodicity in aorta flow analysis with isogeometric discretization", *Computers & Fluids*, **179** (2019) 790–798, doi:10.1016/j.compfluid.2018.05.025.

[4] 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, doi:10.1016/j.compfluid.2016.11.012.

[5] K. Takizawa and T.E. Tezduyar, “Multiscale space–time fluid–structure interaction techniques”, *Computational Mechanics*, **48** (2011) 247–267, doi:10.1007/s00466-011-0571-z.

[6] 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, doi:10.1142/S0218202515400126.

[7] K. Takizawa, T.E. Tezduyar, T. Kuraishi, S. Tabata, and H. Takagi, “Computational thermo-fluid analysis of a disk brake”, *Computational Mechanics*, **57** (2016) 965–977, doi:10.1007/s00466-016-1272-4.

[8] 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, doi:10.1007/s00466-013-0935-7.

[9] K. Takizawa, T.E. Tezduyar, A. Buscher, and S. Asada, “Space–time fluid mechanics computation of heart valve models”, *Computational Mechanics*, **54** (2014) 973–986, doi:10.1007/s00466-014-1046-9.

[10] K. Takizawa, B. Henicke, A. Puntel, T. Spielman, and T.E. Tezduyar, “Space–time computational techniques for the aerodynamics of flapping wings”, *Journal of Applied Mechanics*, **79** (2012) 010903, doi:10.1115/1.4005073.

[11] 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, doi:10.1016/j.compfluid.2016.02.021.

[12] 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, doi:10.1016/j.compfluid.2017.04.017.

[13] 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, ISBN 978-3-319-96468-3, doi:10.1007/978-3-319-96469-0_10.

Date

① NOV 21 (THU) 12:00–12:50

② NOV 22 (FRI) 10:00–10:50

— — — — — — —

50-Minute Lectures

The Motion of a Heavy Rigid Body with a Liquid-Filled Interior Cavity

GIOVANNI P. GALDI

MECHANICAL ENGINEERING AND MATERIALS SCIENCE
UNIVERSITY OF PITTSBURGH, USA

Problems involving the motion of a rigid body with a cavity filled with a viscous fluid are of fundamental interest in several applied areas of research, including dynamics of flight, space technology, and geophysical problems. Besides its important role in physical and engineering disciplines, the motion of these coupled systems generates a number of mathematical questions, which are intriguing and challenging at the same time. They are principally due to the occurrence of different and coexisting dynamic properties, such as the dissipative nature of the liquid, and the conservative character of some components of the angular momentum of the coupled system as a whole. One important characteristic of this interaction is that the presence of the liquid can dramatically influence the motion of the rigid body and produce a "stabilizing" effect that, in some cases, can even bring the coupled system to rest. Objective of this talk is to present a rather complete mathematical analysis of the dynamics of such systems and provide a rigorous explanation of some of the most relevant observed phenomena.

— — — — — — —

Local well-posedness of the vacuum free boundary of 3-D compressible Navier-Stokes equations

GUILONG GUI

SCHOOL OF MATHEMATICS, NORTHWEST UNIVERSITY, CHINA

Consideration in this talk is the 3-D motion of viscous gas with the vacuum free boundary. We use the conormal derivative to establish local well-posedness of this system. One of important advantages in the work is that we do not need any strong compatibility conditions on the initial data in terms of the acceleration. This is a joint work with Prof. Chao Wang and Yuxi Wang.

— — — — — — —

Magnetic inhibition on the Rayleigh-Taylor instability and thermal convection in non-resistive magnetohydrodynamics

SONG JIANG

INSTITUTE OF APPLIED PHYSICS AND COMPUTATIONAL MATHEMATICS, BEIJING, CHINA

The Rayleigh-Taylor (RT) instability is well known as gravity-driven instability in fluids when a heavy fluid is on top of a light one, while the thermal convection instability often arises when a fluid is heated from below. Both of them appear in a wide range of applications in science and technology, such as in inertia confinement fusion, Tokamak, supernova explosions.

In this talk, mathematical analysis of the magnetic RT/thermal convection instability in incompressible/compressible fluids will be presented, in particular, effects of (impressed) magnetic fields upon the growth of the RT/thermal convection instability will be discussed and analyzed quantitatively. We shall show that a sufficiently strong (impressed) magnetic field can inhibit the RT/thermal convection instability; otherwise, instability will still occur in the sense that solutions do not continuously depend on initial data.

