International Workshop on Multiphase Flows: Analysis, Modelling and Numerics

Dates: December 10 – 12, 2025

Venue: Waseda University, Green Computing Systems Research and Development Center, (**Access**) Wasedamachi-27, Shinjuku-ku, Tokyo

Website: https://www.waseda.jp/fsci/mathphys/news-en/19692

Registration: https://forms.gle/Xbh5Ey7CJXyuGGfJ6 (Before November 28, 2025)

Timetable

December 10 (Wed.)

13:00 – 13:10	Opening
13:10 – 14:00	Naoki Takada (National Institute of Advanced Industrial Science and Technology)
14:15 – 15:05	Anatole Gaudin (Universitat Duisburg-Essen)
15:05 – 15:30	Coffee Break
15:30 – 15:50	Taichi Eguchi (Waseda University)
15:50 – 16:10	Takahiro Nakamura (Waseda University)
16:20 – 16:40	Jou chun Kuo (Waseda University)
16:40 – 17:00	Zhaojing Xu (Waseda University)

December 11 (Thu.)

10:00 – 10:50	Patrick Tolksdorf (Karlsruhe Institute of Technology)
11:05 – 11:55	Tomohiro Otani (The University of Osaka)
11:55 – 14:00	Lunch Break
14:00 – 14:50	Mitsuo Higaki (Kobe University)
15:05 – 15:25	Kento Sube (Waseda University)
15:25 – 15:50	Coffee Break
15:50 – 16:10	Takahiko Shirai (Waseda University)
16:10 – 16:30	Denghui Ren (Waseda University)
16:40 – 17:00	Masato Terasawa (Waseda University)
17:00 – 17:20	Shunsuke Fujita (Waseda University)
17:30 –	Banquet

December 12 (Fri.)

10:00 – 10:50	Masashi Aiki (Tokyo University of Science)
10:50 – 11:05	Coffee Break
11:05 – 11:55	Alessio Castorrini (Sapienza University of Rome)
11:55 – 12:00	Closing

Wednesday, December 10th, 2025

Phase-field Model and Lattice Boltzmann Model: Mesoscopic Approach for Simulation of Multiphase Flow

Naoki Takada

National Institute of Advanced Industrial Science and Technology, Japan

n this lecture, two mesoscopic numerical approaches, phase-field model (PFM) and lattice Boltzmann model (LBM), are reviewed for computational fluid dynamics (CFD) simulation of multiphase flow. PFM describes an interface between different phases as a finite volumetric zone across which physical properties vary continuously. Such a diffusive interface is formed in a self-organizing way to minimize free energy of the multiphase system. Interfacial tension force is given to the interface by the energy increase per unit area caused by local gradient of order parameter (i.e. density, molar concentration). As a result, PFM simplifies interface-tracking calculation with no use of conventional techniques. In LBM, it is assumed that a continuous fluid comprises fictitious mesoscopic particles that alternately repeat interactions between them and linear translations at isotropic discrete velocities. LBM is useful for high-performance computing on incompressible fluid flow because it executes explicit and simple iterative particle-kinematic operations in conservation form and because it does not need to solve the pressure Poisson equation, which is time-consuming when using other conventional schemes. I will briefly explain PFM and LBM and present applications of the mesoscopic models to CFD simulation of multiphase fluid motion to demonstrate their abilities.

Regularity for the Stokes–Dirichlet problem, 5 new results

ANATOLE GAUDIN

UNIVERSITAT DUISBURG-ESSEN, GERMANY

We discuss here sharp regularity results for the Stokes–Dirichlet (steady) resolvent problem, given $\lambda \in \mathbb{C}^*$, $|\arg(\lambda)| < \mu$, provided $\mu \in [0, \pi)$,

$$\lambda \mathbf{u} - \Delta \mathbf{u} + \nabla \mathfrak{p} = \mathbf{f}, \quad \operatorname{div} \mathbf{u} = 0, \quad \mathbf{u}_{|\partial\Omega} = 0,$$

on $\mathbb{R}^n_+$ or on Ω to be a bounded $\mathrm{C}^{1,\alpha}$-domain, $\alpha \in (0,1)$. We write $\mathbb{A}_\mathcal{D}\mathbf{u} := -\Delta\mathbf{u} + \nabla\mathfrak{p}$, and call $\mathbb{A}_\mathcal{D}$ the Stokes–Dirichlet operator. Given $p, q \in [1, \infty]$, the main new results are as follows:

