Rocky Mountain Fluid Mechanics Research Symposium 2016 - Technical Program

Sustainability, Energy, and Environment Complex, University of Colorado, Boulder

August 9, 2016

Start	End		
7:15 AM	8:00 AM	*Breakfast, Check-In, and Welcome*	
8:00 AM	9:45 AM	Session 1A, Room A: Geophysical Fluid Mechanics	Session 1B, Room B: Computational Fluid Dynamics
9:45 AM	10:30 AM	*Coffee, Poster Session, and Break*	
10:30 AM	12:00 PM	Session 2A, Room A: Flow Control and Modeling	Session 2B, Room B: Industrial Applications
12:00 PM	12:30 PM	*Lunch Provided by Half-Fast Subs*	
12:30 PM	1:30 PM	Keynote Presentation by David R. Williams in Room A (During Lunch)	
1:30 PM	1:45 PM	*Break and transition*	
1:45 PM	3:15 PM	Session 3A, Room A: Numerical Methods	Session 3B, Room B: Experimental Methods
3:15 PM	3:45 PM	*Coffee, Poster Session, and Break*	
3:45 PM	5:15 PM	Session 4A, Room A: Energy Applications	Session 4B, Room B: Complex and Compressible Flows
6:00 PM	8:00 PM	*Dinner at Backcountry Pizza*	

Rocky Mountain Fluid Mechanics Research Symposium 2016: Abstracts for Oral and Poster Presentations

Sustainability, Energy and Environment Complex (SEEC)
University of Colorado, Boulder
August 9$^{\text{th}}$, 2016

Keynote Presentation

Dr. David R. Williams, (12:30 PM - 1:30 PM)
Some Trends and Lessons Learned in Active Flow Control

Oral Presentations

Session 1A: Geophysical Fluid Mechanics
8:00 AM - 9:45 AM (Room A)

8:00 AM Katherine M. Smith (University of Colorado, Boulder, CO)
Effects of Langmuir Turbulence on Upper Ocean Carbonate Chemistry

8:15 AM Gregor Robinson (University of Colorado, Boulder, CO)
Bayesian Hierarchical Models for Initiation and Propagation of the Madden-Julian Oscillation

8:30 AM Amrapalli Garanaik (Colorado State University, Fort Collins, CO)
Mixing Efficiency of Turbulent Patches in Stably Stratified Flows

8:45 AM Jian Zhou (Colorado State University, Fort Collins, CO)
Intrusive Gravity Currents Interacting with a Bottom-Mounted Obstacle in a Continuously Stratified Ambient

9:00 AM Benjamin A. Toms (Colorado State University, Fort Collins, CO)
Lower-Tropospheric Gravity Waves: A Case for Atmospherically Inclined Fluid Mechanics

9:15 AM Ying Pan (National Center for Atmospheric Research, Boulder, CO)
Matching Across Scales: Insights into the Local Integral Length Scale in Shear-Dominated Flows Near Solid Boundaries

9:30 AM Shelby A. Mason (University of Colorado, Boulder, CO)
Effects of Climate Oscillations on Burning Index Variability in the United States

Session 1B: Computational Fluid Dynamics
8:00 AM - 9:45 AM (Room B)

8:00 AM Ryan W. Skinner (University of Colorado, Boulder, CO)
Bi-Fidelity Modeling of Geometric Impact on NACA Airfoil Performance

8:15 AM Riccardo Balin (University of Colorado, Boulder, CO)
The Effects of Time Step on Reynolds Averaged Navier-Stokes Simulations of High-Lift Flows

8:30 AM Caelan Lapointe (University of Colorado, Boulder, CO)
Turbulent Couette Flow in OpenFOAM

8:45 AM Dane M. Nelson (University of Colorado, Colorado Springs, CO)
Investigation of Kestrel Flow Solver in Predicting Aerodynamic Properties of a Generic Lambda Wing Configuration

9:00 AM Paul Hasegawa (United States Air Force Academy, Colorado Springs, CO)
Effects of Angle of Attack and Side Slip for HIFiRE-6

9:15 AM Christopher Shelmidine (United States Air Force Academy, Colorado Springs, CO)
USM3D CFD Indicial Flows Over T-Tail Configurations

9:30 AM Mary K. Weidman (United States Air Force Academy, Colorado Springs, CO)
CFD to Replace Wind Tunnel Grid Data: Comparison of Aero Forces and Moments between Kestrel's Static and Dynamics Cases

Session 2A: Flow Control and Modeling
10:30 AM - 12:00 PM (Room A)

10:30 AM Dylan J. Brown (University of Colorado, Colorado Springs, CO)
Open-loop Flow Control on a NACA0018 Airfoil Undergoing Prescribed Sinusoidal Pitching

10:45 AM Keaton Turner (University of Colorado, Colorado Springs, CO)
Open-Loop Flow Control on a Torsionally Flexible Wing

11:00 AM Jacob Allen (University of Colorado, Colorado Springs, CO)
Forced Motion Design to Estimate Aerodynamic Coefficients for a Generic Missile Configuration

11:15 PM Joseph Straccia (University of Colorado, Boulder, CO)
Application of Biot-Savart Solver to Predict Axis Switching Phenomena in Finite-Span Vortices Expelled from a Synthetic Jet

11:30 AM Fabien Niel (United States Air Force Academy, Colorado Springs, CO)
Reduced Order Modeling of a Dynamically Pitching NACA 0018 Airfoil

11:45 AM Benjamin Kee (Colorado School of Mines, Golden, CO)
Flow and Pressure Characteristics in Rectangular Channels with Internal Cylindrical Bodies

Session 2B: Industrial Applications
10:30 AM - 12:00 PM (Room B)

10:30 AM Carly Blaes (Colorado School of Mines, Golden, CO)
Computational Fluid Dynamics in the Steel Manufacturing Industry: A Study of Oscillation Mark Formation in the Continuous Casting Process

10:45 AM Steven Isaacs (University of Colorado, Boulder, CO)
Experimental Development and Computational Optimization of Flat Heat Pipes for CubeSat Applications

11:00 AM Gandhali Kogekar (Colorado School of Mines, Golden, CO)
Coupling Complex Microchannel Heat Exchanger Simulations with Plug and Poiseuille Flow Models

11:15 AM Nicholas T. Wimer (University of Colorado, Boulder, CO)
Effects of Inlet Uniformity on Downstream Statistics for Forced Buoyant Plumes

11:30 AM Scott Jarriel (Colorado School of Mines, Golden, CO)
An Experimental Study on Foam Generation in Tunneling Industry

11:45 AM Nicholas S. Campbell (University of Colorado, Boulder, CO)
Nonequilibrium Aerothermodynamics for the In-Space Transportation Industry

Session 3A: Numerical Methods
1:45 PM - 3:15 PM (Room A)

1:45 PM Eric Peters (University of Colorado, Boulder, CO)
A Stabilized Finite Element Methodology for Compressible Flows on Moving Domains

2:00 PM Charles Morgenstern (Colorado School of Mines, Golden, CO)
An Efficient Non-Standard FEM and Iterative Method for Acoustic Wave Propagation

2:15 PM Ryan King (University of Colorado, Boulder, CO)
Turbulence Model Discovery with Data-Driven Learning and Optimization

2:30 PM Christopher Coley (University of Colorado, Boulder, CO)
Variational Multiscale Methods for Turbulence Modeling

2:45 PM Navid Shervani-Tabar (University of Colorado, Boulder, CO)
Stabilized Conservative Level Set Method with Adaptive Wavelet-based Mesh Refinement

3:00 PM Kenneth Pratt (University of Colorado, Boulder, CO)
Coalescence and Chaotic Stirring: The Role of Lagrangian Coherent Structures in the Mixing of Initially Distant Scalars

Session 3B: Experimental Methods
1:45 PM - 3:15 PM (Room B)

1:45 PM Naveen Penmetsa (University of Colorado, Boulder, CO)
Experimental and Computational Investigation of a Dual-Throat Thrust Vectoring Nozzle

2:00 PM Jason D. Christopher (University of Colorado, Boulder, CO)
Parameter Estimation for a Turbulent Buoyant Jet Using Approximate Bayesian Computation

2:15 PM Jean Hertzberg (University of Colorado, Boulder, CO)
Vorticity for the Assessment of Right Ventricular Diastolic Dysfunction Using 4D Flow CMR

2:30 PM Amanda S. Makowiecki (University of Colorado, Boulder, CO)
Double Wavelength Modulation Spectroscopy for Measurements in Fluctuating Pressure Environments

2:45 PM Torrey R.S. Hayden (University of Colorado, Boulder, CO)
Characterization of Catalytic Combustion in Film Processing Using Wavelength Modulation Spectroscopy

3:00 PM Roger Laurence (University of Colorado, Boulder, CO)
Nonlinear Regression for sUAS Distributed Pressure Wind Sensing

Session 4A: Energy Applications
3:45 PM - 5:00 PM (Room A)

3:45 PM Eliot Quon (National Renewable Energy Laboratory, Golden, CO)
Evaluation of Turbulence Metrics for Mesoscale-Microscale Coupling Methodologies

4:00 PM Nicola Bodini (University of Colorado, Boulder, CO)
Three-Dimensional Structure of Wind Turbine Wakes as Measured by Scanning Lidar

4:15 PM Jessica M. Tomaszewki (University of Colorado, Boulder, CO)
Do Wind Turbines Pose Roll Hazards to Light Aircraft?

