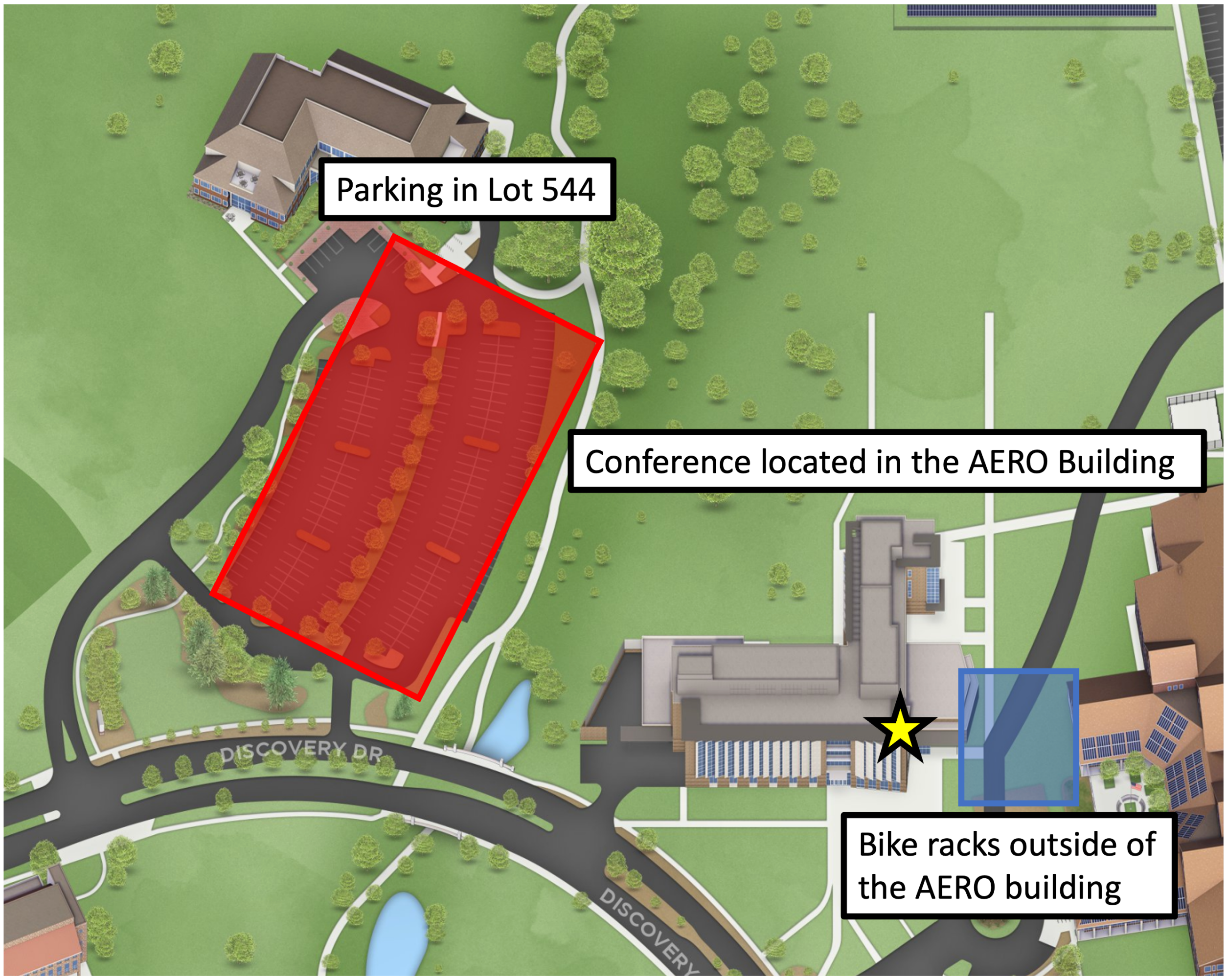
Rocky Mountain Fluid Mechanics Research Symposium 2025: Technical Program

August 8th, 2025

Keynote Presentation

Dr. Jacob Turner, (12:30 PM - 1:30 PM)
AERO 120
Unveiling Aeroacoustic Source Mechanisms with High-Fidelity CFD

Conference Location and Parking Information



The 2025 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 544 - 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: High-Speed Flows and Modeling I
9:00 AM – 10:00 AM (AERO 111)

9:00 AM Kalvin Monroe (University of Colorado Boulder)
Trajectory Optimization of Hypersonic Vehicles with Electron Transpiration Cooling Systems

9:15 AM Nick Rovito (University of Colorado Boulder)
FLATiron: A Hierarchical, Modular, Finite Element Library for Flow Physics and Transport Phenomena

9:30 AM Sarah Kinney (University of Colorado Boulder)
Exploring RANS Turbulence Modeling Deficiencies in Hypersonic Flows

9:45 AM Niyati Shah (Colorado State University)
High-Order Simulations of Supersonic Cylinder Wake Dynamics

Session 1B: Geophysical and Environmental Flows I
9:00 AM – 10:00 AM (AERO 114)

9:00 AM Laura Shannon (University of Colorado Boulder)
Influence of Ground Slope on Fire Spread in a Heading Crossflow

9:15 AM Kari Perry (Montana State University)
Flow separation affects local melt rates of ice cylinders in a cross-flow

9:30 AM Sandip Gautam (University of New Hampshire)
Free-Surface Response to Counter-Rotating Vortex Pairs

9:45 AM Ruby Gans (University of Colorado Boulder)
Characteristic Spacing in Granular Fingering Instabilities

Session 2A: Aerodynamic Flows
10:30 AM – 12:00 PM (AERO 111)

10:30 AM Mostafa Ojaghloo (University of Wyoming)
Experimental Investigation of the Swirling Wake

10:45 AM Jonathan Crider (University of Wyoming)
Comparing Hot-Wire Calibration Approaches in a Free Stream Jet to Support a Comprehensive Swirling-Jet Measurement Campaign

11:00 AM David Nelson (Montana State University)
Modification of Secondary Vortex Structures arising from Vortex-Ground Interactions by Axial Velocity

11:15 AM Cade Pugh (University of Wyoming)
Measuring Viscous Drag on Airfoil Geometry

11:30 AM Jaylon McGhee (University of Colorado Boulder)
Experimental Investigation of Surging Motion Effects on the Unsteady Load Response of an FFA-W3-301 Airfoil

11:45 AM Benjamin Savino (University of Mississippi)
Improving separation prediction by an IDDES turbulence model with a pressure-gradient sensor

Session 2B: Micro, Porous, and Multiphase Flows
10:30 AM – 12:00 PM (AERO 114)

10:30 AM Ward Cereck (Montana State University)
Passive Phase Separations in Microgravity: The ISS Capillary Channel Flow Experiments as a Benchmark for OpenFOAM Simulations

10:45 AM Garrett Mackey (Colorado School of Mines)
CFD simulations of unsteady flow through porous media

11:00 AM Brandon Hayes (University of Colorado Boulder)
Ultra-Fast Micro-Actuation using Thermal Bubble-Driven Micro-Pumps

11:15 AM Henry Lutz (University of Colorado Boulder)
Gravitational Capture of a Drop in a Channel Cavity with Stokes Flow

11:30 AM Madison Lytle (Colorado School of Mines)
A New Approach to Simulating Colloidal Suspensions Driven by AC Electric Fields

11:45 AM David Montgomery (National Renewable Energy Laboratory)
Development and application of high-fidelity models for heterogeneous CO2 frost formation

Session 3A: High-Speed Flows and Modeling II
2:00 PM – 3:00 PM (AERO 111)

2:00 PM Ahmet Kula (Colorado School of Mines)
Compressibility effects in viscous subsonic to supersonic flow past an adiabatic sphere at a Reynolds number of 100

2:15 PM Mitchell Wall (University of Colorado Boulder)
Hybrid Particle-Continuum Chemistry Modeling for Transitional Knudsen Number Flow

2:30 PM Tomoki Koike (National Renewable Energy Laboratory)
Streaming Operator Inference: Data-Driven Nonlinear Operator Learning for Large-Scale Dynamical Systems

2:45 PM Jacob McCallum (Colorado State University)
Numerical Analysis of Supersonic Shock Control Using NRP-Induced Plasmas

Session 3B: Geophysical and Environmental Flows II
2:00 PM – 3:00 PM (AERO 114)

2:00 PM Colin Beyers (Colorado School of Mines)
Where Do Things Go When the Stokes Drift is Patchy?

