2013 SPRING PROGRESS IN MATHEMATICAL AND COMPUTATIONAL STUDIES ON SCIENCE AND ENGINEERING PROBLEMS

WORKSHOPS

CASTS

理論科學研究中心

Center for Advanced Study in Theoretical Sciences

2013. 5 . 6-8

Seminar Title: Direct Modeling for Computational Fluid Dynamics and Unified Gas-kinetic Scheme

Prof. Kun Xu

All partial differential equations (PDEs) for the description of flow motion have their intrinsic valid physical modeling scales, and these scales may not be compatible with the numerical mesh size scales. Theoretically, it is problematic in the numerical PDE methodology, where the PDE is directly discretized in the hope of developing reliable and consistent CFD methods. In this part, I will first introduce a new concept about direct modeling for the development of CFD methods, where the physical modeling will be done directly in a discretized space. The construction of the unified gas-kinetic scheme (UGKS) is an example under the above CFD principle. With discretized particle velocity space, a UKGS for the entire Knudsen number flow has been constructed based on the model Boltzmann equation. In comparison with other existing kinetic schemes for the Boltzmann equation, the current method has no difficulty to get accurate Navier-Stokes solutions in the continuum flow regime with the time step being much larger than the particle collision time, and to capture rarefied flow phenomena, even the free molecular flow. The unified scheme is an extension of the gas-kinetic BGK method for the Navier-Stokes solution to the rarefied flow regime with the discretization of particle velocity space. The success of the method is due to the coupling of particle transport and collision in the evaluation of local time-dependent gas distribution function at a cell interface. Due to this coupling, the gas distribution function will evolve from the free particle transport in the kinetic regime to the accumulating wave interaction in the hydrodynamic regime. Therefore, through the direct modeling, a whole continuum spectrum of fluid dynamic “PDEs” from the collisionless Boltzmann equation to the Navier-Stokes have been recovered, and the local solution used for the flow evolution depends on the ratio of numerical cell size to the molecular mean free path. Many numerical examples from the UGKS will be presented.

Tutorial Title: Introduction of Computational Fluid Dynamics and Gas-kinetic Schemes for Viscous Flow Computations

Prof. Kun Xu

In the first part, we will introduce the basic recipes of CFD methods for the compressible flow computations and the main reason for the capturing of discontinuous solutions. Then, the high-order CFD methods and their design principles will be presented. Some critical questions will be raised. At the same time, the gas-kinetic schemes for the Navier-Stokes flow computation will be introduced. The underlying physical process will be analyzed and compared with the traditional CFD methods. Then, the high-order gas-kinetic schemes will be presented. Other possible ways to construct high-order methods will also be discussed. Many numerical examples will be used to demonstrate the performance of the gas-kinetic schemes.

Seminar Title: Aerodynamics of Flexible, Flapping Wings: Computational Fluid-Structure Interactions and Scaling Analysis

Prof. Wei Shyy

Effects of flexibility on the force generation and propulsive efficiency of flapping wings are elucidated. For a moving body immersed in viscous fluid, different types of forces, as a function of the Reynolds number, reduced frequency (k), and Strouhal number (St), acting on the moving body are identified based on a scaling argument. We will present a computational framework for simulating structural models of varied fidelity and a Navier-Stokes solver, aimed at simulating flapping and flexible wings. Based on the order of magnitude and energy balance arguments, a relationship between the propulsive force and the maximum relative wing tip deformation parameter (γ) is established. The parameter depends on the density ratio, St, k, natural and flapping frequency ratio, and flapping amplitude. The established scaling relationships can offer direct guidance for micro air vehicle design and performance analysis. Systematic comparisons with rigid wings illustrate that the nonlinear response in wing motion results in a greater peak angle compared to a simple harmonic motion, yielding higher lift. Moreover, the compliant wing streamlines its shape via camber deformation to mitigate the nonlinear lift-degrading wing-wake interaction to further enhance lift. These aeroelastic mechanisms broaden our understanding of insect flight aerodynamics and can contribute to the development of flapping wing micro-robots.