4:30 PM Stephen Burke (Colorado State University, Fort Collins, CO)
Distillation-based Droplet Modeling of Non-ideal Oxygenated Gasoline Blends

4:45 PM Colin Gould (Colorado State University, Fort Collins, CO)
Autoignition of Liquid Hydrocarbon Droplets in Lean, High Pressure Natural Gas Mixtures Using a Rapid Compression Machine

Session 4B: Complex and Compressible Flows 3:45 PM - 5:15 PM (Room B)

3:45 PM Richard Martin (University of Colorado, Boulder, CO)
Numerical Calculations of the Rheology and Droplet Behavior of Emulsions in the Presence of Surfactants

4:00 PM Eduard Benet (University of Colorado, Boulder, CO)
The Effect of Particle Deformability on the Transport of Soft Colloids Through Porous Media

4:15 PM Sanli Movafaghi (University of Colorado, Boulder, CO)
Tunable Superomniphobic Surfaces for Sorting Droplets by Surface Tension

4:30 PM Ezio Iacocca (University of Colorado, Boulder, CO)
Thin Film Ferromagnets Acting Like a Compressible Fluid

4:45 PM Marc T. Henry de Frahan (University of Michigan, Ann Arbor, MI)
Blast Wave Induced Hydrodynamic Instabilities

5:00 PM Scott Wieland (University of Colorado, Boulder, CO)
The Vorticity Equation and the Compressible Rayleigh-Taylor Instability

Poster Presentations

(Session 1: 9:45 AM - 10:30 AM, Session 2: 3:15 PM - 3:45 PM)

1. Pengyu Cao (Colorado State University, Fort Collins, CO)
Cells Tracking for Optimization of Sparge Mixing of Synechocystis sp. PCC 6803: Understanding the Fluid Dynamics of Flat Photobioreactors for Cyanobacterium-Based Biofuel Production

2. Ben Gallman (Colorado School of Mines, Golden, CO)
Experimental Study of Unsteady Flow Through Porous Media

3. Andrew Kerr (Roccor, Longmont, CO)
Thermal Test Stand and Display for Flat Thin Heat Pipes

4. Kanghyeon Koo (University of Colorado, Boulder, CO)
Transport of Deformable Particle in Porous Media

5. Ali Moradi (Colorado School of Mines, Golden, CO)
Experimental and Numerical Study of Heat Transfer Through Unsaturated Soil from Pore Scale to Intermediate Scale with an Application to Soil Thermal Energy Storage (SBTES) Aystems

6. Robert Rhoads (University of Colorado, Colorado Springs, CO)
Optimized Fuel Composition and Operating Parameters for Ethanol-Gasoline Direct Injection Spark Ignition Engines

Keynote Presentation

David R. Williams[1] **(12:30 PM - 1:30 PM)**

[1] Mechanical and Aerospace Engineering, Illinois Institute of Technology, Chicago, IL

Some Trends and Lessons Learned in Active Flow Control

An overview of some trends and lessons learned from active flow control research will be presented. Prandtl's seminal work on boundary layer theory in 1904 laid the foundation for "Boundary Layer Control" by steady state actuation methods. Large (2-3X) gains in steady state lift coefficients were demonstrated in flight tests in the 1930s, but at a large actuator power cost. The observation of coherent structures in turbulent shear flows, and demonstrations in the 1980s that unsteady actuation could control the development of coherent structures motivated the "Modern Active Flow Control" approach. In the 1980s unsteady actuation methods, such as acoustic excitation or "synthetic jet" actuators were recognized to be significantly more efficient at controlling flow separation than steady state approaches. In the last 20 years, researchers explored ways in which unsteady actuation can be used to control forces resulting from dynamically varying flows, such as, maneuvering aircraft and wind turbine blades. The integration of control theory with the fluid mechanics of flow control was a requirement for efficient control of the dynamically varying flows. Not only was "dynamic" flow control made possible by control theory, but some limitations of achievable performance were identified, which will be discussed.

Speaker Biography:
David Williams received his Ph.D. from Princeton in 1982, and a postdoctoral position as an Alexander von Humboldt Research Fellow in Germany. He joined the faculty of Illinois Institute of Technology (IIT) in 1983. His research is in experimental fluid dynamics with an emphasis on closed-loop flow control. Since 1999 he has been the director of the Fluid Dynamics Research Center at IIT. Professor Williams is a Fellow of the American Physical Society, and an Associate Fellow of the AIAA.

Back to table of contents

Oral Presentations

Session 1A: Geophysical Fluid Mechanics
8:00 AM - 9:45 AM (Room A)

8:00 AM - Katherine M. Smith[1], Nikki Lovenduski[2], Baylor Fox-Kemper[3], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]Institute of Arctic and Alpine Research, Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO
[3]Department of Earth, Environmental, and Planetary Sciences, Brown University, Providence, RI

Effects of Langmuir Turbulence on Upper Ocean Carbonate Chemistry

Reactive tracers such as carbonate chemical species play important roles in the oceanic carbon cycle, allowing the ocean to hold 60 times more carbon than the atmosphere. However, the uncertainties in regional ocean sinks for anthropogenic CO_2 are still relatively high. Many carbonate species are non-conserved, flux across the air-sea interface, and react on time scales similar to those of turbulent processes in the ocean, such as small-scale Langmuir turbulence. All of this complexity can give rise to heterogeneous tracer distributions that are not fully understood and can greatly affect the rate at which CO_2 fluxes across the air-sea interface. In order to more accurately model the biogeochemistry of the ocean in earth system models (ESMs), a better understanding of the fundamental interactions between these reactive tracers and relevant turbulent processes is required. In this talk, we present results from large eddy simulations of carbonate chemical species in the presence of realistic mixed layer ocean turbulence. The simulations explore the effects of wave-driven Langmuir turbulence by solving the wave-averaged Boussinesq equations with an imposed Stokes drift velocity. By examining different surface forcing scenarios, we connect the coupled turbulence-reactive tracer dynamics with spatial, spectral, and statistical properties of the resulting tracer fields. These results along with implications for development of reduced order tracer models will be discussed.

Back to table of contents

8:15 AM - Gregor Robinson[1], Ian Grooms[1], Keith Julien[1], Ralph Milliff[2], Jeffrey Weiss[3], and Christopher Wikle[4]

[1]Department of Applied Mathematics, University of Colorado, Boulder, CO
[2]Cooperative Institute for Research in Environmental Sciences, Boulder, CO
[2]Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO
[4]Department of Statistics, University of Missouri, Columbia, MO

Bayesian Hierarchical Models for Initiation and Propagation of the Madden-Julian Oscillation

The Madden-Julian Oscillation (MJO) is a complex multiscale dynamical phenomenon primarily associated with moist convection over the tropical Indian and western Pacific oceans, but affecting weather and climate across the globe. Thorough and concerted research has elucidated the dynamical mechanisms associated with much of the lifetime of a single MJO event, but the mechanisms of initiation and propagation remain poorly understood. Some multiscale operational weather models accurately predict the propagation, but they are too complex to pinpoint simple causal phenomena. It remains unclear which, if any, of the many proposed mechanisms is correct. We are seeking understandable models for the dynamics underlying MJO initiation and propagation using the framework of Bayesian Hierarchical Models (BHM). We are designing a Markov Chain Monte Carlo (MCMC) sampler, where each sample is a simulation of crude phenomenological models, in order to estimate the relative weight of observational evidence in favor of each model. Such a high-dimensional and computationally expensive application BHM requires special MCMC techniques, new computational approaches, and delicate design of the Bayesian hierarchical model.

Back to table of contents

8:30 AM - Amrapalli Garanaik[1] and Subhas Karan Venayagamoorthy[1]

[1]Department of Civil and Environmental Engineering, Colorado State University, Fort Collins, CO

Mixing Efficiency of Turbulent Patches in Stably Stratified Flows

A quantitative description of turbulent mixing in stably stratified geophysical flow is challenging due to the complexity introduced into the flow by the density stratification. A key quantity that is essential for estimating the turbulent diapycnal (irreversible) mixing in stably stratified flow is the mixing efficiency, which is a measure of the amount of turbulent kinetic energy that is irreversibly converted into background potential energy. In particular, there is an ongoing debate in the oceanographic mixing community regarding the utility of the buoyancy Reynolds number Re_b, particularly with regard to how mixing efficiency and diapycnal diffusivity vary with this parameter. Specifically, is there a universal relationship between the intensity of turbulence and the strength of the stratification that supports an unambiguous description of mixing efficiency based on Re_b? The focus of the present study is to investigate the variability of mixing efficiency by considering oceanic turbulence data obtained from microstructure profiles in conjunction with data from laboratory experiments and direct numerical simulations. Field data analysis has done by identifying turbulent patches using Thorpe sorting method for potential density. The analysis clearly shows that high mixing efficiencies can persist at high buoyancy Reynolds numbers. This is contradiction to previous studies which predict that mixing efficiency should decrease universally for buoyancy Reynolds numbers greater than O(100).

Back to table of contents

8:45 AM - Jian Zhou[1] and Subhas Karan Venayagamoorthy[1]

[1]Department of Civil and Environmental Engineering, Colorado State University, Fort Collins, CO

Intrusive Gravity Currents Interacting with a Bottom-Mounted Obstacle in a Continuously Stratified Ambient

The flow dynamics of intrusive gravity currents past a surface-mounted obstacle was investigated using large eddy simulations. The propagation dynamics of a classical intrusive gravity current in the absence of an obstacle was first simulated to validate the numerical simulations. The numerical results showed good agreement with experimental measurements. An obstacle with a dimensionless height of $\tilde{D} = D/H$ (H the total fluid depth) was then introduced and acted as a controlling factor of the downstream flow pattern. It is found that for short obstacles, the intrusion re-established itself downstream in a form similar to the classical intrusion (in the absence of an obstacle). However, for tall obstacles, the downstream flow was found to be a joint effect of horizontal advection, overshoot-springback phenomenon, and the Kelvin-Helmholtz instability. Three regimes of downstream obstacle-affected propagation were identified depending on values of $\tilde{D}$, i.e. a retarding regime ($\tilde{D} \simeq 0 \sim 0.3$), an impounding regime ($\tilde{D} \simeq 0.3 \sim 0.6$), and a choking regime ($\tilde{D} \simeq 0.6 \sim 1.0$).

Back to table of contents

9:00 AM - Benjamin A. Toms[1], Jessica M. Tomaszewski[2], David D. Turner[3], and Steven S. Koch[3]

[1]Department of Atmospheric Science, Colorado State University, Fort Collins, CO

[2]Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO

[3]National Severe Storms Laboratory, Norman, OK

Lower-Tropospheric Gravity Waves: A Case for Atmospherically Inclined Fluid Mechanics

This case study offers insight into the influences of lower-tropospheric bores and solitons on the nocturnal boundary layer using observations and theory. Bores and solitons influence the evolution of moist convection, and serve as a critical component to the cross-scale energy cascade. Hydraulic theory was applied to a gravity wave-train using observationally derived pre- and post-wave boundary layer characteristics. Results of the theoretical analysis were applied to gravity wave characteristics derived from remote sensing observations to corroborate observationally derived conclusions. The observational data

suggest the bore propagated on a low-level inversion associated with the nocturnal stable surface layer, while the posterior soliton propagated within an increasingly elevated stably stratified layer induced and maintained by vertical motions of the preceding gravity wave(s). This unique waveform suggests continued collaborations between foundational fluid mechanists and atmospheric scientists are necessary to understand the scope of influences of lower-tropospheric gravity waves. Original insights are offered regarding gravity wave characteristics, with a primary suggestion of future studies on the sensitivity of bores to environmental variables.

