Thermal & Fluid Sciences Industrial Affiliates and Sponsor Conference

February 1-3, 2017

Book of Abstracts

TFSA17

DAY 1	
	SESSION I – PSAAP PROGRAM
➢	OVERVIEW: Prof. Gianluca Iaccarino
Title:	**Modeling of particle-laden wall turbulence**
Authors:	Mahdi Esmaily, Hoora Abdehkakha
Abstract:	The study of particle-laden wall turbulence is of prime importance because of its relevance to numerous engineering and scientific applications. We are particularly interested in this problem because of its relevance to characterization and simulation of particle-based solar receivers. The concept behind this new generation of solar receivers is to collect radiative energy volumetrically through dispersed particles rather than the conventional approach of absorption via a surface. In the design under investigation at our center, we explore this concept by seeding nickel particles in a turbulent flow of air in a square duct and exposing the mixture to radiative heat flux. In this setting, particles are heated through radiation, warming the surrounding air that is used for power generation. To make this design practical, the temperature in the near wall region must be maintained below a certain limit. However, due to a phenomenon called turbophoresis, particles interacting with turbulence migrate toward the solid boundaries and produce very high temperatures at the wall by absorbing a significant amount of radiation in a small region. This high local temperature depends primarily on the particle concentration in the viscous sublayer, which its correct prediction relies on proper modeling of turbophoresis. Hence, in this study, we investigate the effect of various parameters on turbophoresis through direct numerical simulations of turbulent flow laden with Lagrangian point-particles. We consider a flow of air in a square duct at a bulk Reynolds number of 5,000 to 20,000 dispersed with 4- to 16-micron nickel particles. We examine the effect of the Stokes and Reynolds numbers as well as collision modeling on the near wall concentration. We discuss two collision modeling approaches: 1) hard sphere and 2) stochastic model. We show that accounting for the collisions can change the near-wall concentration by orders of magnitude. The sensitivity of the results to the coefficient of restitution will be demonstrated.
Bio:	Mahdi Esmaily-Moghadam received his B.S. and M.Sc. in Mechanical Engineering from the Sharif University of Technology, Tehran, Iran, and his Ph.D. from the University of California, San Diego, working on the development of multiscale methods for optimization of surgical techniques for single ventricle heart patients. He is currently a postdoctoral scholar at the Center for Turbulence Research at Stanford University, studying particle-based solar receivers. He has an interdisciplinary background in areas of computational and cardiovascular mechanics, particle-laden flows, finite-element analysis, shape optimization methods, high-performance computing, and linear algebraic solvers.
Title:	**High-fidelity simulations of radiation transport**
Authors:	Ari Frankel, Hilario Torres, Ali Mani, Gianluca Iaccarino
Abstract:	In particle-based solar receivers, a concentrated beam of sunlight is focused on a dispersed particle phase falling in air, and the particles absorb and scatter the sunlight and warm up. The particles can also shadow each other, effectively attenuating the amount of radiation that penetrates through the receiver. The governing equation to describe the sunlight-particle interaction is the radiative transfer equation, a differo-integral equation in 3 spatial and 2 angular variables. Solving the radiative transfer equation in conjunction with the equations for fluid motion and Lagrangian particle tracking is expensive and requires careful treatment of the particle properties. In this talk we provide an overview of radiation-particle modeling in the context of state-of-the-art high-fidelity simulations and application to particle-based solar receivers. We provide an outline of a solution strategy for the radiative transfer equation in a discrete particle phase and radiation-particle energy coupling using the discrete

	ordinates method. We then turn to the treatment of illumination sources using the method of first collisions and demonstrate verification against Monte Carlo ray tracing. Finally, results from ongoing simulation campaigns of a particle-based solar receiver prototype are demonstrated, comparing temperature results using the discrete ordinates method against the optically thin approximation.
Bio:	Ari Frankel is a 5th year PhD student at Stanford University in the Mechanical Engineering Department, working under Profs. Gianluca Iaccarino and Ali Mani. He received his BS from MIT in 2012 and his MS from Stanford in 2013. His research interests include radiation transport, heat transfer, fluid dynamics, and uncertainty quantification.
Title:	**Subgrid-scale modeling and wavelet analysis of particle-laden turbulent flows**
Authors:	Maxime Bassenne, Javier Urzay, George I. Park, Parviz Moin
Abstract:	A new dynamic model is proposed for large-eddy simulations (LES) of small inertial particles in turbulent flows. The model is simple, involves no significant computational overhead, and is flexible enough to be deployed in any type of flow solvers and grids, including unstructured setups. The approach does not require any tunable parameters and is based on the use of elliptic differential filters. The performance of the model is tested in LES of isotropic turbulence laden with particles, where improved agreement with direct numerical simulation (DNS) results are observed in the dispersed-phase statistics, including particle acceleration, local carrier-phase velocity, and preferential-concentration metrics. Additionally, a method referred to as coherent cluster extraction (CCE) is described that decomposes a Eulerian particle number-density field into the sum of a coherent (organized) and an incoherent (disorganized) components, which are computed using wavelet filtering. The CCE method is applied to instantaneous snapshots of number-density fields obtained from DNS of isotropic turbulence laden with Lagrangian inertial particles. The coherent component structurally bears the clusters of particles at a strikingly low compression rate. On the contrary, the incoherent component tends to homogeneously fill the space and shows no clear spatial correlation. A direct application of the CCE method to radiative heat-transfer simulations is described in the form of a grid-adaptation algorithm, in such a way that the particles within each control volume follow an approximate random spatial distribution. The adaptation algorithm reduces the grid size by two orders of magnitude and might enable faster computation of radiation heat-transfer problems.
Bio:	Maxime is currently a Ph.D. student in Mechanical Engineering at Stanford. Prior to joining Stanford, Maxime earned a M.Sc. in Engineering at Ecole Centrale Paris. He now works on developing predictive computational tools to investigate particle-laden turbulent flows. Maxime is passionate about education, which fueled his vision to complete a Ph.D. degree in order to become a faculty.
Title:	**Towards predictive point-particle models**
Authors:	Jeremy Horwitz, Swetava Ganguli, Sanjiva Lele, Ali Mani, Mohammad Mehrabadi, Shankar Subramaniam
Abstract:	The point-particle method is a tool for tracking pieces of dispersed phase material and their respective interactions with a carrier fluid. As particles move through a fluid, they exchange momentum and energy (kinetic and thermal) with the surrounding phase. In the point-particle method, the Navier-Stokes equations are solved for the fluid phase and each particle is tracked in its own Lagrangian frame. Coupling between the phases however is not directly enforced through the boundary conditions at particle surfaces but rather through regularized point-sources of momentum and energy. Some questions with regard to point-particle methods are firstly, what should be the model forms of these point-sources, meaning how do they depend on particle and fluid parameters (,,…)? Secondly, what is the numerical implementation required to achieve verifiability, that is, in a regime where the point-source model is known to be valid, what numerical methods are required to ensure the computational results match theory? Thirdly, how good are verifiable models in regimes where they were not derived? The latter is the validation question. This work explores simulations of point-particles and fully resolved particles with and without heat transfer in an effort to address the former questions. Verification of the point-particle algorithm for momentum and energy exchange in the absence of heat transfer is explored in the context of a particle settling in an otherwise unbounded fluid. For the validation case, the verifiable procedure is then used to simulate decaying particle-laden homogeneous isotropic turbulence. The second part of the study is concerned with extracting point-source models from