1. Full (partially) new resolvent estimates (possibly assuming $\operatorname{div} \mathbf{u} = g$) on $\dot{\mathrm{B}}^s_{p,q}(\mathbb{R}^n_+)$ and $\dot{\mathrm{H}}^{s,p}(\mathbb{R}^n_+)$, including $p = 1, \infty$ for Besov spaces, whenever $-1 + 1/p < s < 1/p$:
$|\lambda| \|\mathbf{u}\|_{\dot{\mathrm{B}}^s_{p,q}(\mathbb{R}^n_+)} + \|(\nabla^2\mathbf{u},\, \nabla\mathfrak{p})\|_{\dot{\mathrm{B}}^s_{p,q}(\mathbb{R}^n_+)} \lesssim_{p,n,s,\mu} \|\mathbf{f}\|_{\dot{\mathrm{B}}^s_{p,q}(\mathbb{R}^n_+)} + |\lambda| \|g\|_{\dot{\mathrm{B}}^{s-1}_{p,q}(\mathbb{R}^n_+)} + \|\nabla g\|_{\dot{\mathrm{B}}^s_{p,q}(\mathbb{R}^n_+)}$;
2. On $\mathrm{L}^\infty_{\mathfrak{n},\sigma}(\mathbb{R}^n_+) = \mathrm{L}^\infty(\mathbb{R}^n_+)^n \cap \{ \operatorname{div} \mathbf{v} = 0,\, \mathbf{v} \cdot \mathfrak{e}_{n|\partial\mathbb{R}^n_+} = 0\}$, one has the following description
$$\mathbb{A}_\mathcal{D}\mathbf{u} = \mathbb{P}_{\mathbb{R}^n_+}(-\Delta_\mathcal{D}\mathbf{u}),$$
$$\mathrm{D}_\infty(\mathbb{A}_\mathcal{D}) = \{ \mathbf{u} \in \mathrm{W}^{1,\infty}_{0,\sigma} \cap \mathrm{B}^2_{\infty,\infty}(\mathbb{R}^n_+)^n \;:\; \mathbb{P}_{\mathbb{R}^n_+}(-\Delta_\mathcal{D}\mathbf{u}) \in \mathrm{L}^\infty(\mathbb{R}^n_+)^n \},$$
with a "natural" resolvent estimate, even for the pressure term $\nabla\mathfrak{p}$.
3. On $\mathrm{L}^p(\Omega)$, $1 < p < \infty$, $\mathbb{A}_\mathcal{D}$ has bounded $\mathbf{H}^\infty$-functional calculus (in particular is 0-sectorial). One has, as closed subspaces
$$\mathrm{D}_p(\mathbb{A}_\mathcal{D}^{\frac{s}{2}}) \hookrightarrow \mathrm{H}^{s,p}(\Omega)^n, \quad -1 + 1/p < s < 1 + \alpha + 1/p.$$
A similar results holds for Besov spaces including $p = 1, \infty$ (not deduced from interpolation).
4. On $\mathrm{L}^\infty(\Omega)$, *i.e.* if $\mathbf{f} \in \mathrm{L}^\infty(\Omega)^n$, $\operatorname{div}(\mathbf{f}) = 0$, $\mathbf{f} \cdot \nu_{|\partial\Omega} = 0$, one has $\mathbf{u} \in \mathrm{C}^{1,\alpha}(\overline{\Omega})$, and the estimate
$$(1 + |\lambda|)\|\mathbf{u}\|_{\mathrm{L}^\infty(\Omega)} + (1 + |\lambda|)^{\frac{1-\alpha}{2}} \|(\nabla\mathbf{u}, \mathfrak{p})\|_{\mathrm{C}^{0,\alpha}(\Omega)} \lesssim^{\Omega}_{n,\alpha,\mu} \|\mathbf{f}\|_{\mathrm{L}^\infty(\Omega)}.$$
5. On $\mathrm{L}^1(\Omega)$, if $\mathbf{f} \in \mathrm{L}^1(\Omega)^n$, $\operatorname{div}(\mathbf{f}) = 0$, $\mathbf{f} \cdot \nu_{|\partial\Omega} = 0$, one has $\|\nabla e^{-t\mathbb{A}_\mathcal{D}}\mathbf{f}\|_{\mathrm{L}^1(\Omega)} \leqslant C_\Omega\, t^{-\frac{1}{2}} \|\mathbf{f}\|_{\mathrm{L}^1(\Omega)}$.