2:15 PM Mija Jovchevska (University of Colorado Boulder)
Biodegradable tracer particles for underwater particle image velocimetry

2:30 PM Samantha Preuss (Colorado State University)
Planning for the Hydrogen Highway: A Case Study of Enabling Hydrogen Refueling Infrastructure Along Colorado's I-25 Corridor

2:45 PM Tina Geller (University of Colorado Boulder)
Circulation in Arctic Coastal Lagoons: A Numerical Modeling Study with Lagrangian Particle Tracking

Session 4A: Biological Flows
3:30 PM – 5:00 PM (AERO 111)

3:30 PM Federico Municchi (National Renewable Energy Laboratory)
Modeling Bioreactors for Lunar and microgravity applications

3:45 PM Adiba Ashrafee (University of Colorado Boulder)
Benchtop Modeling Of Cerebrovascular Flow Networks and Cerebral Collateral Circulation Pathways

4:00 PM J. Scott Malloy (University of Colorado Boulder)
Non-linear dynamics of emboli transport in arterial flows with applications to Left Ventricular Assist Devices

4:15 PM Evan Williams (University of Colorado Boulder)
Vortex interactions in Tiny Insect Flight

4:30 PM Elle Stark (University of Colorado Boulder)
Synchronous stereo PIV and PLIF for quantifying low-Re, low-Sc, chaotic odor plumes

Session 4B: Chemical and Reacting Flows
3:30 PM – 5:00 PM PM (AERO 114)

3:30 PM Zachary Kinzler (Colorado State University)
Experimental Characterization of a Natural Gas Burner for Nanosecond Pulsed Discharge Integration

3:45 PM Liad Habot (National Renewable Energy Laboratory)
A Multi-Physics Simulation Tool to Predict Melting Times in Electric Arc Furnaces

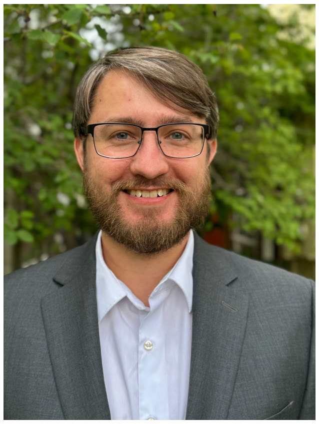
4:00 PM Aaron Phillips (Colorado State University)
Resolving Flow Abnormalities to Improve Stability in CO2 Flow Electrolyzer using Pulse Dampening: An experimental and numerical study

4:15 PM Jack Bicksler (Colorado State University)
An Experimental and Numerical Study on the Combustion of Ammonia/Hydrogen Fuel Mixtures in a Rapid Compression Machine

4:30 PM Mozhdeh Hooshyar (Colorado State University)
State-to-state Modeling of Femtosecond Filaments at Atmospheric Conditions in Air

4:45 PM Maryam Ahmadi (Colorado State University)
Understanding Potential Losses in Electrochemical Nitrate Reduction Reaction to Ammonia Using Multi-Phase Fluid Modeling

Keynote Presentation



Dr. Jacob Turner (12:30 PM - 1:30 PM)

Assistant Professor at Colorado State University
Group Leader, Computational Aeromechanics Group
https://www.engr.colostate.edu/me/people/dr-jacob-turner/

Unveiling Aeroacoustic Source Mechanisms with High-Fidelity CFD

Minimizing aerodynamic noise is critical for the certification and broad adoption of emerging transportation technologies. For example, unmanned aerial vehicles and advanced air mobility platforms offer exciting opportunities for economic growth but require a deeper understanding of noise generation to enable integration into urban environments. Similarly, the renewed interest in supersonic commercial flight demands the development of new technologies for quieter sonic booms. The successful development of noise mitigation strategies hinges on a deep understanding of flow-induced noise mechanisms, attainable through the application of high-fidelity numerical simulations. In this talk, I will present several recent applications of time-accurate numerical simulations to aeroacoustics problems. First, I will discuss the origins of leading-edge noise occurring in rotor systems, and the mechanisms for noise reduction through bio-inspired serrations. Next, I will examine the generation of secondary nonlinear noise sources arising in airfoil vortex interaction driven by near-wall dynamics. Finally, I will share recent insights into the significance of direct quadrupole sound from separated flows at subsonic speeds, relevant to both aerospace and automotive applications.

Speaker Biography:
Jacob Turner is an Assistant Professor in the Department of Mechanical Engineering at Colorado State University. He leads the Computational Aeromechanics Group, which focuses on developing and applying novel CFD approaches to simulate compressible fluid flows, unsteady aerodynamics, aeroacoustics, and aeroelasticity problems. His current projects leverage high-order immersed boundary methods to investigate fluid-structure interactions, supersonic bluff body flows, and noise generation in both aircraft and automotive applications. He obtained his PhD in 2019 from the Aerodynamics and Flight Mechanics Group at the University of Southampton, where he investigated airfoil-turbulence interaction noise and explored noise mitigation through bio-inspired leading-edge serrations. Following his PhD, he held postdoctoral positions at the Institute of Sound and Vibration Research and at Johns Hopkins University in the Flow Physics and Computation Lab.

Back to table of contents

Presentation Abstracts

Session 1A: High-Speed Flows and Modeling I
9:00 AM – 10:00 AM (AERO 111)

Trajectory Optimization of Hypersonic Vehicles with Electron Transpiration Cooling Systems

Kalvin Monroe, Aerospace Engineering, University of Colorado Boulder

Electron transpiration cooling (ETC) is a candidate thermal management strategy for hypersonic vehicle surfaces, utilizing thermionic emission to dissipate heat. However, an operational understanding of the regimes of hypersonic flight where ETC is applicable, as well as the power requirements ETC systems entail, is incomplete. To further this understanding, an model to estimate the composition of a hypersonic environment is constructed. This model is coupled to a one-dimensional ETC model, and thermionic cooling effectiveness is assessed across various hypersonic trajectory parameters. The role of material choices is investigated, and flight regimes where ETC is effective, as well as ineffective, are identified. Finally, ETC is incorporated into the trajectory optimization of a hypersonic vehicle. Integrated parameters, such as system energy requirements and heat loads, are computed for trajectories subject to constraints such as a maximum wall temperature. Results suggest that the cooling performance predicted for an ETC system can successfully reduce peak surface temperatures and improve the performance of a hypersonic vehicle for reasonable power requirements.