Back to table of contents

9:15 AM - Ying Pan[1] and Marcelo Chamecki[2]

[1]National Center for Atmospheric Research, Boulder, CO

[2]Department of Atmospheric and Oceanic Sciences, University of California, Los Angeles, CA

Matching Across Scales: Insights into the Local Integral Length Scale in Shear-Dominated Flows Near Solid Boundaries

Turbulence near solid boundaries consists of motions at three distinct ranges of scales: (1) a universal range in which motions are uniquely determined by kinematic viscosity and the dissipation rate of turbulent kinetic energy (TKE), (2) a local production range in which motions are scaled by local integral length scale and (3) an inactive range in which motions are scaled by boundary-layer thickness. We use dimensional analysis and asymptotic matching to gain insights into the behavior of local integral length scale, which is not well determined for flows above canopies of large roughness elements or in non-neutral atmospheric surface layers. The matching between universal and local production ranges indicates that the local integral length scale is proportional to a length scale defined by friction velocity and TKE dissipation rate for neutral conditions, regardless of the presence of canopies. In shear-dominated atmospheric surface layer, the ratio between these two length scales only depends on the ratio between buoyancy production/destruction and the dissipation of TKE. The resulting scaling laws for turbulent motions in the overlap region of local production and inactive ranges are confirmed by large-eddy simulation results and field experimental data.

Back to table of contents

9:30 AM - Shelby A. Mason[1], Peter E. Hamlington[1], Benjamin D. Hamlington[2], William M. Jolly[3], and Chad M. Hoffman[4]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

[2]Department of Ocean, Earth and Atmospheric Sciences, Old Dominion University, Norfolk, VA

[3]Fire, Fuel, and Smoke Science Program, Rocky Mountain Research Station, Missoula, MT

[4]Department of Forest and Rangeland Stewardship, Colorado State University, Fort Collins, CO

Effects of Climate Oscillations on Burning Index Variability in the United States

It has been widely observed that climate has effects on wildland fires at many different spatial and temporal scales. This presentation will relate wildland fires to a dimensionless burning index (BI), which is calculated monthly across the continental United States from 1979 to 2014 using NOAA's North American Regional Reanalysis weather data set. Cyclo-stationary empirical orthogonal function (CSEOF) analysis is used to reconstruct and isolate the signals from the modulated annual cycle (MAC) and the El Nino Southern Oscillation (ENSO). Both the MAC and ENSO signals are identified, and the CSEOF analysis demonstrates the spatial and temporal variability in the mean burning index that accompanies these climate oscillations. The results presented here allow for burning index prediction and, consequently, wildland fire risk assessment and resource allocation depending on the current strength and stage of an oscillatory, long-term weather cycle.

Back to table of contents

Session 1B: Computational Fluid Dynamics 8:00 AM - 9:45 AM (Room B)

8:00 AM - Ryan W. Skinner[1], Alireza Doostan[1], and Kenneth E. Jansen[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Bi-Fidelity Modeling of Geometric Impact on NACA Airfoil Performance

Bi-fidelity modeling has the potential to substantially accelerate design optimization of unsteady aerodynamic systems. This bi-fidelity investigation explores the effect of variation in geometry and angle of attack on the performance of a 2-D NACA 4412 airfoil, with applications to geometric optimization. The flow is incompressible and is characterized by Re $\sim$1.5-3M. All simulations are steady and rely on the Spalart-Allmaras (SA) RANS turbulence closure. Low-fidelity simulations use a coarse mesh with unresolved boundary layers (BLs), whereas high-fidelity simulations employ a refined mesh that fully resolves the near-wall BL. Predictive capacity of the bi-fidelity approach is analyzed with pressure coefficient data.

Back to table of contents

8:15 AM - Riccardo Balin[1] and Kenneth E. Jansen[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

The Effects of Time Step on Reynolds Averaged Navier-Stokes Simulations of High-Lift Flows

The PHASTA flow solver was employed to perform RANS simulations of the DLR F11 model, representative of a commercial aircraft in landing configuration. The effects of time and spatial resolution of the numerical computation on the predicting capability of the solver were investigated for the geometry at two angles of attack; namely 7° and 21°. At 7° the model is in the linear section of the lift curve, however 21° is a post-stall stall condition. A time step study was conducted with RANS and unsteady RANS simulations, followed by a grid refinement study with both uniform and local, error-based refinement.
Analysis of the results obtained from the time step study showed that at 7° angle of attack, time step does not have a significant effect on the predicting capability of the solver. However, for the F11 geometry at 21° angle of attack, time accuracy for both the transient phase and the limit cycle is important in order to avoid significant modeling errors in the solution. Grid refinement was shown to be a valuable tool to improve flow prediction, however overprediction of separation regions cannot be corrected with increased spatial resolution. Furthermore, error-based, local adaptation resulted to be a more efficient approach compared to uniform refinement of the entire grid.

Back to table of contents

8:30 AM - Caelan Lapointe[1], Nicholas T. Wimer[1], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Turbulent Couette Flow in OpenFOAM

OpenFOAM (Field Operation and Manipulation) is open source computational software with a diverse set of solvers, including of pre and post-processing utilities. A robust soft- ware, OpenFOAM used to create CFD (computational fluid dynamics) models in academic and industrial settings. Before models can be trusted and deemed physically accurate, soft- ware must be validated against experimental (or otherwise acceptable) data. Shear driven fluid motion – Couette flow – was chosen as a test case due to its theoretical simplicity and availability of direct numerical simulation (DNS) data. A large eddy simulation (LES) for Reynolds number of 12,800 was created in OpenFOAM to match DNS results from Kawamura *et al.* (`http://murasun.me.noda.tus.ac.jp/turbulence/cou/text/Cou12800_A.dat`). Mean horizontal velocity profiles are shown to be in good agreement with DNS results. Non-dimensional mean velocity profiles are also shown to be in good agreement with the DNS as well as profiles from boundary layer theory.

Back to table of contents

8:45 AM - Dane M. Nelson[1], Mehdi Ghoreysh[2], and Andrew J. Lofthouse[2]

[1]University of Colorado, Colorado Springs, CO

[2]High Performance Computing Research Center, United States Air Force Academy, Colorado Springs, CO

Investigation of Kestrel Flow Solver in Predicting Aerodynamic Properties of a Generic Lambda Wing Configuration

The development of new aircraft heavily depends upon the initial research and testing stages of the design process. These stages consist of quantifying and evaluating the aerodynamic properties of the aircraft and developing flight dynamics models. The common method in obtaining the data required to produce these models is using the results from physical experiments. However, as the costs of production continue to be a major driving force in the development scheme, a more efficient alternative to physical testing is desired. This oral report investigates the effectiveness of Kestrel in replicating the aerodynamic data from wind tunnel experiments of a generic lambda wing UCAV with four control surfaces. In order to validate it's use in developing flight dynamics models. Kestrel demonstrated very good results from angles of attack of -5 to 10 degrees, and -10 to 10 degree control surface deflections using both a single and overset grid setup. This data seems promising in that the use of Kestrel may be the answer to replacing high-volume physical experiments in order to develop flight dynamics models. The next step in this investigation will be to use the Kestrel and experimental data in a stability and control program to compare the system behavior.

Back to table of contents

9:00 AM - Paul Hasegawa[1]

[1]United States Air Force Academy, Colorado Springs, CO

Effects of Angle of Attack and Side Slip for HIFiRE-6

This study was conducted under the DoD HPCMP Frontier Project on one of the possible configurations and several flight regimes of Hypersonic International Flight Research and Experimentation program (HIFiRE-6), using an Implicit Large Eddy Simulation (ILES). The external and internal geometry were simulated and investigated using the DoD High Performance Computing systems Lightning and Thunder. Three possible flight regimes were investigated which included insertion, cruise, and late cruise conditions. In the three flight regimes angle of attack and sideslip sweeps were conducted to determine the effects of off- design flight conditions, which could be present during maneuvers and unexpected adverse conditions. In total, thirteen cases were run to 900,000 iterations, which were examined to reveal any affects from the off design flight conditions. The primary effects investigated were analyzing turbulence and inlet behavior by visualizing pressure and temperature over the full geometry. At angles of attack zero and below the inlet displayed unsteady behavior that moves along the length of the inlet. While this behavior does not unstart the engine flow path, the preliminary results warrant additional investigation to determine if the observed unsteady state degrades engine performance. The unsteady phenomenon, as seen in Figure 1, decreases in speed as the angle of attack becomes more negative, so in order to resolve this behavior much longer simulations are required. Angle of attack also effects whether the upper surface has turbulent flow. Positive angles of attack have a primarily laminar upper surface but negative angles of attack cause earlier transition leading to turbulent flow over a larger area. Sideslip creates an imbalance over the geometry as a whole which effects flow on the outer surface as well as inner flow path. Sideslip creates asymmetric turbulent flow over the outer geometry of HIFiRE-6, and in certain cases shows turbulent flow on the windward wing but laminar flow on the opposing wing. The overall effects of this turbulence have yet to be fully investigated but the presence of turbulence could have significant implications that affect controls as well as required materials. Turbulence creates larger gradients on the surfaces and materials for this vehicle would have to withstand these forces. Inlet behavior is also altered by sideslip causing the position of the unsteady phenomena to shift toward the direction of the sideslip leading to an asymmetric flow cross section at the entrance of the internal flow path. Although flow entering the internal flow path is asymmetric, shock interactions within the flow path effectively eliminate the asymmetry leading to almost perfectly symmetric flow in the combustor. Due to the use of a relatively coarse grid, higher fidelity simulations of the internal flow path should be conducted in order to confirm this behavior. In conclusion this study shows that sideslip and angle of

attack have an impact on the flow over the full geometry of HIFiRE-6. The study also shows that additional investigation is required for some of the cases to be fully resolved, as these cases show significant alterations to the optimal aerodynamics of this geometry.