For the results on bounded domains, the strategy relies on a non-trivial localisation procedure using the Sobolev-Besov multiplier theory by Maz'ya and Shaposhnikova [**2011, Springer**].

— — — — — — —

Onsager's singular set and energy equality for the Navier–Stokes equations in Lebesgue spaces with variable exponents

TAICHI EGUCHI

WASEDA UNIVERSITY, JAPAN

We prove the energy equality for weak solutions of the Navier–Stokes equations under a new condition involving variable Lebesgue spaces, where the integrability exponent varies with each point $(x, t) \in \mathbb{R}^3 \times (0, T)$. Our variable exponent refines the classical Shinbrot–Taniuchi–Lions exponent on a suitable space-time subset. In particular, by choosing the subset appropriately, the required regularity in our condition is automatically satisfied by Leray–Hopf weak solutions. Consequently, the energy equality holds without imposing any additional assumptions on the subset.

Computational Analysis of Multiscale Cavitating Flows Based on the Navier–Stokes–Korteweg Equations

Takahiro Nakamura

Waseda University, Japan

Propellers in fluid machinery such as ships, submarines, and pumps generate a phenomenon called cavitation. Cavitation damages the propellers and causes vibrations and noise in the machinery. Therefore, understanding and analyzing cavitation is important. Numerical analysis is one of the methods used to understand cavitation. Various cavitation models have been proposed, and they are generally classified into two types: the sharp interface model and the diffuse interface model. In this research, we use the Navier–Stokes–Korteweg (NSK) equations in the diffuse interface model. These equations describe the dynamics of two-phase flows. However, there are several challenges in computing cavitation. Here, we introduce three of them. First, cavitation is a multiscale problem. We formulate a residual-based variational multiscale method based on the NSK equations with a consistent form. Second, the governing equations include third-order derivative terms. To evaluate these terms, we formulate the equations using the space – time method and discretize them using smooth basis functions such as non-uniform rational B-splines, which are used in isogeometric analysis. Third, to stablize the method, we include Streamline-Upwind/Petrov–Galerkin stabilization and a discontinuity-capturing method.

Finally, we perform simulations for several test cases, including flow past a cylinder, flow past a wedge, Venturi flow, and the two-bubbles problem.

— — — — — — —

L_1 approach to the compressible viscous fluid flows in general domains

Jou-Chun Kuo

Waseda University, Japan

This talk is based on a joint work with Professor Emeritus Yoshihiro Shibata (Waseda University).

This talk is devoted to proving the L_1 in time and $B^s_{q,1}$ in space maximal regularity for the Stokes equations obtained by linearized procedure of the Navier-Stokes equations describing the compressible viscous fluid motion. Here, $N - 1 < q < 2N$, $-\min(N/q, N/q') < s < N/q$ and $-1 + 1/q < s < 1/q$, where N is the space dimension. The approach is by means of the spectral analysis of Lamé equations based on the real interpolation arguments. An application of our theorem is to prove the local well-posedness of the Navier-Stokes equations with non-slip boundary conditions in uniform C^3 domains, whose boundary is compact.

Computational Thermo-Fluid Analysis of a Car Tire with Road Contact, Tire Deformation, and Heat Exchange

Zhaojing Xu

Waseda University, Japan

In the pursuit of carbon neutrality in the automobile industry, improving aerodynamic efficiency has become a critical objective in research and development. For passenger cars, improving aerodynamic efficiency usually emphasizes reducing aerodynamic drag. This can be achieved by optimizing various vehicle components such as body panels, undertrays, and spoilers. However, the tire remains a geometrically fixed element due to its function and regulatory constraints. As a result, the relative aerodynamic influence of the tire grows increasingly significant as other parts of the vehicle are refined.

For a running car, its tires undergo complex dynamics. It is deforming because of the load, rotating at high rotational speeds, and contacting the road. To add more complexity to the system, for normal passenger cars, tires usually contain complex patterns to add mechanical grip and channel water. In addition, tires during operation usually maintain a surface temperature higher than that of the surrounding ground and ambient air because of hysteresis effect. Altogether, these factors make airflow over the tire a complex thermal-fluid-structure interaction problem.