FLATiron: A Hierarchical, Modular, Finite Element Library for Flow Physics and Transport Phenomena

Nick Rovito, Mechanical Engineering, University of Colorado Boulder
Jessica Holmes, Mechanical Engineering, University of Colorado Boulder
Debanjan Mukherjee, Mechanical Engineering, University of Colorado Boulder

Computational flow and transport simulations are crucial in many research applications, with several high and low-level tools emerging to aid simulation development. Commercially available high-level tools are powerful and easy to use, but are rigid and do not allow for multiphysics coupling or customized problem definitions. Conversely, low-level symbolic tools offer greater flexibility at the cost of increased complexity. Developing coupled multiphysics models in these low-level packages requires custom weak forms that lead to long and complex scripts. Even minor changes to the problem definition, such as altering boundary conditions, can be difficult or non-intuitive to implement. We present a new mid-level wrapper for FEniCSx designed to overcome these limitations. Our library allows users to create staggered or monolithic coupled multiphysics models, customize boundary conditions, and generate custom computational endpoints, without sacrificing the flexibility native to FEniCSx. We simplify and expedite the workflow for building coupled physics simulations and offer built-in support for problems, including transport-reaction, Stokes, and Navier-Stokes. We demonstrate the efficacy of our approach with included scripts that model physics coupling in flow-reaction simulations.

Exploring RANS Turbulence Modeling Deficiencies in Hypersonic Flows

Sarah Kinney, Aerospace Engineering, University of Colorado Boulder
John Evans, Aerospace Engineering, University of Colorado Boulder

Iain Boyd, Aerospace Engineering, University of Colorado Boulder

Accurate modeling of turbulent boundary layers is critical for simulating hypersonic flows. Vehicle heating and drag are significantly affected by turbulence and flow separation can impact hypersonic engine performance. This presentation investigates the performance of two Reynolds Averaged Navier Stokes (RANS) turbulence models for simulating hypersonic internal flows. Specifically, two variations of the Menter Shear Stress Transport (SST) turbulence model—the SST-V and SST-V2003 models—are considered. These models are evaluated against data measured in an impinging shock generator experiment and using high-fidelity Direct Numerical Simulation (DNS) data for a curved wall flow case. The goal is to assess how well each model captures key flow features such as wall heating and separation under high Mach number conditions. Both models exhibit notable deficiencies when compared to benchmark data. These findings highlight current limitations in RANS-based turbulence modeling for hypersonic applications and motivate future efforts in model refinement and development.

High-Order Simulations of Supersonic Cylinder Wake Dynamics

Niyati Shah, Mechanical Engineering, Colorado State University
Jacob Turner, Mechanical Engineering, Colorado State University

Flow past a circular cylinder serves as a fundamental test case for understanding vorticity generation, flow separation, and drag characteristics around immersed bodies. Low-speed flow around a cylinder has been explored thoroughly; however, critical gaps persist in our understanding of cylinder flows at supersonic Mach numbers. Addressing these gaps is essential for applications such as re-entry vehicles, munitions, low-density flight (e.g., Mars), and high-speed particle flows. Previous studies have shown that compressibility can significantly alter wake topology by shifting the onset of vortex shedding, modifying pressure recovery, and influencing unsteady forces. However, the underlying mechanisms governing vortex shedding in the far wake and their dependence on flow parameters remain unclear. To address this, we investigate Mach 1.2–5 flow over a 2D circular cylinder at Re = 200,000 using high-order immersed boundary simulations, offering new insights into compressibility-driven wake dynamics and shedding behavior. We speculate that downstream vortex shedding originates from the propagation of instabilities through the separated shear layer, driven by vortices in the recirculation region that exhibit significant variations with Mach number.

Session 1B: Geophysical and Environmental Flows I
9:00 AM – 10:00 AM (AERO 114)

Influence of Ground Slope on Fire Spread in a Heading Crossflow

Laura Shannon, Mechanical Engineering, University of Colorado Boulder
Annamarie Guth, Mechanical Engineering, University of Colorado Boulder
Sean Coburn, Mechanical Engineering, University of Colorado Boulder
Greg Rieker, Mechanical Engineering, University of Colorado Boulder
John Farnsworth, Aerospace Engineering, University of Colorado Boulder

Wildfires are dangerous and complicated processes whose dynamics are governed by numerous environmental variables. Two of these environmental variables, wind and ground slope, impact both the flame structure and rate of spread. While previous experimental research has looked at the individual effects of these two variables on fire dynamics, few studies have considered the

concurrent effects of crossflow speed and ground slope. This study aims to investigate the effects of positive and negative ground slope with a concurrent heading crossflow on fire spread within a solid fuel array. The burns were carried out in a well-characterized, inclinable wind tunnel that provided ground slope angles of -10, 0, and +10 degrees and a crossflow speed of 1 m/s. Solid fuel arrays were created using 50 mm tall Douglas fir pegs. Two cameras were placed perpendicular to each other to capture side and top views of the flames moving through the arrays. Additionally, an array of thermocouples was embedded into the base of the solid fuel arrays to capture temperature time histories. Flame angle, flame spread rate, shape of the fire front, and downstream heating are examined.

Flow separation affects local melt rates of ice cylinders in a cross-flow

Kari Perry, Mechanical Engineering, Montana State University
Sarah Morris, Mechanical Engineering, Montana State University

Icebergs released from ice sheets account for half of the total freshwater discharged into the ocean annually. However, present-day iceberg melt models focus on global melt rates, and further study into local melting dynamics is needed. Previous studies show that separation of the meltwater plume from an ice body is associated with higher total melt rate as relatively warm water replenishes at the ice surface. Consequently, local melt rates vary with respect to the orientation of relative flow. This work investigates the effect of flow separation on local melt rates of ice cylinders in a cross-flow. Experiments are completed with cross-flow speeds ranging from U=0-4 cm/s. Flow structures are measured using flow visualization, and melt rates are found via cross-sectional area measurements. Results show that total melt rates increase with flow speed; however, local melt rates vary across the shape and deviate from current melt models. Melting is higher on the front face where ambient flow impinges, and lower along the rear face where the recirculating meltwater plume is present. This leads to distinct changes in ice curvature that are associated with boundary layer detachment from the ice surface. This material is based upon work supported by the NSF GRFP under Grant No. 2043105.

Free-Surface Response to Counter-Rotating Vortex Pairs

Sandip Gautam, Mechanical Engineering, University of New Hampshire
Tracy Mandel, Mechanical Engineering, University of New Hampshire
David Nelson, Mechanical Engineering, Montana State University
Sarah Morris, Mechanical Engineering, Montana State University

Free surface signatures offer a valuable means to predict the underlying sub-surface dynamics in the ocean. Momentum sources such as submarines generate counter-rotating vortex pairs that propagate downstream with self-induced velocity, creating a detectable surface footprint when interacting with the surface. Several qualitative experimental studies and numerical works have been conducted on studying the signature of a single vortex pair but lack spatially resolved, quantitative information at the free surface. In this study, we investigated the free surface signatures of a counter-rotating vortex pair, generated by two wing sections placed side by side with some tip separation distance in an open-channel recirculating flume. We combine the results from free surface elevation maps and the free surface thermal field, obtained using free surface synthetic schlieren and an infrared camera, to quantitatively study the signatures and relate them with the subsurface dynamics. We present results for varying wing submergence to explore primary vortex dynamics. The results of this study help in stealth navigation and sub-surface object detection in the ocean.

Characteristic Spacing in Granular Fingering Instabilities

Ruby Gans, Mechanical Engineering, University of Colorado Boulder
Nathalie Vriend, Mechanical Engineering, University of Colorado Boulder

Free-surface flows of mixed granular media over rough inclined surfaces are known to develop distinct lobate fingers at the flow front. Finger formation depends upon a distribution of both size and roughness in the granular mixture and is driven by granular segregation (Pouliquen et al., 1997). These instabilities are frequently observed in geophysical contexts, including pyroclastic flows and debris avalanches.