Back to table of contents

9:15 AM - Christopher Shelmidine[1] and Neal Frink[2]

[1]United States Air Force Academy, Colorado Springs, CO
[2]NASA Langley, Hampton, VA

USM3D CFD Indicial Flows Over T-tail Configurations

NASA currently uses the flow solver USM3D as well as FUN3D. For my projects we used USM3D to solve Indicial movements with specific bodies including a generic T-tail configuration and a missile that was used in a tunnel at NASA. These Indicial movements allowed us to isolate specific stability derivatives that would be used in the simulators. A bug was found to be in an older version of the USM3D code that many of the branch members were using. It gave bad readings during Indicial movements of and did not settle out to a steady state value. This meant that much of the time was spent making sure that the new code was debugged and would give us correct data for Indicial runs. The missile grid was used mainly to check the debugging process. The T-tail configuration was used as well to debug. Once the problem was solved it was run with Indicial runs to help increase the information about the stall region within simulators that to help prepare flight crews in commercial aircraft.

Back to table of contents

9:30 AM - Mary K. Weidman[1]

[1]United States Air Force Academy, Colorado Springs, CO

CFD to Replace Wind Tunnel Grid Data: Comparison of Aero Forces and Moments between Kestrel's Static and Dynamics Cases

Grid data has been traditionally measured via translation of a model aircraft within a wind tunnel. Using CFD models to collect grid data is more time efficient and cost effective. CFD modeled grid data can be collected using either static or dynamic cases. A static case performs iterations on a fixed store while a dynamic case performs iterations simultaneously while the store departs from the aircraft. In the past, only static solutions have been trusted. Yet, dynamic cases are more efficient than static cases when collecting large data sets. This project explores the feasibility of using dynamic cases to collect grid data using the flow solver, Kestrel. Various aero forces and moments were compared between static and dynamic runs to understand which combination of flow solver parameters were favorable. The configuration used in this experiment included a generic aircraft with a single store carried under a wing. Static cases were run at 0, 1, 3, 6, 9, 18, and 36 in from wing and dynamic cases were run at 1, 2, and 4 ft/s. The flow solving parameters that were varied included turbulence model, sweeps convergence criteria, time step, and Newton subiteration. Results showed potential in using dynamic cases to solve grid data. Lift and pitch forces matched very accurately independent of the flow solver parameters used, while drag, side, roll, and yaw moments showed otherwise. Changing the turbulence model, cutting time step in half, and increasing the Newton subiterations improved the accuracy of the dynamic cases overall.

Back to table of contents

Session 2A: Flow Control and Modeling
10:30 AM - 12:00 PM (Room A)

10:30 AM - Dylan J. Brown[1], Casey Fagley[2], Jürgen Seidel[2], and Thomas McLaughlin[2]

[1]University of Colorado, Colorado Springs, CO

[2]Department of Aeronautics, United States Air Force Academy, Colorado Springs, CO

Open-loop Flow Control on a NACA0018 Airfoil Undergoing Prescribed Sinusoidal Pitching

Three-dimensional simulations were performed on three different variations of a NACA- 0018 airfoil undergoing a prescribed sinusoidal pitching motion using HPCMP CREATE-AV Kestrel. These simulations were employed in order to analyze the effect of open- loop forcing via mass injection on the flowfield around the airfoil. Dynamically pitching oscillations are of interest to understand the unsteady aerodynamic behavior including dynamic stall, lift hysteresis, and vortex shedding. These unsteady processes are important for the formulation of aeroelastic instabilities. The sinusoidal motion was parameterized by the following equation, $\alpha(t) = \alpha_0 + \alpha_1 \sin(\omega t)$. Two base angles of attack (α_0 = 5,10), three sinusoidal magnitudes (α_1 = 5,10,15), and three different reduced frequencies ($k = \omega c/U_\infty$ = 0.1,0.15,0.2). A total of 18 simulations were performed for each campaign. The three campaigns included an airfoil with no actuator slot, an airfoil with an open slot, and an airfoil with an open slot with mass addition to the flowfield. A number of mass flow rates were simulated to asses the influence on the dynamic stall behavior. Differences in unsteady aerodynamics are shown with lift, drag, and pitching moment data sets for all three campaigns.

Back to table of contents

10:45 AM - Keaton Turner[1], Casey Fagley[2], Jürgen Seidel[2], Thomas McLaughlin[2]

[1]University of Colorado, Colorado Springs, CO

[2]Department of Aeronautics, United States Air Force Academy, Colorado Springs, CO

Open-Loop Flow Control on a Torsionally Flexible Wing

Fully coupled CFD-CSD computations were run on a torsionally flexible NACA0018 wing with finite aspect ratio at prescribed angles of tip twist using HPCMP CREATE-AV Kestrel^{TM}. The primary goal of these simulations was to characterize two and three dimensional fluidic instabilities by prescribing angles of tip twist and control these unstable effects with the addition of flow control. Two unforced cases were run at a static angle of attack for the prescribed angles of tip twistone with an actuator slot at 75% span and one without. The actuator slot was used to induce mass flow in a third case to view the effects of open-loop forcing. In comparison to rigid simulations, deformation of the unforced wings induced flow separation patterns at lower angles of attack due to increased local angles of attack along the span. The resulting flow field showed a pocket of separated flow bounded by the wing tip vortex on the outboard location and attached flow on the inboard location. The presence of the slot on the unforced open slot wing caused localized separation at the slot for angles of tip twist lower than those of the non-slot geometry. Addition of mass flow through the actuated slot, then, further influenced the formation of instabilities. Ultimately, this research shows the ability to augment unsteady, three dimensional flow features due to spanwise twist deformations through open-loop actuation.

Back to table of contents

11:00 AM - Jacob Allen[1], Mehdi Ghoreysh[2], and Andrew J. Lofthouse[2]

[1]University of Colorado, Colorado Springs, CO

[2]High Performance Computing Research Center, United States Air Force Academy, Colorado Springs, CO

Forced Motion Design to Estimate Aerodynamic Coefficients for a Generic Missile Configuration

Current techniques used to estimate aerodynamic coefficients using computational fluid dynamics (CFD)

simulations are computationally expensive. Many current CFD solvers incorporate the ability to superimpose six-degrees-of-freedom rigid body mesh motions during transient simulations, also known as forced motion or prescribed motion, which offers a potential solution. This research explored forced motion designs to reduce the computational expense of estimating aerodynamic coefficients. The geometry used for this research was the Army Navy Basic Finner, which consists of a 20-degree, 2.84 caliber long nose cone, a one caliber diameter cylindrical body, and four, 1 caliber square fins. Experimental free flight test results were compared against CFD results. The forced motions studied were mesh translations that swept mach ranges from 0.5 to 4.5, as well as simultaneous multisine translations and rotations. Additionally, reduced order models (ROM) were created using linear regression from forced motions. Test motions were computed in CFD and then compared against the ROMs. The forced motions reproduced aerodynamic coefficients with good accuracy and with less computational cost. Also, the ROMs reproduced the coefficients of the test motions with reasonable accuracy. This submission is for an oral presentation.

Back to table of contents

11:15 AM - Joseph Straccia[1] and John A. Farnsworth[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Application of Biot-Savart Solver to Predict Axis Switching Phenomena in Finite-Span Vortices Expelled from a Synthetic Jet

The Biot-Savart Law is a simple yet powerful inviscid and incompressible relationship between the velocity induced at a point and the circulation, orientation and distance of separation of a vortex line. The authors have developed an algorithm for obtaining numerical solutions of the Biot-Savart relationship to predict the self-induced velocity on a vortex line of arbitrary shape. In this work the Biot-Savart solver was used to predict the self-induced propagation of non-circular, finite-span vortex rings expelled from synthetic jets with rectangular orifices of varying aspect ratios. The solver's prediction of the time varying shape of the vortex ring and frequency of axis switching was then compared with Particle Image Velocimetry (PIV) data from a synthetic jet expelled into a quiescent flow i.e. zero cross flow condition. Conclusions about the effectiveness and limitations of this simple, inviscid relationship are drawn from this experimental data.

Back to table of contents

11:30 AM - Fabien Niel[1], Casey Fagley[2], Jürgen Seidel[2], Thomas McLaughlin[2]

[1]CNRS, Toulouse, France

[2]Department of Aeronautics, United States Air Force Academy, Colorado Springs, CO

Reduced Order Modeling of a Dynamically Pitching NACA 0018 Airfoil

This study takes place in a PhD work concerning the Active flow control of a wing using a blowing actuator. Nevertheless, before being able to develop some advanced control strategies for such a system, it is necessary to obtain an accurate dynamic model of the aerodynamic forces and moments, capturing the nonlinear behavior at high angles of attack. This presentation aims at introducing a modeling of a dynamically pitching airfoil. This model extends up the model proposed by David R. Williams based on Goman-Khrabrov work, by using the ONERA BH model to include the oscillatory behavior encountered during its movement, especially during the downstroke. Indeed, this extended model benefits from the efficiency of William's model to predict the hysteresis of the pitching moment while reproducing the unsteadiness which may develop. Dynamic pitching experiments and Computational Fluid Dynamics (CFD) simulations about mid chord are performed for different reduced frequencies and amplitudes on a 2-D NACA 0018 Airfoil at low speed ($M = 0.068$) with a Reynolds number $Re_c = 190,000$. These data are used to train the developed model and identify the different coefficients. The results from model calculations are compared to experimental data and CFD simulations and show some satisfying agreements.

Back to table of contents

11:45 AM - Benjamin Kee[1], Hendrik Gossler[2], Huayang Zhu[1], Matthias Hettel[2], Olaf Deutschmann[2], and Robert Kee[1]

[1]Department of Mechanical Engineering, Colorado School of Mines, Golden, CO

[2]Karlsruhe Institute of Technology, Karlsruhe, Germany

Flow and Pressure Characteristics in Rectangular Channels with Internal Cylindrical Bodies

This presentation develops models for steady-state, fully developed, laminar flow in rectangular channels with internal coaxial solid cylinders. By casting and solving the parallel-flow momentum equation in a dimensionless setting, correlations are derived for the friction factor f and represented as Ref where Re is the Reynolds number. The correlations consider three positions for the internal cylinder. The cylinder may be in the center of the rectangular channel, in the corner of the channel, or resting at the middle of the channel floor. The correlations incorporate channel aspect ratios in the range $0.1 \leq \alpha \leq 1.0$. The cylinder-diameter aspect ratios β range from being vanishingly small to being large enough to touch the channel walls. Although the results are general, the study is motivated by considering the effects of diagnostic probes within small channels of catalytic monoliths.