	particle-resolved simulation. We examine how steady drag and heat transfer between a particle and fluid change in the presence of gravity and variable fluid density. We first quantify these effects on a single particle. We take our study further by adding interactions between particles placed in a periodic lattice whose parameters we control. We use particle resolved simulations which are agnostic to the Boussinesq regime and use a fully unstructured, node-based, low-Mach, variable density solver to study the low-Mach response. Defining the Boussinesq parameter (λ) as the ratio of the difference of the particle temperature and the far-field fluid temperature to the far-field fluid temperature, the heating of the fluid near the particle affects the drag significantly which can be characterized in a parameter space where the variation in Reynolds number (Re), λ and the Froude number can be collapsed to a single parameter. Despite the large (> 50%) drag changes, the pressure and viscous fractional contributions do not vary with λ. In the low Re limit, a semi-analytical low Mach perturbation expansion has significant (> 90%) explanatory power. For a single particle, these variations can be captured with 95% accuracy by developing correlations based on physical insights from the semi-analytical model. When particles are placed within a lattice, depending on the lattice parameter, the individual wakes of the particles interact. The drag felt by the particles increases or decreases depending on its position in the lattice.
Bio:	Jeremy Horwitz received a B.S. in Mechanical Engineering from Cornell University in 2011 and an M.S. in Mechanical Engineering from the University of Illinois at Urbana-Champaign in 2013. He is a 4th year Ph. D. student working with Professor Ali Mani on verification and validation of point-particle models for numerical simulation of particle-turbulence interaction. He is a recipient of a 2013 National Science Foundation Graduate Research Fellowship.
	SESSION II – REACTIVE FLOWS
➢	OVERVIEW: Prof. Matthias Ihme
Title:	**Low-emission combustion in porous media burners**
Authors:	Sadaf Sobhani, Danyal Mohaddes, Priyanka Muhunthan, Emeric Boigne, Jared Dunnmon, Matthias Ihme
Abstract:	As emission regulations become increasingly more stringent and policies evolve to combat global climate change impacts, reducing pollutant emissions emerges as one of the most important goals of combustion research. Low emissions, enhanced flame stabilization, and improved fuel efficiency in industrial applications can be achieved through the implementation of advanced combustion concepts, such as porous media combustion. In this study, the operational stability of combustion in porous media burners (PMBs) is examined computationally and experimentally through conventional and advanced diagnostics. Using a 1D volume-averaged model, a novel design is proposed for the porous structure that results in a further extension of flame stability and decrease in pressure drop. These results reinforce concepts in PMB design and optimization and demonstrate the potential of PMBs to overcome technological barriers associated with conventional free-flame combustion technologies.
Bio:	Sadaf is currently a 3rd year Ph.D. student developing computational tools to investigate alternative, low-emission matrix-stabilized combustion systems. During her 6 years at Stanford so far, she has earned Bachelor's and Master's degrees in Mechanical Engineering. She is a recipient of the Stanford Enhancing Diversity in Graduate Education (EDGE) Fellowship, the Graduate Public Service (GPS) Fellowship, and the National Science Foundation (NSF) Graduate Research Fellowship.
Title:	**Adaptive and efficient combustion modeling for turbulent reacting flows**
Authors:	Hao Wu, Qing Wang, Jeff Labahn, Matthias Ihme

Abstract:	The utilization of detailed kinetics in the LES of turbulent reacting flows has attracted increasing interest in recent years. It enables further improvement in the prediction of emission quantities in the combustion process and helps eliminate the restriction often imposed by models using lower-dimensional manifolds, such as variants of flamelet models. However, the cost of such approaches remains a great concern despite the continuing growth of computational capacity, as they are often orders-of-magnitude more expensive than flamelet-type calculations. The present series of work focuses on three key areas to address these issues: adaptive modeling for combustion, efficient computation for detailed kinetics, and general-purpose closure for sub-grid-scale turbulence-combustion interaction. Adaptive modeling for combustion is realized through the Pareto-efficient combustion (PEC) framework. The PEC framework allows the strategic usage of finite-rate kinetics in combination with lower-cost models such as the flamelet/progress variable model (FPVA). For complex chemically reacting flows, the PEC framework makes sub-model assignments under consideration of user-specific input about quantities of interest, desired simulation accuracy and computational cost. As a result, the local combustion process are modeled with adequately fidelity avoiding excessive cost. Efficient computation for detailed kinetics is essential for combustion simulations with high chemical fidelity with and without PEC adaptation. High efficiency needs to be achieved for all computational tasks related to the detailed kinetic scheme, including the evaluation of transport properties and chemical source term as well as the integration of stiff chemical systems. A semi-implicit (Rosenbrock-Krylov) scheme is designed and found to be particularly effective in achieving explicit-like cost, i.e. linear in space and time, while being sufficiently stable in face of the chemical stiffness arise in high-fidelity simulations of turbulent reacting flows. The final area of development is on closure models for sub-grid-scale turbulence-combustion interaction, of which the applicability is not limited by the type of combustion models. A regularized deconvolution method (RDM) is proposed for this purpose. With this technique, filtered scalars are reconstructed through deconvolution, while ensuring essential conservation and boundedness conditions. The chemical source terms that are computed directly from the deconvolved scalars are filtered explicitly to represent the turbulence-chemistry interaction.
Bio:	Hao Wu received his B.S. in Mechanical Engineering from Purdue University and Shanghai Jiao Tong University in 2012 and his M.S. from Stanford in 2014. He is currently in the 5th year of his PhD working with Professor Matthias Ihme on modeling and simulations of turbulent reacting flows.
Title:	**Advances on the fundamental understanding and modeling of trans- and supercritical flows**
Authors:	Daniel Banuti, Peter Ma, Hao Wu, Muralikrishna Raju, Matthias Ihme
Abstract:	Propellant injection at trans- and supercritical pressures is a technology prevalent in Diesel engines, gas turbines, and rocket engines. Beyond the critical point, the surface tension vanishes, and the subcritical break-up into ligaments and droplets is replaced by a continuous diffuse mixing process, resulting in steep gradients in temperature and variation in thermodynamic properties. This presentation provides an overview about recent research progress on the modeling of these transcritical flow regimes. Continuum-scale modeling in large-eddy-simulations is hindered by the large variations of density and other fluid properties across a thin transitional zone. Here, we discuss our approach, based on three main constituents: (i) an entropy-stable flux formulation ensures positivity across large density gradients; (ii) a double-flux method for transcritical flows minimizes pressure oscillations inherent in any conservative Godunov-type scheme; (iii) a transcritical flamelet/progress variable approach with partially tabulated thermodynamics allowing for efficient evaluation of real fluid reactive and inert mixtures. The model is successfully validated against canonical test cases of transcritical mixing, before an outlook towards technical applications is given.
Bio:	Dr. Daniel Banuti studied Mechanical Engineering at RWTH Aachen University, Germany. After spending 2005 as a Graduate Research Associate at the University of Tennessee Space Institute, he returned and received his MSc in 2006, specializing in space propulsion and computational fluid dynamics. Daniel then joined the Institute of Aerodynamics and Flow Technology of the German Aerospace Center (DLR) in Gottingen as a Research Scientist, where he was involved in various research projects in hypersonics and numerical propulsion. Pursued in parallel, he received his PhD in Aerospace Engineering from Stuttgart University in 2014 for his work on injection in cryogenic rocket engines. A year ago, he left DLR as a Senior

