RMFM 2024 Rocky Mountain Fluid Mechanics Research Symposium

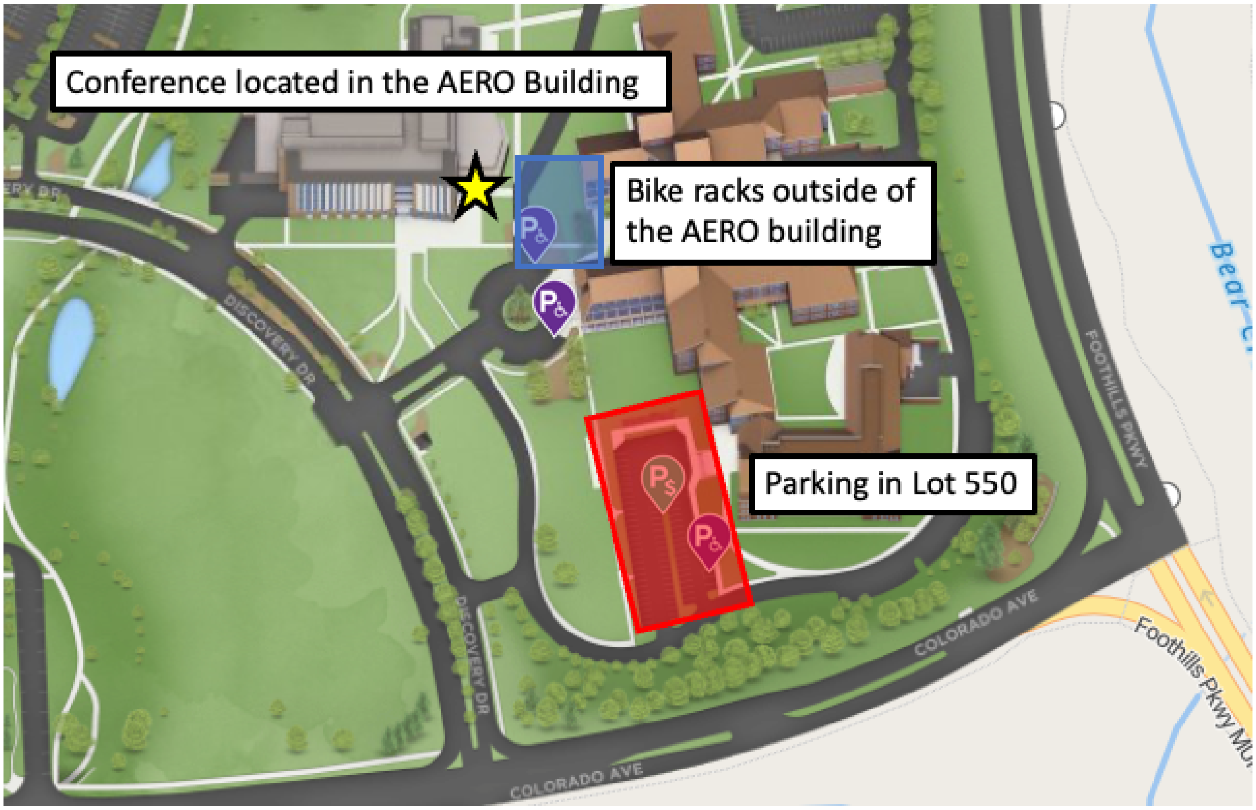
Rocky Mountain Fluid Mechanics Research Symposium 2024: Technical Program

August 6th, 2024

Keynote Presentation

Dr. Marsha Berger, (1:15 PM - 2:30 PM)
AERO 120
Modeling and Simulation of Tsunamis

Conference Location



The 2024 Rocky Mountain Fluid Mechanics Symposium takes place in the Aerospace Engineering Sciences (AERO) building on the CU Boulder East Campus. Parking is available in Lot 550 - denoted by the red box, and public bike racks are available and plentiful just outside of the conference building, shown by the blue box.

Presentation Schedule

Session 1A: Biological Flows
08:45 AM – 10:30 AM (AERO 111)

8:45 AM Nick Rovito (University of Colorado Boulder)
An In Silico Model of Flow-Mediated Fibrinolysis in Acute Ischemic Stroke

9:00 AM Josh Gregory (University of Colorado Boulder)
Recreating in vivo unsteady blood clot flow dynamics via an imaging-informed in silico model

9:15 AM David Montgomery (National Renewable Energy Laboratory)
Modeling shear-dependent platelet aggregation under flow

9:30 AM Brysen Mitchell (Montana State University)
Vortex Ring Formation through Flexible Bio-Inspired Nozzles

9:45 AM Laura Sunberg (University of Colorado Boulder)
Impact of Transport on Microplastics' Exposure to Biofouling-Favorable Conditions

10:00 AM Arkava Ganguly (University of Colorado Boulder)
Unified mobility expressions for externally driven and self-phoretic propulsion of particles

10:15 AM Elle Stark (University of Colorado Boulder)
Structures in smells: Coupling fluid dynamic cues to odor signals in olfactory landscapes

Session 1B: Geophysical Flows
08:45 AM – 10:30 AM (AERO 114)

8:45 AM Brianna Undzis (University of Colorado Boulder)
Open Water Sediment Dynamics on the Alaskan Beaufort Sea Shelf: A Numerical Modeling Study

9:00 AM Tina Geller (University of Colorado Boulder)
Currents in Arctic Alaskan lagoons: a numerical modeling study

9:15 AM Kari Perry (Montana State University)
Melt dynamics of ice cylinders in a cross-flow

9:30 AM Troy Johnson (University of Colorado Colorado Springs)
Approximation of rogue waves using Malmquist-Takenaka functions

9:45 AM Sara Tro (University of Colorado Boulder)
Asymptotic approximations for convection onset with Ekman pumping

10:00 AM Joseph Pugh (Colorado State University)
Characteristics of Curvilinear Flows Over Tilting Weirs

10:15 AM Peter Bevington (University of Colorado Boulder)
Large Eddy Simulation of Turbulent Fire Spread in a Douglas Fir Fuel Array

Session 2A: Computational Methods and Modeling
10:45 AM – 12:30 AM (AERO 111)

10:45 AM Abuajaila Kowas (University of Colorado Denver)
Lattice Boltzmann Method with Nonuniform Grids Based on Orthogonal Coordinate Transformation for Axisymmetric Flow Simulations

11:00 AM William Schupbach (University of Colorado Denver)
Central Moment Lattice Boltzmann Method for Efficient Simulations of Thermal Convective Flows using Orthogonal Curvilinear Coordinates

11:15 AM Kimmo Koponen (Colorado School of Mines)
A second-order, direct forcing, immersed boundary method for conjugate heat transport

11:30 AM Julia Ream (National Renewable Energy Laboratory)
Computational Fluid Dynamics for Microenvironment Development in Bioreactors

11:45 AM Daniel Abdulah (National Renewable Energy Laboratory)
Adaptive computing: macroscale surrogate modeling with on-the-fly training from microscale simulations

12:00 PM Ryker Fish (Colorado School of Mines)
A fast GPU library for fluctuating particle suspensions

12:15 PM Federico Municchi (Colorado School of Mines)
A Multiphase Particle-In-Cell method for simulating particle dispersion in fluidized beds

Session 2B: Wind Energy and Aerodynamics
10:45 AM – 12:30 AM (AERO 114)

10:45 AM Alexandre Cortiella (National Renewable Energy Laboratory)
Generative AI for wind-wave inflows

11:00 AM Omar Sallam (National Renewable Energy Laboratory)
Title: On the Architecture and Performance of a Variational Autoencoder (VAE) for high-fidelity compression of Offshore Wind-Wave Inflows

11:15 AM Jaylon McGhee (University of Colorado Boulder)
Influence of Wake Turbulence and Atmospheric Stability on the Fatigue Loads of Large-Scale Wind Turbines

11:30 AM Cameron Baird (National Renewable Energy Laboratory)
Machine Learning for Aeroacoustics of Generalized Wind Turbine Technologies