Tutorial Title: Computational Methods for Fluid Flows with Moving Boundaries

Prof. Wei Shyy

Many biomedical and engineering problems related to fluid flows involve moving boundaries. For such cases, although the governing laws and computational procedures for the fluid and solid domains are well developed, treatment of the whole physical system. From the continuum mechanics viewpoint, the phase interface is a discontinuity and must be addressed as an integral part of the solution procedure. Within the context of finite grid resolution, this discontinuity needs to be accurately tracked in time and in space. In this tutorial talk, the Eulerian-Lagrangian method will be emphasized. Specifically, one can consider the interface as either a sharp discontinuity, consistent with the continuum theory, or a smooth transition zone, reducing numerical difficulties in tackling distinct regions which move and change shapes. Computationally, both approaches can be devised using similar concepts, namely, the interface is represented by marker points and advected in a Lagrangian framework, and the mass, momentum, and energy conservation equations are solved on a fixed (Eulerian) Cartesian grid. The main difference lies in the way to account for the interfacial conditions, communication across the interface, and the resulting impact on the data structure and assembly of the discretized equations. The sharp interface method is more demanding computationally because the field equations in each zone need to be coupled while explicitly tracking the interfacial conditions via matching procedures. In return, better accuracy can be attained compared to that of the continuous interface method. Overall, both approaches are valuable in handling multi-fluid problems with moving boundaries. Selected examples from multiphase dynamics, biomechanics and other engineering applications will be presented to illustrate salient features of the numerical techniques and physical characteristics.

Seminar Title: Some Advances in Development and Applications of Immersed Boundary Method

Prof. Chang Shu

Immersed boundary method (IBM) is an efficient approach for simulation of flows around complex geometries/moving objects. Its major advantage is simplicity and easy implementation. The strong coupling between the solution of governing equations and implementation of boundary conditions is considered through a forcing term in the governing equations and the iterative process. Currently, the application of IBM is limited to the incompressible viscous flows. The numerical results may also encounter flow penetration problem due to unsatisfying of no-slip boundary conditions. This talk will report our recent progress in enforcing no-slip boundary conditions in IBM, and IBM for Neumann boundary conditions and compressible inviscid flows. Some applications of IBM for moving boundary flows, multiphase flows and thermal flows are presented.

Tutorial Title: Progress in Development and Aplication of Lattice Boltzmann Method and Lattice Boltzmann Flux Solver

Prof. Chang Shu

As an alternative computational fluid dynamics (CFD) approach, lattice Boltzmann method (LBM) receives more and more attention in recent years. LBM is a particle-based approach, which does not involve the solution of partial differential equations and their resultant algebraic equations. Thus, its implementation and coding are very simple. Currently, LBM has been widely applied to simulate various fluid flow problems. This talk will report our progress in the development and application of LBM. It covers almost all areas of LBM. It includes application of LBM for simulation of isothermal and thermal flows, single phase and multiphase flows, incompressible and compressible flows, and micro flows. In particular, the talk will discuss how to develop various LBM models for problems with complex geometry, micro flow, multiphase flow and compressible flow simulation. Recently, we presented a new solver, that is, lattice Boltzmann flux solver (LBFS), for simulation of incompressible viscous and inviscid flows. The new solver combines the advantages of conventional Navier-Stokes solver and lattice Boltzmann solver, in which the conservative flow variables at the cell center are obtained from the solution of macroscopic governing equations while the fluxes at the cell interface are evaluated by local reconstruction of LBM solution. LBFS overcomes all drawbacks of LBM, which can be easily applied to solve viscous and inviscid flows with non-uniform mesh and curved boundary. The development and application of LBFS will also be addressed in the talk.

Seminar Title: Large- and small-scale dynamics in turbulent Rayleigh-Bénard convection