— — — — — — —

Duality Approach to Local Regularity Theory for Navier-Stokes Equations

Grigory Seregin

University of Oxford, Oxford, United Kingdom
PDMI, St. Petersburg, Russia

In this talk, it will be explained how the problem of the local regularity of weak solutions to the Navier-Stokes equations can be reduced to the time decay of certain Lebesgue's norms of solutions to the Cauchy problem for the Stokes system with a drift. The corresponding drift appears as a result of the rescaling of the original weak solution around a potential singularity. Some interesting cases related to potential Type I singularities are going to be discussed.

— — — — — — —

FLUID DYNAMICS IN CARDIOVASCULAR PROBLEMS

Hiroshi Suito[1]

[1] Advanced Institute for Materials Research, Tohoku University

In blood vessels with congenital heart diseases, characteristic flow structures are formed, in which pulsating flows affect strongly on wall shear stresses and energy dissipation patterns. In this talk, we present computational analyses for blood flows in patient-specific cases, through which we aim at understanding the relationships between differences in geometries and in energy dissipation patterns. Our present targets include an aortic coarctation case and a Norwood surgery for hypoplastic left heart syndrome. On the other hand, although such kind of patient-specific simulations are extremely useful for grasping the flow/stress distributions and for patient-specific treatment planning, they remain insufficient to elucidate the general mechanisms of targeted diseases. We introduce a geometrical characterization of blood vessels, which vary widely among individuals. Through close collaboration between mathematical science and clinical medicine, these analyses yield deeper understandings. This work was supported by JST CREST Grant Number JPMJCR15D1, Japan.

[1] T. Ueda, H. Suito, H. Ota, and K. Takase, "Computational fluid dynamics modeling in aortic diseases", *Cardiovascular Imaging Asia*, **2** (2018) 58–64.

[2] H. Suito, K. Takizawa, V.Q.H. Huynh, D. Sze, and T. Ueda, “FSI analysis of the blood flow and geometrical characteristics in the thoracic aorta”, *Computational Mechanics*, **54** (2014) 1035–1045, doi:10.1007/s00466-014-1017-1.

[3] H. Suito, T. Ueda, and D. Sze, “Numerical simulation of blood flow in the thoracic aorta using a centerline-fitted finite difference approach”, *Japan Journal of Industrial and Applied Mathematics*, **30** (2013) 701–710.

— — — — — — —

INFLAMMATION CAUSED BY BLOOD FLOW

TADASHI YAMAMOTO[1]

[1] HOKKAIDO CARDIOVASCULAR HOSPITAL

A coronary blood flow is not seen directly, however we can see the flow with simulation. The simulation is computational fluid dynamics (CFD). From around 10 years ago, prediction of the progress of plaque was started. Recently, researches on the relationship between arteriosclerosis and coronary blood flow are progressing, And I think we must understand the correlation with arteriosclerosis, thrombus and coronary blood flow [1]. We presented this prediction by demonstrating low WSS sites that cause endothelial cell damage using CFD. WSS is obtained from the product of blood viscosity and blood flow velocity. The analysis of atherosclerosis is performed by WSS.A plaque develops with very low WSS, and more lipid rich and with inflammation [2]. Many reports have shown that CFD matched with pathology, IVUS images and coronary CT images. The superiority of CFD is where we may predict the progression and rupture of plaque [3]. Turbulent flow makes low WSS and high strain zone. Endothelial cell damage occurs, endothelial cell sequence changes, gradually becoming vulnerable. And low WSS makes cap weak, high WSS makes cap thin. High and low WSS weakens the cap and drives it to a situation that is likely to rupture [4] A plaque rupture may cause myocardial infarction and trigger sudden death.

[1] H. Samady, P. Eshtehardi, M.C. McDaniel, J. Suo, S.S. Dhawan, C. Maynard, L.H. Timmins, A.A. Quyyumi, and D.P. Giddens, “Coronary artery wall shear stress is associated with progression and transformation of atherosclerotic plaque and arterial remodeling in patients with coronary artery disease”, **124** (2011) 779–788.

