Rocky Mountain Fluid Mechanics Research Symposium 2020: Technical Program

(https://cuboulder.zoom.us/j/99659848218)

August 4th, 2020

Keynote Presentation

(https://cuboulder.zoom.us/j/99659848218)

Prof. Melissa Green, (10:30 AM - 11:30 AM)
Performance effects of vortex formation and shedding characterized by FTLE

Panel Discussion

(https://cuboulder.zoom.us/j/99659848218)

Prof. Hope Michelsen, Dr. Nazanin Hoghooghi, Katie Lockwood, and Skyler Kern (9:00 AM - 10:00 AM)
Diversity and Inclusion in the Fluid Dynamics Community

Presentation Schedule

Session 1A: Reacting Flows
8:30 AM – 8:55 AM (https://cuboulder.zoom.us/j/91217406920)

8:30 AM Nate Malarich (University of Colorado, Boulder)
Spatially resolved temperature measurements from a single-beam laser.

8:35 AM Matthew Brown (University of Wyoming)
Assessment of Hencken Burner Platform for Measurement of Unstretched Flame Speeds

8:40 AM Charlie Callahan (University of Colorado, Boulder)
WindCline: A Wind Tunnel for Characterizing Flame Behavior Under Variable Wind Velocity and Slope

8:45 AM Caelan Lapointe (University of Colorado, Boulder)
Progress Towards Efficient Simulation of Fire Suppression

8:50 AM Ryan Darragh (University of Colorado, Boulder)
Vorticity Dynamics in a Turbulent Premixed Flame

Session 1B: Multiphase Flows
8:30 AM – 8:55 AM (<https://cuboulder.zoom.us/j/93965198075>)

8:30 AM Samantha Ausman (University of Colorado, Boulder)
Oblique Collisions of Two and Three Wetted Spheres

8:35 AM Jacob Gissinger (University of Colorado, Boulder)
Contaminated emulsions flowing through porous media near critical conditions

8:40 AM Andrew J. Gibson (University of Colorado, Colorado Springs)
Application of Koopman theory and dynamic mode decomposition to the analysis of nonlinear bubble dynamics

8:45 AM Arsalan Taassob (North Carolina State University)
Droplet Manipulation via Triboelectrification

8:50 AM Fathia Arifi (University of Colorado, Colorado Springs)
Optimal control of the nonspherical oscillation of encapsulated microbubbles for biomedicine

Session 1C: Engineering Applications
8:30 AM – 8:55 AM (<https://cuboulder.zoom.us/j/96234600699>)

8:30 AM Abhishek Kumar (University of Colorado, Boulder)
Visualization of Flows from Musical Instruments

8:35 AM Sravanthi Vallabhuneni (Colorado State University and North Carolina State University)
Superhydrophobic Coatings for Improved Performance of Electrical Insulators

8:40 AM Mark Blanco (University of Colorado, Boulder)
Computational Fluid Dynamic Simulations of a Finite NACA 0015 Wing in an Unsteady Flow

8:45 AM Jaylon McGhee (University of Colorado, Boulder)
Impact of Reynolds Number on a Finite Wing in Surging Flow

8:50 AM Dasha Gloutak (University of Colorado, Boulder)
Impact of Reduced Frequency on a Finite Wing in Surging Flow

Session 2A: Biological Applications
10:00 AM – 10:25 AM (<https://cuboulder.zoom.us/j/91217406920>)

10:00 AM Joseph Wilson (University of Colorado, Boulder)
Flow Physics Modeling for Sars-CoV-2 Negative Pressure Isolation Space in a Skilled Nursing Facility

10:05 AM Chayut Teeraratkul (University of Colorado, Boulder)
Understanding Flow-mediated Transport in the Arterial Thrombus Neighborhood

10:10 AM Akshita Sahni (University of Colorado, Boulder)
Assessing Hemodynamics in the Ascending Aorta due to Surgical Anastomosis and Flow Modulation of Left Ventricular Assist Device

10:15 AM Amanda Benjamin (University of Colorado, Anschutz)
Measuring pressure and flow to quantify changes in alveolar recruitment during mechanical ventilation

Session 2B: Modeling - I
10:00 AM − 10:25 AM (<https://cuboulder.zoom.us/j/93965198075>**)**

10:00 AM Keegan Karbach (University of Colorado, Denver and Metropolitan State University of Denver)
Modeling of laminar-turbulent transition in Taylor-Couette system using OpenFOAM in creation of undergraduate CFD tutorial

10:05 AM Basu Parmar (University of Colorado, Boulder)
Data Driven RANS Modeling

10:10 AM Olga Doronina (University of Colorado, Boulder)
Parameter Estimation for Menter SST RANS Model Using Approximate Bayesian Computation

10:15 AM Aviral Prakash (University of Colorado, Boulder)
Invariant Data-Driven Subgrid Stress Closure for Large Eddy Simulation of Turbulence

10:20 AM Prakriti Sardana (University of Colorado, Boulder)
Modeling a non-reacting buoyant turbulent Helium plume computationally using adaptive mesh refinement (AMR)

Session 2C: Instabilities and Waves
10:00 AM − 10:25 AM (<https://cuboulder.zoom.us/j/96234600699>**)**

10:00 AM Riccardo Balin (University of Colorado, Boulder)
Simulations of a Turbulent Boundary Layer with Strong Pressure Gradients

10:05 AM Michael Meehan (University of Colorado, Boulder)
The Reynolds Number Dependence of the Buoyant Jet Puffing Frequency

10:10 AM Samuel Whitman (University of Colorado, Boulder)
Turbulent Structure and Mixing in a Heated Bluff Body Wake

10:15 AM Paris Buedel (University of Colorado, Boulder)
Oblique Dispersive Shock Waves on the Surface of a Steady Supercritical Shallow Water Flow

10:20 AM Yifeng Mao (University of Colorado, Boulder)
Experimental Investigation of Nonlinear Periodic Waves in a Viscous Fluid Conduit

Session 3A: Optimization
11:30 AM − 11:55 AM (<https://cuboulder.zoom.us/j/91217406920>**)**

11:30 AM David Molina (University of Colorado, Colorado Springs)
Optimization of fish kinematics using genetic algorithm

11:35 AM Skyler Kern (University of Colorado, Boulder)
Reducing the Biogeochemical Flux Model for Small Scale Simulations

11:40 AM Julian Quick (University of Colorado, Boulder)
Field Sensitivity Analysis for Wind Energy Modeling

11:45 AM Aleix Lopez Garulo (University of Colorado, Boulder)
Novel Small UAS Flight Control through Active Flow Control

11:50 AM Marzieh Alireza Mirhoseini (University of Notre Dame)
An optimization approach to solve parametrized conservation laws based on reduced order models

Session 3B: Modeling - II
11:30 AM − 11:55 AM ()

11:30 AM Andrew Glaws (National Renewable Energy Laboratory)
Turbulent flow style transfer for fast wind farm inflow generation

11:35 AM John Patterson (University of Colorado, Boulder)
Assessing and Improving the Accuracy of Synthetic Turbulence Generation