We investigate the granular fingering instability through laboratory experiments. We avalanche bidisperse mixtures using a novel tool, developed in-house, to trigger fingering at a fixed perturbation width and explore the dependence of finger geometry on the imposed spacing. We find that fingers tend to develop within a bounded range of widths, even when the trigger spacing lies outside this range. Resulting fingers become more uniform as the trigger spacing approaches a certain value, supporting the existence of a most unstable wavelength. We also showcase an untriggered experiment, in which the observed finger widths fall within the same bounded range observed in the triggered runs. Fourier analysis of these spontaneous finger geometries reveals a dominant frequency similar to the characteristic wavelength identified in the triggered experiments.

Session 2A: Aerodynamic Flows
10:30 AM – 12:00 PM (AERO 111)

Experimental Investigation of the Swirling Wake

Mostafa Ojaghloo, Mechanical Engineering, University of Wyoming
Jonathan Naughton, Mechanical Engineering, University of Wyoming

Understanding the effects of swirl on axisymmetric shear flows, such as jets and wakes, is important due to their extensive applications ranging from industrial mixing to wind turbine wakes in wind farms. The addition of swirl, even at low intensities, significantly alters turbulence characteristics and the downstream evolution of the flow. Previous studies on the impact of swirl on these flows, while limited, indicate that its presence increases turbulence intensity and enhances flow growth and wake recovery. In the present study, a controlled swirling wake is produced using a custom-designed wake generator, which produces the swirling wake by spinning a porous disk in a wind tunnel. Flow measurements are conducted using stereoscopic PIV at cross-planes downstream of the disk for varying swirl strengths. The analysis emphasizes first- and second-order statistics to gain a deeper understanding of how swirl affects the flow.

Comparing Hot-Wire Calibration Approaches in a Free Stream Jet to Support a Comprehensive Swirling-Jet Measurement Campaign

Jonathan Crider, Mechanical Engineering, University of Wyoming

It has been established that swirl accelerates jet growth, yet the mechanisms that drive this enhancement remain unresolved. The dynamics responsible for accelerated jet growth will be addressed through a measurement campaign that uses a single cross-wire anemometer to map axial velocities in a non-swirling and swirling jet. We will then deploy an array of eight single hot wires distributed across the radial axis of the jet to capture multi-point velocity records for a temporally rich baseline intended for structure identification. This baseline coupled with particle-image velocimetry, whose spatial rich fields will be merged with the hot-wire data via stochastic estimation, will provide time and spatially resolved velocity field estimations and expose swirl induced changes in turbulence structure. To build confidence in the hot wire measurements, two calibration techniques will be used. Both a temperature-compensated regression fit, and a look-up table that captures directional sensitivity across the full velocity and angle profile will be considered. Applying both calibrations to an identical data set and comparing the flow fields will reveal each method's advantages and drawbacks, allowing us to quantify and build a clear error analysis before further testing begins.

Modification of Secondary Vortex Structures arising from Vortex-Ground Interactions by Axial Velocity

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

Counter-rotating vortex pairs (CRVPs) are unavoidably produced by fixed-wing aircraft and present a hazard to trailing aircraft, especially in terminal phases of flight. Naval runways also contend with a dynamic oceanic surface featuring high-amplitude waves. Understanding how this environment interacts with CRVPs is crucial to improving the safety of aircraft carrier operations. This is explored experimentally via a delta wing towed through a quiescent water tank, generating a CRVP with circumferential and axial velocity and a non-zero descent angle. The CRVP impinges on a ground plane with static sinusoidal perturbations representative of the ocean surface. Particle Image Velocimetry (PIV) quantifies circulation decay while laser-induced dye fluorescence illuminates 3D topological changes and production of Secondary Vortex Structures (SVS) resulting from wall interaction. Experiments are also conducted with the ground plane angled as to isolate the effect of axial velocity. SVS are shown to respond strongly to axial velocity, while PIV analysis shows increased decay is correlated to increasing amplitude and spatial frequency of ground perturbations. This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-24-1-0211.

Measuring Viscous Drag on Airfoil Geometry

Cade Pugh, Mechanical Engineering, University of Wyoming

Drag reduction has been an emphasis of research for decades with many applications in racing and aeronautics. If the drag force on the surface of a vehicle is reduced, the vehicle becomes more efficient, conserving fuel, and increasing range. This would result in lower shipping costs, faster cruising speeds, and more efficient aircraft. However, effects from viscous drag have been notoriously difficult to measure, leading to a limited number of studies. Fortunately, techniques for measuring viscous drag involving thin oil films have been developed and utilized over the past 50 years. This research builds on the recent development of thin-oil-film techniques, namely oil-film interferometry (OFI), and applies these techniques to irregular geometries, such as an airfoil. The main problems with obtaining accurate OFI data on irregular geometries are orienting the camera with respect to the surface in 3D space and adequately illuminating the surface from a range of angles. These issues are addressed by the creation of a calibration block with reference points to ease setup and shorten the time required to take measurements and the design of a flexible light source capable of illuminating the entire leading-edge of an airfoil.

Experimental Investigation of Surging Motion Effects on the Unsteady Load Response of an FFA-W3-301 Airfoil

Jaylon McGhee, Aerospace Engineering, University of Colorado Boulder
John Farnsworth, Aerospace Engineering, University of Colorado Boulder

Many unsteady aerodynamic models account for pitching motions but neglect unsteady freestream effects or assume they can be captured through quasi-steady Reynolds number changes. This study quantifies the impact of surging motions on aerodynamic performance of an FFA-W3-301 airfoil relevant to wind energy. The airfoil was tested in the University of Colorado Boulder unsteady wind tunnel at a chord Reynolds number of 250,000 under two conditions: (1) static/stationary baseline and (2) surge-only motions. All cases were tested under 5% freestream turbulence with sinusoidal surge trajectories over reduced frequencies from 0.01 to 0.1 and normalized surge amplitudes from 0.05 to 0.15. Time-resolved surface pressure distributions were integrated to quantify unsteady aerodynamic lift, drag, and pitching moment coefficients. Statistical and modal analyses provided insight into surging motion contributions to load variance. Initial findings demonstrate that surging motions produce significant dynamic load responses that cannot be captured through quasi-steady modeling, highlighting the importance of accounting for unsteady freestream effects in aerodynamic load prediction.

Improving separation prediction by an IDDES turbulence model with a pressure-gradient sensor

Benjamin Savino, Mechanical Engineering, University of Mississippi
Kevin Griffin, National Renewable Energy Laboratory
Bumseok Lee, National Renewable Energy Laboratory
Ganesh Vijayakumar, National Renewable Energy Laboratory
Wen Wu, Mechanical Engineering, University of Mississippi
Michael Sprague, National Renewable Energy Laboratory

Hybrid RANS/LES turbulence models often fail to predict flow separation caused by adverse pressure gradients (APGs), as opposed to geometric singularities. This is due to the underlying RANS model's limitations in capturing separation onset near the wall. This work aims to improve separation prediction capabilities of the Improved Delayed Detached Eddy Simulation (IDDES) turbulence model (Gritskevich et al., 2012), which is based on the canonical k-ω SST RANS model. To achieve this, we employ a pressure gradient sensor recently developed by Griffin et al. (2025), which greatly improved separation prediction by the k-ω SST RANS model. A correction is applied to the eddy-viscosity calculation when the sensor detects the boundary layer to be separated. To ensure the well-tuned LES regions away from the wall are unaffected, we only modify the eddy-viscosity in the near-wall RANS portion by use the IDDES RANS indicator blending function. Model performance is evaluated on a NACA-0012 airfoil across angles of attack from 0 to 90 degrees, permitting assessment for attached flow, APG-induced separation near stall inception, and geometry-induced separation in deep stall. Results are compared with the conventional IDDES and k-ω SST RANS turbulence models.