11:45 AM David Nelson (Montana State University)
Interaction of Spatially Evolving Vortex Pairs with Perturbed Walls

12:00 PM Yunxing Su (University of Colorado Boulder)
Bioinspired flapping foils for renewable energy harvesting and underwater propulsion

12:15 PM Adam Harris (University of Colorado Boulder)
3D Printable Phononic Subsurfaces for Passive Tollmien-Schlichting Wave Attenuation

Session 3A: High-Speed, Reacting, and Nonequilibrium Flows
2:45 PM – 4:30 PM (AERO 111)

2:45 PM Kelsea Souders (University of Colorado Boulder)
Evaluation of Sub-Grid Energy Characteristics in a Turbulent Reacting Flow

3:00 PM Denis Aslangil (Colorado School of Mines)
Coupled effects of variable density and isothermal background stratification on compressible Rayleigh-Taylor instability

3:15 PM Ahmet Kula (Colorado School of Mines)
Compressible Flow Over a Heated Cylinder with High Surface Temperatures

3:30 PM Elijah House (Colorado State University)
Numerical Modeling of Plasma-Assisted Shock Control in Supersonic Flow

3:45 PM Connor Morency (University of Colorado Boulder)
Modeling Thermochemical Nonequilibrium with the Continuous-Galerkin Finite Element Flow Solver PHASTA

4:00 PM Katie Plese (Colorado State University)
Integration of a High Enthalpy Source in the CSU Indraft Wind Tunnel

4:15 PM Mozhdeh Hooshyar (Colorado State University)
Femtosecond Laser-Induced Continuous Optical Discharge in Air: A Numerical Investigation

Session 3B: Wind Energy and Aerodynamics
2:45 PM – 4:30 PM (AERO 114)

2:45 PM Derek Goulet (University of Colorado Boulder)
The effects of macro- & micro-scale morphology on drag and odor capture around honey bee antennae

3:00 PM Ritu Raj (University of Colorado Boulder)
Exploring the chemical and mechanical modes of swimming of a bent rod actuator

3:15 PM Amogh Meshram (National Renewable Energy Laboratory)
Modeling DRI Pellet Reduction on Laboratory Scale and Pilot Scale using Multiphysics Modeling

3:30 PM Bashir Elbousefi (University of Colorado Denver)
Numerical Study of Thermocapillary Flows in Self-Rewetting Drop Impingement over Nonuniformly Heated Fluid Layers and in Enclosed Cavities Using the Lattice Boltzmann Method

3:45 PM Ward Cereck (Montana State University)
Simulations of Passive Bubble Separations Analogous to Capillary Nucleate Boiling in Microgravity

4:00 PM Souradeep Roychowdhury (University of Colorado Boulder)
An experimental and numerical investigation of collisions of wet particles

4:15 PM Rajarshi Chattopadhyay (University of Colorado Boulder)
Dynamics of deformable droplets in a straight rectangular channel: An experimental and numerical study

Keynote Presentation

Dr. Marsha Berger (1:15 PM - 2:30 PM)

Group Leader, Modeling and Simulation, CCM, Flatiron Institute
https://cs.nyu.edu/~berger/

Modeling and Simulation of Tsunamis

We describe our work on modeling and simulation of tsunamis. Tsunamis can have several causes, which lead to different issues in their numerics. We present the equations typically used, and discuss the assumptions behind them. For simulations involving the whole ocean, adaptive mesh refinement is important. The computation experiments use GeoClaw, an open-source software project, which will also be described.

Speaker Biography:
Marsha Berger is the Group Leader for Modeling and Simulation, and a Senior Research Scientist, having joined Flatiron Institute's Center for Computational Mathematics in January 2021. Berger's research interests include computational fluid dynamics, adaptive methods, and parallel scientific computing. Berger has been the recipient of the National Science Foundation Presidential Young Investigator Award and the Faculty Award for Women. She has also received the NASA 2002 Software of the Year Award for Cart3D with her team, and the IEEE Sidney Fernbach Award. In 2019, she was awarded the Norbert Wiener Prize in Applied Mathematics for her contributions to adaptive mesh refinement (AMR) and to Cartesian mesh techniques for automating the simulation of compressible flows in complex geometry. She has been elected to the National Academy of Sciences, the National Academy of Engineering, and the American Academy of Arts and Sciences. In addition, she was a fellow at the Society for Industrial and Applied Mathematics. Berger has a Ph.D. in Computer Science from Stanford University.

Biography taken from *https://www.simonsfoundation.org/people/marsha-berger/*

Back to table of contents

Presentation Abstracts

Session 1A: Biological Flows
08:45 AM – 10:30 AM (AERO 111)

An In Silico Model of Flow-Mediated Fibrinolysis in Acute Ischemic Stroke

Nick Rovito, Mechanical Engineering, University of Colorado Boulder
Debanjan Mikherjee, Mechanical Engineering, University of Colorado Boulder

Acute Ischemic Stroke (AIS) occurs when a brain artery is blocked by a thrombus. Standard AIS treatments use tissue plasminogen activator (tPA) to dissolve fibrin fibers in the clot. Despite widespread usage of tPA, treatments are often unsuccessful. Failures are largely due to the effect of clot-flow interactions on drug delivery to the thrombus, though detailed characterization of flow-mediated lysis remains challenging.

We address this by developing an anatomically realistic finite element (FE) model of the Circle of Willis to study the effect of flow on drug delivery to a fully occluding clot. Our model couples blood flow with advection, diffusion, and reactions of five key lysis proteins. A Brinkman term is applied to a fictitious clot domain, capturing hindered transport in a porous thrombus. The multiphyics FE model uses prior simulation data and clinical treatment protocol to inform boundary conditions. The FE models are coupled with a compartmental biochemical reaction model to describe thrombolysis and spatiotemporally evolving clot porosity. Our model captures flow-transport-reaction coupling, can generate detailed spatiotemporal predictions of drug delivery and lysis, and provide treatment descriptors that are challenging to obtain in vivo or in vitro.

Recreating in vivo unsteady blood clot flow dynamics via an imaging-informed in silico model

Josh Gregory, Mechanical Engineering, University of Colorado Boulder
Chayut Teeraratkul, Mechanical Engineering, University of Colorado Boulder
Timothy Stalker, Hematology, Thomas Jefferson University
Maurizio Tomaiuolo, Wills Eye Hospital
Debanjan Mukherjee, Mechanical Engineering, University of Colorado Boulder

The flow environment around a blood clot following vascular injury is complex and dynamic. Intravital microscopy is commonly used to study blood clot mechanics in vivo, attempting to capture some of these complexities. The resulting data show how a clot may form, grow, and embolize, but gaining quantifiable, deeper insights (like flow-induced forces) has remained a challenge. To address this, our group previously created an in silico method (IVISim) that allows us to gain these deeper insights. Here, we employ IVISim to compute the clot-hemodynamics interactions on six diYF knockout mice. These mice have a clotting factor removed, inhibiting their clot contraction abilities compared to wild type. Therefore, we seek to quantify the clot-hemodynamic interplay for a population of diYF mice, highlighting trends in clot deformation under impaired contraction. The simulation results for clot-hemodynamics interactions for these mice will be presented. Each simulation represents approximately two and a half minutes. Simulation endpoints include the time-varying hemodynamic loading on the clot as it grows, deforms, and embolizes. A novel description of the force-deformation behavior that combines the growth and embolization events together will be elucidated based on the diYF clot data.