11:40 AM Charlotte Clapham (National Renewable Energy Laboratory)
Artificial Intelligence for Turbulent Flow Data Recovery

11:45 AM Alex Rybchuk (University of Colorado, Boulder)
A Comparison of Trace Gas Dispersion from WRF-LES to Project Prairie Grass

11:50 AM Colin Towery (University of Colorado, Boulder)
A Scaling Law for the Required Transition Zone Depth in Hybrid LES-DNS

Session 3C: Advanced CFD
11:30 AM − 11:55 AM ()

11:30 AM Gesse Roure (University of Colorado, Boulder)
Capture of Fine Particles By Expanding Drops in Linear Flows

11:35 AM Bashir Alnajar (University of Colorado, Colorado Springs)
Numerical modeling of the dynamics of bubbles and droplets with the coupled level set and volume-of-fluid (CLSVOF) method

11:40 AM William Schupbach (University of Colorado, Denver)
Central Moment Lattice Boltzmann Method based on Fokker-Planck Collision for Computational Fluid Dyanmics

11:45 AM Corey Wetterer-Nelson (University of Colorado, Boulder)
Interactive Geometry Modification for Massively Parallel CFD Simulations

11:50 AM James Wright (University of Colorado, Boulder)
Grid Resolution Requirements for High-Fidelity STG Development in Finite Element Simulation

Keynote Presentation

Professor Melissa Green (10:30 AM - 11:30 AM)

Department of Mechanical and Aerospace Engineering, Syracuse University

Performance effects of vortex formation and shedding characterized by FTLE

We study the formation and shedding of vortices in vortex-dominated flows in order to detect the formation of coherent structures objectively (i.e., in a frame invariant fashion), and to determine how the formation and shedding of those structures relates to pressure and force in the different applications. We specifically employ the Lagrangian technique of the finite-time Lyapunov exponent (FTLE). The FTLE analysis yields ridges of Lagrangian repulsion and attraction that have been shown to identify transport barriers in a range of flow fields, and vortex boundary features in vortex-dominated flows. In recent work, we have shown that using FTLE to identify and track material ridges associated with forming and shedding vortices can indicate a shedding or reattachment time that correlates well to surface pressure and body force extrema on cylinders and wings in cross-flow. In a study of satellite data of the Gulf Stream, FTLE ridges aligned with transport of other known scalars in the North Atlantic, and highlight regions of material mixing across that strong jet. These results prompt us to question whether robust modeling and control of performance metrics (force, efficiency, mixing) may be accessible by exploiting the evolving flow structure available from the FTLE results.

Speaker Biography:
Melissa Green is an Associate Professor in the Department of Mechanical and Aerospace Engineering at Syracuse University. She received her BS in Aerospace Engineering from the University of Notre Dame and her PhD in Mechanical and Aerospace Engineering from Princeton University. She was an NAS/NRC Postdoctoral Research Associate at the Naval Research Laboratory from 2009 to 2011 before starting her faculty position at SU in 2012. She received the Air Force Office of Scientific Research Young Investigator Award in 2014. Her research interests are primarily in the field of experimental fluid dynamics, particularly in vortex-dominated and bio-inspired applications.

Presentation Abstracts

An optimization approach to solve parametrized conservation laws based on reduced order models

Marzieh Alireza Mirhoseini, *Mechanical Engineering, University of Notre Dame*
Matthew Zahr, *Mechanical Engineering, University of Notre Dame*

Define an optimization problem whose solution is a reduced-order, discontinuity-aligned approximation of PDE solution, together corresponding mesh grid. Since, the strategy is based on reduced order methods, we decompose computational work into offline and online parts. Make low-dimensional basis by linearly compression of tracked snapshots on the coarse mesh to improve the decay of Kolomogorov N-widths, at the offline part that is not expensive like as previous works in this area. Then, we track features associate with new parameters in an online phase with solving an error based optimization problem. To solve faster and reduce the computational costs at the online part, we apply empirical quadrature procedure (EQP) that provides hyper-reduction of these bases. We show the ability of the method by solving equations contain discontinuities like linear advection and Burger together smooth Burger equation.

Numerical modeling of the dynamics of bubbles and droplets with the coupled level set and volume-of-fluid (CLSVOF) method

Bashir Alnajar, *Mechanical Engineering, University of Colorado, Colorado Springs*
Indrajit Chakraborty, *Applied Mathematics, University of Warwick*
Michael Calvisi, *Mechanical Engineering, University of Colorado, Colorado Springs*

The coupled level set and volume-of-fluid (CLSVOF) method is an efficient method used to simulate multiphase flows in which fluids of different phases (e.g., bubbles and droplets) are separated by a complex, evolving interface. This method leverages the advantages of both the level set (LS) and volume-of-fluid (VOF) approaches by combining the strong mass conservation properties of the VOF method, while retaining the accurate, straightforward representation of the interface curvature and normal offered by the LS method. In the present work, the flow field is discretized by a single-field, finite difference formulation of the incompressible, immiscible Navier-Stokes equations on a stationary grid. The advection VOF formulation is based on the second-order split algorithm scheme. The fluxes across the mesh cell depend on the orientation of the interface, which is reconstructed using the least squares volume-of-fluid interface reconstruction algorithm (LVIRA). A numerical code has been developed for 2D and axisymmetric cases, and its performance has been evaluated through a series of numerical tests, such as the simulation of a gas bubble rising with large density ratio, on the order of 1:1000.

Optimal control of the nonspherical oscillation of encapsulated microbubbles for biomedicine

Fathia Arifi, *Mechanical Engineering, University of Colorado, Colorado Springs*
Michael Calvisi, *Mechanical Engineering, University of Colorado, Colorado Springs*

Encapsulated microbubbles (EMBs) are on the order of 1-10 microns in diameter and are typically coated with a lipid, polymer, or protein to provide stability. EMBs were originally developed as contrast agents for ultrasound imaging, but more recently are being developed as vehicles for intravenous drug and gene delivery. Ultrasound can be used to incite EMB rupture and promote drug and gene delivery at targeted sites (e.g., tumors) within the circulatory system. Nonspherical oscillations, or shape modes, strongly effect both the acoustic signature and stability of microbubbles. Therefore, the ability to control shape modes can improve the efficacy of diagnosis and treatment mediated by EMBs. This work uses optimal control theory to determine the ultrasound input that maximizes a desired nonspherical EMB response (e.g., rupture), while minimizing the total acoustic input in order to enhance patient safety and reduce unwanted side effects. The optimal control problem is applied to models of both an EMB and a free gas bubble for small nonspherical oscillations, which are solved numerically through pseudospectral collocation methods using commercial

optimization software. Single frequency and broadband acoustic forcing schemes are explored and compared.