	Research Scientist to join the Center for Turbulence Research as Postdoctoral Fellow, where he is researching the fundamentals of supercritical fluid injection.
Title:	**HyChem Approach to Combustion Chemistry of Jet Fuels**
Authors:	Rui Xu, Hai Wang
Abstract:	Jet fuels are multicomponent hydrocarbon mixtures that may contain over thousands of molecular components. The composition can be complex and cannot be precisely defined. In this work, we introduce an unconventional approach to unraveling the high-temperature combustion chemistry of multicomponent jet fuels. In flames, large hydrocarbon fuels will undergo oxidation in two steps. In the preheat zone of the flame, the fuel decomposes to a small set of low-molecular weight fragments in an endothermic process, regardless how complex the composition of the initial fuel is. The fragments then enter into the flame zone and are oxidized to combustion products in an exothermic step, which is rate limiting. Because of the fast kinetic rate of the overall pyrolysis as compared to the rate of oxidation process, the distribution of the pyrolysis products impacts radical buildup and heat release. In the current work, the hybrid chemistry (HyChem) approach decouples the fuel pyrolysis process from the oxidation of fuel decomposition intermediates. The thermal decomposition and oxidative thermal decomposition processes are modeled by several lumped reactions in which the stoichiometric and reaction rate coefficients can be derived from experiments, while the oxidation process is described by a detailed foundational fuel chemistry model. Modeling results supports that the HyChem models are able to predict a wide range of key combustion properties, and is by far the most accurate and efficient approach to predicting multicomponent jet fuel combustion chemistry.
Bio:	Rui Xu is currently a 3rd year Ph.D. student working with Professor Hai Wang. Before coming to Stanford University, he received his B.S. in Mechanical Engineering from Shanghai Jiao Tong University (Shanghai, China) in 2012, and his M.S. in Mechanical Engineering from Northwestern University in 2014. His current research is focused on developing chemical kinetics models for multicomponent real fuels.
	SESSION III – CTR SUMMER PROGRAM
Title:	2016 Center for Turbulence Research (CTR) Summer Program
Presenter:	Prof. Parviz Moin
	The sixteenth biennial Summer Program of the Center for Turbulence Research was held from June 26 to July 22, 2016. CTR hosted eighty-one participants from thirteen countries, including eighteen U.S. institutions. Thirty-two CTR staff members, including graduate students, postdoctoral fellows and faculty, worked alongside the participants and contributed to forty-four projects spearheaded during the Summer Program. The participants were selected based on their research proposals and their synergy with current scientific interests of CTR. The role of CTR continues to be that of providing a forum for the fundamental study of multi-physics turbulent flows for engineering analysis. The participants were divided into five groups: Multi-phase Flows, Combustion, Turbulence and Transition Physics, Large-Eddy Simulation (LES), and Uncertainty Quantification. The participants presented their accomplishments on July 22nd. This final event was attended by several colleagues from industry, academia, and government. The book of proceedings of the Summer Program is available in print for our Industrial Affiliates, and at http://ctr.stanford.edu. The 2016 Summer Program was sponsored by the US Air Force Office of Scientific Research (AFOSR), National Science Foundation (NSF), National Aeronautics and Space Administration (NASA), the Advanced Simulation and Computing Program of the Department of Energy's National Nuclear Security Administration, and the Office of Naval Research (ONR). The simultaneous commitment of five different federal agencies to funding the 2016 CTR Summer Program underscores the importance of understanding and modeling turbulent flows for addressing outstanding engineering challenges.

Title:	**On the use of LES for engineering design**
Author:	Sanjeeb Bose
Abstract:	The continued growth of supercomputing capabilities has allowed for the use of large-eddy simulation for aid the engineering design process. Simulation of realistic engineering systems introduce additional challenges associated with mutli-physics flows and complex geometries as well as computational cost restrictions stemming from the need to perform many calculations to explore the design space. We review the consequences of these constraints on how calculations are performed and analyzed, and the impact on closure modeling for LES. Some recent developments made to address these challenges will also be presented.

DAY 2	
SESSION IV – RENEWABLE ENERGY SYSTEMS	
➢	OVERVIEW: Prof. John Dabiri
Title:	**Large-scale field measurements of wind turbine aerodynamics**
Authors:	Ian D. Brownstein, John O. Dabiri
Abstract:	Laboratory studies and numerical simulations of wind turbines are typically constrained in how they can inform operational turbine behavior. Laboratory experiments are usually unable to match both pertinent parameters of full-scale wind turbines, the Reynolds number (Re) and tip speed ratio, using scaled-down models. Additionally, numerical simulations of the flow around wind turbines are constrained by the large domain size and high Re that need to be simulated. When these simulations are performed, turbine geometry is typically simplified resulting in flow structures near the rotor not being well resolved. In order to bypass these limitations, a quantitative flow visualization method was developed to take in situ measurements of the flow around wind turbines at the Field Laboratory for Optimized Wind Energy (FLOWE) in Lancaster, CA. The apparatus constructed was able to seed an approximately 9m x 9m x 5m volume in the wake of the turbine using artificial snow. Quantitative measurements were obtained by tracking the evolution of the artificial snow using a four-camera setup. The methodology for calibrating and collecting data, as well as preliminary results detailing the flow around a 2kW vertical-axis wind turbine (VAWT), will be presented.
Bio:	Ian Brownstein received his B.S. in Mechanical Engineering and Egyptian Archeology from Brown University in 2013 and his M.S. in Space Engineering from Caltech in 2014. He is currently in the 4th year of his PhD working for Professor John Dabiri studying aerodynamic interactions in arrays of vertical-axis wind turbines.
Title:	**Large-Eddy Simulation of vertical-axis wind turbine wakes**
Authors:	Mahdi Abkar and John O. Dabiri
Abstract:	In this study, large-eddy simulation (LES) combined with a turbine model is used to investigate the structure of the wake behind a vertical-axis wind turbine (VAWT). In the simulations, a recently developed minimum dissipation model is used to parameterize the subgrid-scale stress tensor, while the turbine-induced forces are modeled with an actuator line technique. The LES framework is first tested in the simulation of the wake behind a model straight-bladed VAWT placed in the water channel and then used to study the wake structure downwind of a full-scale VAWT sited in the atmospheric boundary layer. In particular, the self-similarity of the wake is examined, and it is found that the wake velocity deficit is well characterized by a two-dimensional elliptical Gaussian distribution. By assuming a self-similar Gaussian distribution of the velocity deficit, and applying mass and momentum conservation, an analytical model is developed and tested to predict the maximum velocity deficit downwind of the turbine.
Bio:	Mahdi Abkar is currently a postdoctoral scholar at the Center for Turbulence Research (CTR) at Stanford University. He received his Ph.D. in Mechanical Engineering from Ecole Polytechnique Federale de Lausanne (EPFL), Switzerland, in 2014. His research interests lie on environmental fluid mechanics, atmospheric turbulence, wind engineering and renewable energy.
Title:	**Machine Learning for Scalar Transport Modeling in Film Cooling Applications**
Authors:	Pedro Milani, Julia Ling, John Eaton
Abstract:	Film cooling is a widely used technique to reduce the temperature of gas turbine blades and increase their lifespan. Relatively cool air flows from an interior plenum through small holes in the blade, creating a protective layer on its outer surface. In these applications, it is important to know the temperature distribution resulting from the interaction between a hot main flow and a colder jet. However, high-fidelity simulations such as