Session 2B: Micro, Porous, and Multiphase Flows
10:30 AM – 12:00 PM (AERO 114)

Passive Phase Separations in Microgravity: The ISS Capillary Channel Flow Experiments as a Benchmark for OpenFOAM Simulations

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

On Earth, passive phase separations are easily achieved as gravity causes bubbles to rise and droplets to fall. In the microgravity environments aboard orbiting spacecraft, however, this simple process becomes complex. Multiphase systems (e.g., fuels, thermal control, life support) are instead governed by capillarity—the combined effects of surface tension, wetting, and geometry. The Capillary Channel Flow (CCF-EU2) experiments aboard the ISS demonstrate passive bubble separation using an open wedge channel. Bubbles introduced into the flow are asymmetrically confined, creating a capillary pressure gradient that drives them toward their spherical inscribed locations. If these locations lie above the free surface, separation occurs; if below, separation is unlikely. Some bubbles, however, overshoot their expected position, leading to unanticipated phase separations. This study investigates such "bubble overshoot" using CFD simulations to resolve local flow dynamics. NASA's PSI archive for the ISS CCF-EU2 Video Database (https://psi.nasa.gov) is employed as a benchmark for the OpenFOAM incompressible VOF CFD model. A convergence study assesses flow stability, bubble paths, and overshoot with rare direct comparisons to ISS data.
This work is supported by NASA Grant No. 80NSSC25K7768

CFD simulations of unsteady flow through porous media

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

Fluid flows through porous media are often modeled on a "macroscopic scale," using models such as Darcy's law. While this approach is relatively well understood for steady flows, fundamental questions remain about how to best model unsteady flows through porous media. To investigate this, we perform computational fluid dynamics simulations of incompressible fluid flow through periodic arrays of cylinders using a finite volume method with immersed boundaries. We consider flows that are driven by a pressure gradient that oscillates in time, and we systematically vary the frequency and amplitude of the pressure gradient. Using our CFD data, we volume-average the velocity field and explore the validity of both the quasi-steady Darcy Equation and a popular unsteady form of the Darcy equation.

Ultra-Fast Micro-Actuation using Thermal Bubble-Driven Micro-Pumps

Brandon Hayes, Mechanical Engineering, University of Colorado Boulder
Heiko Kabutz, Mechanical Engineering, University of Colorado Boulder
Kaushik Jayaram, Mechanical Engineering, University of Colorado Boulder
Robert MacCurdy, Mechanical Engineering, University of Colorado Boulder

Taking inspiration from the Mantis shrimp, we seek to develop a new class of micro-actuators for micro-robots that are both (a) ultra-fast and (b) high force. To date, these two criteria are difficult to achieve in micro-robotic actuators but have applications in both legged and winged micro-robots. In this work, we explore the use of a new class of micro-actuation technology, thermal bubble-driven micro-pumps, as an ultra-fast and high force micro-actuator. Thermal bubble-driven micro-pumps are essentially high-power thermal inkjet resistors. A current pulse heats the surface of the resistor to 300 °C in microseconds causing explosive boiling of an interfacial fluid layer which forms a high pressure (10's atm) vapor bubble. This vapor bubble is then harnessed to perform mechanical work. When the high-power resistor is actuated, the high-pressure vapor bubble causes the thin membrane to deflect. We characterize the system using high-speed imaging and computational fluid dynamics.

Gravitational Capture of a Drop in a Channel Cavity with Stokes Flow

Henry Lutz, Chemical and Biological Engineering, University of Colorado Boulder
Souradeep Roychowdhury, Chemical and Biological Engineering, University of Colorado Boulder
Robert Davis, Chemical and Biological Engineering, University of Colorado Boulder

Channel geometries and fluid/particle properties have been studied for use in microfluidic separation techniques. Neutral buoyancy is often assumed in these analyses. However, in this project, the balance of gravitational and viscous forces under Stokes flow conditions was investigated by measuring the critical Archimedes (Arc) number required to achieve droplet entrapment in a cavity. The system consisted of a well (cavity) at the bottom of a rectangular channel. Aqueous glycerol drops of varied glycerol concentration were studied in bulk castor oil. The decrease of castor oil's viscosity with increasing temperature was assessed to improve consistency of calculated nondimensional parameters. Arc was measured as a function of drop diameter, cavity geometry, and viscosity ratio. Arc was found to increase for larger drop diameters, indicating that they were more difficult to trap. The sensitivity to drop diameters was greater for viscosity ratios at or below 0.01 compared to viscosity ratios of 0.1 or 1. The cavity with a length to depth ratio of 1 had Arc values that were twice those of a cavity with a ratio of 2. Computational simulations to compare and investigate the impact of individual parameters under the same system parameters are ongoing.

A New Approach to Simulating Colloidal Suspensions Driven by AC Electric Fields

Madison Lytle, Applied Mathematics, Colorado School of Mines
Brennan Sprinkle, Applied Mathematics, Colorado School of Mines
Ning Wu, Chemical and Biological Engineering, Colorado School of Mines
Laura Alvarez, University of Bordeaux
Vivien Willems, University of Bordeaux

I will present a numerical method to simulate micron-scale particles subject to an AC electric field and driven by the resulting electrohydrodynamic flows. To solve for the AC-field driven dynamics of suspended particles, this approach couples the time-averaged electrostatics and surface velocities from the Debye layer using the Method of Fundamental Solutions paired with the rigid multiblob method. I will expand on a diverse set of experimental applications by not only reproducing particle dynamics, but also providing insight into the fundamental physics driving these systems. Specifically, I will discuss the electrohydrodynamics of spherical particle suspensions, the frequency-dependent motion of asymmetric dumbbell-like particles, and the dynamics of phase-separated lipid vesicles.

Development and application of high-fidelity models for heterogeneous CO2 frost formation

David Montgomery, Scalable Algorithms, Modeling and Simulation, National Renewable Energy Laboratory
Jeffrey Young, Carbon America
Sreejith Nadakkal Appukuttan, Scalable Algorithms, Modeling and Simulation, National Renewable Energy Laboratory
Bruce Perry, Scalable Algorithms, Modeling and Simulation, National Renewable Energy Laboratory

Carbon America has developed a cryogenic carbon capture technology ("FrostCC") that separates CO2 from point source emissions by solidifying it at cold temperatures through preferential desublimation. Cooling is achieved through a series of interlinked compression, heat exchange, and expansion operations. In the current system, frosting of CO2 happens in heat exchangers, followed by CO2 recovery in a separate extraction step. In this work, multiphysics computational fluid dynamics (CFD) models are developed and validated for compressible and low Mach flows to simulate the formation of solid CO2 in flue gas flowing in a heat exchanger geometry. The models track the mass transfer rate of CO2 from gas phase to solid phase, heat released from desublimation, and the evolution of the solid CO2 layer. Simulations are used to answer scientific questions related to the angle of heat exchanger pipes, where buoyancy effects from flow velocity and pipe orientation influence CO_2 frosting. Results show that upwardly angled pipes produce notably different flow structures compared to horizontal or vertical configurations, and that carbon capture efficiency correlates with buoyancy effects for pipe angles within ±23° of horizontal.

Session 3A: High-Speed Flows and Modeling II
2:00 PM – 3:00 PM (AERO 111)

Compressibility effects in viscous subsonic to supersonic flow past an adiabatic sphere at a Reynolds number of 100

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

Wall-resolved simulations are performed to investigate the aerodynamics and shock formation in compressible flow over an adiabatic sphere. Fully compressible Navier—Stokes equations are discretized conservatively using second-order accurate finite difference methods, hybridized with a fifth-order accurate shock capturing scheme applied only in regions with shocks. Boundary conditions on the sphere surface are imposed using a second-order accurate ghost-cell immersed boundary method. Compressibility effects are investigated with parametric variation of free-stream Mach numbers (Ma) from 0.70 (subsonic) to 1.40 (supersonic) at a Reynolds number of 100. Sphere aerodynamics are analyzed using aerodynamic coefficients as well as pressure distributions over the sphere surface. The results indicate that the total drag increases rapidly with Ma in the subsonic regime, primarily due to a rise in pressure drag. However, this increasing trend becomes less pronounced in the supersonic regime. Shock structures are examined through the local Mach number and pressure coefficient fields around the sphere. For supersonic cases, both shock stand-off distances and Mach angles are found to decrease sharply with increasing Ma.