Oblique Collisions of Two and Three Wetted Spheres

Samantha Ausman, *Chemical and Biological Engineering, University of Colorado, Boulder*
Jacob Sitison, *Chemical and Biological Engineering, University of Colorado, Boulder*
Robert Davis, *Chemical and Biological Engineering, University of Colorado, Boulder*

Wet granular flows can be observed both in nature with geological flows and in major industrial practices such as the release of drugs, particle coating, and fertilizer production. While dry granular media has been heavily studied, the systems and mechanisms of wetted granular media are still poorly understood. In this work, a combined analytical and numerical model is first used to simulate and analyze the oblique collisions of two wetted particles and is later extended to collisions of three particles. Previous work by the Davis group has shown that the Stokes number, a dimensionless ratio of viscous forces and particle inertia, greatly affects the outcome of collisions as the relative motion of the particles is resisted by the viscous forces. Analysis of two-particle collisions confirm three outcomes previously observed (rebound, stick, and stick-rotate-separate). In the extension to three-particle collisions, it is expected that, depending on the Stokes number, outcomes such as stick-rotate-full separation or a stick-rotate-reverse Newton's cradle may occur.

Simulations of a Turbulent Boundary Layer with Strong Pressure Gradients

Riccardo Balin, *Aerospace Engineering, University of Colorado, Boulder*

The turbulent boundary layer over a Gaussian bump is computed by direct numerical simulation (DNS), wall-modeled large eddy simulation (WMLES), and Reynolds-Averaged Navier Stokes (RANS). The two-dimensional bump causes a rapid succession of favorable-to-adverse pressure gradients that leads to shallow separation on the downstream side. At the inflow, the momentum thickness Reynolds number is approximately 1,000, and the boundary layer thickness is 1/8 of the bump height. DNS results exhibit significant effects from the strong pressure gradients on the near-wall turbulence, resulting in atypical skin friction, velocity, and stress profiles. WMLES shows significant dependence on the extent of the near-wall modeling performed, and RANS lacks the essential pressure gradient physics entirely.

Measuring pressure and flow to quantify changes in alveolar recruitment during mechanical ventilation

Amanda Benjamin, *Bioengineering, University of Colorado, Anschutz*
Bradford Smith, *Bioengineering, University of Colorado, Anschutz*

Mechanical ventilation is a necessary intervention when an illness, such as COVID-19, prevents spontaneous breathing. However, mechanical ventilation may cause Ventilator Induced lung Injury (VILI) that leads to worse outcomes. Current clinical practice for mechanical ventilation involves the adjustment of ventilator settings to protect the already injured lung from ventilator-induced injury caused by alveolar collapse or overdistension. However, determining the safest settings is challenging even for the most highly skilled clinicians as these settings depend on the patient-specific alveolar dynamics which currently cannot be inferred at the bedside. To address this unmet need we have developed a software and hardware system, including custom-fabricated bidirectional flowmeters, to record pressures and flows during the mechanical ventilation of mice. A computational model of the lung that includes alveolar recruitment and tissue distension was then fit to the data. We found that VILI causes more alveoli to collapse, and this collapse occurs at higher pressures with increasing injury severity. Incorporating such a model into the clinical setting may lead to the development of personalized lung-protective ventilation and thus a decreased prevalence of ARDS.

Computational Fluid Dynamic Simulations of a Finite NACA 0015 Wing in an Unsteady Flow

Mark Blanco, *Aerospace Engineering, University of Colorado, Boulder*

Delayed-Detached Eddy simulations (DDES) are carried out to study the effects of an unsteady flow on a finite NACA 0015 Wing. The simulations demonstrate significant time varying and three-dimensional flow structure and are correspondent to experimental work currently being done at CU Boulder's Experimental Aerodynamics Laboratory.

Assessment of Hencken Burner Platform for Measurement of Unstretched Flame Speeds

Matthew Brown, *Mechanical Engineering, University of Wyoming*
Marcus Brown, *Mechanical Engineering, University of Wyoming*
Erica Belmont, *Mechanical Engineering, University of Wyoming*

Due to the significance of laminar flame speed in combustion, great effort has gone into accurately measuring reactant mixture flame speeds. However, flame speeds can be affected by the measurement platform, and particularly by aerodynamic stretching of the flame front. An experimental Hencken burner platform has been used in recent years to measure propagation speeds of hot and cool flames, with some past assessment of the stretch rate in this platform. This study further assessed the stretch rate by studying laminar, pre-mixed, flat methane-air flames in the Hencken platform. Using particle image velocimetry, axial strain and flame speeds of stretched flames were measured. Linear and non-linear extrapolation methods were applied to experimental data to derive unstretched flame speeds, with both extrapolation methods producing similar results due to the Hencken burner platform generating minimal strain in the flow field. Additionally, a chemiluminescence technique was used to derive unstretched adiabatic flame speeds using a less intensive lift-off height technique. Reasonable agreement between the PIV and chemiluminescence methods was found, supporting the use of the Hencken burner platform and relatively simple lift-off height technique to determine unstretched flame speeds.

Oblique Dispersive Shock Waves on the Surface of a Steady Supercritical Shallow Water Flow

Paris Buedel, *Mechanical Engineering and Applied Mathematics, University of Colorado, Boulder*
Mark Hoefer, *Applied Mathematics, University of Colorado, Boulder*
Samuel Ryskamp, *Applied Mathematics, University of Colorado, Boulder*

Motivated by the hydraulic analogy, oblique dispersive shock waves (DSWs) are studied for the interaction of shallow, supercritical water flow and a wedge obstacle. It has been shown theoretically that these steady DSW surface wave patterns can be approximated by a modulated nonlinear wave-train in the near-field and small amplitude harmonic waves in the far-field. Previous theoretical research has only been conducted in the long-wave shallow water approximation which, neglects the effect of the bottom boundary layer on the surface waves. Through experiment and CFD modeling, existence of the theorized DSWs will be validated with a water table and a 3D Navier-Stokes solver. The experimental set-up consists of a large, recirculating water-table with a wedge obstacle, which aims to produce the DSWs in question. Additionally, the free surface will be digitally reconstructed using the Free Surface Synthetic Schlieren (FSSS) Method. The open source CFD software, Ocellaris will be used to simulate the flow. Ultimately, experiment and numerics will be compared to theory. This presentation focuses on the preliminary, yet promising results obtained so far.

WindCline: A Wind Tunnel for Characterizing Flame Behavior Under Variable Wind Velocity and Slope

Charlie Callahan, *Mechanical Engineering, University of Colorado, Boulder*

Wildfire spread rates depend on many variables, including fuel type and conditions, atmospheric conditions, and terrain. Wildfire experiments and computations must control and systematically vary these parameters in order to understand and predict their impact on wildfire spread. We present the design, construction and characterization of a novel sloping wind tunnel for the purpose of studying flame behavior under varying wind speed and inclination. Initial experiments of a non-premixed methane-air flame issuing from a slot burner in the flow are presented to show key characteristics of slope and wind velocity on flame structure. Future work will incorporate the characterization measurements into computational fluid dynamics simulations for the same conditions to compare the average geometries of flame structures in the simulated flames to measured averages derived from optical images.