Prof. Penger Tong

Turbulent thermal convection, where warm fluid rises and cold fluid falls and their mixing produces convective turbulence, is a common phenomenon which often occurs at large scales. Natural convections in the atmosphere and oceans and in industrial heat exchangers are pertinent examples. In the laboratory, one conducts controlled experiments in an idealized Rayleigh-Bénard convection (RBC) cell, where a fluid layer is heated from below and cooled from the top. In this lecture, I will review the recent development in this area and report our recent experimental studies of the large-scale dynamics of turbulent RBC under different geometry [1,2] and small-scale properties of turbulent RBC [3,4]. *Work supported by the Research Grants Council of Hong Kong SAR. [1] “Scaling laws in turbulent Rayleigh-Benard convection under different geometry,” Hao Song and Penger Tong, Europhys. Letters 90, 44001 (2010). [2] “Coherent oscillations of turbulent Rayleigh-Bénard convection in a thin vertical disk,” Hao Song, E. Villermaux, and Penger Tong, Phys. Rev. Lett. 106, 184504 (2011). [3] “Small-scale turbulent fluctuations beyond Taylor’s frozen-flow

hypothesis," X.-Z He, G.-W He, and P. Tong, Phys. Rev. E 81, 065303(R) (2010). [4] "Locally-averaged thermal dissipation rate in turbulent thermal convection: A decomposition into contributions from different temperature gradient components," Xiaozhou He, Emily S. C. Ching, and Penger Tong, Phys. Fluids 23, 025106 (2011).

Tutorial Title: Large- and small-scale dynamics in turbulent Rayleigh-Bénard convection

Prof. Penger Tong

Turbulent thermal convection, where warm fluid rises and cold fluid falls and their mixing produces convective turbulence, is a common phenomenon which often occurs at large scales. Natural convections in the atmosphere and oceans and in industrial heat exchangers are pertinent examples. In the laboratory, one conducts controlled experiments in an idealized Rayleigh-Bénard convection (RBC) cell, where a fluid layer is heated from below and cooled from the top. In this lecture, I will review the recent development in this area and report our recent experimental studies of the large-scale dynamics of turbulent RBC under different geometry [1,2] and small-scale properties of turbulent RBC [3,4]. *Work supported by the Research Grants Council of Hong Kong SAR. [1] “Scaling laws in turbulent Rayleigh-Benard convection under different geometry,” Hao Song and Penger Tong, Europhys. Letters 90, 44001 (2010). [2] “Coherent oscillations of turbulent Rayleigh-Bénard convection in a thin vertical disk,” Hao Song, E. Villermaux, and Penger Tong, Phys. Rev. Lett. 106, 184504 (2011). [3] “Small-scale turbulent fluctuations beyond Taylor’s frozen-flow

hypothesis," X.-Z He, G.-W He, and P. Tong, Phys. Rev. E 81, 065303(R) (2010). [4] "Locally-averaged thermal dissipation rate in turbulent thermal convection: A decomposition into contributions from different temperature gradient components," Xiaozhou He, Emily S. C. Ching, and Penger Tong, Phys. Fluids 23, 025106 (2011).

Tutorial Title: Space–Time Formulation and Fluid–Structure Interaction Techniques

Prof. Kenji Takizawa

Since its introduction in 1991 for computation of flow problems with moving interfaces, the Deforming- Spatial-Domain/Stabilized Space–Time (DSD/SST) formulation [1, 2, 3, 4] has been applied to a diverse set of challenging problems. The classes of problems computed include free-surface and two-fluid flows, fluid–object, fluid–particle and fluid–structure interaction (FSI), and flows with mechanical components in fast, linear or rotational relative motion. The DSD/SST formulation, as a core technology, is being used for some of the most challenging FSI problems, including parachute modeling and arterial FSI. Versions of the DSD/SST formulation introduced in recent years serve as lower-cost alternatives. More recent variational multiscale (VMS) version, which is called DSD/SST-VMST (and also ST-VMS) [5], has brought better computational accuracy and serves as a reliable turbulence model. The VMS components are from [6, 7]. Special ST FSI techniques introduced for specific classes of problems, such as parachute modeling and arterial FSI, have increased the scope and accuracy of the FSI modeling in those classes of computations. This lecture provides an overview of the core ST FSI technique and its recent versions [8]. References [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] T.E.