Back to table of contents

Session 2B: Industrial Applications
10:30 AM - 12:00 PM (Room B)

10:30 AM - Carly Blaes[1], Brian Thomas, and Xiaolu Yan

[1]Department of Mechanical Engineering, Colorado School of Mines, Golden, CO

Computational Fluid Dynamics in the Steel Manufacturing Industry: A Study of Oscillation Mark Formation in the Continuous Casting Process

The steel industry is one of the world's largest energy consumers. However, in order to prevent excessive energy use a great amount of research is performed in the steel industry that focuses on limiting defects and increasing yields in steel making processes. The continuous casting process is the most commonly used process in the world to make steel bars and plates. In the continuous casting process a series of steps are performed in order to supply molten steel to a water-cooled copper mold. As the steel comes near the water-cooled mold a solidified shell is formed. This mold oscillates in order to prevent the solidifying steel shell from sticking to the mold walls. Additionally, to prevent the shell from sticking, mold powder is supplied to the top of the molten steel, which sinters, melts, and then flows between the mold and the steel shell, which provides lubrication. The steel shell is then pulled out of the mold by a series of rollers.

The combination of the oscillation of the mold and the contact between the slag and steel shell can cause a defect called an oscillation mark. Oscillation marks are depressions in the steel shell that can lead to cracking and can decrease the final yield of the batch because to create a smooth surface on the steel some material must be removed from the final product. Therefore, it is important to know what causes the oscillation marks and how they can be minimized. In this research project a fluid flow simulation has been created using the commercial package FLUENT to study oscillation mark formation.

Back to table of contents

10:45 AM - Steven Isaacs[1,2], Greg Shoukas[1], Diego Arias[1], and Peter E. Hamlington[2]

[1]Roccor, LLC., Longmont, CO

[2]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Experimental Development and Computational Optimization of Flat Heat Pipes for CubeSat Applications

CubeSats are small, modular satellites that provide a cost-effective means of conducting a wide range of orbital missions. Due to the dense, compact form factors involved in CubeSat design, however, heat dissipation and thermal management of on-board electronics can be a challenge. Moreover, the dynamic and low pressure orbital environments in which CubeSats operate introduce additional design considerations for the thermal management system.

In order to address this challenge, a flat, conformable heat pipe has been developed. Using an all- copper construction, the total thickness of current prototypes is less than 1 mm. Preliminary testing has resulted in effective thermal conductivity measurements greater than 1,000 W/mK using acetone as a working fluid (giving roughly 5x improvement over pure copper). Additionally, current prototypes have been able to withstand internal vapor pressures as high as 135 psig. A closed-form mathematical model has been developed to predict heat pipe performance. This model was coupled with a gradient-free optimization algorithm to demonstrate how an optimization method can be used to aid in the design of flat heat pipes.

Back to table of contents

11:00 AM - Gandhali Kogekar[1], Richard Nguyen[1,2], and Robert Kee[1]

[1]Department of Mechanical Engineering, Colorado School of Mines, Golden, CO

[2]Seagate, Longmont, CO

Coupling Complex Microchannel Heat Exchanger Simulations with Plug and Poiseuille Flow Models

This presentation demonstrates incorporation of Plug and Poiseuille models into three- dimensional models of microchannel heat exchangers. The full three-dimensional simulation of microchannel heat exchangers can be computationally expensive because of the large number of mesh cells required to accurately resolve flow fields within the microchannels. However, because the flow inside individual channels is (to a very good approximation) fully developed, the velocity profiles can be represented by low-dimensional and exact solutions. An algorithm has been developed to couple these relatively simple low-dimensional models onto large-scale CFD for the heat exchanger as a whole. The intent is to substantially reduce computational effort, but with little loss of accuracy. The approach has been implemented and demonstrated in the ANSYS Fluent software.

Back to table of contents

11:15 AM - Nicholas T. Wimer[1], Caelan Lapointe[1], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Effects of Inlet Uniformity on Downstream Statistics for Forced Buoyant Plumes

Velocity initial conditions of turbulent, forced, buoyant plumes are examined in an effort to understand how velocity and buoyant forcing affect the evolution of the conserved fields. The simulations of the forced, buoyant plumes are conducted in OpenFOAM using Large Eddy Simulation (LES) with a variety of initial velocity distributions: uniform, pseudo-random, and Gaussian with different characteristics. The resulting turbulent statistics of velocity and temperature are then compared to the initial distributions with focus on spatial and temporal evolution. Results are presented with an effort to glen insight into the mechanisms by which the different fields evolve, with an emphasis on turbulence.

Back to table of contents

11:30 AM - Scott Jarriel[1], Dhrupad Parikh[1], Yuanli Wu[1], Mike Mooney[1], and Nils Tilton[1]

[1]Department of Mechanical Engineering, Colorado School of Mines, Golden, CO

An Experimental Study on Foam Generation in Tunneling Industry

Underground tunneling involves the treatment and transportation of large amounts of soil using various conditioning agents. Foam, the most common conditioning agent, is injected into the soil at the cutter head of the tunnel boring machine. The foam transforms the excavated material into a deformable soil paste that provides a homogenous pressure to the surrounding ground, improves stability of the tunnel face, and minimizes friction and wear on metallic parts. Extensive studies have been done on the properties of conditioned soil; however, little literature exists on the mechanical system that generates the foam. In the tunneling industry, foam is generated by flowing a mixture of air, water, and surfactant through a porous medium such as packed beads. We present an experimental study of how operating parameters, such as pressures, flow rates and the type of porous material, influence foam bubble size distribution, stability and compressibility. Our long term objective is to eventually tailor foam properties to varying soil conditions and allow for more efficient and effective underground tunneling.

Back to table of contents

11:45 AM - Nicholas S. Campbell[1] and Brian M. Argrow[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Nonequilibrium Aerothermodynamics for the In-Space Transportation Industry

Aerobraking refers generally to the use of a planet's atmosphere to decelerate a body traveling at orbital velocities. While this concept has been around for a long time, recent developments in the commercial space industry have motivated a new application for the technique. In a flourishing Cis-Lunar economy, it has recently been estimated that the trade-route between Lunar and Earth orbits could constitute a multi-billion US dollar a year market, based on propellant cost. Saving propellant on this route means the transportation sector can offer more per kg of propellant, making the resource mining sector more fiscally attractive. Regularly performing hybrid propulsive-aerobraking maneuvers to decelerate into a low Earth orbit is a fundamentally new implementation of aerobraking. In my talk, this motivation will be reviewed and the associated gas-dynamic complexities that arise will be covered. These complexities require high-altitude, hypersonic flight to be modeled through simulating the Boltzmann Equation. SPARTA (Stochastic PArallel Rarefied-gas Time-accurate Analyzer), an open source Direct Simulation Monte Carlo code developed out of Sandia National Laboratory, is used to address this need. Verification of particle and chemical kinetics will be presented and preliminary modeling results will be discussed.

Back to table of contents

Session 3A: Numerical Methods
1:45 PM - 3:15 PM (Room A)

1:45 PM - Eric Peters[1] and John Evans[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

A Stabilized Finite Element Methodology for Compressible Flows on Moving Domains

We present a new stabilized finite element methodology for compressible fluid flows on moving domains that may be easily implemented into existing finite element codes. Our methodology solves the compressible Navier-Stokes equations in the current con- figuration, and it exploits the Arbitrary Eulerian-Lagrangian approach for moving domains. As a consequence, our formulation introduces an extra source term to account for artificial compressibility introduced by the mesh motion. We show that our methodology satisfies the discrete geometric conservation law in that it preserves constant flow states even under severe mesh deformation. Moreover, even though our formulation is not written in conservation form, we prove that our methodology is conservative in the semi-discrete case and, if a suitable time-integration scheme is employed, in the fully discrete case. We demonstrate the accuracy and stability of our methodology using a suite of numerical tests performed on unstructured grids.

Back to table of contents

2:00 PM - Charles Morgenstern[1] and Mahadevan Ganesh[1]

[1]Department of Applied Mathematics and Statistics, Colorado School of Mines, Golden, CO

An Efficient Non-Standard FEM and Iterative Method for Acoustic Wave Propagation

We consider the time-harmonic acoustic wave propagation governed by the Helmholtz partial differential equation (PDE) with absorbing boundary conditions. The standard Galerkin variational formulation of the Helmholtz model and the associated finite element method (FEM) discretization provide a robust computational framework for simulation of acoustic wave propagation in general media with curved and non-smooth boundaries. However, in order to obtain accurate solutions the degrees of freedom in the resulting system must be increased as the frequency of the problem increases, and even if a high-order basis is implemented, the FEM system becomes very large for high-frequency problems. Thus we require iterative methods of solution and an efficient preconditioner for the problem of interest. We develop a non-standard FEM computer model to simulate the Helmholtz problem and demonstrate the accuracy of solutions and efficiency of iterative methods with the approach compared to the standard formulation.

Back to table of contents

2:15 PM - Ryan N. King[1] and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Turbulence Model Discovery with Data-Driven Learning and Optimization

Data-driven techniques have emerged as a useful tool for model development in applications where first-principles approaches are intractable. In this talk, data-driven multi-task learning techniques are used to discover flow-specific optimal turbulence closure models. We use the recently introduced autonomic closure technique to pose an online supervised learning problem created by test filtering turbulent flows in the self-similar inertial range. The autonomic closure is modified to solve the learning problem for all stress components simultaneously with multi-task learning techniques. The closure is further augmented with a feature extraction step that learns a set of orthogonal modes that are optimal at predicting the turbulent stresses. We demonstrate that these modes can be severely truncated to enable drastic reductions in computational costs without compromising the model accuracy. Furthermore, we discuss the potential universality of the extracted features and implications for reduced order modeling of other turbulent flows.

Back to table of contents

2:30 PM - Christopher Coley[1] and John Evans[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Variational Multiscale Methods for Turbulence Modeling

Large eddy simulation (LES) of turbulence has been a successful methodology for simulating turbulence, but challenges still remain which have so far prevented LES from becoming a mature technology for complex engineering flow analysis. These challenges include difficulties associated with the use of filters and numerical inconsistency of the resolved scales which results from the use of an eddy-viscosity. The use of variational multiscale (VMS) methods for subgrid-scale modelling in LES of incompressible turbulence flows was first introduced in order to replace the use of filters with variational projections. More recently, residual-based variational multiscale methods have been developed in order to capture all scales consistently, eliminating the eddy viscosity. In these VMS methods for turbulent flows, the fine scales are driven by the residual of the coarse scales. We present VMS methods for incompressible turbulent flow and apply these methods to benchmark transport problems in conjunction with structure-preserving isogeometric discretizations. The resulting formulation is demonstrated to have excellent stability and accuracy.