Modeling shear-dependent platelet aggregation under flow

David Montgomery, National Renewable Energy Laboratory

Blood clotting is an intricate biological response that is triggered following vascular injury to prevent excessive bleeding at the site of an injured blood vessel. The hemostatic system is composed of three fundamental components: platelet aggregation, coagulation, and fibrinolysis. Platelet aggregation involves a primarily physical process wherein platelets adhere to the injured vessel wall, become chemically activated, recruit additional platelets, and form a platelet plug. In recent years, increased attention has been given to the pivotal role of von Willebrand Factor (vWF), a multimeric glycoprotein in the bloodstream, in mediating platelet aggregation. Understanding the intricate mechanisms of vWF-platelet interactions is essential for comprehending the complex dynamics of clot formation and identifying potential targets for therapeutic interventions in bleeding disorders such as von Willebrand disease. This work presents a novel shear-dependent platelet aggregation model that incorporates shear-dependent activation, adhesion, and cohesion mechanisms mediated by vWF. The model is validated with data produced from in-vitro microfluidic experiments where agonists such as ADP and Thromboxane A2 were systematically inhibited to determine the parameters associated with vWF-mediated adhesion and cohesion.

Vortex Ring Formation through Flexible Bio-Inspired Nozzles

Brysen Mitchell, Mechanical Engineering, Montana State University
Sarah Morris , Mechanical Engineering, Montana State University

Vortex rings are found over many scales across nature, including the efficient locomotion of jellyfish and squid. These animals move by contracting their bodies to expel fluid out of a flexible orifice, producing a series of vortex rings. Inspired by these animals, this study focuses on the fluid-structure interaction of vortex rings generated with flexible circular nozzles of varying stiffness. A piston-cylinder setup ejects different volumes of fluid to create vortex rings. Particle image velocimetry is used to quantify thrust via hydrodynamic impulse, and Finite-time Lyapunov Exponent fields are quantified to study vortex ring pinch off. The nozzle deformation is tracked and used to find the damped natural frequency of each nozzle to predict peak deformation timing. Our results show that decreasing nozzle stiffness up to an optimal point maximizes impulse for the same kinematic input. However, surpassing this optimal flexibility causes a decline in impulse for the given flow conditions. Synchronizing the peak nozzle deformation with the end of the fluid acceleration maximizes output impulse. The largest relative increase in impulse occurs for the lowest ejected volume due to the importance of the nozzle deformation behavior in the startup of flow generation.

Impact of Transport on Microplastics' Exposure to Biofouling-Favorable Conditions

Laura Sunberg, Institute of Arctic and Alpine Research (INSTAAR), University of Colorado Boulder
Julia Moriarty, Department of Atmospheric and Oceanic Sciences, INSTAAR, University of Colorado Boulder
Melissa Moulton, National Center for Atmospheric Research, University of Washington

Microplastics are a growing problem in the oceans. The transport of microplastics is complicated by biofouling, i.e., the growth organisms adhering to their surfaces. Biofouling increases the size and density of microplastics, changing their settling velocity and thereby their transport. We will present results from modeling the transport of microplastics that are subject to biofouling in the northern Gulf of Mexico. We find the amount of biofouling follows a strong seasonal cycle with more biofouling occurring in the summer, but that the amount of biofouling also varies non-trivially at timescales of weeks. This variation is dependent on how the microplastics sample the flow, as phytoplankton concentrations show significant spatiotemporal heterogeneity. Our results highlight the importance of accounting for changing microplastic properties in response to exposures to local environmental conditions when predicting microplastic transport.

Unified mobility expressions for externally driven and self-phoretic propulsion of particles

Arkava Ganguly, Chemical and Biological Engineering, University of Colorado Boulder
Ankur Gupta, Chemical and Biological Engineering, University of Colorado Boulder
Souradeep Roychowdhury, Chemical and Biological Engineering, University of Colorado Boulder

Technologies involving artificial micro-swimmers are advancing for targeted drug delivery, diagnostics, and environmental cleanup, yet face unique challenges in vivo compared to controlled in vitro environments. Understanding microswimmer propulsion across different conditions is crucial. The mobility of external and self-propulsion of particles is evaluated by simultaneously solving the solute conservation equation, interaction potential equation, and the Stokes equation with a body force. This method, though accurate, becomes complex, especially at finite interaction length scales. Inspired by Brady J. Fluid Mech. (2021), we obtain unified mobility expressions with arbitrary interaction potentials. Firstly, we show that these expressions can recover well-known mobility relationships in external electrophoresis and diffusiophoresis for arbitrary double-layer thickness. Secondly, at the thin interaction length limit, these equations reduce to the slip velocity expressions for spherical microswimmers well-known in active particle literature. Finally, we study the motion of an autophoretic Janus particle through an interplay of active and passive diffusiophoresis. We investigate how surface heterogeneities, external gradients, and interaction length scales affect swimming speeds.

Structures in smells: Coupling fluid dynamic cues to odor signals in olfactory landscapes

Elle Stark, CEAE, University of Colorado Boulder
Aaron True, CEAE, University of Colorado Boulder
John Crimaldi, CEAE, University of Colorado Boulder

Animals exploit olfactory cues for survival, requiring navigation of complex odor plumes in diverse environments. As organisms navigate, they encounter a rich set of fluid dynamic cues (e.g. accelerations, strains, vorticity) alongside odor cues, owing to the coupling of the Navier-Stokes and Advection-Diffusion equations. Lagrangian coherent structures (LCS) provide an intuitive framework for investigating coupled flow and odor structure; they are derived from the flow deformation field and have been shown to drive the spatial organization and temporal evolution of scalar fields in chaotic flows. We investigate this coupled structure using 2D numerical simulations of chaotic plume dispersion with passive scalars released downstream of an array of interacting cylinder wakes. We use event-based statistics centered on whiff occurrences to quantify relative timing between flow

and odor cues across plume regions and candidate flow cues. We find significant differences in relative timing across plume regions, and provide a physical interpretation based on the LCS framework and considerations of local mixing regimes. Spatial structure in relative timing variations between flow and odor cues could inform fundamental olfactory search tasks like edge detection or source localization.

Session 1B: Geophysical Flows
08:45 AM – 10:30 AM (AERO 114)

Open Water Sediment Dynamics on the Alaskan Beaufort Sea Shelf: A Numerical Modeling Study

Brianna Undzis, Atmospheric and Oceanic Sciences , University of Colorado Boulder
Julia Moriarty, Atmospheric and Oceanic Sciences, University of Colorado Boulder
Irina Overeem, Geological Sciences, University of Colorado Boulder
Emily Eidam, College of Earth, Ocean, and Atmospheric Sciences, Oregon State University

Sediment dynamics on continental shelves can impact coastal geomorphology and biogeochemical cycling. In the coastal Arctic, changing sediment dynamics have implications for carbon sequestration and primary production. Thus, this study aims to better understand hourly to monthly variability in coastal sediment transport. We implemented a coupled hydrodynamic – sediment transport numerical model, the Regional Ocean Modeling System (ROMS) - Community Sediment Transport Modeling System (CSTMS) within the Coupled Ocean-Atmosphere-Wave-Sediment Transport (COAWST) Modeling System, for the 2019 open water (nearly ice-free) season on the Alaskan Beaufort Sea shelf. Results showed that wave- and current-induced bed shear stresses frequently exceeded the critical stress for erosion, causing resuspension. Bed stress was dominated by waves in areas shallower than 10 m and by currents in areas deeper than 20 m. When averaged over the open water season, modeled suspended sediment fluxes were westward, despite prevailing eastward currents, due to fast westward current events. We expect these events may become more important for future sediment fluxes as storm frequency is projected to increase.

Currents in Arctic Alaskan lagoons: a numerical modeling study

Tina Geller, Atmospheric and Oceanic Sciences, University of Colorado Boulder
Julia M Moriarty, Atmospheric and Oceanic Sciences, University of Colorado Boulder

Lagoon hydrodynamics determine when and where nutrients from rivers are transported, impacting the growth of phytoplankton and marine food webs. We implemented a numerical model, the Regional Ocean Modeling System, to characterize currents in Arctic Alaskan lagoons that vary in river influence and shelf connectivity from June to September of 2019. The model accounts for winds, rivers, shelf circulation, tides, and bathymetry. Preliminary results suggest that winds and shelf currents often drive the direction and magnitude of lagoon currents, but rivers are also important when discharge is high. During times of high river discharge, westward winds cause freshwater entering Arey Lagoon to be more directly transported to the shelf; freshwater entering Jago Lagoon is instead retained longer in the lagoon system. The opposite is true during eastward wind and shelf currents. Kaktovik Lagoon, which has low shelf connectivity and low river influence, has relatively weak currents. Nutrient retention in Arctic Alaskan lagoons therefore likely depends on the direction and strength of wind and shelf currents as well as the location of rivers and lagoons relative to shelf inlets.