[2] Y.S. Chatzizisis, M. Jonas, A.U. Coskun, R. Beigel, B.V. Stone, C. Maynard, R.G. Gerrity, W. Daley, C. Rogers, E.R. Edelman, C.L. Feldman, and P.H. Stone, “Prediction of the localization of high-risk coronary atherosclerotic plaques on the basis of low endothelial shear stress”, **117** (2008) 993–1002.

[3] Y. Fukumoto, T. Hiro, T. Fujii, G. Hashimoto, T. Fujimura, J. Yamada, T. Okamura, and M. Matsuzaki, “Localized elevation of shear stress is related to coronary plaque rupture: A 3-dimensional intravascular ultrasound study with in-vivo color mapping of shear stress distribution”, *Journal of the American Colleg of Cardiology*, **51** (2008) 645–650.

[4] B.R. Kwak, M. Bäck, M.L. Bochaton-Piallat, G. Caligiuri, M.J. Daemen, P.F. Davies, I.E. Hoefer, P. Holvoet, H. Jo, R. Krams, S. Lehoux, C. Monaco, S. Steffens, R. Virmani,

C. Weber, J.J. Wentzel, and P.C. Evans, “Biomechanical factors in atherosclerosis: mechanisms and clinical implications”, *European Heart Journal*, **35** (2014) 3013–3020.

— — — — — — — —

Short Talks

Global existence and stability of large solutions to isentropy compressible Navier-Stokes equations

JINGCHI HUANG

SCHOOL OF MATHEMATICS, SUN YAT-SEN UNIVERSITY, GUANGZHOU, CHINA

In this joint work with Lingbing He and Chao Wang, we will briefly review some recently result of global existence of large solutions to isentropy compressible Navier-Stokes equations. And then we will introduce a new mechanism to obtain a new kind of global large solutions.

— — — — — — — —

SPACE–TIME VARIATIONAL MULTISCALE METHOD AND ISOGEOMETRIC ANALYSIS WITH TOPOLOGY CHANGE

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

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 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 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 gives us the ST-SI-TC-IGA method [7]. This increases flow solution accuracy while keeping the element density in narrow spaces near contact areas at a reasonable level. In computational analysis of fluid films [7, 8], the ST-SI-TC-IGA enables solution with a computational cost comparable to that of the Reynolds-equation model for the comparable solution quality [7]. We give several examples of challenging computations carried out with the ST-SI-TC-IGA.

[1] K. Takizawa and T.E. Tezduyar, “Multiscale space–time fluid–structure interaction techniques”, *Computational Mechanics*, **48** (2011) 247–267, doi:10.1007/s00466-011-0571-z.

[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, doi:10.1142/S0218202515400126.

[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, doi:10.1007/s00466-013-0935-7.

[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, doi:10.1016/j.compfluid.2016.02.021.

[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, doi:10.1016/j.compfluid.2016.05.006.

[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, doi:10.1016/j.compfluid.2016.11.012.

[7] T. Kuraishi, K. Takizawa, and T.E. Tezduyar, “Space–Time Isogeometric flow analysis with built-in Reynolds-equation limit”, *Mathematical Models and Methods in Applied Sciences*, **29** (2019) 871–904, doi:10.1142/S0218202519410021.

[8] T. Kuraishi, K. Takizawa, and T.E. Tezduyar, “Space–time computational analysis of tire aerodynamics with actual geometry, road contact, tire deformation, road roughness and fluid film”, *Computational Mechanics*, published online, DOI: 10.1007/s00466-019-01746-8, July 2019, doi:10.1007/s00466-019-01746-8.

— — — — — — —

MITTRAL VALVE AND CHORDA TENDINEAE MODELING WITH T-SPLINES

Megumi Minamihara[1], Takuya Terahara[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

Modeling of the mitral valve is one of the most challenging cases in cardiovascular modeling. That is because it involves contact, the valve has more modes of motion than the other valves, and the motion is confined by "cables" called "chorda tendineae." T-Splines provide one of the most flexible freeform surface discretizations, and it is suitable for analysis based on shell formulations, such as the formulation introduced in [1]. One of the challenges in modeling the valve and chorda tendineae is combining shells and cables. Because the shell control points are not on the shell, it becomes difficult to attach a cable at the actual, physical attachment point (see [5] for a similar challenge faced in modeling of a ram-air parachute). Here we introduce a simple but effective method for attaching the cables at the actual attachment points.