Back to table of contents

2:45 PM - Navid Shervani-Tabar[1] and Oleg V. Vasilyev[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Stabilized Conservative Level Set Method with Adaptive Wavelet-based Mesh Refinement

This study addresses one of the main challenges of the conservative level set method, namely the ill-conditioned behavior of the normal vector away from the interface. An alternative formulation for reconstruction of the interface is proposed. Unlike the commonly used methods which rely on the unit normal vector, Stabilized Conservative Level Set (SCLS) uses a modified renormalization vector with diminishing magnitude away from the interface. With the new formulation, in the vicinity of the interface the reinitialization procedure utilizes compressive flux and diffusive terms only in the normal direction to the interface, thus, preserving the conservative level set properties, while away from the interfaces the directional diffusion mechanism automatically switches to homogeneous diffusion. The proposed formulation is robust and general. It is especially well suited for use with adaptive mesh refinement (AMR) approaches due to need for a finer resolution in the vicinity of the interface in comparison with the rest of the domain. All of the results were obtained using the Adaptive Wavelet Collocation Method, a general AMR-type method, which utilizes wavelet decomposition to adapt on steep gradients in the solution while retaining a predetermined order of accuracy.

Back to table of contents

3:00 PM - Kenneth Pratt[1] and John Crimaldi[1]

[1]Department of Civil and Environmental Engineering, University of Colorado, Boulder, CO

Coalescence and Chaotic Stirring: The Role of Lagrangian Coherent Structures in the Mixing of Initially Distant Scalars

Anyone that has ever looked at a field of wheat blowing in the wind understands that turbulence has structure. For initially distant reactive scalars that are separated by a third non-reactive fluid, this structure facilitates the coalescence process, resulting in reaction enhancement. To investigate the role of Lagrangian coherent structures (LCS) in the coalescence of scalars, an interacting Taylor vortex model is used to simulate chaotic mixing that is ubiquitous in turbulent flows. To show the importance of structured stirring, coalescence processes are also studied for scalars that only undergo non-structured diffusive mixing. We find that chaotic flows, renowned for their ability to efficiently dilute scalars, have the competing effect of organizing initially distant scalars at timescales shorter than that required for dilution, leading to reaction enhancement. Initially distant filaments are shown to coalesce on LCS at early times, while concentrations are orders of magnitude greater than the long term, well-mixed state.

Therefore, the instantaneous structures embedded within turbulent flows are essential in the coalescence process, and cannot be approximated as a macroscopic eddy diffusivity.

Back to table of contents

Session 3B: Experimental Methods
1:45 PM - 3:15 PM (Room B)

1:45 PM - Naveen Penmetsa[1], John A. N. Farnsworth[1], and Ryan P. Starkey[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Experimental and Computational Investigation of a Dual-Throat Thrust Vectoring Nozzle

The dual-throat fluidic thrust vectoring nozzle is of particular interest because of its ability to provide large vector angles with minimal losses in thrust. This work investigated the performance of a dual-throat fluidic thrust vectoring nozzle for three secondary injection geometries: two spanwise oriented rectangular slots of two thicknesses, and a single spanwise oriented array of circular holes. Initial testing of the nozzles at a nozzle pressure ratio of two showed that the presence of the injection geometry alone influenced the baseline vector angle of the flow. With the introduction of secondary injection, the thinner rectangular slot was found to outperform the two other configurations at low injection percentages, while secondary injection through an array of holes trended higher at higher injection percentages. Using the experimental and computational data collected during this study, a method was developed to predict vector angle from the wall static-pressure distributions internal to the nozzle. The predicted thrust-vector angle matched the angles measured from Schlieren photographs to within measurement uncertainty across the range of injection mass flow rates tested.

Back to table of contents

2:00 PM - Jason D. Christopher[1], Nicholas T. Wimer[1], Torrey R. S. Hayden[1], Caelan Lapointe[1], Ian Grooms[2], Gregory B. Rieker[1], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

[2]Department of Applied Mathematics, University of Colorado, Boulder, CO

Parameter Estimation for a Turbulent Buoyant Jet Using Approximate Bayesian Computation

Approximate Bayesian Computation (ABC) is a powerful tool that allows sparse experimental or other "truth" data to be used for the prediction of unknown model parameters in numerical simulations of real-world engineering systems. In this presentation, I first introduce the ABC approach. I then use ABC to predict unknown inflow conditions in simulations of a two-dimensional (2D) turbulent, high-temperature buoyant jet. For this test case, "truth" data are obtained from a simulation with known boundary conditions and problem parameters. Using spatially- sparse temperature statistics from the 2D buoyant jet "truth" simulation, I show that the ABC method provides accurate predictions of the "true" jet inflow temperature. The success of the ABC approach in the present test suggests that ABC is a useful and versatile tool for engineering fluid dynamics research.

Back to table of contents

2:15 PM - Jean Hertzberg[1], James Browning[2], and Brett Fenster[3]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

[2]Northeastern University, Boston, MA

[3]National Jewish Health Center, Denver, CO

Vorticity for the Assessment of Right Ventricular Diastolic Dysfunction Using 4D Flow CMR

Right ventricular diastolic dysfunction (RVDD) may represent a precursor to systolic dysfunction and is an important prognostic indicator in pulmonary arterial hypertension (PAH). RV vortex rings have been observed in healthy subjects, but their significance in RV dysfunction is unknown. Vorticity, the local spinning motion of an element of fluid, is expected to be a sensitive measure of right heart cardiac fluid dynamics, and we hypothesize that it correlates with other indices of RVDD. Thirteen (13) subjects with right heart catheterization-proven PAH and 10 age-matched normal controls underwent same-day

4D flow CMR and echocardiography. RV and left ventricular (LV) echocardiographic parameters were assessed using trans-tricuspid valve (TV) and mitral valve (MV) E, A, and e' velocities as well as E/A ratios. RV and right atrial (RA) integrated mean vorticities were calculated for E and A wave filling periods using anti-aliased and noise-corrected 4D flow datasets. Our initial study of volume-integrated RA and RV vorticities indicates a significant correlation with both RV and LV echo-derived diastolic dysfunction markers. Ongoing investigations involve detailed analyses of atrial and ventricular flow and vorticity fields, and their relationships to specific anatomical features including valve leaflets, trabeculae and papillary muscles. A deeper understanding of the physics is a prerequisite to identification of specific features that can be linked to specific pathologies, and lead to improved diagnoses and treatments. This talk will present an overview of our subject cohort and workflow, and an in-depth visual tour of fluid dynamics in two subjects, one normal and one RVDD patient.

Back to table of contents

2:30 PM - Amanda S. Makowiecki[1], Torrey R.S. Hayden[1], and Gregory B. Reiker[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Double Wavelength Modulation Spectroscopy for Measurements in Fluctuating Pressure Environments

In situ pressure measurements are often critical to determining the performance and operating conditions of combustion systems. While commercial pressure transducers provide accurate pressure measurements, they can be intrusive and cannot reliably operate at temperatures in excess of 400°C. Wavelength modulation spectroscopy (WMS) has previously been used to measure pressure in these high temperature environments by measuring absorption lineshape parameters. This method requires accurate knowledge of both concentration and temperature within the system, which is rarely possible in harsh combustion environments. We demonstrate a double modulation spectroscopy technique which allows for pressure tracking, while reducing temperature and concentration dependence. We have successfully demonstrated this technique on a known mixture of water vapor in air at pressures ranging from 0.1 atm to atmospheric pressure at 1030 K in a high temperature furnace at CU Boulder. We show that there is good agreement between the data and a simulation across this range of pressures. This agreement suggests that this double modulation WMS technique has the potential to be a sensitive pressure diagnostic in fluctuating pressure systems.

Back to table of contents

2:45 PM - Torrey R.S. Hayden[1], Nicholas T. Wimer[1], Caelan Lapointe[1], Jason D. Christopher[1], Peter E. Hamlington[1], and Gregory B. Rieker[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Characterization of Catalytic Combustion in Film Processing Using Wavelength Modulation Spectroscopy

We are characterizing the combustion of a catalytic burner with Wavelength Modulation Spectroscopy (WMS) to aid with the optimization of heat treatments integral to some industrial processes. Because of the high temperatures, system noise, and large temperature gradients, laser diagnostics are an ideal tool. WMS is a technique in which one rapidly tunes the wavelength of a laser across an absorption feature of a species of interest, generating absorption-induced harmonic signals at high frequencies where various noise sources are lower. Using WMS to measure the thermodynamic properties of the product gases allows for fast and sensitive characterization of the fluid flow between the burner and the film. We can explore the thermodynamic properties of the gases that the film will be exposed to by coupling the WMS measurements with a CFD model.

Back to table of contents

3:00 PM - Roger Laurence[1] and Brian Argrow[1]

[1]Department of Aerospace Engineering Sciences, University of Colorado, Boulder, CO

Nonlinear Regression for sUAS Distributed Pressure Wind Sensing

Typically, RECUV has used a multi-hole probe to gather in-situ wind measurements. While these probes allow for high precision relative wind measurements, their costs can exceed that of the airframe and avionics by several times. In addition, small unmanned aircraft systems (sUAS) tend to not have landing gear, which increases the risk of damage to the probe during landing. To address these issues, an alternative method of wind sensing using distributed pressure sensors is currently under development (for a more in depth analysis of the project, see AIAA Papers 2015-1425 and 2016-4199).

Results from wind tunnel tests are presented here, using two forms of nonlinear regression. Nonlinear least squares was initially used to provide estimates of the angle of attack (α) and the sideslip (β) from the pressure measurements. These results are compared to results from using neural networks (the foundation of "deep learning"). In both cases, the mean errors are less than 1° for the test cases, with the neural networks giving more accurate results. The variance in the α errors are also noticeably smaller than for the β errors. Future work involves implementing this method on a flightworthy aircraft and performing calibration outside of a wind tunnel.

Back to table of contents

Session 4A: Energy Applications
3:45 PM - 5:00 PM (Room A)

3:45 PM - Eliot Quon[1]

[1]National Renewable Energy Laboratory, Golden, CO

Evaluation of Turbulence Metrics for Mesoscale-Microscale Coupling Methodologies

Wind plants operate within the atmospheric boundary layer (ABL) under inherently non-equilibrium conditions. State-of-the-art large-eddy simulations of the turbulent microscale ABL typically assume a quasi-equilibrium inflow, generated by recycling methods. In reality the inflow is susceptible to mesoscale forcings that, for instance, may arise from the diurnal cycle or frontal passages. The unsteady nature of the inflow motivates development of advanced turbulence generation techniques that are able to rapidly develop turbulence at the microscale from a smooth, time-varying mesoscale mean flow. This presentation will briefly describe a synthetic turbulence generation technique based on velocity perturbations, and then discuss various physical and statistical metrics for performance of this technique.

