Rocky Mountain Fluid Mechanics Research Symposium 2015: Abstracts for Oral and Poster Presentations

Discovery Learning Center, University of Colorado, Boulder
August 4th, 2015

Keynote Presentation

Dr. Diego A. Donzis, Texas A&M University, (1:00 - 2:00 PM)
Extreme events in turbulence: scaling and direct numerical simulations

Oral Presentations

Session 1: Biological and Environmental Fluid Mechanics (8:05 - 9:01 AM)

8:05 AM David Bark Jr. (Colorado State University, Fort Collins, CO)
Hemodynamic and thrombotic properties of a superhydrophobic prosthetic heart valve

8:19 AM James Browning (University of Colorado, Boulder, CO)
Right heart 3-dimensional flow in the normal and RVDD disease state

8:33 AM Kenny Pratt (University of Colorado, Boulder, CO)
Impacts of non-divergence-free behavior on the coalescence of initially distant buoyant scalars on a turbulent free surface

8:47 AM Claire Strebinger (Colorado School of Mines, Golden, CO)
High-accuracy numerical simulations of concentration polarization in reverse osmosis systems

Session 2: Biofuels and Reacting Flows (9:01 - 9:57 AM)

9:01 AM Carlos Quiroz (Colorado State University, Fort Collins, CO)
Scalability of photobioreactors based on cyanobacteria by fluid dynamics approaches incorporated in growth kinetic models

9:15 AM Jordan Kennedy (National Renewable Energy Laboratory, Golden, CO)
Rheology of concentrated algal slurrier

9:29 AM Jack Ziegler (National Renewable Energy Laboratory, Golden, CO)
3D multiphase gas/particle flow modeling for reactor-scale biomass conversion and upgrading simulations: heat transfer, mixing, and basic deactivation modeling in fixed, bubbling, and fluidized bed reactors

9:43 AM Paul Schroeder (University of Colorado, Boulder, CO)
Dual frequency comb spectroscopy of high temperature water vapor absorption: Testing and improving spectral databases for use in coal gasification

Session 3: Colloidal Systems (10:40 - 11:22 AM)

10:40 AM Xingfu Yang (Colorado School of Mines, Golden, CO)
Propulsion of colloidal dimers by breaking the symmetry in electrohydrodynamic flow

10:54 AM Yang Guo (Colorado School of Mines, Golden, CO)
Colloid transport in a microfluidic soil analog: population dynamics and single particle trajectories

11:08 AM John Mersch IV (University of Colorado, Boulder, CO)
Fouling reduction through nano-imprint lithography (NIL) on ultrafiltration (UF) membranes

Session 4: Fundamentals (11:22 AM - 12:18 PM)

11:22 AM Michelle Maiden (University of Colorado, Boulder, CO)
Nonlinear wave trains in viscous conduits

11:36 AM Ian May (Colorado State University, Fort Collins, CO)
Universality of dissipation scales in turbulence

11:50 AM Colin Towery (University of Colorado, Boulder, CO)
The dynamics of strongly compressible turbulence

12:04 AM Benjamin Miquel (University of Colorado, Boulder, CO)
Multiscale analysis of the equatorial dynamics of protoplanetary disks

Session 5: Aerodynamics and Wind Energy (2:00 - 3:24 PM)

2:00 PM Jennifer Annoni (National Renewable Energy Laboratory, Golden, CO))
Reduced-order modeling for control of complex fluid systems

2:14 PM Alec Kucala (University of Colorado, Boulder, CO)
Phononic subsurfaces for laminar flow control

2:28 PM William Liller (United States Air Force Academy, Colorado Spring, CO)
Designing small propellers for optimum efficiency and low noise footprint

2:42 PM Ella Sidor (United States Air Force Academy, Colorado Spring, CO)
RANS of HIFiRE-6 flow path

2:56 PM Ethan Culler (University of Colorado, Boulder, CO)
Spanwise variation of stall flutter on a flexible NACA 0018 finite span wing

3:10 PM Srinivas Gunter (National Renewable Energy Laboratory, Golden, CO))
Observations in MW scale wind turbine aerodynamic tests

Session 6: Numerical Methods (4:06 - 5:30 PM)

4:06 PM Patrick French (United States Air Force Academy, Colorado Spring, CO)
Reduced order model development for aircraft dynamics using indicial grid movement

4:20 PM Eric Brown-Dymkoski (University of Colorado, Boulder, CO)
A hybrid adaptive simulation method for bounded turbulent flows

4:34 PM Landon Owen (Colorado State University, Fort Collins, CO)
Lid-driven cavity flow simulation using fourth-order, compressible solver with mesh refinement

4:48 PM Eric Peters (University of Colorado, Boulder, CO)
A local kinematic/kinetic coupling algorithm for non-matching isogeometric/finite element and isogeometric/finite volume discretizations