Melt dynamics of ice cylinders in a cross-flow

Kari Perry, Mechanical Engineering, Montana State University
James Luo, Aerospace Engineering, Cornell University
C.H.K. Williamson, Aerospace Engineering, Cornell University
Sarah Morris, Mechanical Engineering, Montana State University

The formation and release of icebergs into the ocean accounts for half of the total freshwater discharged from ice sheets (Depoorter, 2013). As icebergs float and melt far from their sources, they impose adverse effects on both local and global marine ecosystems. Present-day ice melt models frequently underpredict iceberg melt dynamics, necessitating further research into the fundamental fluid mechanics of icebergs. In this work, we study the bluff body dynamics of ice to understand the effect of a cross-flow on iceberg melt dynamics and resultant meltwater spread. Experiments are completed in a closed-loop water channel for initial Reynolds numbers ranging from Re=0 to 885. Flow structures are measured using particle image velocimetry and melt rates are found from surface area measurements. It is found that the melt rate increases with higher Re; however, across all Re it is found that as the ice melts, the ice shape does not remain symmetric. Rather, the resultant vortex wake leads to different local melt rates between the front and back faces, creating significant shape changes. As a result of the time-varying ice cylinder shape, preliminary results show that the formation length, wake width and Strouhal number all vary through the melt process as well as with Re.

Approximation of rogue waves using Malmquist-Takenaka functions

Troy Johnson, Mathematics, University of Colorado Colorado Springs
Justin Cole, Mathematics, University of Colorado Colorado Springs

Rogue waves are fascinating large amplitude coherent structures that abruptly appear and then disappear soon after. In certain partial differential equations these waves are modeled by rational solutions. In this talk we will discuss approximating rogue wave solutions in a basis of orthogonal functions known as the Malmquist-Takenaka (MT) functions. This family of rational functions can be directly mapped to a modified Fourier series, allowing the fast Fourier transform computation of the spectral MT coefficients from which spectral differentiation matrices are derived. The approximation of the various rogue wave solutions in the nonlinear Schrödinger (NLS) equation is explored.

Asymptotic approximations for convection onset with Ekman pumping

Sara Tro, Applied Mathematics, University of Colorado Boulder
Ian Grooms, Applied Mathematics, University of Colorado Boulder

Ekman pumping is a phenomenon induced by no-slip boundary conditions in rotating fluids. In the context of Rayleigh-Bénard convection, Ekman pumping causes a significant change in the linear stability of the system compared to when it is not present (that is, stress-free). Motivated by numerical solutions to the marginal stability problem of the incompressible Navier-Stokes (iNSE) system, we seek analytic asymptotic solutions which describe the departure of the no-slip solution from the stress-free. The substitution of normal modes into a reduced asymptotic model yields a linear system for which we explore analytic solutions for various scalings of wavenumber. We find very good agreement between the analytic asymptotic solutions and the numerical solutions to the iNSE linear stability problem with no-slip boundary conditions.

Characteristics of Curvilinear Flows Over Tilting Weirs

Joseph Pugh, Civil and Environmental Engineering, Colorado State University
S. Karan Venayagamoorthy, Civil and Environmental Engineering, Colorado State University
Timothy K. Gates, Civil and Environmental Engineering, Colorado State University

In the context of water resources management, the need for dual-function hydraulic structures that offer both flow measurement and water-level control has become of increasing importance in arid regions experiencing population growth and a greater occurrence of climatic extremes. The tilting weir is one such type of structure that only recently has been examined as a means for obtaining discharge measurement. In light of this end, we examine the fundamentals of the discharge equation for a sharp-crested weir, its implicit assumptions and their validity, and the feasibility of applying this same approach to a weir that is tilted downstream. Experimental results using PIV of flow over scaled tilting weir models show that the assumptions of inviscid and irrotational flow in the vicinity of the weir are sufficiently true, leading to the application of the same mathematical approach for a tilting weir discharge equation. Additionally discussed are the development of a in-house solver for the Poisson equation that can be used to estimate the pressure field given 2D PIV data, and comparisons between flow structures found using experimental techniques vs. numerical methods with varying turbulence closure schemes.

Large Eddy Simulation of Turbulent Fire Spread in a Douglas Fir Fuel Array

Peter Bevington, Mechanical Engineering, University of Colorado Boulder
Peter Hamlington, Mechanical Engineering, University of Colorado Boulder
John Farnsworth, Aerospace Engineering, University of Colorado Boulder
Greg Rieker, Mechanical Engineering, University of Colorado Boulder
Laura Shannon, Aerospace Engineering, University of Colorado Boulder
Sean Coburn, Mechanical Engineering, University of Colorado Boulder

Computational simulations are proving to be an essential tool in the study of wildland terrain fires, due to their relatively low cost and low risk. Such simulations span a large range of spatial and temporal scales, from the minuscule chemistry-turbulence interactions of combustion to the large-scale eddies that govern atmospheric flows. While wildfire research is typically focused on the largest scales, bench-scale simulations are more practical from a validation perspective, and aim to inform the development of terrain-scale models. Here, we perform a large eddy simulation (LES) with adaptive mesh refinement to model the multi-phase pyrolysis and combustion of Douglas fir pegs in a turbulent crossflow. We compare simulation results with experimental measurements captured in a wind tunnel, using optical imaging and a multi-diagnostic suite. This work represents the initial steps in the application of state-of-the-art computational and experimental techniques to develop a computational tool for the prediction of wildland fire spread across a range of scales.

Session 2A: Computational Methods and Modeling
10:45 AM – 12:30 AM (AERO 111)

Lattice Boltzmann Method with Nonuniform Grids Based on Orthogonal Coordinate Transformation for Axisymmetric Flow Simulations

Abuajaila Kowas, Mechanical Engineering, University of Colorado Denver
William Schupbach, Mechanical Engineering, University of Colorado Denver

kannan Premnath, Mechanical Engineering, University of Colorado Denver

Axisymmetric flow equations represent a dimensional reduction of three-dimensional flows where axial symmetry can be exploited. If such flows involve boundary layers or shear layers, they can be more efficiently resolved by the clustering of grids that follow the flow features. However, the standard lattice Boltzmann (LB) methods are generally restricted to uniform Cartesian grids. To overcome this limitation, we develop a new LB method that solves the axisymmetric Navier-Stokes equations based on general orthogonal coordinates in a computational domain. We construct a collision model whose equilibria as well as the geometric body forces depend on the metric coefficients and their spatial derivatives arising from the variable grids used in the physical domain. The swirl effects in such flows are accounted for by computing the azimuthal momentum that satisfies a convection-diffusion type equation with a source term in general orthogonal coordinates using another LB scheme. Both these LB methods are designed to accommodate the usual collide-and-stream steps while still effectively allowing the use of variable grids, and their collision operators are based on central moments. We show the efficacy of our approach for various canonical axisymmetric flows including swirl effects.

Central Moment Lattice Boltzmann Method for Efficient Simulations of Thermal Convective Flows using Orthogonal Curvilinear Coordinates

William Schupbach, Mechanical Engineering, University of Colorado Denver
Eman Yahia, Mechanical Engineering, University of Colorado Denver
Kannan Premnath, Mechanical Engineering, University of Colorado Denver

The lattice Boltzmann method (LBM) is a drastically simplified discretization of the Boltzmann equation for computational fluid dynamics (CFD). We construct two central moment-based LBMs for recovering the Navier-Stokes equations and the energy equation. The use of such central moment formulations in modeling the collision step offers significant improvements in numerical stability. For efficient simulations of thermal flows involving thin boundary layers and curved boundaries, we develop our formulation via orthogonal curvilinear coordinate transformations, which allow the use of nonuniform body-fitted meshes with grid clustering/stretching and thus significantly overcomes the limitations of the standard LBMs that are restricted to uniform Cartesian grids. We demonstrate the utility of our approach for a case study involving thermal convective buoyancy-driven flows.