Back to table of contents

4:00 PM - Nicola Bodini[1,2]**, Julie K. Lundquist**[2]**, Dino Zardi, and Mark Handschy**

[1]University of Trento, Italy

[2]Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO

Three-Dimensional Structure of Wind Turbine Wakes as Measured by Scanning Lidar

The slower wind speeds and increased turbulence characteristic of turbine wakes have considerable consequences on large wind farms: turbines located downwind generate less power and experience increased turbulent loads. Wake characterization can provide important insights for turbine layout optimization and eventually decrease the cost of wind energy. While wake models exist, many studies focus on isolated turbines, while the interaction of multiple wakes must be captured in studies of large wind farms.

Accordingly, the CWEX-13 field campaign, which took place between June and September 2013 in a wind farm in Iowa, was designed to explore the interaction of multiple wakes in a range of atmospheric stability conditions. We apply a quantitative algorithm to assess wake parameters such as the velocity deficits, the size of the wake boundaries, and the location of the wake centerlines from wind speed data measured with a scanning lidar.

Using multiple horizontal scans at different elevations, a 3-D structure of wakes from the row of four turbines we focus on can be created. We find that wakes erode quickly during unstable conditions, and can in fact be detected primarily in stable conditions. Important differences emerge between the wakes of inner turbines and the wakes of outer turbines. Further, in strongly stable conditions, wind veer occurs and results in a stretching of the wake structures. These insights can be incorporated into wake models for wind farm layout optimization or for wind power forecasting.

Back to table of contents

4:15 PM - Jessica M. Tomaszewki[1]**, Julie K. Lundquist**[1]**, Matthew J. Churchfield**[2]**, and Patrick J. Moriarty**[2]

[1]Department of Atmospheric and Oceanic Sciences, University of Colorado, Boulder, CO

[2]National Renewable Energy Laboratory, Golden, CO

Do Wind Turbines Pose Roll Hazards to Light Aircraft?

Wind energy development has increased rapidly to now account for 3.3% of all electricity generation in the United States. Much of this development has occurred in rural locations, where open spaces are also home to numerous local or "general aviation" airports. Almost 40% of all wind turbines in the US exist within 10 km of a small airport. Wind turbines generate electricity by extracting momentum from the atmosphere, thereby creating wakes characterized by a wind-speed deficit and increased turbulence downwind. Recently, the concern that wind turbine wakes pose hazards for small aircraft has been used to limit wind farm development; however, conflicting estimates on the extent of the wake's hazardous influence still exist. As such, we use large-eddy simulations to examine the wake of a single turbine to assess hazards to small aircraft. Atmospheric measures of rotation (e.g. vorticity) were first examined, followed by wind-generated lift forces and subsequent rolling moments on hypothetical aircraft transecting the wake. Stable, unstable, and neutral cases are explored, with preliminary findings of the neutral case showing that discernible wake characteristics and roll hazards diminish by 10 rotor diameters downstream.

Back to table of contents

4:30 PM - Stephen Burke[1], Robert Rhoads[2], Matthew A. Ratcliff[3], Robert L. McCormick[3], Bret Windom[1]

[1]Colorado State University, Fort Collins, CO
[2]University of Colorado, Colorado Springs, CO
[3]National Renewable Energy Laboratory, Golden, CO

Distillation-based Droplet Modeling of Non-ideal Oxygenated Gasoline Blends

A relationship has been observed between increasing ethanol content in gasoline and particulate matter emissions (PM) in direct injection spark ignition (DISI) engines. Ethanol features a greater enthalpy of vaporization (HOV) than gasoline and is commonly used in gasoline blends. In addition to increasing HOV, ethanol also influences vaporization by altering the liquid and vapor composition throughout the distillation process. A droplet vaporization model is being prepared to explore causes for this PM generation. The evolving composition is being modeled as a distillation process, with non-ideal interactions between oxygenates and hydrocarbons accounted for using UNIFAC group contribution theory. Predicted composition and distillation curves are validated by experiments. Detailed hydrocarbon analysis was applied to fuel samples taken before and during distillation, and used as input for the initial droplet composition. With composition calculated throughout the distillation, the changing HOV and other physical properties can be found using reference data. The droplet can thus be modeled in terms of energy transfer, which in turn provides the transient mass transfer, droplet temperature, and droplet diameter.

Back to table of contents

4:45 PM - Colin Gould[1], Siddhesh Bhoite[1], and Anthony Marchese[1]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO

Autoignition of Liquid Hydrocarbon Droplets in Lean, High Pressure Natural Gas Mixtures Using a Rapid Compression Machine

The combustion of two fuels with disparate reactivity in internal combustion engines has been demonstrated as a means to increase fuel efficiency, reduce fuel costs and reduce pollutant formation in comparison to traditional diesel or spark ignited engines. However the natural gas substitution percentage is often limited by uncontrolled fast combustion or engine knock. To increase the natural gas substitution percentage, a greater understanding of the processes that lead to uncontrolled combustion rates in dual fuel engines is needed. A rapid compression machine (RCM) provides a controlled, high-pressure, high-temperature environment in which both the kinetics and chemistry of dual-fuel combustion processes can be studied. While RCMs are typically employed to study ignition delay of homogeneous gaseous mixtures, the RCM at CSU has been modified to incorporate suspended droplets in mixtures of methane/air to enable observations of ignition and combustion. The experimental results will be compared against computational modeling using the CONVERGE™ chemically reacting flow model, which incorporates detailed chemical kinetics. Using the combined experimental and computational approach, we will develop accurate reduced chemical kinetic mechanisms for dual fuel internal combustion engine simulations.

Back to table of contents

Session 4B: Complex and Compressible Flows 3:45 PM - 5:15 PM (Room B)

3:45 PM - Richard Martin[1], Alexander Zinchenko[1], and Robert Davis[1]

[1]Department of Chemical and Biological Engineering, University of Colorado, Boulder, CO

Numerical Calculations of the Rheology and Droplet Behavior of Emulsions in the Presence of Surfactants

Emulsions, which are defined as a mixture of two immiscible liquids, are often found in nature and industrial processes. When surfactants are present (whether added intentionally, or existing as impurities), they reduce the interfacial tension of the drops, which affects the deformations of the drops and the resulting rheology of the bulk fluid. Prior studies on surfactant-covered drops have focused primarily on the drop deformation and modes of drop breakup in shear and extensional flows, with little focus on the rheology of the system. In this work, we numerically study the rheology of emulsions in the presence of surfactants by solving the boundary-integral (BI) equation for the evolution of the drop surface, coupled with the convection-diffusion equation for the distribution of the surfactant on the surface of the drops. The general approach to constitutive modeling of non-Newtonian liquids based on the Oldroyd equation with variable coefficients, which was developed in our prior work (Martin et al. 2014), is applied to this system. The generalized Oldroyd equation involves a constitutive modeling technique that requires the use of two base flows – simple shear and planar extension – to predict the stress in a broad range of kinematics. Results for the rheology in planar-mixed flow will be presented. Another important consideration for surfactant-covered drops is the relationship between the surfactant concentration and the interfacial tension, which is related by an equation of state. Different equations of state are explored, including the linear, Langmuir (or Szyszkowski), and Frumkin equations of state. This work expands on previous work of Pawar and Stebe (1996) and Bazhlekov et al. (2006) to determine how the selection of the equation of state affects the spatial distribution of the surfactant on the drop's surface and the resulting deformation.

Back to table of contents

4:00 PM - Eduard Benet[1], Franck Vernerey[1], John Pellegrino[1] and Aly Badran[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

The Effect of Particle Deformability on the Transport of Soft Colloids Through Porous Media

To further the ability to design membranes for separation/fractionation of deformable (soft) particles (cells, liposomes, vesicles, oil-in-water suspensions), we have developed a (2-d) multiscale computational approach. We use a combination of the extended finite element method to describe the creeping fluid flow (Re $\sim$ 0) inside a portion of a filtration membrane with an embedded interface, coupled with a particle method that interpolates the particle's membrane (interfacial) position and corresponding fields using least square fitting. We have now modeled how the combination of a porous network geometry (porosity, pore size, and connectivity) and a soft particle's deformation-related properties (elasticity or surface tension) influence the particles' sieving coefficients in model porous domains made up of circular obstacles. First, we present the base case of two immiscible fluids (such as an oil droplet in water), and show that the transport of such particles consistently scale with the concepts of a critical pressure gradient across the pore being determined by the ratio of the particle and pore diameter and the interfacial tension. The base case of a single pore and droplet is then extended to include arrays of obstacles. In this case, the applied trans-domain pressure gradient is not the same pressure drop each droplet experiences when it needs to deform to pass between obstacles (a pore throat). These cases can provide a set of scaling rules to guide membrane design for fractionation purposes.

Back to table of contents

4:15 PM - Sanli Movafaghi[1], Wei Wang[1], Ari Metzger[1], Desiree D. Williams[1], John D. Williams[1] and Arun K. Kota[1,2]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO

[2]School of Biomedical Engineering, Colorado State University, Fort Collins, CO

Tunable Superomniphobic Surfaces for Sorting Droplets by Surface Tension

Manipulation of liquid droplets on super-repellent surfaces (i.e., surfaces that are extremely repellent to liquids) has been widely studied because droplets exhibit high mobility on these surfaces due to the ultra-low adhesion, which leads to minimal sample loss and contamination. Although droplet manipulation has been demonstrated using electric fields, magnetic fields, guiding tracks and wettability gradients, to the best of our knowledge, there are no reports of droplet manipulation methods that can sort droplets by surface tension on super-repellent surfaces. In this work, we utilized tunable superomniphobic surfaces (i.e., surfaces that are extremely repellent to virtually all liquids) to develop a simple device with precisely tailored solid surface energy domains that, for the first time, can sort droplets by surface tension. Droplet sorting occurs on our device entirely due to a balance between the work done by gravity and the work expended due to adhesion, without the need for any external energy input. Our devices can be fabricated easily in a short time and each device can be reused many times to sort droplets by surface tension. Further, this device is particularly useful for in-the-field and on-the-go operations, where complex analysis equipment is unavailable. We envision that our methodology for droplet sorting will enable inexpensive and energy-efficient analytical devices for personalized point-of-care diagnostic platforms, lab-on-a-chip systems, biochemical assays and biosensors.