[1] K. Takizawa, T.E. Tezduyar, and T. Sasaki, "Isogeometric hyperelastic shell analysis with out-of-plane deformation mapping", *Computational Mechanics*, **63** (2019) 681–700, doi:10.1007/s00466-018-1616-3.

[2] K. Takizawa, T.E. Tezduyar, and T. Terahara, "Ram-air parachute structural and fluid mechanics computations with the space–time isogeometric analysis (ST-IGA)", *Computers & Fluids*, **141** (2016) 191–200, doi:10.1016/j.compfluid.2016.05.027.

— — — — — — —

A GENERAL-PURPOSE NURBS MESH GENERATION METHOD FOR COMPLEX GEOMETRIES AND ELEMENT LENGTH EXPRESSIONS

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

To increase the scope of isogeometric discretization, NURBS volume mesh generation needs to be easier and as automated as possible. To that end, we present a general-purpose NURBS mesh generation method [1]. The method is based on multi-block structured mesh generation with existing techniques, projection of that mesh to a NURBS mesh made of patches that correspond to the blocks, and recovery of the original model surfaces to the extent they are suitable for accurate and robust fluid mechanics computations. It is expected to retain the refinement distribution and element quality of the multi-block structured mesh that we start with. The flexibility of discretization with the general-purpose mesh generation is supplemented with the Space–Time (ST) Slip Interface method, which allows, without loss of accuracy, C^1 continuity between NURBS patches and thus removes the matching requirement between the patches. We present a test computation for a turbocharger turbine and exhaust manifold, which demonstrates that the general-purpose mesh generation method proposed makes the isogeometric analysis use in fluid mechanics computations even more practical. We also present directional element-length expressions designed for ST computations with isogeometric discretization. The element length expressions are needed in calculating the stabilization parameters embedded in widely used stabilized methods such as the VMS method. They are also needed in calculating the discontinuity-capturing (DC) parameters if the method is supplemented with a DC term. Various well-performing element length expressions and stabilization and DC parameters were introduced for stabilized 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 element length expressions we present here for isogeometric discretization are in the context of the advection–diffusion equation and the Navier–Stokes equations of incompressible flows. The expressions are outcome of an easy to understand derivation [2]. 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 present versions of the element length expressions that are applicable to complex geometries [1, 3] and we use these expressions in computational flow analysis of a turbocharger turbine [4, 5].

[1] 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.

[2] 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*, **62** (2018) 1169–1186.

[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*, **179** (2019) 764–776.

[5] Y. Otoguro, K. Takizawa, T.E. Tezduyar, K. Nagaoka, R. Avsar, and Y. Zhang, "Space–time VMS flow analysis of a turbocharger turbine with isogeometric discretization: Computations with time-dependent and steady-inflow representations of the intake/exhaust cycle", *Computational Mechanics*, **64** (2019) 1403–1419.

— — — — — — —

VENTRICLE-VALVE-AORTA FLOW ANALYSIS WITH THE SPACE–TIME ISOGEOMETRIC DISCRETIZATION AND TOPOLOGY CHANGE

Takuya Terahara[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

Heart valve flow analysis requires accurate representation of boundary layers near moving surfaces, even when the leaflets come into contact, and handling high geometric complexity. We address these computational challenges with a space–time (ST) method that integrates 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, 5]. The ST-SI-TC-IGA [6, 7] improves the flow solution accuracy by dealing with the contact between the leaflets while maintaining high-resolution representation near the leaflet surfaces, including an accurate representation of the wall shear stress (WSS) distribution over the two sides of the leaflet. Here we apply the ST-SI-TC-IGA to a model consisting of the left ventricle, aortic valve and the aorta. The NURBS meshes for the three parts are generated separately and the SIs accurately connect the three solution parts. The meshes for the valve and aorta are from [8, 9], and the mesh for the ventricle is generated from the medical images. The computation presented shows the effectiveness of the ST-SI-TC-IGA and how the flow in the ventricle influences the WSS on the leaflet surfaces and the flow in the aorta.

[1] K. Takizawa and T.E. Tezduyar, “Multiscale space–time fluid–structure interaction techniques”, *Computational Mechanics*, **48** (2011) 247–267, doi:10.1007/s00466-011-0571-z.