	direct numerical simulations (DNS) and large-eddy simulations (LES) are too costly to be used in complex industrial geometries and current Reynolds-averaged Navier-Stokes (RANS) models yield poor temperature predictions. A novel approach for RANS modeling of the turbulent scalar flux is proposed, in which the simple gradient diffusion hypothesis (GDH) is assumed and a machine learning algorithm is used to infer an improved turbulent diffusivity field. The machine learning framework allows the use of high-fidelity simulation results in canonical configurations to inform RANS models. This approach is implemented using three distinct data sets: two are used to train the machine learning algorithm and the third is used for validation. The results show that the proposed method produces significant improvement compared to the common RANS model, especially in the prediction of film cooling effectiveness.
Bio:	Pedro Milani received his B.S. in Mechanical Engineering with a minor in Computer Science from Stanford University in 2015. He is currently a second year PhD student at Stanford University, working with Prof. John Eaton and Dr. Julia Ling, who is currently at Sandia National Labs. He is excited to be exploring a very new field, the intersection of machine learning and fluid mechanics. His work consists of using data driven algorithms to improve turbulence modeling, particularly in configurations relevant to the turbomachinery community. Previously, Pedro also worked with Prof. Ali Mani on two-phase flow problems.
Title:	**Modeling and simulations of vertically-staggered multi-rotor wind farms**
Authors:	<u>Niranjan S. Ghaisas</u>, Aditya S. Ghate and Sanjiva K. Lele
Abstract:	We investigate a vertically-staggered arrangement of wind turbine rotors using large-eddy simulation (LES) and previously developed single-column top-down analytical models. The LES code uses a pseudo-spectral discretization in the horizontal directions, a sixth-order compact scheme in the vertical direction, the Sigma eddy-viscosity subgrid-scale model, and actuator representations for modeling the forces exerted by the wind turbine rotors. The top-down models assume that classical similarity theory holds away from the turbine rotors, and that the turbine-rotor regions can be described using an added wake-eddy-diffusivity, to predict the mean velocity in the fully-developed wind-turbine array boundary layer, as a function of the surface roughness and the applied geostrophic forcing. A conventional, non-staggered arrangement of turbines (henceforth referred to as a '1-layer' wind farm), is compared to a vertically-staggered (or '2-layer') configuration, comprised of smaller turbine rotors packed twice as closely in the horizontal directions as in the 1-layer case. The top-down models indicate that for appropriate choices of the geometric design variables, the 2-layer configuration can generate significantly more power than the 1-layer wind farm, by employing a larger number of rotors that are up to half the size of the turbine rotors in the 1-layer wind farm. LES of several 1-layer and equivalent 2-layer configurations operating in the deep-array regime are carried out and the results are found to be qualitatively similar to the top-down model predictions. The LES results are further used to investigate wake recovery and interactions between the wakes of the vertically-staggered multi-rotor wind turbines.
Bio:	Niranjan Ghaisas is a CTR postdoc since July 2015, working with Prof. Sanjiva Lele, on Eulerian modeling of solids coupled to fluids and wind farm simulations. He completed his Bachelor's degree from Indian Institute of Technology Kharagpur in 2006 and PhD from Purdue University in 2013.
	SESSION V – LARGE EDDY SIMULATIONS
➢	OVERVIEW: Prof. Parviz Moin
Title:	**Wall modeling in Large-Eddy Simulation**
Authors:	<u>George Park</u>, Parviz Moin

Abstract:	Turbulence is the rule, not the exception, in many complex engineering systems. Accurate prediction of turbulent flows over complex wall geometries allows engineers to design fuel-efficient aircrafts, quieter drones, and less expensive, more productive turbomachineries and wind farms. Despite the fact that we know the exact governing equations of fluid motion, and that the techniques to solve them numerically in complex geometries have matured enough over the past decades, first-principle-based prediction methods (such as large-eddy simulation (LES)) have long been in disfavor for solving industry-strength problems due to the cost consideration. The purpose of this talk is to introduce such cost-related issues in high-fidelity numerical simulation of practical turbulent flows, and then to describe the speaker's coordinated activities toward predictive and affordable LES in geometrically flexible computational framework. Wall modeling does not attempt to resolve the computationally demanding near-wall region of turbulent flows directly, but instead it provides an alternative boundary closure that can augment the near-wall turbulence, represented with very coarse grids, to the correct state. In this talk, I will convey and expose the basic concepts and methodologies of wall modeling to the general audience. I will first introduce the basic philosophy of wall modeling in LES, and walk the audience through the state-of-the-art wall modeling techniques, showing examples of its application ranging from simple academic flows to an industry-strength problem involving a full 3-D aircraft.
Bio:	Dr. George Park is an engineering research associate in the Center for Turbulence Research (CTR) at Stanford University. He earned an undergraduate degree (B.S. in ME, 2009) at Seoul National University in Korea, and received his M.S. /PhD in Mechanical Engineering from Stanford University in 2011/2014 with Prof. Parviz Moin, specializing in unstructured-grid simulation of complex wall-bounded turbulent flows. Dr. Park subsequently conducted a postdoctoral research at CTR in LES modeling of particle-laden turbulent flows. His current research activities include LES wall modeling, particle-laden turbulent flows, non-equilibrium turbulent boundary layer, and surfactant transport in two-phase flows.
Title:	**Physics-based Enrichment of High Reynolds Number LES**
Authors:	<u>Aditya S. Ghate</u> and Sanjiva K. Lele
Abstract:	A new multiscale simulation methodology is introduced to facilitate efficient simulations of very high Reynolds number wall bounded flows. The two-simulation, one-way coupled, scale splitting methodology combining a) Non-linear wave space model using the Gabor Transform and spectral eddy-viscosity, b) Representation of the subfilter fields via a set of random modes, and c) Large Eddy Simulation using a robust subgrid scale model, is introduced. The viability of the methodology is investigated using two idealizations for the Planetary Boundary layer (PBL). In the first idealization, the surface layer is approximated using a uniform shear and a positive (stable) temperature gradient which makes the problem homogeneous. The high latitude Stable PBL used in GABLS1 intercomparison study (Beare et. al. BLM 2006) serves as the second idealization for the PBL, and it introduces Coriolis and Stratification effects, along with the inhomogeneity in the wall-normal direction. These idealizations help validate the two-simulation methodology, where comparisons are made in terms of statistics such as two-point space-time correlations, k-omega spectra and profiles of second order, one-point correlations.
Bio:	Aditya Ghate received his B.S. in Aerospace Engineering with a minor in Mathematics from the University of Kansas in 2012 and his M.S. in Aeronautics and Astronautics from Stanford in 2014. He is currently in the 5th year of his PhD working for Professor Sanjiva Lele on Large Eddy Simulations of Atmospheric flows, along with applications in Wind Energy.
Title:	**Investigation of slip wall boundary condition for wall-modeled Large-Eddy Simulation**
Authors:	<u>A. Lozano-Durán</u>, H. J. Bae and P. Moin
Abstract:	Wall models for large-eddy simulation are necessary to overcome the resolution requirements near the wall for high Reynolds number turbulent flows. In the present study, the slip wall boundary condition is examined (Bose and Moin, Phys. Fluids, 2014). The optimal slip length and its dependence on Reynolds number, grid size, subgrid scale model, etc. is investigated in turbulent channel flows up to Re_τ=4200. A new model based on the average streamwise momentum balance is tested and compared to the optimal slip length and the filtered direct numerical simulation results.