A second-order, direct forcing, immersed boundary method for conjugate heat transport

Kimmo Koponen, Mechanical Engineering, Colorado School of Mines
Nils Tilton, Mechanical Engineering, Colorado School of Mines

We propose a direct forcing method for simulating discontinuous Dirichlet and Neumann conditions at the interface between two materials. The method is motivated by applications to conjugate heat transfer and electrohydrodynamics. We consider a 2D Poisson equation in a square domain in which an object made of one material is immersed in a second material. The Poisson equation is solved in both materials using finite-volume methods on a Cartesian grid. To extend the direct forcing method of Fadlun et al. (2000), we identify pairs of adjacent grid points, called forcing pairs, that lie on opposite sides of the interface. We approximate the fields about the forcing pairs using 2D Taylor polynomials. We show that Neumann conditions generally require a 12 point stencil, with 6 points in each material. The stencils and coefficients are easily implemented using the signed distance function. We verify second-order spatial accuracy by comparing with analytical solutions for two-phase problems of varying geometries. We also explore cases where complex geometries require reduced stencils and/or smoothing of the local interface.

Computational Fluid Dynamics for Microenvironment Development in Bioreactors

Julia Ream, National Renewable Energy Laboratory
Mohammad Rahimi, National Renewable Energy Laboratory
Nicholas Wimer, National Renewable Energy Laboratory
William Cordell, National Renewable Energy Laboratory
Hariswaran Sitaraman, National Renewable Energy Laboratory
Marc Day, National Renewable Energy Laboratory
Davinia Salvachua, National Renewable Energy Laboratory

Large-scale bioreactors play a critical role in enhancing the bioeconomy and advancing biotechnology-related industries. In transitioning from the laboratory- to the industrial-scale, the complexity of these fluid systems increases, making microbial performance harder to predict. Computational fluid dynamics is a vital tool in exploring how the change in dynamics during scale-up affects these bioprocesses. We investigate the formation and impact of microenvironments within bioreactors of increasing size and their influence on microbe cell production. The investigation is based on a multiphase solver we developed within OpenFOAM that couples multi-component transport and reaction processes specific to the organism P. putida, incorporating experimental data through metabolic model parameters. We discuss reactor geometry implementation and various coupling strategies explored so far through the development process.

Adaptive computing: macroscale surrogate modeling with on-the-fly training from microscale simulations

Daniel Abdulah, National Renewable Energy Laboratory
Kevin Griffin, National Renewable Energy Laboratory
Marc Henry de Frahan, National Renewable Energy Laboratory
Marc Day, National Renewable Energy Laboratory

Adaptive computing (AC) methods leverage a surrogate model to resolve small scale processes over a large scale domain. We consider solar cell modeling, which requires coupling a fluid dynamical model for gas flow with a molecular-scale Kinetic Monte Carlo (KMC) model for the development of perovskite crystal lattices. Within each grid cell of the boundary layer, the fluid dynamical driver uses the AC framework to determine lattice formation. The AC framework determines when to call the KMC code or use the computationally inexpensive surrogate model estimate. We present two approaches using different constraints. First, a computational budget fixes the number of KMC model runs while minimizing uncertainty in the surrogate model training. Second, an accuracy budget limits uncertainty in the result while maximizing use of the surrogate.

A fast GPU library for fluctuating particle suspensions

Ryker Fish, Applied Mathematics, Colorado School of Mines
Brennan Sprinkle, Applied Mathematics, Colorado School of Mines

Raúl Pérez Peláez, Universitat Pompeu Fabra
Aleksandar Donev, Applied Mathematics, Courant Institute

We'll introduce a fast and modular GPU library, libMobility, that solves the hydrodynamic mobility problem for Stokesian particle suspensions in open, periodic, and confined geometries. In particular, we've implemented fast routines to apply mobility matrices and their principle square root, ensuring that the library is immediately available for use in Brownian dynamics applications. In addition to being a high-performance library, libMobility is designed to have a streamlined installation and be easy to integrate into existing codes. We include a testing framework which verifies that important thermodynamic properties are maintained by each method.

A Multiphase Particle-In-Cell method for simulating particle dispersion in fluidized beds

Federico Municchi, Mechanical Engineering, Colorado School of Mines
Keaton Brewster, Mechanical Engineering, Colorado School of Mines
Lei Fuqiong, Mechanical Engineering, Colorado School of Mines
Gregory Jackson, Mechanical Engineering, Colorado School of Mines

In this work, we present a Multiphase Particle-In-Cell (MPPIC) method capable of capturing particle dispersion in fluidized beds. Thanks to the use of a continuum particle stress instead of detailed particle pair collisions, this method allows to simulate large number of particles. However, it also presents drawbacks such as overpacking (particles overlapping) and lack of stability. We discuss the issue of overpacking and we show that, while it can be circumvented, this is an intrinsic characteristics of MPPIC methods. We also discuss how to compute the gradient of the particle volume fraction for calculating the particle stress. Finally, we compare our method against the standard MPPIC solver implemented in OpenFOAM and CFD-DEM simulations. We also describe an approach to extract the axial particle dispersion coefficient from experimental measurements of pressure oscillations, and compare it against results from MPPIC simulations.

Session 2B: Wind Energy and Aerodynamics
10:45 AM – 12:30 AM (AERO 114)

Generative AI for wind-wave inflows

Alexandre Cortiella, Computer Science, National Renewable Energy Laboratory

Designing floating offshore wind turbines (FOWTs) and their platforms involves costly aeroelastic simulations with coupled wind-wave inflow conditions that are challenging to create in a self-consistent manner. Existing wind-wave inflow precursors require expensive coupled, multiphysics simulations that make extreme event exploration and uncertainty quantification tasks inviable. Generative models have emerged as valuable tools for producing synthetic samples from limited data, thus making uncertainty quantification and design space exploration more feasible. This work proposes a Latent Diffusion Model (LDM) based on spatial-temporal neural network transformer architectures capable of efficiently generating consistent, coupled wind-wave inflows conditioned on turbine loading cases. This approach will dramatically reduce the computational barriers to performing floating offshore wind turbine simulations, studying extreme load conditions, and enhancing current FOWT designs.

Title: On the Architecture and Performance of a Variational Autoencoder (VAE) for high-fidelity compression of Offshore Wind-Wave Inflows

Omar Sallam, Computer Science, National Renewable Energy Laboratory
Alex Cortiella, Computer Science, National Renewable Energy Laboratory
Alex Rybchuk, Computer Science, National Renewable Energy Laboratory
Andrew Glaws, Applied Mathematics, National Renewable Energy Laboratory
Ryan King, Applied Mathematics, National Renewable Energy Laboratory

Autoencoders (AEs) have numerous applications in high-fidelity fluid dynamics, numerical simulations, and fluid flow data synthesis. These include fluid flow reconstruction from sparse representations, flow super-resolution, and the compression and latent space reconstruction of fluid fields data. In this study, we introduce a Variational Autoencoder (VAE) designed for constructing the latent space of free-surface wind-wave flow fields as a compression stage for a diffusion-based generative model.

Our VAE is trained on 2D inflow planes of high-fidelity wind-wave flow data under turbulent offshore inflow wind conditions, generated by AMR-Wind solver. This research explores the impact of incorporating Physics-Informed (PI) loss terms into the VAE loss function, adjusting the network depth using Residual Blocks (RBs), employing Dual Scale Residual Blocks (DS-RBs), and integrating a discriminator into the network architecture. These experiments were conducted across various compression ratios to thoroughly evaluate their effects.

Reconstruction performance was assessed using metrics such as the reconstruction loss of velocity fields and the free surface, the statistical distribution of physical quantities, and the reconstruction of the spatial turbulent kinetic energy spectrum.