This research focuses on the computational flow analysis of a car tire. Computational flow analysis can complement wind tunnel studies, but doing that for an operating tire is challenging because of the aforementioned aspects. While some of the previous studies addressed those challenges, the full thermal-fluid-structure interactions are rarely accounted for. In order to obtain deeper understanding of flow, we believe it is essential to perform computational flow analysis of the tire with full detail of its dynamics.

Thus, the main goal of the study is to conduct the computational thermal – fluid analysis of a car tire with fully detailed complex patterns undergoing rotation, contact, and heat exchange. This is achieved through the following methods. 1) Space–Time Isogeometric Analysis (ST-IGA) [1, 2] is used as a flow computation framework which enables accurate tracking of moving objects like tires and representation of the flow field. 2) ST-IGA with Slip Interface and Topology Change (ST-SI-TC-IGA) [3] is used to model rotation, deformation and contact. 3)Thermo-Fluid ST [4] method is used to enable a simultaneous computation of fluid and thermal transport, with coupling between flow. The tire, the ground, and the ambient air can be given different temperatures. 4) Other techniques are used for mesh generation and its 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, 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.

[3] 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.

[4] K. Takizawa, T.E. Tezduyar and T. Kuraishi, "Multiscale space–time methods for thermo-fluid analysis of a ground vehicle and its tires", Mathematical Models and Methods in Applied Sciences, **25** (2015). 2227–2255.

Thursday, December 11th, 2025

L^p-bounds of Riesz transforms associated to generalized Stokes operators

Patrick Tolksdorf

Karlsruhe Institute of Technology, Germany

In this work, we study Riesz transforms associated to the generalized Stokes operator A given by

$$Au = f \quad \Leftrightarrow \quad \begin{cases} -\mathrm{div}\,(\mu \nabla u) + \nabla \phi = f & \text{in } \mathbb{R}^d, \\ \mathrm{div}\,(u) = 0 & \text{in } \mathbb{R}^d. \end{cases}$$

Besides ellipticity of μ, we only assume that the coefficients are bounded and measurable. We show that the associated Riesz transform $\nabla A^{-1/2}$ is bounded on $L^2(\mathbb{R}^d)$ which is an extension of the resolution of Kato's square root problem for elliptic operators in divergence form [1] to generalized Stokes operators. In addition, we study lower and upper bounds of the Riesz transforms in $L^p(\mathbb{R}^d)$, *i.e.*,

$$\|\nabla A^{-1/2} u\|_{L^p} \leq C \|u\|_{L^p} \quad \text{as well as} \quad \|\nabla A^{-1/2} u\|_{L^p} \geq c \|u\|_{L^p}$$

and provide ranges of p for which such an upper or lower bound hold in general. This part can be seen as an extension of results in the monograph [2] of Auscher.

This is joint research with Luca Haardt.

References

[1] P. Auscher, S. Hofmann, M. Lacey, A. McIntosh, and Ph. Tchamitchian, The solution of the Kato square root problem for second order elliptic operators on $\mathbb{R}^n$, *Ann. of Math.* **156** (2002), no. 2, 633–654.

[2] P. Auscher, On necessary and sufficient conditions for L^p-estimates of Riesz transforms associated to elliptic operators on $\mathbb{R}^n$ and related estimates, *Mem. Amer. Math. Soc.* **186** (2007), no. 871, xviii+75 pp.

— — — — — — —

Theoretical and numerical modeling for interpreting flow magnetic resonance velocimetry

Tomohiro Otani

The University of Osaka, Japan

Magnetic resonance imaging (MRI) is a well-established clinical tool for the noninvasive acquisition of internal human body information, and has been extended to spatiotemporal flow velocity imaging, such as phase-contrast and four-dimensional flow MRI. This flow velocimetry technique is highly attractive for the clinical assessment of cardiac, cerebrovascular, and neurofluid dynamics. However, MRI-based flow data contain various artefacts, and their quantitative validity remains to be fully clarified. In this talk, we will briefly review the background of MRI flow velocimetry and introduce theoretical and numerical modeling approaches that aim to interpret the underlying flow characteristics embedded in MRI data.