Bio:	Dr. Adrián Lozano-Durán received his PhD from the Technical University of Madrid in 2015 at the Computational Fluid Mechanics Lab. headed by Prof. J. Jiménez. His main research has focused on Computational Fluid Mechanics and the fundamental physics of Turbulence. Currently, he is a postdoctoral fellow at the Center for Turbulence Research at Stanford University working on Large Eddy Simulation and wall-modeling.
Title:	**Minimum-dissipation subgrid scale model for scalar transport and its applications**
Authors:	H. Jane Bae and Mahdi Akbar
Abstract:	Minimum-dissipation models are a simple alternative to the Smagorinsky-type approaches to parameterize the sub-filter scale turbulent fluxes in large-eddy simulation. A recently derived minimum-dissipation model for sub-filter stress tensor is the AMD model (Rozema et al., Phys. Fluids, 2015) and has many desirable properties. It is more cost effective than the dynamic Smagorinsky model, it appropriately switches off in laminar and transitional flows, and it is consistent with the theoretic sub-filter stress tensor on both isotropic and anisotropic grids. In this study, an extension of this approach to modeling the sub-filter scalar flux is proposed. The performance of the AMD model is tested in the simulation of a high Reynolds number, rough wall, boundary layer flow with a constant and uniform surface scalar flux. The simulation results obtained from the AMD model show good agreement with well-established empirical correlations and theoretical predictions of the resolved flow statistics. In particular, the AMD model is capable to accurately predict the expected surface-layer similarity profiles and power spectra for both velocity and scalar concentration.
Bio:	H. Jane Bae is a Ph. D. student in the ICME department studying under Professor Parviz Moin. She graduated from Caltech in 2012 with a B.S in Mathematics with honors. Her main research interests are modeling and optimization, with emphasis on time-parallel integration and wall-modeled large-eddy simulations.
Title:	**Modeling pre-transitional flow using PSE**
Authors:	Philipp Hack, Adrian Lozano-Durán, and Parviz Moin
Abstract:	The modeling of the laminar region and the prediction of the point of transition remain key challenges in the numerical simulation of boundary layers. The issue is of particular relevance for wall-modeled large eddy simulations which require 10 to 100 times higher grid resolution in the thin laminar region than in the turbulent regime. Our study examines the potential of the nonlinear parabolized stability equations (PSE) to provide an accurate, yet computationally efficient treatment of the growth of disturbances in the pre-transitional flow regime. The PSE capture the nonlinear interactions that eventually induce breakdown to turbulence, and can as such identify the onset of transition without relying on empirical correlations. Since the local PSE solution at the point of transition is a solution of the Navier-Stokes equations, it provides a natural inflow condition for large eddy and direct simulations and avoids unphysical transients. We show that in a classical H-type transition scenario, a combined PSE/DNS approach can reproduce the skin-friction distribution obtained in reference direct numerical simulations at significantly lower computational cost.
Bio:	Dr. Philipp Hack studied mechanical engineering with focus on fluid mechanics and numerical methods at TU Munich and ETH Zurich. He received his PhD from Imperial College London in 2014. His theoretical and computational work on transitional flows was awarded in 2013 with the EPSRC Doctoral Prize Fellowship and in 2015 with a DFG Research Fellowship. Philipp Hack joined the group of Parviz Moin at the beginning of 2015. Current research interests include flow stability, optimization and machine learning.
	SESSION VI - EXPERIMENTAL FLUID MECHANICS AND HEAT TRANSFER
➢	OVERVIEW: Prof. John Eaton
Title:	**Measuring 3D Particle Concentration in Complex Flows Using MRI**
Authors:	Daniel D. Borup, Christopher Elkins, John K. Eaton