Influence of Wake Turbulence and Atmospheric Stability on the Fatigue Loads of Large-Scale Wind Turbines

Jaylon McGhee, Aerospace Engineering, University of Colorado Boulder
Brooke Stanislawski, National Renewable Energy Laboratory
John Farnsworth, Aerospace Engineering, University of Colorado Boulder
Ganesh Vijayakumar, University of Colorado Boulder

Atmospheric stability, which primarily affects vertical velocity fluctuations within the atmospheric boundary layer (ABL), also influences the vertical shear and turbulence intensity at wind turbine operational heights. As a result, these inflow parameters can significantly impact the fatigue loads on various wind turbine components. The ExaWind software suite was used to simulate the ABL and the response of two IEA 15-MW wind turbines separated by seven rotor diameters under a range of atmospheric stability conditions from neutral to highly unstable. Analysis was carried out to better understand how atmospheric stability affects the upstream turbine's wake properties and the downstream turbine's fatigue loads. The results show that despite experiencing lower wind speeds, the downstream wind turbine generally undergoes increased rotor-based fatigue loads across all stability conditions. In contrast, the upstream turbine's blade root fatigue loads are consistently higher. These results provide insights into the complex interactions between atmospheric stability and wake effects in large-scale wind farms, which could help improve comprehensive wind farm design strategies and long-term turbine reliability predictions under variable atmospheric conditions.

Machine Learning for Aeroacoustics of Generalized Wind Turbine Technologies

Cameron Baird, Computer Science, National Renewable Energy Laboratory
Andrew Glaws, Applied Mathematics, National Renewable Energy Laboratory

As wind energy deployment increases, sound ordinances are being implemented to address social concerns about their impact on residents. However, ordinances also limit the available locations for wind turbine site selection, impacting wind energy potential across the U.S. Aeroacoustics models can leverage site-specific wind resources to help understand the effects of local sound laws on wind energy in different regions across the country. However, such models are too computationally expensive to characterize large-scale wind energy potential. Previous work proposed a machine learning model to translate sound ordinances into building setback distances that provide useful spatial constraints for wind turbine placement. This approach was limited by the assumption of a fixed type of turbine which is quickly becoming outdated. In this work we explore methods for extending surrogate modeling capabilities to multiple turbine technologies, developing a neural network to predict sound pressure levels within a five-kilometer radius around a given turbine. The network is trained on synthetic aeroacoustic data, including variable environmental conditions and wind turbine technologies. We demonstrate the model's ability to generalize to new turbine technologies not seen during training.

Interaction of Spatially Evolving Vortex Pairs with Perturbed Walls

David Nelson, Mechanical Engineering, Montana State University
Sarah Morris, Mechanical Engineering, Montana State University

Counter-rotating vortex pair wakes (CRVP) are an unavoidable by-product of aircraft lift generation. Aircraft encountering these wakes can be subjected to unexpected hazardous rolling moments. This is especially true in terminal flight phases where there is insufficient altitude to recover. Naval aircraft in terminal flight phases must also contend with approaches above a highly dynamic oceanic free surface. This investigation uses a towed delta wing to generate a spatially-evolving CRVP that descends towards the ground plane under its own self-induced velocity. A static wavy wall is placed at the ground, representative of a perturbed oceanic surface. Laser-induced fluorescence and particle-image velocimetry (PIV) measurements are taken to assess the vortex-wall interaction and quantify the rate of CRVP decay. The spatially-evolving CRVP comprises of both axial and circumferential flows and approaches the ground plane at an angle. Preliminary results suggest that the inclined vortex pair produces time-dependent secondary vortex structures at the wall, increasing the decay rate of the primary CRVP. Future work will use a motorized, dynamic ground plane to further investigate the influence of time-dependent secondary vortex structures on the decay of the primary CRVP.

Bioinspired flapping foils for renewable energy harvesting and underwater propulsion

Yunxing Su, Mechanical Engineering, University of Colorado Boulder
Nicole Xu, Mechanical Engineering, University of Colorado Boulder

Renewable energy is sourced from naturally replenishing resources such as sunlight, wind, tides, waves, and geothermal heat. Increasing concerns about climate change and global warming underscore the need for greater reliance on renewable energy. Using flapping foils to harvest tidal current energy, we will highlight the tandem-foil interaction for energy harvesting and propose a global phase model to predict energy harvesting performance. In addition to energy harvesting, tandem flapping fins are also commonly found in animal locomotion. We will investigate two tandem flapping fins for underwater propulsion to design a highly efficient and maneuverable propulsion system. Particle image velocimetry (PIV) is used to visualize the fluid flow around the fins. However, commercial PIV particles often pose environmental and health risks for live animal experiments. We propose using biodegradable particles in our PIV experiments, which are nontoxic, sustainable, accessible, and cost-effective. PIV experiments across various scales (jellyfish (10 cm) and brine shrimp (5 mm)) were conducted to test their feasibility. By exploring energy harvesting and propulsion of flapping foils, we aim contribute to the development of more sustainable and efficient energy and propulsion systems.

3D Printable Phononic Subsurfaces for Passive Tollmien-Schlichting Wave Attenuation

Adam Harris, Materials Science and Engineering, University of Colorado Boulder
Jaylon McGhee, Aerospace Engineering, University of Colorado Boulder
John Farnsworth, Aerospace Engineering, University of Colorado Boulder
Mahmoud Hussein, Aerospace Engineering, University of Colorado Boulder

A phononic subsurface (PSub) is an elastic structure that passively interacts with a flow through linear vibrations at the fluid-structure interface. Elastic waves stemming from flow in-stabilities excite the PSub, meanwhile the PSub is tuned to resonate out of phase with these waves causing their suppression by destructive interference. This leads to favorable effects such as attenuation of instabilities, transition delay, and reduced skin-friction drag. PSubs are de-signed as elastic metamaterials (MMs) which utilize local resonance. Varying the material pa-rameters of the PSub in addition to manipulating the geometry of the PSub's unit cell enables the designer to tune the PSub for operation at a specific frequency. Here, a MM-based PSub was designed to target a Tollmien-Schlichting (T-S) wave near 370 Hz in an otherwise laminar sub-sonic flow. Frequency response (FRF) characteristics were obtained computationally via finite-element analysis in COMSOL and verified using a laser vibrometer. FRF results promise that the PSub will attenuate a T-S wave anywhere between 350-400 Hz. Future work towards experi-mental validation will involve installation of the PSub into the low-speed wind tunnel in the Ex-perimental Aerodynamics Laboratory at the University of Colorado Boulder.

Session 3A: High-Speed, Reacting, and Nonequilibrium Flows
2:45 PM – 4:30 PM (AERO 111)

Evaluation of Sub-Grid Energy Characteristics in a Turbulent Reacting Flow

Kelsea Souders, Mechanical Engineering, University of Colorado Boulder
Peter Hamlington, Mechanical Engineering, University of Colorado Boulder

The cascade of energy in most realistic turbulent flows occurs in the forward direction, where energy is injected at large scales of motion and is transferred non-dissipatively to progressively smaller scales of motion before dissipating at the smallest scales. In some cases, such as flows featuring combustion, energy release from the chemical reactions occurs near the smallest scales of motion and may be transferred to large scales of motion in a phenomenon known as energy "backscatter". This can lead to inaccuracies and numerical instabilities in large-eddy simulations (LES), where many popular sub-grid scale closure models assume that the transfer of energy between the resolved and sub-grid scales is exclusively in the forward direction.

We have used PeleC, a fully compressible reacting flows code, to simulate a bluff-body stabilized premixed methane flame in an environment subjected to both free-stream turbulence and a mean background pressure gradient. We apply spatial filters to the fully-resolved dataset in order to mimic the filters used in LES, thereby allowing us to directly compute the sub-grid scale energy flux, which allows us to characterize the turbulent energy backscatter.