Artificial Intelligence for Turbulent Flow Data Recovery

Charlotte Clapham, *Applied Mathematics and Computer Science, National Renewable Energy Laboratory*
Andrew Glaws, *Applied Mathematics and Computer Science, National Renewable Energy Laboratory*
Ryan King, *Applied Mathematics and Computer Science, National Renewable Energy Laboratory*

The collection of wind and solar data is crucial in the research of these renewable energy resources. Environmental conditions, such as airborne particulates and cloud coverage, can disrupt data collection from satellite imagery. Such conditions result in random noise across the image data. Hardware failures during simulation and data storage are also a concern, resulting in the loss of large blocks of image data. Rather than lose the entirety of these partially corrupted datasets, we explore methods to effectively recover the lost data using artificial intelligence and deep learning models. Specially, we employ a deep fully convolutional neural network and study the role and performance of two different loss-functions within these models, through which the model evaluates its performance as it is trained. The two loss functions are the mean square error (MSE) and a novel loss function based on scientific perceptual loss networks (SPLN). These performance of these two losses are compared through preservation of key turbulent statistics. Through the evaluation of this key component in model training, this research aims to optimize model performance in order to most effectively recover data. Such data recovery techniques can result in significant computational and monetary savings.

Vorticity Dynamics in a Turbulent Premixed Flame

Ryan Darragh, *Aerospace Engineering, University of Colorado, Boulder*
Colin Towery, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

The vorticity dynamics of premixed flames differ from non-reacting flows primarily due to the increased importance of dilatation and baroclinic torque. As fluid combusts, the flow expands leading to an increase in dilatation. Combustion also results in misaligned pressure and density gradients which results in baroclinic torque that can either enhance or diminish vorticity. In this study we simulate both a forced and a decaying highly turbulent premixed methane-air flame using the second order code Athena-RFX. A Lagrangian analysis approach is then used to examine the enstrophy budget terms and determine what causes large enstrophy. Additionally, the effects of numerical diffusion in the simulation are propagated through to the enstrophy budget equation in order to determine their effects on the study of vorticity.

Parameter Estimation for Menter SST RANS Model Using Approximate Bayesian Computation

Olga Doronina, *Mechanical Engineering, University of Colorado, Boulder*
Scott Murman, *NASA Ames Research Center*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

In this study, we demonstrate the use of the approximate Bayesian computation with Markov chain Monte Carlo (ABC-MCMC) technique for calibration of the RANS Menter Shear-Stress Transport (Menter SST)

model parameters. ABC approximates the posterior distribution of unknown model parameters, but it does not require the direct computation or estimation of a likelihood function. Thus, ABC provides a substantially faster estimation of unknown parameters than a full Bayesian analysis. The unknown Menter SST model parameters are estimated using experimental data for the Bachalo-Johnson axisymmetric transonic bump. We provide simulation results for estimated parameters as well as a detailed description of the calibration step for ABC-MCMC and the choice of summary statistics.

Application of Koopman theory and dynamic mode decomposition to the analysis of nonlinear bubble dynamics

Andrew J. Gibson, *Mechanical Engineering and Aerospace Engineering, University of Colorado, Colorado Springs*
Xin Yee, *Mechanical Engineering and Aerospace Engineering, University of Colorado, Colorado Springs*
Michael L. Calvisi, *Mechanical Engineering and Aerospace Engineering, University of Colorado, Colorado Springs*

Volume and shape oscillations of gas bubbles in liquids form a central area of study in multiphase fluids, with important applications to intravenous drug delivery, contrast-enhanced ultrasound imaging, and cavitation-induced flow instabilities and damage in turbomachinery. In this study, we use emerging tools from Koopman theory to analyze the Rayleigh-Plesset equation governing spherical bubble oscillations. In particular, we apply the dynamic mode decomposition (DMD) and the Hankel alternative view of Koopman (HAVOK) analysis to numerically-generated time series in MATLAB. These methods can extract coherent spatio-temporal structures from data and provide a globally linear representation of strongly nonlinear periodic and even chaotic dynamics. Such a Koopman embedding allows for future state prediction and admits the application of classical control techniques, including optimal control. The resulting framework is generic, and, while here it is applied to simulated data, it may be equally applied to experimental measurements of encapsulated microbubbles or other systems. The issue of multi-scale dynamics is also explored.

Contaminated emulsions flowing through porous media near critical conditions

Jacob Gissinger, *Chemical Engineering, University of Colorado, Boulder*
Alexander Zinchenko, *Chemical and Biological Engineering, University of Colorado, Boulder*
Robert Davis, *Chemical and Biological Engineering, University of Colorado, Boulder*

A pressure-driven, periodic emulsion flowing at small Reynolds number through a cubic array of spheres is modeled using a three-dimensional boundary-integral algorithm, by considering one droplet in a repeating unit cell. The solid phase is fixed in space and fills 50

Turbulent flow style transfer for fast wind farm inflow generation

Andrew Glaws, *Computer Science, National Renewable Energy Laboratory*
Ryan King, *Computer Science, National Renewable Energy Laboratory*

As wind energy continues to grow in the U.S. and abroad, wind farm models play a critical role in plant siting and operation. However, accurate wind farm models depend on inflow fields that can vary significantly with mesoscale factors and can be difficult to generate. Common approaches to wind farm inflow generation include running a periodic precursor simulation to spinup fully developed turbulence and stochastic inflow generators. Precursor simulations require additional an additional computation while stochastic approaches don't scale well to large domains and struggle to preserve higher order turbulent statistics. In this work, we employ deep learning style transfer methods from the field of image processing. These techniques have been used to recast photographs into the artistic style of Renaissance painters. We consider the canonical channel flow problem as an analog to wind farm inflow and adapt style transfer techniques built on customized perceptual loss networks to inject small-scale turbulent structures into the smooth data. We evaluate the performance of this methodology using classical turbulent statistical quantities.

Impact of Reduced Frequency on a Finite Wing in Surging Flow

Dasha Gloutak, *Aerospace Engineering, University of Colorado, Boulder*
Jaylon McGhee, *Aerospace Engineering, University of Colorado, Boulder*
John Farnsworth, *Aerospace Engineering, University of Colorado, Boulder*

The effects of surging flow parameters are examined for a NACA 0015 semi-span finite wing. Varying the reduced frequency of the unsteady wind tunnel flow reveals pronounced changes in instantaneous lift coefficient profiles. The impact of reduced frequency depends on both AR and angle of attack.

Modeling of laminar-turbulent transition in Taylor-Couette system using Open-FOAM in creation of undergraduate CFD tutorial

Keegan Karbach, *Applied Mathematics and Physics, University of Colorado, Denver and Metropolitan State University of Denver*

OpenFOAM is attractive for academic use due to the fact that it is provided as an open-source library of computational fluid dynamics packages making it beneficial for students both from a financial standpoint as well as a from the fact that it is heavily customizable. There is, however, a substantial learning curve when compared with other commercially available CFD software. This has prompted the creation of an advanced undergraduate tutorial for the software that models an existing Taylor-Couette lab experiment to introduce CFD at an undergraduate level. Before this tutorial could be deployed, the computational model required verification, the results of which are presented here. To this end, the flow characteristics of the computational model were examined both immediately above and below the experimentally validated critical Reynolds Number (Re_c). Visualizations of the field variables and the flow structure are presented for both circular Couette flow and Taylor vortex flow, as well as for the flow structure. Additionally, the numerical solution for circular Couette flow in the laminar regime is confirmed with the analytical solution.