Abstract:	Magnetic Resonance Imaging (MRI) is well suited for the study of fluid mechanics in complex flows where optical access is not possible. Current MRI-based techniques allow for the measurement of 3D, 3-component velocity and scalar concentration fields. The current work aims to develop and validate a technique for measuring the concentration of a dispersed solid microspheres in a turbulent water flow. Such a technique would provide a novel diagnostic to study the transport of small particles (e.g., atmospheric dust) in arbitrarily complicated biological, engineering, or natural flows. Examples of these flows include atmospheric dust ingested into gas turbine engine cooling systems, noxious or medicinal particles inhaled into the human airways, and sediment carried in coastal and estuarine waterways. MRI signal decays more rapidly in the presence of small paramagnetic particles than it does for pure water due to small disturbances in the magnetic field, and the decay rate can be obtained using existing MRI pulse sequences. Tests using static gel suspensions at known particle concentration produced good agreement with theoretical predictions of a linear relationship between particle concentration and decay rate. Three-dimensional concentration data were measured for a particle streak injected into a ribbed serpentine channel flow representing a simplified version of a gas turbine cooling passage. The particles were small enough that the behavior of the particle streak was similar to the behavior of a passive scalar tracer injected in the same manner. However, the data show that a non-uniform particle distribution is generated between the ribs as the particles deviate from the rapidly turning flow in these regions. Streaks of high particle concentration are generated and persist even as the flow turns away from the ribs. This result is perhaps the first example of quantitative 3D particle concentration measurements being used to discern particle transport phenomena in a complex turbulent flow, and showcases the potential of the new diagnostic as a tool for research, design engineering, and computational model validation.
Bio:	Daniel D. Borup received his B.S. in Engineering Mechanics from the University of Illinois at Urbana-Champaign in 2013, graduating with Highest Honors and also completing a minor in Computer Science. He received his M.S. in Mechanical Engineering from Stanford University in 2015. Daniel is currently a 4th year Ph.D. student working with Prof. John Eaton on the development of a new MRI-based method for particle concentration measurement, as well as other MRI-based experiments in turbulent flows.
Title:	**Conjugate Heat Transfer Measurements for a Film Cooling Jet**
Authors:	Pablo Vasquez Guzman, Chris Elkins, & John Eaton
Abstract:	Thermal measurements are essential for numerous industrial and scientific applications, especially in analyzing the heat transfer characteristics of a system to improve its performance, reliability, and efficiency. Conventional thermal measurement techniques, which are limited to providing data at discrete points or planes, provide accurate information, but are either intrusive or require direct optical access. Obtaining full-field temperature measurements using existing techniques is difficult and seldom done. For complex thermal-fluid flows, an accurate and detailed full-field understanding of the heat transfer characteristics is needed to optimize device performance and/or test computational models. This experimental study describes a quantitative measurement technique to analyze heat transfer based on Magnetic Resonance Imaging (MRI). This technique called MRT exploits the weak temperature dependence of the hydrogen proton resonant frequency to measure the 3D temperature field. The present work develops new methods to use MRT to study conjugate heat transfer (coupled convection and conduction) in turbulent flows. Additional MRI-based measurements of the 3D velocity field provide a complete picture of the thermal-fluid behavior. The methodology is applied to study conjugate heat transfer for a fully turbulent inclined jet mixing with a cross flow. A heated jet is fed into a cool mainstream through a hole in a thermally conductive wall. This causes local warming of the wall and cooling of the jet. The temperature field measurements are the first ever recorded for turbulent heat transfer with strong conduction effects. This heat transfer configuration has direct relevance to gas turbine cooling and will have application in testing of advanced models.
Bio:	Pablo A. Vasquez Guzman received his B.S in Mechanical Engineering from Columbia University in 2012 and M.S. in Mechanical Engineering from Stanford University in 2014. He is currently a 5th year Ph.D. candidate working under guidance of Professor John K. Eaton focusing on the

	application of modern medical imaging techniques in providing noninvasive detailed three-dimensional full-field information of complex thermal-fluid flows. His general research interest is in the development of technology for clean and efficient energy systems.
Title:	**Unsteady Measurements in 3D Separated Flows**
Authors:	David Ching, Christopher Elkins, John Eaton
Abstract:	A new method for examining unsteady flows using MRI techniques is used to examine the flow structure and geometric sensitivity of a turbulent boundary layer flowing over a skewed bump. The skewed bump exhibits sensitivity to the angle of the bump with respect to the flow. Using Magnetic Resonance Velocimetry (MRV), the mean flow at several bump angles is examined. When the bump is in a symmetric position, two counter-rotating vortices are formed in the separation bubble, but farther downstream there is a weak counter-rotating vortex pair in the opposite sense. For asymmetric cases, one of the vortices is stronger and overwhelms the other vortex leaving a single dominant vortex. A new technique called spiral MRV was developed to acquire a single component of velocity in a plane at short time intervals. The spiral MRV shows that the wake is highly unsteady and has periodic structures as well as broadband turbulence. The periodic structures are isolated and studied using a Proper Orthogonal Decomposition (POD) of the images. The spiral MRV shows that the magnitude of fluctuations is sensitive to the angle of the bump. The shedding frequency was also calculated using the time resolved results and compared to shedding frequencies found with hot-wire anemometry.
Bio:	David Ching is a third year graduate student working with Professor John Eaton on experimental studies of geometric sensitivity and jet film cooling. He received his B.S in Mechanical Engineering at New Jersey Institute of Technology in 2014 and his M.S. in Mechanical Engineering at Stanford in 2016.
Title:	**Effects of clustering on heat transfer in particle-laden turbulence**
Authors:	Hadi Pouransari, Ali Mani
Abstract:	Particle-laden flows are ubiquitous in variety of natural and industrial phenomena. Rain droplets in clouds, protoplanetary disks, and combustion chambers are examples in which particles are interacting with background turbulence. It is well known that interaction of particles and turbulent flow results in preferential concentration. The extent of preferential concentration depends on ratio of particle relaxation time and turbulent eddies time scale. In this work, we consider particle-laden turbulent flows, in which particles are heated. This is the case for example in the particle-based solar receivers where particles absorb external radiation and heat the background gas. We use three-dimensional variable density direct numerical simulations for the turbulent flow and Lagrangian point-particle tracking to study the implication of particle clustering in particle-to-gas heat transfer. We investigate variety of non-dimensional numbers including particle Stokes number, Reynolds number, and mass loading ratio. Using our statistical analyses we introduce a model to correct the particle-to-gas heat transfer to account for particle clustering. This can be employed in Reynolds average Navier Stokes (RANS) computations.
Bio:	Hadi Pouransari graduated from Sharif University of Technology in 2011 after obtaining his dual degree Bachelor of Science in Computer Science and Mechanical Engineering. He earned a Master of Science in Mechanical Engineering from Stanford University in 2013. Currently, he is a 6th year PhD student working with Profs. Darve and Mani on fast linear algebra techniques and applications in computational flow physics with a PhD minor in Computer Science.