Hybrid Particle-Continuum Chemistry Modeling for Transitional Knudsen Number Flow

Mitchell Wall, Aerospace Engineering, University of Colorado Boulder

Modeling flows in the transitional Knudsen number regime is a challenge for hypersonic entry vehicle design. At these altitudes, DSMC methods [Bird, 1994] can become computationally expensive and Navier-Stokes methods become inaccurate as continuum assumptions break down. Critical events such as peak heating can occur in this flight regime, motivating better methods to model transitional flows. This work seeks to use a code known as the Modular Particle-Continuum (MPC) method [Schwartzentruber et al. 2007], to model these transitional flows with continuum methods when accurate, and with particle methods when necessary.

One shortfall is that MPC cannot include chemical reactions in the flow. This work aims to develop and implement a consistent set of chemistry models for MPC. First, work will explore the flow regime to understand under what conditions chemistry becomes important for transitional flows. An approach is presented to mitigate the inconsistencies between CFD and DSMC chemistry models. This work will represent progress towards a significant increase in fidelity for this particle-continuum method.

Streaming Operator Inference: Data-Driven Nonlinear Operator Learning for Large-Scale Dynamical Systems

Tomoki Koike, Aerospace Engineering, National Renewable Energy Laboratory
Prakash Mohan, Computational Science , National Renewable Energy Laboratory
Marc Henry de Frahan, Computational Science, National Renewable Energy Laboratory
Julie Bessac, Computational Science, National Renewable Energy Laboratory
Elizabeth Qian, Aerospace Engineering, Georgia Institute of Technology

Modeling and simulation of real-world applications often involve dynamical systems with large degrees of freedom, requiring substantial computational time and resources. Model reduction addresses this challenge by constructing low-dimensional surrogate models that can be evaluated more efficiently. Operator Inference (OpInf), a data-driven projection-based model reduction method, forms a reduced subspace from simulation data and learns a reduced model by fitting operators to the projected data without requiring access to the underlying code. While effective across a broad class of problems, OpInf requires storing all data in memory and inverting large data matrices, creating prohibitive storage and memory requirements for extremely large datasets. We propose Streaming OpInf, an efficient method that learns reduced models from data streams. Our approach incrementally constructs a reduced basis in a single pass over the data, then recursively learns the reduced operators in a second pass. Our current results demonstrate that our streaming method achieves model accuracy comparable to standard OpInf while significantly reducing memory requirements.

Numerical Analysis of Supersonic Shock Control Using NRP-Induced Plasmas

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

This study investigates the use of plasma to modify the reflection angle of shocks in Mach 2.5 flow using numerical simulations. The motivation for this study is to assess the viability of using nanosecond repetitively pulsed (NRP) laser-induced plasmas as a dynamic shock control mechanism to prevent unstart in scramjet engines. The simulations were performed using APDL-CFD with a 10° triangular wedge in a rectangular channel to generate a shock train. Results show that NRP-induced plasma has minimal control on the shock operating at lower frequencies (¡100kHz) in Mach 2.5 flow. Unlike a continuous electrical discharge of plasma, the resulting change in reflection angle via NRP-induced plasma is insignificant. Increasing the frequency of pulses past what is currently possible does create significant results. At a 300kHz pulsing frequency with a pulse duration of 10ns, results were 33% as effective compared to continuous electrical discharge. Future work involves investigating other laser systems that could succeed in dynamic shock control and adding parallelization to the APDL-CFD software to run more expansive simulations in reasonable time.

Session 3B: Geophysical and Environmental Flows II
2:00 PM – 3:00 PM (AERO 114)

Where Do Things Go When the Stokes Drift is Patchy?

Colin Beyers, Geophysics, Colorado School of Mines
Bia Villas Bôas, Geophysics, Colorado School of Mines

Particles in ocean surface gravity waves follow an open orbital pattern, which results in a net transport in the direction of wave propagation called the Stokes drift. We aim to quantify the role that winds and currents have in driving the spatial variability of the Stokes drift. To do so, we used the wave model WAVEWATCHIII (WW3), which was forced with a combination of low-variability winds, high-variability winds, and currents. We analyzed the Stokes drift fields produced by these models using spectral analysis. We found that when WW3 was forced with winds, the spatial variability of the Stokes drift closely followed the spatial variability of the winds. When WW3 is forced with winds and currents, non-local spatial gradients in the sub-meso-to-mesoscale (1-300 km) range are introduced that can not be explained by the winds. These enhanced spatial gradients have implications for upper ocean processes such as transport and wave-induced mixing.

Biodegradable tracer particles for underwater particle image velocimetry

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

We present biodegradable particles as a sustainable and low-cost alternative to synthetic tracers for visualizing flow in biological and underwater systems using particle image velocimetry (PIV). These natural alternatives address key challenges posed by conventional tracers (e.g., silver-coated glass beads), including environmental impact, potential toxicity, and high financial cost. To evaluate their suitability for PIV, we characterized several biodegradable particle candidates, focusing on density, size distribution, and light-scattering properties. Both corn and arrowroot starch closely matched the densities of deionized and artificial seawater (35 ppt), which minimized buoyancy effects and emerged as the most promising candidates. These particles also exhibited uniform size distributions, with mean diameters of 10.9 (corn) and 8.7 µm (arrowroot). PIV experiments with a translating airfoil, brine shrimp, and jellyfish showed that starch particles reliably captured key flow structures, such as velocity and vorticity fields. Their nontoxic, digestible nature makes them ideal for biological and ecological studies. Thus, this work highlights starch as an eco-friendly

replacement for synthetic tracers, enabling sustainable flow visualization without compromising performance.

PLANNING FOR THE HYDROGEN HIGHWAY: A CASE STUDY OF ENABLING HYDROGEN REFUELING INFRASTRUCTURE ALONG COLORADO'S I-25 CORRIDOR

Samantha Preuss, Mechanical Engineering, Colorado State University
Bret Windom, Mechanical Engineering, Colorado State University

The decarbonization of heavy-duty transportation is critical to reducing greenhouse gas emissions and strengthening U.S. energy security. Fuel cell electric vehicles (FCEVs), powered by hydrogen, offer a promising solution due to their long range, fast refueling times, and zero tailpipe emissions. However, deployment of hydrogen infrastructure in the United States faces economic and logistical challenges. This analysis presents a comprehensive case study of enabling hydrogen refueling infrastructure along Colorado's Interstate 25 corridor—a high traffic north-south freight route experiencing rapid population growth and increased transportation demand. Three hydrogen infrastructure deployment scenarios are evaluated, emerging, developing, and mature, as the infrastructure rollout and projected demand increases. The emerging scenario reflects actively planned installations by Colorado State University and New Day Hydrogen, with the developing and mature scenarios describing future expansions to include regional passenger rail and airport logistics hubs. The findings plan to identify fuel and infrastructure cost to evaluate station economics, highlight areas where more incentives are needed and suggest strategic actions to align state incentives with infrastructure development.