Reducing the Biogeochemical Flux Model for Small Scale Simulations

Skyler Kern, *Mechanical Engineering, University of Colorado, Boulder*
Katherine Smith, *Los Alamos National Laboratory*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*
Nadia Pinardi, *University of Bologna*
Marco Zavatarelli, *University of Bologna*

Modeling the global climate systems is extremely complex due to the wide range of spatial and temporal scales which cannot be simultaneously resolved due to current computational limitations. In order to perform studies on the global scale, Earth System Models parameterize the small scales. The interactions taking place on the small scales are therefore important to understand in order to formulate useful sub-grid scale models. An important area of investigation is the interaction between upper ocean dynamics and biogeochemical processes. The biogeochemical models available are not ideal for coupling with small scale physical models, such as those used to study upper ocean dynamics. To confront this issue a reduced Biogeochemical Flux Model with 17 state variables (BFM17) has been developed. To reduce the computational cost and focus on open ocean settings, the full BFM was reduced by eliminating certain processes - including benthic, silicate, and iron influences- and the parameterizing others, such as the bacterial loop. BFM17 was coupled with the one-dimensional Prince Ocean Model for validation against observational data from the Sargasso Sea. The coupled model is now being used to preform a parameter optimization before coupling BFM17 to a 3D physical model.

Visualization of Flows from Musical Instruments

Abhishek Kumar, *Mechanical Engineering, University of Colorado, Boulder*
Tehya Stockman, *Environmental Engineering, University of Colorado, Boulder*
Jean Hertzberg, *Mechanical Engineering, University of Colorado, Boulder*

The COVID pandemic has created a great deal of fear and uncertainty around whether aerosols from singers and brass and woodwind instruments can transmit the virus. As part of a larger study to quantify aerosol emissions from performers we have been conducting schlieren and laser sheet visualizations of flow from singers and musicians playing brass and woodwind instruments, with and without mitigation devices such as masks and bags. Limited hot wire anemometry is also being done. Preliminary results indicate a wide range of jet behaviors. For example, although most of the flow from a clarinet exits through the bell at low velocity, small higher speed jets can be seen from finger holes, particularly in the higher note registers. In contrast, a concert tuba is held close to the chest with the bell pointing up, above the musician's head, so the heated plume from the musician's body swamps the emissions from the instrument in the schlieren videos, despite the relatively large volume of flow. One example of a mitigation device is pantyhose stretched over the bell of a trombone. The exit jet was diffused and sound emission was not reduced, but the musician reported a difference in the feel of the instrument. However, the impact on aerosol emission has not yet been studied.

Progress Towards Efficient Simulation of Fire Suppression

Caelan Lapointe, *Mechanical Engineering, University of Colorado, Boulder*
Jeffrey Glusman, *Mechanical Engineering, University of Colorado, Boulder*
Prakriti Sardana, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

Large fires — in natural or urban environments — can be socioeconomically devastating. It is therefore desirable to be able to predict, for example, where a fire will spread and optimize its suppression. This has motivated the development of a computational tool to efficiently model fire spread and suppression. To address this, adaptive mesh refinement (AMR) has been incorporated into the OpenFOAM solver for complex fire phenomena, fireFoam. The new solver, wildFireFoam, will be demonstrated for a simple test case in which a sprinkler is employed as the mode of water delivery.

Novel Small UAS Flight Control through Active Flow Control

Aleix Lopez Garulo, *Aerospace Engineering, University of Colorado, Boulder*
John Farnsworth, *Aerospace Engineering, University of Colorado, Boulder*
Joseph Straccia, *Aerospace Engineering, University of Colorado, Boulder*

The use of a novel active flow control (AFC) system to replace conventional moving control surfaces for circulation control is investigated on a tailless Small UAS (sUAS). Control is effected by a pair of opposed and independently-controlled zero-mass-flux synthetic jet actuators (SJA) located near the trailing edge at both wingtips. The jets generated by the SJAs emanate from aft-facing, narrow slots (AR = 18) at a small backwards step that extends into a Coanda surface. The jet then attaches to the curved surface, entraining fluid from the surroundings and therefore vectoring the base flow. Hotwire anemometry was conducted to calibrate the SJAs and optimize their geometry. A full-sized wing fitted with conventional control surfaces was mounted on a low-speed, open-return wind tunnel to measure the aerodynamic forces and moments. The experimental conditions were matched to the typical flight conditions of the DataHawk sUAS. These results will be used for a future system level analysis of the performance of the AFC system once the wing fitted with the AFC module is tested on the wind tunnel.

Spatially resolved temperature measurements from a single-beam laser.

Nate Malarich, *Mechanical Engineering, University of Colorado, Boulder*
Greg Rieker, *Mechanical Engineering, University of Colorado, Boulder*

Combustion systems are commonly heterogeneous due to fluid mixing layers, boundary layers, etc. We present a laser diagnostic to resolve temperature nonuniformities in a combustion flow, based on the Boltzmann distribution of molecules in the gas. We then demonstrate this technique with a dual frequency comb

laser on a laboratory tube furnace, additionally showing how traditional thermocouple measurements are biased in this system.

Experimental Investigation of Nonlinear Periodic Waves in a Viscous Fluid Conduit

Yifeng Mao, *Applied Mathematics, University of Colorado, Boulder*
Mark Hoefer, *Applied Mathematics, University of Colorado, Boulder*

Conduits generated by the buoyant dynamics between two miscible, Stokes fluids with high viscosity contrast have been studied due to their remarkable nonlinear wave dynamics. Applications are known in many geological and geophysical systems including the dynamics of magmatic and glacial systems. In this talk, laboratory measurements of glycerin viscous fluid conduits compared with theoretical predictions will be presented. Periodic interfacial waves are generated by periodically varying the flow rate of dyed, diluted glycerin injected into the bottom of a pure glycerin reservoir. Measurements of wave profiles and the wavenumber-frequency dispersion relation quantitatively agree with the wave solutions to the conduit equation in the linear regime. In the large-amplitude nonlinear regime, higher harmonics in the wave profile are observed, yielding the quantitative relationship between wave frequency, wavenumber, amplitude and mean via a weakly nonlinear analysis. Arrested wave propagation above a critical injection frequency is observed due to the bounded, nonlocal conduit dispersion relation. This study is an important precursor to the experimental investigation of more complex nonlinear wave dynamics including envelope solitons and dispersive shock waves.

Impact of Reynolds Number on a Finite Wing in Surging Flow

Jaylon McGhee, *Aerospace Engineering, University of Colorado, Boulder*
Dasha Gloutak, *Aerospace Engineering, University of Colorado, Boulder*
John Farnsworth, *Aerospace Engineering, University of Colorado, Boulder*

The effects of surging flow parameters are examined for a NACA 0015 semi-span finite wing. Increasing the Reynolds number above 50,000 demonstrates a fundamental change in lift coefficient behavior at low angles of attack. The instantaneous lift highlights the shift from quasi-steady to unsteady wing behavior.