Tezduyar, M. Behr, and J. Liou, “A new strategy for finite element computations involving moving boundaries and interfaces – the deforming-spatial-domain/space–time procedure: I. The concept and the preliminary numerical tests”, Computer Methods in Applied Mechanics and Engineering, 94 (1992) 339–351, doi: 10.1016/0045-7825(92)90059-S. [3] T.E. Tezduyar, M. Behr, S. Mittal, and J. Liou, “A new strategy for finite element computations involving moving boundaries and interfaces – the deforming-spatial-domain/space–time procedure: II. Computation of free-surface flows, two-liquid flows, and flows with drifting cylinders”, Computer Methods in Applied Mechanics and Engineering, 94 (1992) 353–371, doi: 10.1016/0045- 7825(92)90060-W. [4] 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. [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] T.J.R. Hughes, “Multiscale phenomena: Green’s functions, the Dirichlet-to-Neumann formulation, subgrid scale models, bubbles, and the origins of stabilized methods”, Computer Methods in Applied Mechanics and Engineering, 127 (1995) 387–401. [7] T.J.R. Hughes, A.A. Oberai, and L. Mazzei, “Large eddy simulation of turbulent channel flows by the variational multiscale method”, Physics of Fluids, 13 (2001) 1784–1799. [8] Y. Bazilevs, K. Takizawa, and T.E. Tezduyar, Computational Fluid–Structure Interaction: Methods and Applications. Wiley, 2013.

Seminar Title: Applications of Space–Time Fluid–Structure Interaction Techniques

Prof. Kenji Takizawa

We provide an overview of the real-world computations [1] by the Team for Advanced Flow Simulation and Modeling (TFAFSM). The two main examples are parachute and arterial FSI. Computer modeling of spacecraft parachutes, which are quite often used in clusters of two or three large parachutes, involve FSI between the parachute canopy and the air, geometric complexities created by the construction of the parachute from "rings" and "sails" with hundreds of gaps and slits, and the contact between the parachutes. A number of special FSI techniques were introduced in [2, 3, 4, 5], which very much increased the scope of our parachute modeling. Many of these special techniques are in the category of interface projection techniques, such as the FSI Geometric Smoothing Technique (FSI-GST) [2], Separated Stress Projection (SSP) [3], Homogenized Modeling of Geometric Porosity (HMGP) [3], "symmetric FSI" technique [5], accounting for fluid forces acting on structural components (such as parachute suspension lines) that are not expected to influence the flow [5], and other interface projection techniques [4]. We also present an extensive comparative study based on patient-specific FSI modeling of cerebral aneurysms. A number of special techniques were developed also for arterial FSI. These include techniques for calculating an estimated zero-pressure (EZP) arterial geometry [4], a special mapping technique for specifying the velocity profile at an inflow boundary with non-circular shape [6], techniques for using variable arterial wall thickness [4], mesh generation techniques for building layers of refined fluid mechanics mesh near the arterial walls [4], and techniques [4] for the projection of fluid– structure interface stresses, calculation of the wall shear stress (WSS) and calculation of the oscillatory shear index (OSI). References [1] Y. Bazilevs, K. Takizawa, and T.E. Tezduyar, Computational Fluid–Structure Interaction: Methods and Applications. Wiley, 2013. [2] T.E. Tezduyar and S. Sathe, “Modeling of fluid–structure interactions with the space–time finite elements: Solution techniques”, International Journal for Numerical Methods in Fluids, 54 (2007) 855–900, doi: 10.1002/fld.1430. [3] T.E. Tezduyar, S. Sathe, J. Pausewang, M. Schwaab, J. Christopher, and J. Crabtree, “Interface projection techniques for fluid–structure interaction modeling with moving-mesh methods”, Computational Mechanics, 43 (2008) 39–49, doi: 10.1007/s00466-008-0261-7. [4] K. Takizawa, C. Moorman, S. Wright, J. Christopher, and T.E. Tezduyar, “Wall shear stress calculations in space–time finite element computation of arterial fluid–structure interactions”, Computational Mechanics, 46 (2010) 31–41, doi: 10.1007/s00466-009-0425-0. [5] T.E. Tezduyar, K. Takizawa, C. Moorman, S. Wright, and J. Christopher, “Space–time finite element computation of complex fluid–structure interactions”, International Journal for Numerical Methods in Fluids, 64 (2010) 1201–1218, doi: 10.1002/fld.2221. [6] K. Takizawa, J. Christopher, T.E. Tezduyar, and S. Sathe, “Space–time finite element computation of arterial fluid–structure interactions with patient-specific data”, International Journal for Numerical Methods in Biomedical Engineering, 26 (2010) 101–116, doi: 10.1002/cnm.1241.