DAY 3	
	SESSION IX – MULTIPHASE FLOWS
➤	OVERVIEW: Prof. Ali Mani
Title:	**Interfacial robustness of gas pockets and design criteria for superhydrophobic surfaces in turbulent flows**
Authors:	<u>Jongmin Seo</u>, Ricardo Garcia-Mayoral, and Ali Mani
Abstract:	Superhydrophobic surfaces (SHS) have been highlighted as one of skin drag reduction schemes since it can capture gas bubbles inside the microscale structures when submerged in liquid, suppressing direct contact with solid surfaces. However, when these surfaces are exposed to turbulent flows, the gas bubbles are depleted due to high shear and pressure fluctuations and in this case the skin friction drag increases. In this study, we aim to identify mechanisms of gas depletion on superhydrophobic surfaces that lead to drag reduction failure in turbulent flows. We conduct direct numerical simulations of turbulent flows on superhydrophobic surfaces including the dynamics associated with gas-liquid interfaces. The solid textures on superhydrophobic surfaces are modeled as patterned no-slip boundary conditions on the overlying turbulent liquid flow. The liquid-gas interface is modeled as shear-free boundary conditions with a linearized Young-Laplace equation, which is coupled with pressure from overlying turbulence. Our analysis identifies a mechanism for failure in retention of gas pockets by upstream traveling capillary waves. A semi-analytical inviscid model is developed to help understanding of underlying physics associated with the capillary waves. The scaling and magnitude of speeds of capillary waves observed in DNS are well predicted by the semi-analytical model. From DNS data, the magnitude of capillary pressure fluctuation is found to be scaling with the fourth power of slip velocity and linearly scaling with the Weber number, when all quantities are represented in wall units. Based on these scaling the failure mode for drag reduction of superhydrophobic surfaces is developed as a function of flow condition, fluid properties and design parameters including size of textures on SHS and material contact angle. Finally, we show boundary maps that establish stable and unstable zones of drag reduction to provide implications on the design of SHS. We review previously identified failure mechanisms of superhydrophobic surfaces, namely stagnation by slipping flow and shear-driven pressure, and present these additional mechanisms in an overlay map.
Bio:	Dr. Jongmin Seo is a Postdoctoral Scholar in the Department of Pediatric and Bioengineering at Stanford University. He received his Bachelors of Science in mechanical engineering from Seoul National University. Then, he joined Stanford and completed his Master of Science and Ph.D. degree in mechanical engineering working with Prof. Ali Mani. He was recipient of Jeongsong Scholarship and Kwanjeong Scholarship for his graduate studies. In his doctoral program, his research focused on the interaction of turbulent flows with superhydrophobic surfaces through direct numerical simulations. Currently he is working on the uncertainty quantifications in cardiovascular systems.
Title:	**Measurements in a Radiatively Heated Two-Phase Channel Flow**
Authors:	<u>Andrew Banko</u>, Laura Villafañe, Ji Hoon Kim, Chris Elkins, John Eaton
Abstract:	The coupled dynamics of small inertial particles, turbulence, and radiative heating is examined experimentally. A vertically downward airflow with Reynolds number of order 10,000 is laden with disperse Nickel particles which are smaller than all flow length scales. The particles have Stokes numbers of order 10 and the thermal time constant is similar to the aerodynamic time constant. This particle-air mixture is exposed to monochromatic near infrared radiation through one wall of the duct. While the gas and walls are nearly transparent to the incident radiation, the particles absorb energy and heat the gas with a spatial distribution dependent on the instantaneous particle concentrations. The mass loading ratio of particles is varied in order to study the effect of increasing optical depth on the gas temperature rise. A fine wire thermocouple is used to measure the mean gas temperature variation along the full width of the duct, including the near wall region where particle concentrations mildly

	increase. Total energy absorption is inferred from measurements of transmitted light intensity. Comparisons are made to a 1-D model which assumes homogeneity of all flow quantities, low optical depth, and ignores preferential concentration.
Bio:	Andrew Banko received his B.S. in Mechanical Engineering with a minor in Physics from the University of Illinois in 2012 and his M.S. in Mechanical Engineering from Stanford in 2014. He is currently in the 5th year of his PhD working for Professor John Eaton on experimental investigations of heat transfer in particle-laden turbulent flows.
Title:	**High-order AMR Simulations of Shock-induced Mixing**
Authors:	Man Long Wong, Sanjiva K. Lele
Abstract:	An adaptive mesh refinement (AMR) framework with high-order shock capturing scheme is developed for simulating multi-species shock-induced turbulent mixing. A multi-resolution wavelet detector is implemented as an extension of a second order derivative gradient detector in the AMR algorithm to improve the identification of features of interest in multi-species compressible flows. The framework also uses an improved sixth order localized dissipation Weighted Compact Nonlinear Scheme (WCNS) that can suppress spurious numerical oscillations near shocks and better resolve fluctuating features simultaneously. Through different test problems, the framework is shown to be robust and efficient in simulating shock-induced turbulent mixing.
Bio:	Man-Long Wong received his B.Eng. in Mechanical Engineering from the University of Hong Kong in 2011 and his M.S. in Aeronautics and Astronautics from Stanford in 2014. He is currently in the 5th year of his Ph.D. working with Professor Sanjiva Lele on high-order shock-capturing methods, adaptive mesh refinement technique and shock-induced turbulent mixing
Title:	**Computational Modeling of Breaking Waves and Bubbles**
Authors:	Ronald Chan, Javier Urzay, Ali Mani, and Parviz Moin
Abstract:	Multiphase flows often involve a wide range of impact events, such as liquid droplets impinging upon a liquid pool or gas bubbles coalescing in a liquid medium. These events contribute to a myriad of multiscale phenomena, including the production of micro-bubbles in breaking waves on ocean surfaces. In these waves, collisions of liquid bodies often trap thin gaseous films, which then become unstable and break up into smaller tiny bubbles that are later dispersed. These micro-bubbles are important for acoustic and optical ship wake detection, as well as aerosol generation and enhanced albedo. Many recent computational studies have been devoted to the accurate resolution of complex air-water interfaces, including three-dimensional simulations of breaking waves and hydraulic jumps, with sub-Hinze scale spatial resolution. However, the smallest micro-bubbles remain out of reach of these simulations, and would be prohibitively expensive to resolve with today's computational resources. More fundamentally, as impacts between liquid surfaces necessarily occur at isolated points in space, accurate simulation of these impact events requires the resolution of molecular scales near the impact points. Accurate and cost-effective simulations of these multi-scale phenomena necessitate the development of sub-grid impact and breakup models within a multiphase large-eddy simulation (LES) based framework. Here, the following methodology is proposed for the framework: through an exhaustive search, all collisions within the simulation domain are detected every time step. The surface geometry and impact velocities of the colliding interfaces are characterized, following which the film breakup dynamics are computed. Finally, Lagrangian micro-bubbles are inserted into the meso-scale numerical solver for subsequent transport. This talk focuses on the development of the collision detection algorithm, which is adapted from earlier algorithms for cloth animation and fluid flow post-processing. The fluid interfaces are first triangulated, following which the component triangles are tested pairwise for collisions within the computational time interval. The proposed algorithm is designed to function in tandem with unstructured and parallelizable flow solvers. In addition, it is inherently compatible with the