Navier wall law for viscous flows in randomly rough domains

Mitsuo Higaki

Kobe University, Japan

The wall law has been known in engineering as an empirical method for obtaining an effective approximation of viscous fluid motion in a domain with rough surface [Nikuradse (1933)]. For steady laminar flows in a cylindrical rough domain, the wall law yields a velocity field that obeys the Navier-slip boundary condition (Navier wall law). In this talk, we rigorously prove that this is indeed an effective approximation. More precisely, we report that, when considering a sample space of rough domains, the optimal approximation rate can be obtained under certain ergodicity. The key in the proof is the deterministic/probabilistic estimate of boundary layers describing the fluid motion near rough surface. Here, we use ideas from quantitative stochastic homogenization for elliptic equations [Armstrong-Smart, Armstrong-Kuusi-Mourrat, Gloria-Neukamm-Otto, Shen]. Note, however, that we consider the sample space of rough domains rather than of coefficient matrices. The ergodicity assumption mentioned above hinges on the validity of functional inequalities for random variables such as the logarithmic Sobolev inequality and the spectral gap inequality. This talk is based on joint work with Jinping Zhuge (Morningside Center of Mathematics, China) and Yulong Lu (University of Minnesota, USA).

— — — — — — —

Well-posedness and analyticity of solutions to the stationary MHD equations

Kento Sube

Waseda University, Japan

We consider the stationary problem of the MHD equations in $\mathbb{R}^3$. The aim of this talk is to show existence, uniqueness, regularity, and analyticity of solutions in the scaling invariant homogeneous Besov space $\dot{B}_{p,q}^{-1+3/p}$ for $1 \leqslant p < 3$ and $1 \leqslant q \leqslant \infty$. In particular, for analyticity, we make a use of a technique so-called parameter trick. Such a trick is known as an elegant method to prove space-time analyticity of solutions to semilinear or quasilinear parabolic equations. It is clarified that the method of the parameter trick is also useful to the nonlinear elliptic equations such as MHD system.

Computational Analysis of a Cloth inside a Pump

Takahiko Shirai

Waseda University, Japan

Foreign objects may enter pumps, such as those used for flood control in rivers. These objects cause pump clogging, leading to reduced pump efficiency and possible pump failure. Approximately 66% of the foreign objects consist of fibrous materials such as cloth. The behavior of the cloth is not understood in detail, which makes the development of clog-resistant pumps more challenging. The goal of this study is to understand the behavior of the cloth inside pumps through numerical analysis to aid in the development of clog-resistant pumps.

One of the main challenging points in this research is the representation of the Fluid – Structure Interaction (FSI) effect. In the pump, the deformation of the cloth is influenced by the pump flow, while the pump flow is affected by the cloth deformation. Capturing this effect is important for tracking the motion of the cloth in the pump. In this study, we focus on doing FSI computation using the mesh-moving technique and representing the FSI effect.

— — — — — — —

A New Regularity Criterion for the Boussinesq Equations in Vishik Space

Denghui Ren

Waseda University, Japan

In this talk, we provide a new regularity criterion for 3D Boussinesq equations, improving the previous result obtained by Omrane, Gala and Théra (2024). By using duality of Triebel-Lizorkin space and a logarithmic inter- polation inequality, we derive an extension criterion from Besov space to Vishik space. Besides, we provide a proof by using two components of vorticity instead of velocity.

A Fundamental Study on the Effect of Exhaust Gas Temperature and Fast Perturbation on Conversion Rate of the Three Way Catalyst