[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, doi:10.1142/S0218202515400126.

[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, doi:10.1007/s00466-013-0935-7.

[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, doi:10.1016/j.compfluid.2016.02.021.

[5] K. Takizawa, T.E. Tezduyar, and T. Terahara, “Ram-air parachute structural and fluid mechanics computations with the space–time isogeometric analysis (ST-IGA)”, *Computers & Fluids*, **141** (2016) 191–200, doi:10.1016/j.compfluid.2016.05.027.

[6] K. Takizawa, T.E. Tezduyar, T. Terahara, and T. Sasaki, “Heart valve flow computation with the Space–Time Slip Interface Topology Change (ST-SI-TC) method and Isogeometric Analysis (IGA)”, in P. Wriggers and T. Lenarz, editors, *Biomedical Technology: Modeling, Experiments and Simulation*, Lecture Notes in Applied and Computational Mechanics, 77–99, Springer, 2018, ISBN 978-3-319-59547-4, doi:10.1007/978-3-319-59548-1_6.

[7] 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, doi:10.1016/j.compfluid.2016.11.012.

[8] K. Takizawa, T.E. Tezduyar, H. Uchikawa, T. Terahara, T. Sasaki, K. Shiozaki, A. Yoshida, K. Komiya, and G. Inoue, “Aorta flow analysis and heart valve flow and structure analysis”, 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, 29–89, Springer, 2018, ISBN 978-3-319-96468-3, doi:10.1007/978-3-319-96469-0_2.

[9] K. Takizawa, T.E. Tezduyar, H. Uchikawa, T. Terahara, T. Sasaki, and A. Yoshida, "Mesh refinement influence and cardiac-cycle flow periodicity in aorta flow analysis with isogeometric discretization", *Computers & Fluids*, **179** (2019) 790–798, doi:10.1016/j.compfluid.2018.05.025.

— — — — — — —

The stationary Navier-Stokes equations in toroidal Besov spaces

HIROYUKI TSURUMI

WASEDA UNIVERSITY

In this talk, we consider the stationary problem of the Navier-Stokes equations on tori. Following the methods of previous studies in whole space $\mathbb{R}^n$, we first prove the existence and uniqueness of solutions in $\dot{B}_{p,q}^{-1+\frac{n}{p}}(\mathbb{T}^n)$ for given external forces in $\dot{B}_{p,q}^{-3+\frac{n}{p}}(\mathbb{T}^n)$ when $1 \leq p < n$. Moreover, in the case $n \leq p \leq \infty$, we show some examples of external forces causing the ill-posedness (the discontinuity of the map from given external forces to solutions).

— — — — — — —

The local well-posedness of water wave equations

CHAO WANG

SCHOOL OF MATHEMATEICAL SCIENCES, PEKING UNIVERSITY, BEIJING, CHINA

In this talk, I will present our recent results on the water wave equations. First, I give the proof of the local well-posedness of the free boundary problem for the incompressible Euler equations in low regularity Sobolev spaces, in which the velocity is a Lipschitz function and the free surface belongs to $C^{\frac{3}{2}+\varepsilon}$. Second part, I will talk about the water-waves problem in a two-dimensional bounded corner domain Ω_t with an upper free surface Γ_t and a fixed bottom Γ_b. We prove the local well-posedness of the solution to the water-waves system when the contact angles are less than $\frac{\pi}{16}$.

— — — — — — —

Maximal regularity of the Stokes operator in exterior Lipschitz domains

KEIICHI WATANABE

WASEDA UNIVERSITY

We consider the Stokes equations in exterior Lipschitz domains $\Omega \subset \mathbb{R}^n$, $n \geq 3$. Especially, we show that the Stokes operator defined on $L^p_\sigma(\Omega)$ admits the maximal regularity assuming that p satisfies $|1/p - 1/2| < 1/(2n) + \varepsilon$ for some ε. The proof is based on the cut-off techniques and the construction of a parametrix of the solution to the resolvent problem as the sum of solutions to a problem on the whole space and a problem on a bounded Lipschitz domain with suitable chosen data. A crucial point in our analysis is to prove decay with respect to the resolvent parameter of the L^p-norm of the pressure appeared in the resolvent problem on a bounded Lipschitz domain. This talk is based on joint work with Dr. Patrick Tolksdorf (University of Mainz).