	geometric volume-of-fluid (VoF) interface capturing method, which has demonstrated superior conservation properties to traditional level-set (LS) and coupled LS/VoF methods. This work is supported by the Office of Naval Research.
Bio:	Ronald Chan received his B.S. in Engineering with a minor in Energy Studies from the Massachusetts Institute of Technology in 2014. He is currently in the 2nd year of the M.S. /PhD program in the Department of Mechanical Engineering. Ronald is currently working with Professor Parviz Moin on the development of sub-grid scale models for turbulent multiphase flows, with a focus on the modeling of micro-bubbles in breaking waves. Ronald is also affiliated with the Agency of Science, Technology and Research in Singapore.
Title:	**Shock Driven Plastic Flow at Solid-Solid Interfaces**
Authors:	Akshay Subramaniam, Niranjan Ghaisas, Sanjiva Lele
Abstract:	A high-order, fully Eulerian numerical framework is developed for tracking large, elastic-plastic deformations of solids coupled to fluids. Material interfaces are treated numerically using a diffuse-interface approximation. The numerical method is based on a 10th order compact finite difference scheme for spatial discretization, a 4th order Runge-Kutta time stepping method and a localized artificial diffusivity (LAD) method for regularizing shocks and material interfaces. This numerical framework was previously established for ideal gases and is extended in this study to liquids (stiffened gases) and solids. We employ the aforementioned numerical method to solve the problem of shock interaction with a perturbed material interface between Copper and Aluminum. With plasticity, the transport of vorticity away from the interface is reduced. Vorticity at the interface causes it to curl-up non-linearly with large deformations. The effects of different material stiffness parameters on the characteristics of the Richtmyer-Meshkov flow are investigated.
Bio:	Akshay Subramaniam received his B.Tech degree in Aerospace Engineering with a minor in Physics from the Indian Institute of Technology Madras in 2012 and his M.S. in Aeronautics and Astronautics from Stanford in 2014. He is currently in the 5th year of his PhD working with Prof. Sanjiva Lele on compressible multi-material turbulence and numerical methods for multi-phase flows.
SESSION X – OPTIMIZATION & UNCERTAINTY QUANTIFICATION	
➢	OVERVIEW: Prof. Gianluca Iaccarino
Title:	**Algorithm-driven Insights**
Authors:	Z. del Rosario, P. Constantine, G. Iaccarino
Abstract:	The conceptual phase of engineering design requires conceptual understanding; that is, insight into the problem at hand. Traditionally, engineering insights are the hard-won products of analysis and experience. But can similar results be found algorithmically? In this work we do exactly that, combining classical dimensional analysis with modern data-driven dimension reduction – Active Subspaces. The results address long-standing issues with Dimensional Analysis; addressing the uniqueness of Dimensionless Parameters and the detection of Hidden Parameters.
Bio:	Zach del Rosario earned a BS in Mechanical Engineering at Olin College of Engineering in 2014. He is currently pursuing a MS and PhD in the Aero/Astro Department at Stanford University, and his research focuses on Uncertainty Quantification and Dimension Reduction, specifically on extracting physical insight from associated computational techniques.
Tittle:	**Eigenspace perturbations for RANS model uncertainty**
Authors:	Aashwin Mishra

Abstract:	CFD simulations using Reynolds Averaged Navier Stokes models represent the workhorse for industrial applications while dealing with turbulent flows. However, owing to the inherent limitations of such models, a quantification of the uncertainty in their predictions is required to establish them as trustworthy tools in the engineering design process. This quantification accounts for the discrepancy between CFD predictions and the true flow evolution, for different cases and under diverse conditions. We outline a framework for RANS uncertainty estimation using eigenspace perturbations to the modeled Reynolds stress tensor. After a brief overview of the underlying theory, this methodology is applied to a variety of benchmark flows and complex engineering turbulent flows. In all cases, this procedure is able to account for a significant proportion of the discrepancy and provide bounds of engineering utility at a minimal computational cost.
Bio:	Dr. Aashwin Mishra is a postdoctoral fellow at the Center for Turbulence Research. Formerly, Aashwin was part of the faculty of Aerospace Engineering at the Texas A&M University. He is a recipient of the Vice President's Award for Excellence in Research, the Sigma Xi Interdisciplinary Research Award, and the Outstanding Dissertation Award, among others. His research interests include statistical inference and modeling, machine learning and uncertainty quantification.
Title:	**Optimal management of large scale aquifers under uncertainty**
Authors:	<u>Hojat Ghorbanidehno</u>, Amalia Kokkinaki, Peter Kitanidis, Eric Darve
Abstract:	Water resources systems, and especially groundwater reservoirs, are a valuable resource that is being endangered in many places around the world by contamination and overexploitation, threatening a vulnerable supply and its long-term sustainability. Optimal control techniques have been widely applied to groundwater management applications since the early days of dynamic programming. One such popular algorithm for aquifer management problems is the Linear Quadratic Gaussian Control (LQG) method. The objective of such methods is to quantify the controls (e.g. pumping schedule) for systems with uncertain model parameters and uncertain boundary conditions such that a cost function is minimized. However, LQG is limited by its high computational cost for systems with a large number of states and a large number of controls as the computational cost increases quadratically with the latter. This work presents a new method for optimal control in the linear Gaussian case with a linear cost that is applicable for large scale problems where limited uncertain information about the state of the system is available. The new algorithm reduces both the computational and storage cost by harnessing the structure of the large weighting and covariance matrices involved in the computations by using efficient low rank approximation methods. Embedded in the control algorithm is the Spectral Kalman Filter, a fast kalman filtering method that allows the assimilation of available data in order to improve the estimate of the system state and system parameters in the face of uncertainties related to unknown parameters and boundary conditions. The integration of filtering in the optimal control algorithm also allows the estimation of critical parameters such as aquifer properties in real time. Our joint control-estimation algorithm also provides real time estimates of uncertainty. The cost of the method increases linearly with the number of states and the number of controls, a significant improvement compared to the textbook version for KF and LQG methods. Our algorithm provides a practical approach for combined uncertainty quantification and optimal control for linear and weakly non-linear systems. We present a validation case for pumping schedule management in a 2D heterogeneous aquifer. We show an evaluation of our method in terms of accuracy and computational cost.
Bio:	Hojat Ghorbanidehno is a 5th year PhD student and he is currently working for Professor Eric Darve and Professor Peter Kitanidis. His research focus is on development and implementation of fast data assimilation and control methods for large-scale problems. Hojat received his M.S. In Mechanical Engineering from Stanford University in 2014 and his B.S. in Mechanical Engineering from Sharif University in 2012.

Title:	**Multifidelity Supersonic Nozzle Design Under Uncertainty**
Authors:	Rick Fenrich and Juan Alonso
Abstract:	The complex multidisciplinary design problems found in aerospace engineering often contain a large number of uncertainties, thus leading to the growing interest in uncertainty quantification and design under uncertainty methods. By mitigating uncertainties and incorporating them in the design process, engineers can design more robust and reliable applications. A primary difficulty of solving such design problems is the accurate estimation of probabilistic quantities with a reasonable amount of computational power. We introduce a new combination of techniques for reliability-based design optimization applied to the shape design of a military turbofan nozzle. Low- and high-fidelity separable surrogate models are built for chance constraints on quantities of interest using anchored decomposition and polynomial chaos and then embedded in a trust region model management framework. The solution of a reduced optimization problem with a single nonlinear thrust constraint is shown to be tractable and leads to an optimal reliable nozzle design.
Bio:	Rick Fenrich received his B.S. in Mechanical Engineering from Johns Hopkins University in 2013 and his M.S. in Aeronautics and Astronautics from Stanford in 2015. He is currently in the 4th year of his Ph.D. working under Professor Juan Alonso in the Aerospace Design Lab. His research interests include multidisciplinary design optimization and design under uncertainty with a focus on aerospace applications.