The Reynolds Number Dependence of the Buoyant Jet Puffing Frequency

Michael Meehan, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

Buoyant jets display a global instability near the jet exit in which coherent vortical structures are regularly shed due to buoyancy accelerating the lighter fluid. These flow structures are commonly found in nature or as a result of combustion processes; the instability also plays an important role providing oxygen in buoyancy-driven reacting flows. In an effort to characterize the global behavior, many studies, including both experimental and numerical, have shown that the Strouhal number of the oscillation frequency is very well correlated to the Richardson number based on inlet parameters.

In this work, we conduct high-fidelity numerical simulations using adaptive mesh refinement in PeleLM to investigate how the Reynolds number plays a role in the puffing frequency. We find that at lower Reynolds numbers, the observed frequency is slightly below the predicted puffing frequency. As Reynolds number increases, the observed frequency is above the predicted; further increases bring the observed frequency closer to the predicted. This region of over prediction is dependent upon Richardson number. Comparing with experimental observations, we believe that the open ended question of the change of scaling at large Richardson numbers may in fact be due to the Reynolds number effect.

Optimization of fish kinematics using genetic algorithm

David Molina, *Mechanical Engineering, University of Colorado, Colorado Springs*
Hui Wan, *Mechanical Engineering, University of Colorado, Colorado Springs*

Fish generate propulsion through body and caudal (tail) fin undulation. The undulation kinematics, such as the amplitude and the frequency, determines the generated hydrodynamic force, associated acceleration, and therefore the fish velocity. In this study, we use high fidelity computational fluid dynamics (CFD) and genetic algorithm (GA) to investigate how a two-dimensional fish model can generate propelling force in the shortest time. Genetic algorithm is used to generate fish kinematics; CFD simulation provides the corresponding the hydrodynamic force. Iteration between GA and CFD is conducted until the optimized kinematics is obtained.

Data Driven RANS Modeling

Basu Parmar, *Aerospace Engineering, University of Colorado, Boulder*
John Evans, *Aerospace Engineering, University of Colorado, Boulder*
Kenneth Jansen, *Aerospace Engineering, University of Colorado, Boulder*

Reynolds Averaged Navier Stokes (RANS) models are the most popular tool for modeling turbulent flow. RANS models require modeling of an unclosed term called the Reynolds stress tensor, but state-of-the-art Reynolds stress closures such as Linear Eddy Viscosity and Non-Linear Eddy Viscosity models are inadequate for many complex flows of industrial interest, especially those exhibiting flow separation. In this talk, Data Driven RANS modeling is introduced as an alternative to traditional RANS models. Special emphasis is placed on using Generalized Non Linear Eddy Viscosity models to correct deficiencies in current turbulence models. Further challenges to fully couple data driven models along with traditional RANS models will be presented.

Assessing and Improving the Accuracy of Synthetic Turbulence Generation

John Patterson, *Aerospace Engineering, University of Colorado, Boulder*
Kenneth Jansen, *Aerospace Engineering, University of Colorado, Boulder*
Riccardo Balin, *Aerospace Engineering, University of Colorado, Boulder*

With the growing interest in scale-resolving simulations of spatially evolving boundary layers, synthetic turbulence generation (STG) has become a valuable tool for providing stochastic turbulent fluctuations through a sum over a finite number of spatio-temporal Fourier modes with amplitude, direction, and phase determined by a random number set. Recent developments of STG methods are designed to match target profiles for anisotropic and inhomogeneous Reynolds stresses. It is shown that for practical values of the number of modes used, a given set of random numbers may produce Reynolds stress profiles that are 30

Invariant Data-Driven Subgrid Stress Closure for Large Eddy Simulation of Turbulence

Aviral Prakash, *Aerospace Engineering, University of Colorado, Boulder*
Kenneth Jansen, *Aerospace Engineering, University of Colorado, Boulder*
John Evans, *Aerospace Engineering, University of Colorado, Boulder*

Large-Eddy Simulations (LES) are often used to get accurate statistics of turbulent flows. These simulations rely on an accurate model for the Sub-Grid Scale Stress (SGS) tensor to close the filtered Navier-Stokes equations. We propose a data-driven approach to model the SGS tensor. The model form is designed to ensure Galilean, time, frame, and dimensional invariance, thereby ensuring physical consistency with the filtered Navier-Stokes equations. The model form allows for a simplified neural network representation that is trained using a small amount of data obtained from an open-source turbulence database. We show that the data-driven model exhibits superior performance over classical SGS models in both a priori and a posteriori tests. We demonstrate that the data-driven model also appears to have a generalizable nature as it gives accurate results even for cases that are at a different Reynolds number or involve flow physics different than the one that it was trained on.

Field Sensitivity Analysis for Wind Energy Modeling

Julian Quick, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*
Ryan King, *National Renewable Energy Laboratory*
Marc Henry de Frahan, *National Renewable Energy Laboratory*
Shreyas Ananthan, *National Renewable Energy Laboratory*
Michael Sprague, *National Renewable Energy Laboratory*

Wind energy systems are complex, necessitating turbulence models to approximate the true flow dynamics. Misspecified turbulence model parameters can lead to poor airfoil performance, unexpected structural loads, or suboptimal energy production. A field sensitivity analysis can reveal crucial model parameters that introduce simulation errors leading to these undesirable outcomes. This study demonstrates field sensitivity analysis for a turbulent flow simulation relevant to wind energy — flow over a NACA 0015 wing at 12 degrees angle of attack and Reynolds number of 1.5 million — with respect to ten parameters in the 2003 Menter shear-stress transport turbulence model. Sensitivity is quantified using Sobol indices and the mean-squared gradient, which are estimated through polynomial chaos expansion and active subspace models, respectively. Two sets of turbulence model parameters are identified, which are important to flow close to and far away from the wing, respectively. Simultaneous dimension reduction across several quantities of interest is also explored. This sensitivity analysis and dimension reduction will enable efficient model calibration for future wind energy studies.

Capture of Fine Particles By Expanding Drops in Linear Flows

Gesse Roure, *Chemical and Biological Engineering, University of Colorado, Boulder*
Robert Davis, *Chemical and Biological Engineering, University of Colorado, Boulder*

The present work investigates the two-particle dynamics of a solid particle and a semi-permeable spherical drop that expands due to osmosis in an external, pure-extensional flow field. Computational results from numerical integration of trajectories determine a transient collision efficiency, which describes the influence of hydrodynamic interactions and osmotic flow on particle capture. The results show that drop expansion, which decays slowly with time, greatly increases particle capture rates. Moreover, as the engulfment parameter increases, there is a transition from flow-dominated capture to expansion-dominated capture. For the case of a non-expanding droplet, we provide a numerical solution for the transient pair distribution function, which enables us to explain the transient particle capture rate in terms of the microstructure of the suspension. Furthermore, we derive an analytical expression for the initial collision efficiency at zero times, which agrees with our numerical data. The numerical results for non-expanding droplets at long times show increasing collision efficiency as the permeability increases and when the size ratio is near unity, in agreement with previous steady-state calculations. Weak diffusion effects inside of the droplet are also discussed.