Coupled effects of variable density and isothermal background stratification on compressible Rayleigh-Taylor instability

Denis Aslangil, Mechanical Engineering, Colorado School of Mines
Orkun Ustun, Mechanical Engineering, Colorado School of Mines

Rayleigh-Taylor instability (RTI) occurs at the perturbed interface between fluids with different densities, where the acceleration and density gradient are in opposite directions. RTI is observed in nature, such as atmospheric and geological flows, and in engineering applications, such as the mixing stage of ramjets and scramjets and Inertial Confinement Fusion (ICF). We study the coupled effects of the isothermal background stratification strength and large molar mass differences between the mixing fluids through Direct Numerical Simulations of the two-dimensional single-mode and three-dimensional multi-mode RTI by solving compressible multi-species Navier-Stokes equations. From the low Atwood number cases, it is observed that when the molar mass ratio between the mixing fluids is small, the growth of the mixing layer ceases, and the flow becomes molecularly well-mixed as the background stratification increases. However, our high Atwood number simulations suggest that there is a more complex flow evolution where a stronger background stratification leads to a relatively faster mixing growth rate. In addition to the overall flow evolution, we will compare the vorticity transport equation terms for the low and high Atwood numbers under weak and strong background stratifications.

Compressible Flow Over a Heated Cylinder with High Surface Temperatures

Ahmet Kula, Mechanical Engineering, Colorado School of Mines
Denis Aslangil, Mechanical Engineering, Colorado School of Mines

Flow over heated objects is a strongly coupled heat transfer and fluid dynamics problem that arises in various engineering applications, such as hypersonic flight conditions, particle-fluid interactions, and re-entry problems. Due to the high-temperature variations observed in these flows, the constant fluid transport property assumptions may not be sufficient to capture the actual physics. This study compares constant and temperature-dependent transport coefficient models for compressible flows over a 2D cylinder and a 3D sphere using direct numerical simulations (DNS). DNS is performed at different temperature ratios where the object surface temperature is 1.2, 1.5, and 3.0 times the free-stream flow temperature with Reynolds numbers ranging from 20 to 300. DNS suggests that the mean drag coefficient and separation angle are considerably larger for the cases where we account for temperature-dependent fluid transport properties compared to their counterparts with constant fluid transport properties. Additionally, the Nusselt number and total convective heat flux become highly sensitive to the choice of the transport coefficient model. A detailed comparison of the effects of transport coefficient models on aerodynamics, heat transfer, and vortex dynamics will be presented.

Numerical Modeling of Plasma-Assisted Shock Control in Supersonic Flow

Elijah House, Mechanical Engineering, Colorado State University
Ciprian Dumitrache, Mechanical Engineering, Colorado State University

The effect of plasma on shock wave (SW) train dynamics in a supersonic wind tunnel operating at Mach 2.5 are explored. Numerical simulations were performed inside a rectangular channel using our in-house APDL-CFD code. A 10° wedge mounted on the upper wall is used to generate the oblique SW train. A simulated quasi-DC (Q-DC) electrical discharge is generated slightly before the position where the first shock impacts the bottom wall of the test section. By varying the plasma temperature and slightly altering the location of the plasma within the channel, we observed an upstream displacement of the reflected SW by as much as 20 mm at the upper wall impingement point. Following plasma generation, a thickening of the BL on the bottom wall is observed. Once the incident shock interacts with the plasma-affected BL, a separation bubble is observed, and the reflected shock is largely mitigated. Instead, two weaker plasma and flow reattachment shocks are formed at a different incidence angle compared to the original stronger reflected shock. A better understanding of these plasma influenced shock-boundary layer interactions (SWBLI) opens the possibility for the development of a new plasma-based active method of SW control in hypersonic applications.

Modeling Thermochemical Nonequilibrium with the Continuous-Galerkin Finite Element Flow Solver PHASTA

Connor Morency, Aerospace Engineering, University of Colorado Boulder
Kenneth Jansen, Aerospace Engineering, University of Colorado Boulder

This presentation aims to introduce developments within the PHASTA flow solver that are needed for applications in hypersonic CFD research. The PHASTA formulation is described and unit tests are performed that serve to validate new features which enable the simulation of hypersonic flow characterized by thermochemical nonequilibrium. These features include a multi-species reacting flow module, a two-temperature formulation that includes a model for vibrational temperature, a model for mass diffusion, and the Spalart-Allmaras RANS turbulence model. The code accurately predicts the physics for simple problems but requires further testing for more complex flows that represent real-world engineering problems.

Integration of a High Enthalpy Source in the CSU Indraft Wind Tunnel

Katie Plese, Mechanical Engineering, Colorado State University
Ciprian Dumitrache, Mechanical Engineering, Colorado State University

The following contribution presents the design and ongoing developments of the Colorado State University Indraft Wind Tunnel, housed in the Aerospace Propulsion and Diagnostics Laboratory. The wind tunnel operates at Mach 2.4 for approximately 2.0 seconds. The test section has a width of 5.25" and variable heights ranging from 1.57" to 5.25." In response to the need for increased inlet temperatures to facilitate combustion experiments within the test section, research has been conducted to identify suitable high enthalpy source solutions. The primary focus of this presentation revolves around selecting a cost-effective high enthalpy source which will elevate the inlet temperature from ambient conditions to above 1000 K. The selected design is dual stage, where the first stage is an electrically heated plenum chamber that utilizes a high-pressure charge tank

to overcome adiabatic cooling due to rapid evacuation. The second stage is a vitiated-air burner, which operates at a high percentage of excess air to enable secondary combustion in the wind tunnel test section. The following presentation will outline the decision-making process in selecting the high enthalpy source for the CSU Indraft Wind Tunnel and discuss design considerations for future implementation.

Femtosecond Laser-Induced Continuous Optical Discharge in Air: A Numerical Investigation

Mozhdeh Hooshyar, Mechanical Engineering, Colorado State University
Ciprian Dumitrache, Mechanical Engineering, Colorado State University

This contribution represents an investigation on the use of a femtosecond-initiated continuous optical discharge (fs-COD) within the context of plasma-life extension. The proposed COD is generated using a femtosecond pulse to achieve initial gas preionization. This is followed by an energy addition continuous-wave (CW) laser that sustains the original femtosecond filament. To demonstrate the feasibility of such a discharge, we have developed a comprehensive 1-D axis-symmetric model which includes both plasma and chemical kinetics. The model is capable of simulating laser heating of the air mixture and modeling the joule heating of the free electrons. The proposed model self-integrated the fluid flow Navier-Stokes equations with the conservation of electron and vibrational energy in the plasma. The results showed that electron energy is effectively transferred to vibrational and translational modes, leading to heating and gas dissociation. The effect of the CW laser on preventing the complete decay of the fs-filament has been investigated. The proposed technique holds significant promise for a broad spectrum of applications that necessitate long-lived, highly energetic optical discharges, such as plasma-assisted combustion in aerospace engines.

Session 3B: Wind Energy and Aerodynamics
2:45 PM – 4:30 PM (AERO 114)

The effects of macro- & micro-scale morphology on drag and odor capture around honey bee antennae

Derek Goulet, CEAE, University of Colorado Boulder
John Crimaldi, CEAE, University of Colorado Boulder
Aaron True, CEAE, University of Colorado Boulder

Insects explore environments using chemoreceptors on their antennae. Antennae exhibit diverse morphologies at both the macro-scale (antennal lengths and diameters) and micro-scale (pore plates and hairs). Presently, we focus on understanding the effect of these morphologies on capture efficiency (ratio of odor reaching binding sites to supplied odor) and fluid drag forces for a model organism, the honey bee. We built numerical models of honey bee antennae in representative flows transporting a uniform odor field, resolving three-dimensional flow and odor fields around the antenna. We evaluated effects of Reynolds number (a function of wind speed and antennal radius) and pore packing density (ratio of pore plate radius to distance between pores) on capture efficiencies. Increasing Reynolds number increases viscous stresses (contributing to drag) and diminishes the relative strength of diffusive fluxes to the antennal surface (primary mechanism of capture efficiency). Increasing pore packing density does not change viscous stresses but allows more odor to reach binding sites (increasing capture efficiency) asymptotically, suggesting an optimal spatial configuration of odor binding sites on an antennal surface. Future efforts will explore unsteadiness in the flow and odor fields.