Circulation in Arctic Coastal Lagoons: A Numerical Modeling Study with Lagrangian Particle Tracking

Tina Geller, Atmospheric and Oceanic Sciences, University of Colorado Boulder
Julia M. Moriarty, Atmospheric and Oceanic Sciences, University of Colorado Boulder
Irina Overeem, Geological Sciences, University of Colorado Boulder

Arctic coastal lagoons are ice-free in summer, and their circulation patterns affect carbon cycles and water quality. Using the Regional Ocean Modeling System (ROMS), we analyzed circulation in a chain of lagoons (the Kaktovik Lagoon chain) along the Alaskan Beaufort Sea coast from July to September of 2019, when sea ice was not present in the lagoons. These lagoons vary in river influence, shelf connectivity, and water depth. The model includes forcing from local rivers, winds, shelf circulation, tides, and bathymetry, and tracks neutrally buoyant tracers to analyze variability in residence time. Results suggest that the model represents the magnitude and variability of lagoon currents well. System currents oscillated with winds ($r=0.94$) and shelf currents ($r=0.74$), and to a lesser extent river discharge ($r=0.42$). Generally, enclosed lagoons had slower current speeds, more stable salinities, and greater residence times than the lagoons more connected to the shelf. Westward transport events removed the most riverine freshwater from the system compared to eastward and low transport events. These results highlight the importance of accounting for spatially variable residence time and circulation when considering local processes in coastal Arctic lagoons.

Session 4A: Biological Flows
3:30 PM – 5:00 PM (AERO 111)

Modeling Bioreactors for Lunar and microgravity applications

Federico Municchi, Computational Science, National Renewable Energy Laboratory
Hariswaran Sitaraman, Mechanical Engineering, National Renewable Energy Laboratory
Marc Day, Computational Science, National Renewable Energy Laboratory
Davinia Salvachua, Chemical and Biological Engineering, National Renewable Energy Laboratory

Microbes and enzymes are promising biocatalysts to produce fuels, chemicals, and biopolymers in zero and Lunar gravity environments. To produce usable quantities of these products, continuous stirred tank reactor (CSTR)-style vessels are commonly used to ensure efficient mixing and to control key parameters such as pH, temperature, and dissolved gas levels (e.g., oxygen, CO2). Understanding CSTR performance in microgravity will provide critical insights into the feasibility and limitations of conventional Earth-based bioreactor systems in low -gravity environments and will guide future bioreactor designs and predictive models for biomanufacturing created specifically for these conditions. In this work, we present a computational study on the effects of gravity on the mixing process and bacterial growth in continuous stirred tank bioreactors. More specifically, we investigate how key quantities such as the gas holdup, the mass transfer coefficient, and the uptake rate are modified under Lunar and zero gravity conditions. Distribution of gases dissolved in the liquid phase is also a key parameter to ensure efficient bioreactions. Our modeling approach is based on NREL open-source framework BiRD (Bioreactor Design), which leverages the OpenFOAM finite volume library.

Benchtop Modeling Of Cerebrovascular Flow Networks and Cerebral Collateral Circulation Pathways

Adiba Ashrafee, Biomedical Engineering, University of Colorado Boulder
Alena Tucker, Biomedical Engineering, University of Colorado Boulder
Manaswi Pulijala, Mechanical Engineering, University of Colorado Boulder
Debanjan Mukherjee, Mechanical Engineering, University of Colorado Boulder

The Circle of Willis (CoW) is a ring-like arterial network that supplies blood to the brain. Redundant vascular network connections from the CoW regulate cerebral blood flow during ischemia through collateral circulation. Yet, collateral flow pathways remain poorly understood. Presently, no animal models or imaging techniques can fully resolve collateral circulation. Imaging-based qualitative scores can assess collateral perfusion changes during ischemia, but they do not incorporate flow and network details. To address this gap, we have devised a benchtop experimental model for combining proximal and distal collateral pathways in the human brain. Morphometric features were extracted from a patient-specific CoW model to create a planar 3D-printed phantom. Pulsatile physiological inlet flow rate was delivered using a peristaltic pump. A 3D printed pellet was used to block phantom vessels and emulate a stroke. A full-factorial design was used to generate 16 collateral configurations by varying four distal collateral pathways connecting mid-anterior and mid-posterior vessels. Experimental flow distribution measurements illustrated distal collateral flow recovery, which were paired with numerical simulations to understand the proximal-distal collateral network coupling during stroke.

Non-linear dynamics of emboli transport in arterial flows with applications to Left Ventricular Assist Devices

J. Scott Malloy, Mechanical Engineering, University of Colorado Boulder
Debanjan Mukherjee, Mechanical Engineering, University of Colorado Boulder

Stroke remains a leading cause of mortality in patients implanted with Left Ventricular Assist Devices (LVADs). The altered hemodynamics from these devices can promote the formation and transport of emboli (suspended clot fragments). Patient-specific computational models are increasingly used to assess mechanisms of these events, but predictive accuracy is limited by the complex, nonlinear nature of vascular geometries and flow. One key factor underlying this challenge is the inherent nonlinear dynamics of embolus movement through real vascular networks. We computationally characterized how emboli travel relative to their initial release. Our findings reveal dispersive transport dynamics, where emboli rapidly lose correlation with their initial positions and velocities, akin to a random walk. This chaotic advection highlights the need to go beyond deterministic trajectory predictions, towards source-to-destination probabilistic mapping. This mapping captures the nonlinear dynamics of emboli transport, relying on large ensembles of trajectories to estimate outcome distributions, rather than deterministic predictions from individual particle paths. Characterization of this chaotic and nonlinear flow behavior may offer new insights into stroke etiology and adverse patient outcomes.

Vortex interactions in Tiny Insect Flight

Evan Williams, Civil and Environmental Engineering, University of Colorado Boulder
John Murray-Bruce, Computer Science, University of South Florida
David Murphy, Mechanical Engineering, University of South Florida

Tiny insects flying at chordwise Reynolds numbers less than 30 employ U-shaped wing-tip trajectories and the clap-and-fling mechanism to generate lift, yet the underlying aerodynamic mechanisms are not well understood. Here, we present 2D flow measurements around freely flying tobacco whiteflies (Bemisia tabaci) with body length of 0.85 mm flying at speeds of up to 400 body lengths s-1. We used an ultra-high speed brightfield micro-PIV system and 3D photogrammetry system to measure time resolved (10 kHz) flows and wing and body kinematics and analyze the circulation of the vortices formed throughout the wingstroke. During clap, a high-speed jet (500 mm/s) ejected from between the wings forms a viscous vortex ring that rapidly dissipates after t = 5 ms. We model this vortex as an impulsive Stokeslet. During the fling, the leading edge vortex from the previous half-stroke is preserved as it translates to the wing's trailing edge, which may be an energy-saving mechanism. Finally, wing-tip vortices generated during the forward sweep are intersected and weakened by the wings rowing through them on a subsequent stroke, possibly indicating an energy-saving wing-wake interaction. These observations provide new insight into the energetics and impulsive nature of tiny insect flight.

Synchronous stereo PIV and PLIF for quantifying low-Re, low-Sc, chaotic odor plumes

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

Olfactory navigation is a critical skill fundamental to the survival of many animals for tasks such as finding food, mates, and habitat. Emerging evidence shows that animals leverage information from both odor concentration and flow velocity fields when navigating. There is currently a need for datasets that include spatially distributed measurements of both odor and flow signals in naturalistic airborne plumes. The current work utilizes synchronous PLIF and stereo PIV to provide quantitative measurements of the velocity and concentration fields for several configurations of gaseous plumes over a 30 x 30 cm field of view, at a temporal resolution 10 Hz and spatial resolution up to 0.15 mm. Experiments were conducted in a recirculating wind tunnel with wind speeds of 10, 15, and 20 cm/s and a classical turbulence grid to induce chaotic plume behavior. To measure velocity, we use a 532 nm double-pulse Nd:YAG and two sCMOS cameras to resolve three components of velocity. To measure concentration, we use a 266 nm single-pulse Nd:YAG to excite acetone florescence, imaged by one sCMOS camera. The resulting datasets will be shared widely for use in olfaction studies and analyzed to quantify characteristics of mixing and dispersion in low-Re, low-Sc plumes.