Masato Terasawa

Waseda University, Japan

Towards carbon neutrality, PHEVs (plug-in hybrid vehicles) and HEVs (hybrid electric vehicles) are becoming more popular. For hybrid electric vehicles, the spark ignition engine, which determines the cruising range, is an important component. In the future, spark ignition engines will have high thermal efficiency and low exhaust gas temperatures, so three-way catalysts that simultaneously purify CO, NOx, and THC will be required to have high purification performance even at lower exhaust gas temperatures. In addition, to secure space in the engine compartment of hybrid vehicles, it is also necessary to miniaturize the three-way catalyst. To improve the performance of three-way catalysts, a technology called perturbation (dithering, lambda switching) is used. This is a system that efficiently purifies the above three types of exhaust gases by suddenly switching the excess air ratio between rich and lean. In actual engine experiments, attention is paid to the perturbation parameters of frequency and amplitude, but there have been few studies focusing on exhaust gas temperature, and its mechanism has not been clarified. In this study, perturbations were applied using an actual engine, and the effects of frequency f, amplitude A, and temperature T were experimentally investigated. The experiments were carried out using a 4-cylinder 2.4L S.I. engine equipped with the TWC. The experiments were carried out at f=0-1.0 Hz, A=0-0.1, and T=70-350 ℃. The conversion rates of THC, CO, and NO and their conversion mechanisms were investigated at all frequencies with a low amplitude at T=270 ℃. The conversion rate of THC was over 80% at T=350 ℃, and the conversion rate of CO was nearly 100% at A = 0.02-0.08 and T=350 ℃at all frequencies. The effect of frequency and amplitude on the oxidation reaction rates of CO, NO, and THC was also investigated.

— — — — — — —

Generalized Constantin-Lax-Majda-DeGregorio equation

Shunsuke Fujita

Waseda University, Japan

We provide a mathematical analysis of a one-dimensional model of turbulence based on a stochastic generalized Constantin-Lax-Majda-DeGregorio (gCLMG) equation:

$$\omega_t + au\omega_x - u_x\omega = \nu\omega_{xx} + f, \quad u_x = \mathcal{H}(\omega),$$

where $\nu > 0$ describes viscosity, f is a noise and $\mathcal{H}$ is the Hilbert transform. In particular, the parameter a governs the strength of the advection term. We focus on the specific case $a = -2$, where the resulting nonlinearity allows the anomalous cascade of enstrophy, which is an inviscid conserved quantity, and some effective energy estimates. These estimates enable us to prove the global well- posedness of solutions in the mean-zero Sobolev space on the one-dimensional torus. Furthermore, we establish the existence of an invariant measure. The uniqueness of the invariant measure and the exponential mixing are proved under a sufficiently large viscosity condition. This is a joint work with Reika Fukuizumi (Waseda University) and Takashi Sakajo (Kyoto University).

Friday, December 12th, 2025

On the Stability of Arc-shaped and Circular Vortex Filaments

MASASHI AIKI

TOKYO UNIVERSITY OF SCIENCE, JAPAN

We consider a nonlinear model equation, known as the Localized Induction Equation, describing the motion of a vortex filament immersed in an incompressible and inviscid fluid. We investigate the long-time behavior of an arc-shaped vortex filament, which is an exact solution to an initial-boundary value problem for the Localized Induction Equation (LIE). We also study the long-time behavior of a circular vortex filament, which is an exact solution of the initial value problem for the LIE. These two types of filaments travel along a straight axis at a constant speed without change of shape.

Under certain assumptions, we prove that both types of filaments are stable in the Lyapunov sense except along the axis of travel, for which perturbations can grow linearly with respect to time.

We also show numerical evidence which suggest that the assumptions of the current results may be relaxed.

— — — — — — —

Computational methods for particle-laden flow analysis in turbomachinery blade erosion prediction

ALESSIO CASTORRINI

SAPIENZA UNIVERSITY OF ROME, ITALY

This presentation will review the development and application of advanced computational mechanics methods for predicting erosion in turbomachinery blades subjected to particle-laden flows, including both solid and liquid particles. We will first examine the modeling of particle-fluid interactions, from single-particle tracking to statistical cloud-based transport, and their implementation within finite-element-based Computational Fluid Dynamics solvers. The fluid dynamics is computed using stabilized formulations (SUPG/PSPG) for incompressible flows, initially combined with the Reynolds-Averaged Navier–Stokes (RANS) approach for turbulence modeling and later replaced by the Residual-Based Variational Multiscale (RB-VMS) method to improve the accuracy of unsteady turbulent flow simulations. We will then present the integration of particle-transport models with mesh-morphing techniques to capture blade surface evolution due to material removal or deposition. The latest developments will also be outlined, focusing on coupled simulations based on Space-Time VMS formulations of computational fluid dynamics and isogeometric analysis. These approaches enable the simultaneous resolution of the aerodynamic field and the evolving geometry with an improved accuracy in capturing geometrical damage patterns on the blade surface.

Applications to turbomachinery highlight the potential of these methods to enhance blade lifetime prediction, optimize coatings design, plan the maintenance, and support the development of more erosion-resistant components.