A Comparison of Trace Gas Dispersion from WRF-LES to Project Prairie Grass

Alex Rybchuk, *Mechanical Engineering, University of Colorado, Boulder*
Caroline Alden, *CIRES, University of Colorado, Boulder and National Oceanic and Atmospheric Administration*
Julie Lundquist, *Atmospheric and Oceanic Sciences, University of Colorado, Boulder and National Renewable Energy Laboratory*
Gregory Rieker, *Mechanical Engineering, University of Colorado, Boulder*

In recent years, large-eddy simulation (LES) has been applied to study trace gas dispersion in the atmospheric surface layer (¡100 m). Applications range from quantifying emissions from natural gas infrastructure to modeling urban air quality. While LES is a powerful tool that has the potential to accurately model trace gas plumes within the atmospheric surface layer, its accuracy has not been statistically evaluated against measurements for this application. In this study, we statistically evalaute LES from the Weather Research and Forecasting model (WRF-LES) in strongly and weakly convective conditions against observations from

the Project Prairie Grass field campaign. We summarize three main conclusions. First, WRF-LES concentrations are grid-dependent within the first 200 m. Second, WRF-LES accurately models dispersion under strong convection, so long as sufficient grid resolution is used. Finally, under weak convection, WRF-LES substantially deviates from observations.

Assessing Hemodynamics in the Ascending Aorta due to Surgical Anastomosis and Flow Modulation of Left Ventricular Assist Device

Akshita Sahni, *Mechanical Engineering, University of Colorado, Boulder*
Andrew Beiter, *Mechanical Engineering, University of Colorado, Boulder*
Debanjan Mukherjee, *Mechanical Engineering, University of Colorado, Boulder*

A Left Ventricular Assist Device (LVAD) is a blood pump surgically implanted into the heart to serve as a mechanical support for the circulatory function of the left ventricle. LVADs are widely used as bridge-to-transplant as well as destination therapy for heart failure (HF) patients. However, LVAD therapy is commonly associated with complications such as pump thrombosis, ischemic stroke, and gastrointestinal bleeding. These complications are thought to be intimately related to the altered state of hemodynamics induced by LVAD pump flow and outflow cannula anastomosis. Our objective is to quantify the aortic flow features in the vicinity of the anastomosis in terms of key parameters like cannula angle with the aorta, and pulsatile modulation of pump flow. This presentation discusses a computational hemodynamics study for LVAD deployment in a real patient. Image based modeling is used for vascular anatomy reconstruction, and virtual surgical placement of LVAD outflow cannula at different angles. CFD simulations were performed for each case by using a constant, low and high-pulse modulation boundary condition. Using the results, we identify variations in aortic flow helicity and vorticity as a function of mentioned key parameters and discuss potential benefits of flow pulsation.

Modeling a non-reacting buoyant turbulent Helium plume computationally using adaptive mesh refinement (AMR)

Prakriti Sardana, *Mechanical Engineering, University of Colorado, Boulder*
Caelan B. Lapointe, *Mechanical Engineering, University of Colorado, Boulder*
Jeffrey Glusman, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering and Aerospace Engineering, University of Colorado, Boulder*

Buoyancy-driven turbulent plumes are a fascinating area of fluid studies especially when it comes to probing fire dynamics. The first step is to be able to model and understand the contribution of buoyancy to turbulence production in a manner that resolves flow physics sufficiently realistically and at a non-prohibitive computational cost. Towards this end, in this current study, we have been looking at ways of modeling a buoyant Helium plume in a square cavity and resolving the system at sub-millimeter scales by employing large-eddy simulation (LES) sub-grid scale (SGS) models that explicitly account for the buoyant contribution to turbulence production. In addition, we have employed the use of adaptive mesh refinement (AMR) in order to be able to perform LES at a computationally affordable cost. Modeling is being done in OpenFOAM using a novel solver called diffusionFireFoam.

Central Moment Lattice Boltzmann Method based on Fokker-Planck Collision for Computational Fluid Dyanmics

William Schupbach, *Mechanical Engineering, University of Colorado, Denver*
Kannan Premnath, *Mechanical Engineering, University of Colorado, Denver*

Central moments-based lattice Boltzmann method (LBM), a promising more recent approach for flow simulations, is generally based on the relaxation of various central moments to their equilibria under collision. The latter is usually constructed either directly from the Maxwell distribution function or by exploiting its factorization property. We propose a central moment LBM from a different perspective, where its collision

operator is constructed by matching the changes in different discrete central moments under collision to the changes in the corresponding continuous central moments as given by the Fokker-Planck (FP) collision model of the Boltzmann equation. The resulting formulation can be interpreted in terms of the relaxation of the various central moments to "equilibria" that depend only on the adjacent, lower order post-collision moments. We designate such newly constructed chain of equilibria as the Markovian central moment attractors and the relaxation rates are based on scaling the drift coefficient of the FP model by the order of the participating moment. We will demonstrate the accuracy and robustness of our new formulation for simulations of a variety of flows using the standard D2Q9 and D3Q27 lattices and also present a comparison against other collision models.

Droplet Manipulation via Triboelectrification

Arsalan Taassob, *Mechanical Engineering, North Carolina State University*
Wei Wang, *Mechanical Engineering, North Carolina State University*
Hamed Vahabi, *Bioengineering, Duke University*
Sravanthi Vallabhuneni, *Mechanical Engineering, North Carolina State University*
Arun Kota, *Mechanical Engineering, North Carolina State University*

Droplet manipulation on surfaces has large applications such as in lab-on-a-chip systems, energy harvesting, and biomedical devices. To date, various active methods such as the magnetic field and electric field have been developed for controlling the manipulation of liquid droplets. However, these active methods are usually costly and the fabrication of them is a complicated process. In this study, we use a simple and precise method to manipulate liquid droplets on a smooth, slippery surface via triboelectric effect. We demonstrate that both high (e.g., water) and low (e.g., n-hexadecane) surface tension liquids can be controlled by using solely the electrostatic attraction or repulsion force, which is exerted on the droplet by a charged actuator (e.g., Teflon film).

Understanding Flow-mediated Transport in the Arterial Thrombus Neighborhood

Chayut Teeraratkul, *Mechanical Engineering, University of Colorado, Boulder*
Debanjan Mukherjee, *Mechanical Engineering, University of Colorado, Boulder*

Pathological blood clotting (thrombosis) is a primary cause or complication in stroke and other cardiovascular diseases. Understanding blood flow and flow-mediated transport in thrombus neighbourhood under physiologically realistic conditions is crucial for treatment of such diseases. Given that in vivo experiments are significantly invasive and complicated, computational methods can provide useful noninvasive insights into the hemodynamics in the vicinity of a thrombus. This, however, remains a challenge because of the arbitrary shape and heterogeneous microstructure of a realistic thrombus. Using a hybrid particle-continuum based stabilized finite element approach, we conducted a parametric simulation-based study to characterize unsteady hemodynamics and flow-mediated transport in the neighbourhood of an arterial thrombus by varying three key parameters - thrombus shape, microstructures, and extent of wall disease state. Results from 50 different parametric simulations illustrate the influence of these three parameters on the unsteady vortical structures, pressure-gradient across thrombus boundary, and dynamic coherent structures that organize advective transport around the thrombus.