Back to table of contents

4:30 PM - Ezio Iacocca[1,2], Thomas Silva[3], and Mark A. Hoefer[1]

[1]Department of Applied Mathematics, University of Colorado, Boulder, CO

[2]Department of Physics, Division for Theoretical Physics, Chalmers University of Technology, Gothenburg, Sweden

[3]National Institute of Standards and Technology, Boulder, CO

Thin Film Ferromagnets Acting Like a Compressible Fluid

Spin dynamics in ferromagnetic materials are mathematically described by the Landau-Lifshitz equation of motion. Recently, it has been shown that this equation can be exactly rewritten as a system of hydrodynamic equations (arXiv:1606.01565) that are analogues of the isentropic Euler equations of compressible gas dynamics. These equations exhibit intriguing features such as a velocity-dependent pressure law and broken Galilean invariance, implying that the ferromagnet's fluid-like physics are reference-frame dependent. A magnetic Mach number is defined from which subsonic and supersonic conditions are identified. By introducing finite-sized obstacles, we numerically observe laminar flow or the nucleation of ordered vortex-antivortex pairs in the subsonic regime; and the formation of a Mach cone, wavefronts, and irregular vortex-antivortex pairs in the supersonic regime. Our approach identifies a deep connection between ferromagnetism and fluid dynamics, enabling new predictions for thin film ferromagnets and opening up a new paradigm for magnetic research.

Back to table of contents

4:45 PM - Marc T. Henry de Frahan[1] and Eric Johnsen[1]

[1]Department of Mechanical Engineering, University of Michigan, Ann Arbor, MI

Blast Wave Induced Hydrodynamic Instabilities

Understanding the growth of perturbations at interfaces between different fluids is important in many scientific and engineering endeavors. A large localized deposition of energy can lead to the formation of a blast wave, a shock followed by a rarefaction. The shock strength gradually decreases as the rarefaction catches up with it. Through decompression effects and a transient and time-varying acceleration of the interface, blast waves induce hydrodynamic instabilities and fluid mixing when interacting with interfaces. Using a high-order Discontinuous Galerkin code to solve the multifluid Euler equations, we quantify the perturbation growth for the unstable configuration. We also analyze the effect of shock

strength, rarefaction length, and rarefaction strength on baroclinic circulation generation. Finally, we propose scaling relationships for the circulation and perturbation growth after the blast wave interaction with the interface.

Back to table of contents

5:00 PM - Scott Wieland[1], Daniel Livescu[2], Oleg V. Vasilyev[1], and Scott Reckinger[3]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

[2]Los Alamos National Laboratory, Los Alamos, NM

[3]Montana State University, Bozeman, MT

The Vorticity Equation and the Compressible Rayleigh-Taylor Instability

Rayleigh Taylor instabilities (RTIs) arise when a more dense fluid is suspended above a less dense fluid causing the direction of the density gradient and acceleration to be opposite. After applying a perturbation to the interface between these two fluids, the heavier begins falling in a spike as the light begins rising in a bubble. The vorticity generated by this motion causes the instability to grow in an increasingly nonlinear fashion and if the Reynolds number is high enough, this will lead to chaotic growth. Adding in the effects of compressibility in the form of a stratified background density leads to new regimes in the growth of low Atwood number RTIs, namely the exaggeration of bubble and spike asymmetries in the weakly stratified background state and complete suppression of the growth in the strongly stratified scenario. In order to better understand these results, the individual terms of the vorticity transport equation are analyzed and compared to the analogous results for the simplified cases of counter rotating vortex pairs in stratified media.

Back to table of contents

Poster Presentations

Pengyu Cao[1], Chase Fogus[1], Matthew Knopf[1], Carlos Quiroz-Arita[1], and Thomas H. Bradley[1]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO

Cells Tracking for Optimization of Sparge Mixing of Synechocystis sp. PCC 6803: Understanding the Fluid Dynamics of Flat Photobioreactors for Cyanobacterium-Based Biofuel Production

The biofuel industry based on the products of cyanobacteria has a promising future, where five-fold higher biofuel productivities than microalgae-based biofuel technologies have been reported. The net energy balance of cyanobacteria growth in photobioreactors needs to be promoted before the real change comes into reality. By understanding the fluid dynamics of a flat photobioreactor (PBR) and the motion of *Synehocystis sp.* PCC 6803 cells, the mixing rate can be optimized to minimize the energy inputs and maximize co-products. As the Photosynthetic Active Radiation (PAR) is attenuated by cyanobacterium biomass, a certain bio cell cannot always receive the maximum PAR obtained by the surface cells. By analyzing the Photosynthetic Active Radiation (PAR) experienced by certain particles during their mixing process in the PBR, a better understanding will be made to establish mixing strategies used for increasing the outcomes in PBR.

Back to table of contents

Ben Gallman[1] and Nils Tilton[1]

[1]Department of Mechanical Engineering, Colorado School of Mines, Golden, CO

Experimental Study of Unsteady Flow Through Porous Media

Though steady fluid flow through porous media is well understood, fundamental questions remain for the case of unsteady flow through porous media. This is particularly true for high permeability media where inertial, viscous, and unsteady effects can all play competing roles. Though there are several proposed models for unsteady flow through porous media, their validity is not well understood. The objective of this study is to design and build an experiment that will generate the necessary data to explore the validity of these models. The experimental apparatus is a U-shaped tube filled with water and open to the atmosphere on both ends. A porous test section is placed in the center of the U and exposed to damped oscillatory flow by raising the water level on one end of the tube and then releasing. The oscillating water level will be measured with a non-contact ultrasonic level transmitter so as to not disturb the flow, and the pressure gradient across the porous section will be measured with differential pressure transducers.

Back to table of contents

Andrew Kerr[1], Greg Shoukas[1], and Diego Arias[1]

[1]Roccor, LLC, Longmont, CO

Thermal Test Stand and Display for Flat Thin Heat Pipes

Roccor, LLC, is developing thin flat heat pipes for thermal management of power electronics and small satellites. In this project, a portable thermal test stand was developed for product testing, showcasing and advocacy. This test-stand requires being able to securely hold a flat thin heat pipe device and a reference strap, while heating one end of both specimens and cooling the other ends. Temperature and heat flux measurements are taken on the sample and reference part, in order to calculate their effective thermal conductivities. Multiple design options were evaluated for effectiveness, cost, and aesthetic value. The final design was chosen for its ease of manufacturing, requiring only simple two dimensional water jet cutting and some slight lathe-work. The Thermal Display will provide an effective and appealing demonstration to potential heat pipe users.

Back to table of contents

Kanghyeon Koo[1], Franck Vernerey[1], Eduard Benet Cerda[1], and Raghuveer Lalitha Sridhar[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO

Transport of Deformable Particle in Porous Media

The transport of soft particles under the permeable membrane which is consist of its pores and obstacles has been observed in the nature phenomenon or the industry fields. In general, hundreds of deformable particles are passing through the complex porous network, so we will investigate how the multiple particle jamming have an influence on the traffic of particle on the porous media in the macroscopic viewpoints. First of all, the path of the particle in porous media can be idealized by the discretized cylindrical elements subject to the Voronoi cells, then we investigate the laws, which is governed by the surface tension, contact angle and size, for the entry of the soft droplet in a pore. Furthermore, we derive the force balance equations for the particle interactions to understand how they deform when they are clogging at a pore. To verify our study, we perform several benchmark problems, so those results show that the implementations can not only illustrate the transport and jamming of particles, but also capture the deformability of among the particles during the interaction with pores and particles in specific. In addition, we will briefly discuss what experimental approaches need for the verification of our applied theories and implementations.

Back to table of contents

Ali Moradi[1] and Kathleen M. Smits[1]

[1]Department of Civil & Environmental Engineering, Colorado School of Mines, Golden, CO

Experimental and Numerical Study of Heat Transfer Through Unsaturated Soil from Pore Scale to Intermediate Scale with an Application to Soil Thermal Energy Storage (SBTES) Systems

A promising energy storage option is to inject and store heat generated from renewable energy sources in soil to form soil-borehole thermal energy storage (SBTES) systems. SBTES systems involve direct circulation of heated fluid through closed-loop heat exchangers in vertical borehole arrays. Most previous studies assume that soil thermal and hydraulic properties are constant and the heat transfer in soil occurs only in the form of conduction, and neglecting convective and latent heat transfer. An approach to potentially enhance the efficiency of SBTES systems is to install them in the unsaturated zone. In this case, it is possible to take advantage of both latent heat and convective heat transfer to obtain greater heat injection and extraction rates. The goal of our research is to better understand heat transfer processes for SBTES systems installed in the vadose zone through conducting experimental and numerical studies. Three different phases using both experimental and numerical investigations are defined. The experimental study was started with performing 2-D bench scale experiments followed by 3-D experiments. Experimental data were then used to validate a non-isothermal numerical model. Moreover, a set of pore scale simulations are defined to study heat transfer processes at the pore scale through developing a numerical model using a phase field method.

Back to table of contents

Robert Rhoads[1], Stephen Burke[2], Matthew Ratcliff[3], Robert McCormick[3], and Bret Windom[2]

[1]University of Colorado, Colorado Springs, CO

[2]Colorado State University, Fort Collins, CO

[3]National Renewable Energy Laboratory, Golden, CO

Optimized Fuel Composition and Operating Parameters for Ethanol-Gasoline Direct Injection Spark Ignition Engines

In direct injection spark ignition (DISI) engines, the spray/fuel evaporation process can impact engine efficiency, knock resistance, and particulate matter (PM) emissions. In fuels which contain oxygenated compounds or species with high enthalpies of vaporization, these effects on engine performance can be amplified. Work is underway to analyze the phase change behaviors of ethanol-gasoline blends and understand the impact that these properties play during the chemical energy conversion process in a DISI engine. Using an advanced distillation apparatus the distillation curves of multiple ethanol-gasoline mixtures are measured to characterize the volatility of the fuel. Distillate samples are withdrawn at varying points during the distillation and chemically analyzed to track the compositional make-up of the vaporizing fuel. These data are then used to determine the evolution of important physical properties of the fuel as it is distilled, which are concurrently being used as inputs and/or validation of a multi-component droplet simulation. Future studies will employ the same fuel mixtures in a DISI engine to derive correlations between the fuel volatility and engine performance/PM emissions such that optimal ethanol-gasoline mixtures for use in DISI engines may be determined.

Back to table of contents