Session 4B: Chemical and Reacting Flows
3:30 PM – 5:00 PM PM (AERO 114)

Experimental Characterization of a Natural Gas Burner for Nanosecond Pulsed Discharge Integration

Zachary Kinzler, Mechanical Engineering, Colorado State University

This study investigates the use of plasma-assisted combustion (PAC) to improve flame stability and emissions characteristics of a Honeywell SER 450 burner operated using natural gas. Baseline flame stability (without plasma) was studied in the open flame configuration across a wide range of equivalence ratios ($\phi = 0.27$–1.27) and thermal input powers (25–100 kBTU/hr). Flame structure and behavior were assessed using flame imaging, acoustics, and emissions. Results showed an unexpected behavior: the rich blowout were clearly noted at each power level. However, lean blowout was not observed at all. Instead, lean flames tend to transition to low-intensity pilot flame that is always present. For a fixed equivalence ratio, increasing the burner power resulted in a noticeable decrease in flame intensity, suggesting a reduction in combustion efficiency. These trends reflect the burner's staged design, where fuel is injected into a rich primary zone and entrains air through geometric features upstream of the igniter. Once integrated, the PAC burner performance (e.g., flame structure and behavior, emissions and combustion efficiency) will be compared with baseline data to quantify its effectiveness on enhancing flame stability in the lean combustion regime.

A Multi-Physics Simulation Tool to Predict Melting Times in Electric Arc Furnaces

Liad Habot, National Renewable Energy Laboratory

Steelmaking accounts for 8% of global energy use. Computer simulations are a powerful tool for making steelmaking processes more efficient and versatile. Depending on where the ore was sourced from and how it was processed, iron can have a range of thermal properties that affect its melting dynamics. For example, iron pellets processed with hydrogen (Hydrogen Direct Reduced Iron) have distinct properties from those processed using carbon materials (Direct Reduced Iron). This makes it difficult to efficiently use iron from different sources in an electric furnace.

To this end, we develop a multi-physics simulation tool in python that predicts the melting time of iron pellets in electric arc furnaces. The program considers a solid iron pellet surrounded by molten metal. Using a finite-difference solver, partial differential equations for heat transfer in the 1-dimensional spherical system are discretized and solved. We use variable thermodynamic properties and mesh size. The program updates the temperature distribution across the pellet over time, with the pellet reducing in size until fully melting. This program will be used in advanced computer simulations of steel manufacturing processes, accelerating the adoption of more efficient technologies in the industry.

Resolving Flow Abnormalities to Improve Stability in CO2 Flow Electrolyzer using Pulse Dampening: An experimental and numerical study

Aaron Phillips, Mechanical Engineering, Colorado State University

Electrochemical reduction of CO2 emissions from point sources into energy dense hydrocarbon products offers a unique carbon negative strategy for managing industrial emissions. Using flow cell reactors to study CO2 reduction (CO2RR) reactions at the lab scale allows for a better understanding of mass transport phenomenon relative to large scale reactor designs.

Currently, there is no standard configuration for CO2 flow reactor designs. Thus, a need to develop a reactor system with well-defined flow characteristics is necessary to produce reliable results for CO2RR electro-catalysts. The proposed design allows for industrially relevant electrode areas and offers uniform flow across the electrode area. Here, we develop and validate a laminar flow field that maximizes uniformity of fluid velocities across the electrochemically active surface area using computational fluid dynamics (CFD) and dye injection techniques. We demonstrate the necessity for pulse dampening in line with our flow cell to resolve flow abnormalities between computational and experimental flow. The outcome of this work results in the design of an electrochemical reactor with improved flow distribution, enhanced selectivity, activity, and durability to produce desired products from CO2.

An Experimental and Numerical Study on the Combustion of Ammonia/Hydrogen Fuel Mixtures in a Rapid Compression Machine

Jack Bicksler, Mechanical Engineering, Colorado State University

This research aims to extend the experimental and modeling database of ammonia/hydrogen fuel mixtures at engine-relevant conditions. While previous studies have established ignition and flame characteristics at near-atmospheric pressures, data at elevated pressures and temperatures is underrepresented, primarily for flames, limiting the validation of chemical kinetic mechanisms for advanced combustion applications. Preliminary research into ammonia/hydrogen mixtures has been conducted but has largely focused on low temperature and pressure conditions. To address this, a dual-piston Rapid Compression Machine was used to test 100% NH3 at 739K and 828K and 22bar. Complementing these experiments, 1-Dimensional simulations in Cantera were performed to evaluate the capabilities of existing chemical kinetic mechanisms. 3-Dimensional RANS combustion and turbulence simulations in CONVERGE studied flame front behavior and turbulent effects. Results show significant discrepancies between experimental measurements and model predictions, highlighting the need for further mechanism development. chamber This work establishes a foundation for future high pressure laminar flame speed measurements and further 3D modeling.

State-to-state Modeling of Femtosecond Filaments at Atmospheric Conditions in Air

Mozhdeh Hooshyar, Mechanical Engineering, Colorado State University

This contribution investigates state-to-state modeling of femtosecond (fs) filaments. Since the properties of fs filaments are still under active investigation, the main goal of this study is to develop a detailed state-to-state plasma kinetic model that can predict the main pathways for formation and destruction of electronically excited nitrogen species with special attention to N2(C) and N2+(B). As previous experimental studies using optical emission spectroscopy, in our group, have shown that the short-lived early emission is dominated by the N2(C-B) and N2+(B-X) transitions. Numerically, it was found that N2(C) is primarily formed via the dissociative recombination of the intermediate N4+ ion. In contrast, the N2+(B) comes mainly from the electron impact excitation reaction with N2(X), which is generated during the fs ionization process. Experimentally, the lifetime for N2(C-B) emission is observed only within the first 6 ns after the pulse, while N2+(B-X) is limited to the first 2 ns. The model's predictions of species' lifetimes show good agreement with experimental measurements. These results are important for improving the understanding of early-time emissions, with applications such as fs-LIBS, micromachining, directed energy systems, or plasma-assisted combustion.

Understanding Potential Losses in Electrochemical Nitrate Reduction Reaction to Ammonia Using Multi-Phase Fluid Modeling

Maryam Ahmadi, Mechanical Engineering, Colorado State University

Electrochemical nitrate reduction reaction (NO3-RR) to ammonia is a promising route to eliminate one of the major pollutants in surface and groundwater. When powered by renewable electricity, electrolysis provides a sustainable method to generate green ammonia from nitrate ions. Optimizing the physical and chemical properties of electrolysis cells is crucial for making this process economically viable for widespread implementation. We explore how the choice of current density, conductivity, pH, inter-electrode distance, membrane, catalyst, and buffer solution affect nitrate removal performance and efficiency. We develop a modeling framework to investigate the cell characteristics and fluid dynamics during electrochemical NO3-RR using both laminar and bubbly flows. To obtain more precise results, we employed the bubbly flow model (i.e., Multi-phase Fluid) to account for the impact of gas production near the electrode surface on liquid velocity, pH distribution, and, ultimately, potential losses. We exploit mass transfer theory to include the current density effect on migration and diffusion. This modeling platform helps pinpoint the optimal operating conditions of the system, ultimately resulting in the design of a fit-for-purpose electrolyzer for wastewater treatment.