A Scaling Law for the Required Transition Zone Depth in Hybrid LES-DNS

Colin Towery, *Mechanical Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

There is a substantial need to better understand multi-physics interactions in realistic laboratory experiments, yet it is currently unfeasible to perform direct numerical simulations (DNS) of most such problems. As a result, reduced-order models such as large-eddy simulations (LES) are commonly employed for the

study of multi-physics interactions in practical situations. However, using advanced numerical techniques, including adaptive mesh refinement and nested grid structures, it is now possible to selectively refine the computational mesh in a small region of the domain to resolve multi-physics interactions with DNS fidelity. However, rules and guidance for the valid implementation of this multi-fidelity "hybrid LES-DNS" approach have yet to be fully formulated. Within the embedded DNS region, multi-physics interactions will be influenced by large-scale shear gradients or body forces that produce turbulence, LES-regulated advecting turbulence eddies, and sub-LES-scale features that are produced in the DNS region. In this study, we address the need for a scaling law of the required depth of the transition zone surrounding the embedded DNS region within hybrid LES-DNS such that an LES-truncated turbulence cascade can extend into the extra scales resolved by the DNS region.

Superhydrophobic Coatings for Improved Performance of Electrical Insulators

Sravanthi Vallabhuneni, *Mechanical Engineering, Colorado State University and North Carolina State University*
Sanli Movafaghi, *Mechanical Engineering, University of Colorado, Boulder and Colorado State University*
Wei Wang, *Mechanical Engineering, Colorado State University and North Carolina State University*
Arun Kota, *Mechanical Engineering;Chemical and Biological Engineering, Colorado State University and North Carolina State University*

Dielectric strength of electrical insulators is adversely affected by both wet conditions and surface roughness due to accelerated dielectric breakdown. While textured superhydrophobic coatings (i.e., coatings that are extremely repellent towards water) can prevent the accumulation of water and help retain the dielectric strength of insulators under wet conditions, it is unclear whether or not the presence of the texture (i.e., surface roughness) allows the retention of the dielectric strength of insulators. In this work, through a series of systematic experiments and analysis, it is concluded that porous superhydrophobic coatings rather than rough monolithic superhydrophobic surfaces allow the retention of dielectric strength of insulators under wet conditions in spite of the surface roughness. The insights from this work can enable the design of electrical insulators with improved performance and longevity.

Interactive Geometry Modification for Massively Parallel CFD Simulations

Corey Wetterer-Nelson, *Mechanical Engineering, University of Colorado, Boulder*
John Evans, *Aerospace Engineering, University of Colorado, Boulder*
Kenneth Jansen, *Aerospace Engineering, University of Colorado, Boulder*

In the context of high-performance finite element analysis, the cost of iteratively modifying a computational domain via re-meshing and restarting the analysis becomes time prohibitive as the size of simulations increases. In this presentation, we demonstrate a new interactive simulation pipeline targeting high-performance fluid dynamics simulations where the computational domain is modified in situ, that is, while the simulation is ongoing. This pipeline is designed to be modular so that it may interface with any existing finite element simulation framework. A server-client architecture is employed to manage simulation mesh data existing on a high performance computing resource while user-prescribed geometric modifications take place on a separate workstation. We employ existing in situ visualization techniques to rapidly inform the user of simulation progression, enabling computational steering. By expressing the simulation domain in a reduced fashion on the client application, this pipeline manages highly refined finite element simulation domains on the server while maintaining good performance on the client application.

Turbulent Structure and Mixing in a Heated Bluff Body Wake

Samuel Whitman, *Mechanical Engineering, University of Colorado, Boulder*
James Brasseur, *Aerospace Engineering, University of Colorado, Boulder*
Peter Hamlington, *Mechanical Engineering, University of Colorado, Boulder*

A series of high-resolution simulations of airflow around heated bluff bodies are used to study the effects of

thermofluidic variation and strong gradients on complex turbulent wakes. Bluff body temperatures range from 310K to 1500K, at which point the thermal energy transferred to the fluid plays an active role in the field evolution. At the same time, shear layer instabilities drive entrainment and mixing, pulling cold air into the wake and smoothing out the initial gradients. Simulations are carried out on HPC resources using the fully compressible PeleC code, which incorporates adaptive mesh refinement (AMR) via the AMReX block-structured framework to locally resolve the physics of interest at reduced cost compared to static mesh approaches. Results show the effects of heating on turbulent fluctuations and anisotropy throughout the wake, and on specific terms in the vorticity budget which highlight the importance of capturing highly localized and/or transient behavior with AMR.

Flow Physics Modeling for Sars-CoV-2 Negative Pressure Isolation Space in a Skilled Nursing Facility

Joseph Wilson, *Mechanical Engineering, University of Colorado, Boulder*
Shelly Miller, *Mechanical Engineering and Environmental Engineering, University of Colorado, Boulder*
Nicholas Clements, *Mayo Clinic, General Internal Medicine, Rochester, Minnesota*
Cedric Steiner, *The College of Business and Leadership, Eastern University, St. David's, Pennsylvania*
Debanjan Mukherjee, *Mechanical Engineering, University of Colorado, Boulder*

The Sars-CoV-2 pandemic has fundamentally altered societal structures and norms in ways that will continue for years to come. One group considered at high risk for infection and severe complications is individuals of an advanced age, which makes nursing homes locations of particular concern. Nursing home administrations that do not take proper precautions to protect their residents risk high infection and mortality rates. In this study, we implemented a negative pressure isolation ward at a nursing facility to manage COVID-19 patients, informed by flow physics-based models. We constructed a full-scale CAD model of the nursing home wing that would be used as an isolation space for their residents. We then used the available information on building HVAC system to determine boundary conditions at the vents throughout the ward and performed 3D CFD simulations for airflow. This information enabled us to make rapid, low-cost, data-driven HVAC alterations to create the negative pressure isolation space with on-site pressure measurements that conformed to simulation results. The isolation space continues to be operational, and has been successful in preventing any on site transmission amongst staff as well as yielding zero fatalities amongst all residents treated for COVID-19.

Grid Resolution Requirements for High-Fidelity STG Development in Finite Element Simulation

James Wright, *Aerospace Engineering, University of Colorado, Boulder*
Kenneth Jansen, *Aerospace Engineering, University of Colorado, Boulder*

Synthetic Turbulence Generators (STGs) create pseudo-turbulent flow fields from some input quantities about the flow. They're often used as inlet boundary conditions to reduce the domain size of scale resolving simulations (SRS) or transition from non-SRS turbulence models to SRS models. STGs often take a mean velocity profile and simply add velocity fluctuations to it in the form of Fourier wave-modes. These fluctuations mimic the statistics of turbulence, but they do not exhibit the structures of turbulent flow. These statistical fluctuations must transition into so-called "natural" turbulence, in which turbulent structures are present and the flow's pseudo-turbulent history is "forgotten". The transition distance downstream of the STG boundary condition where "natural" turbulence is present is a key-factor in the efficacy of an STG algorithm. The flow during this transition period is found to be quite sensitive to the mesh resolution. In this presentation, results from a series of simulations of different refinements are presented and discussed.