Exploring the chemical and mechanical modes of swimming of a bent rod actuator

Ritu Raj, Chemical and Biological Engineering, University of Colorado Boulder
Arkava Ganguly, Chemical and Biological Engineering, University of Colorado Boulder
Cora Becker, Chemical and Biological Engineering, University of Colorado Boulder
C. Wyatt Shields IV, Chemical and Biological Engineering, University of Colorado Boulder
Ankur Gupta, Chemical and Biological Engineering, University of Colorado Boulder

Microscale swimmers propel via two mechanisms, mechanical or chemical propulsion. To date, limited studies have looked at how these two modes of swimming interact to alter the motion of swimmers. In this work, we used slender body theory and the Lorentz reciprocal theorem to explore these modes of swimming for a bent rod actuator. The bent rod actuator propels chemically via self-diffusiophoresis and mechanically via hinge articulations.

In the mechanical mode, we recovered Purcell's scallop theorem. The trajectories we calculated were influenced by the amplitude of the articulation, the distribution of angular velocities between the two arms, and the geometric asymmetry of the bent rod. When allowed to swim chemically, we found that the bent rod moved in trajectories that were circular. The radius of curvature and speed of the trajectory were dependent on geometric asymmetry and variations in the flux profile. We then explored both modes of swimming in tandem. The two modes of swimming modified the effective forces and torques acting on the bent rod in a way that either impeded or aided propulsion. Additionally, hinge articulations with diffusiophoretic propulsion allowed the bent rod to experience net displace and enabled motion along s-shaped trajectories.

Modeling DRI Pellet Reduction on Laboratory Scale and Pilot Scale using Multiphysics Modeling

Amogh Meshram, National Renewable Energy Laboratory
Ronald J. OMalley, Missouri University of Science and Technology
Sridhar Seetharaman, SEMTE , Arizona State University

Reducing CO2 emissions from iron-making processes can significantly contribute towards the goals of the USA to fight climate change. Using H2 as a reducing agent instead of CO can drastically reduce the carbon footprints of steel production. This talk presents the modeling of the iron ore reduction process on laboratory scale and pilot scale using Multiphysics modeling. The pellet scale model was used for extraction of kinetic parameters that were used in the pilot scale model for obtaining the steady state flow profiles. The pilot scale model was used extensively in commissioning the pilot plant and to troubleshoot the plant operations.

Numerical Study of Thermocapillary Flows in Self-Rewetting Drop Impingement over Nonuniformly Heated Fluid Layers and in Enclosed Cavities Using the Lattice Boltzmann Method

Bashir Elbousefi, Mechanical Engineering, University of Colorado Denver
William Schupbach, Mechanical Engineering, University of Colorado Denver

Kannan Premnath, Mechanical Engineering, University of Colorado Denver

Self-rewetting fluids (SRFs), such as long-chain alcohol solutions, belong to a special class of liquids with surface tension that varies quadratically with temperature. These fluids display markedly different thermocapillary flows compared to normal fluids (NFs) and have been applied in various fields recently. We present two case studies to demonstrate the distinct interfacial transport phenomena in SRFs versus NFs. We employ a central moment-based lattice Boltzmann method (LBM) featuring three distribution functions: one for simulating two-fluid motion with high-density ratios and Marangoni stresses, another for capturing interfaces using the conservative Allen-Cahn equation, and a third for solving the energy equation. In the first case study, we perform axisymmetric simulations of a drop impacting on a SRF liquid layer subject to nonuniform heat fluxes modeled by a Gaussian profile, focusing on coalescence and pinch-off processes and their dependence on various parameters. The second case study investigates thermocapillary convection in SRF layers within a square cavity subjected to localized heat flux on one of its sides.

Simulations of Passive Bubble Separations Analogous to Capillary Nucleate Boiling in Microgravity

Ward Cereck, Mechanical Engineering, Montana State University
Joshua McCraney, Cornell University
Kjell Lindgren, NASA JSC
Mark Weislogel, IRPI LLC
Sarah Morris, Mechanical Engineering, Montana State University

Steady nucleate boiling is readily achieved on Earth as vapor bubbles are removed from heated surfaces by buoyancy forces due to gravity. Can steady nucleate boiling take place in microgravity? In 2022, NASA astronaut Kjell Lindgren injected air into the vertex of a partially-liquid-filled rhombic channel aboard the ISS. The wetting liquid is driven toward the vertex along the container's interior corners by capillary forces. The liquid flows break up the gas into asymmetric ullages that are driven passively away from the vertex. The bubbles 'rise' to near-inscribed locations where they coalesce, flowing into new inscribed locations and merging with the liquid free surface. Analogous to bubbles rising during the boiling process due to gravity, these bubbles 'rise' in microgravity due to the effects of surface tension, wetting, and conduit geometry. The maximum stable gas flow rate may be considered analogous to the maximum vapor production rate for steady nucleate boiling in microgravity. Our research aims to develop a numerical tool to predict such bubble migrations using the open-source CFD software OpenFOAM. These isothermal simulations are benchmarked against the rare experimental results catalogued by bubble volumes, gas flow rate, conduit geometry, and fluid properties.

An experimental and numerical investigation of collisions of wet particles

Souradeep Roychowdhury, Chemical and Biological Engineering, University of Colorado Boulder
Robert H. Davis, Chemical and Biological Engineering, University of Colorado Boulder
Griffin Hayrynen, Chemical and Biological Engineering, University of Colorado Boulder
Cole Thomas, Biomedical Engineering, University of Colorado Boulder

Collisions and flows of wet particles (solid particles coated with thin liquid layers) play crucial roles in natural and industrial processes. Wetted collisions differ significantly from dry ones and involve coupling of fluid dynamics with solid mechanics. While past studies have focused on normal and oblique collisions between two spheres and head-on collisions of three spheres, there have been no experimental investigations on oblique collisions of three wetted spheres. Here, we investigate the oblique collision dynamics of two and three wetted spheres, both experimentally and theoretically. We perform experiments using wetted spheres mounted on pucks on an air table and validate them numerically with the two-sphere model of Davis & Sitison (Phys. Rev. Fluids, 2020) and three-sphere model of Davis (Phys. Fluids, 2023). The collision process is primarily governed by the Stokes number (St), which measures the inertia of the colliding spheres relative to viscous forces in the thin fluid layer between them. For two spheres, low St leads to agglomeration, high St to rapid bounce, and intermediate St to a slow, stick-rotate-separate outcome. For three spheres, outcomes vary from full agglomeration at low St to full separation at high St, with partial agglomeration at intermediate St.

Dynamics of deformable droplets in a straight rectangular channel: An experimental and numerical study

Rajarshi Chattopadhyay, Chemical and Biological Engineering, University of Colorado Boulder
Aditya Vepa, Chemical and Biological Engineering, University of Colorado Boulder
Gesse Roure, Chemical and Biological Engineering, University of Colorado Boulder
Robert Davis, Chemical and Biological Engineering, University of Colorado Boulder

The motion and deformation of drops in channels have applications in targeted drug delivery, mixing in micro-reactors, and producing mono-disperse emulsions, requiring knowledge of the physics behind drop motion to construct efficient microfluidic systems. Here, we investigate liquid drops with sizes comparable to the channel height moving along a straight, rectangular channel. We use a boundary-integral method for simulating deformable drops in a pressure-driven flow at a low Reynolds number. Moreover, we experimentally investigate the motion of aqueous glycerol or PDMS drops in background flow of viscous castor oil. The drop experiences deformation because of viscous forces, but shape-preserving surface tension forces eventually cause it to reach a steady state. We focus on the steady-state velocity of the drop and its response to various parameters, including the capillary number (Ca), aspect ratio of the rectangular channel, drop size, and drop-to-bulk fluid viscosity ratio (λ). General trends are that the drop steady-state velocity decreases with increasing drop size and λ but increases with increasing Ca. For low-viscosity drops, however, the steady-state velocity increases with increasing size, attaining higher speeds than the maximum background flow velocity.