5:02 PM Joshua Christopher (Colorado State University, Fort Collins, CO)
Anisotropic patch-based adaptive mesh refinement for finite-volume method

5:16 PM Meredith Purser (University of Colorado, Boulder, CO)
Coupled level set volume of fluid method for incompressible two-phase flow

Poster Presentations
(Session 1: 9:57 - 10:40 AM, Session 2: 3:24 - 4:06 PM)

1. David Bark Jr. (Colorado State University, Fort Collins, CO)
   The pressure field and flow environment in the valveless embryonic zebrafish heart

2. Brandon Blakeley (Colorado School of Mines, Golden, CO)
   Autothermal reforming and catalytic partial oxidation of methane in a novel ceramic microchannel reactor

3. Alexander Bulk (Colorado State University, Fort Collins, CO)
   Mechanisms influencing retrograde flow through the atrioventricular canal during embryonic cardiogenesis

4. Thomas Cooper (United States Air Force Academy, Colorado Spring, CO)
   Evaluation of Kestrel at hypersonic speeds

5. Cesar Galan (University of Colorado, Boulder, CO)
   Evaluation of turbulence closures for rotating turbulence

6. Torrey Hayden (University of Colorado, Boulder, CO)
   Large amplitude wavelength modulation spectroscopy for sensitive measurements of broad absorbers

7. Felix Jimenez (University of Colorado, Boulder, CO)
   Inter-operator variability in the analysis of time resolved cardiac magnetic resonance imaging

8. Ryan King (University of Colorado, Boulder, CO)
   Autonomic closure for large eddy simulation

9. Caelan Lapointe (National Renewable Energy Laboratory, Golden, CO)
   An advanced actuator line method for wind energy applications and beyond

10. Marcus Lehmann (Colorado School of Mines, Golden, CO)
    Microfluidic flow assays in the diagnosis of Type I Von Willebrand Disease

11. Patrick McArdle (Colorado State University, Fort Collins, CO)
    A numerical model for the determination of biomass ignition from a hotspot

12. Monique McClain (University of Colorado, Boulder, CO)
    Lagrangian coherent structures in high-speed premixed turbulent reacting flows

13. Eric Miller (National Renewable Energy Laboratory, Golden, CO)
    Semi-implicit methods for solving the 1D momentum equation at finite volume junctions

14. David Nieves (University of Colorado, Boulder, CO)
    Asymptotically reduced equations for rapidly rotating and strongly stratified flow

15. Ryan Oba (Colorado State University, Fort Collins, CO)
    In Vitro pulsatile flow loop using human blood to mimic physiological Flow conditions

16. Meredith Plumley (University of Colorado, Boulder, CO)
    Simulations of rotating Rayleigh-Bérnard convection with Ekman pumping

17. Eliot Quon (National Renewable Energy Laboratory, Golden, CO)
    CFD investigation of wind turbine tower fairing geometries

18. Clayton Reid (United States Air Force Academy, Colorado Spring, CO)
    Suitability of CFD in place of wind tunnel data for use in ATLAS simulation

19. Jennifer Rinker (National Renewable Energy Laboratory, Golden, CO)
    Nonstationary stochastic turbulence simulator via temporal coherence

20. Trevor Roberts (University of Colorado, Boulder, CO)
    Effect of off shore wind turbines on wave height

21. Ryan Skinner (University of Colorado, Boulder, CO):
    Active flow control in aggressive subsonic diffuser

22. Aaron True (University of Colorado, Boulder, CO)
    The fluid mechanics of siphon flows

23. Natalie Wisniewski (United States Air Force Academy, Colorado Spring, CO)
    The influence of airfoil shape, tip geometry, Reynolds number and chord length on small propeller performance and noise

24. Kyle Woolwine (University of Colorado, Boulder, CO)
    Reduced order modeling of an external compression supersonic engine inlet

Keynote Presentation

Diego A. Donzis[1] **(1:00 - 2:00 PM)**

[1]Department of Aerospace Engineering, Texas A&M University, College Station, TX

Extreme events in turbulence: scaling and direct numerical simulations

Turbulence at high Reynolds numbers is replete with strong fluctuations in vorticity, dissipation and other features of small-scale motion, which can be thousands times their respective mean values. Even at low Reynolds numbers, gradients can be extreme in compressible turbulence where shock waves appear. In order to study these very localized fluctuations in space and time, sufficient resolution (higher than commonly used) is required if all the details are to be captured faithfully in direct numerical simulations (DNS). This poses extraordinary challenges in simulations at high Reynolds or Mach numbers. We will discuss current computational challenges in simulations using hundreds of thousands of processors as well as a possible path towards simulations at even higher levels of parallelism.

Using a large DNS database of incompressible and compressible turbulence with resolutions up to 4096^3 we study different aspects of extreme events including universality at high Reynolds numbers, scaling and forecasts. In compressible turbulence, additional dilatational motions are related to the appearance of shocks and shocklets (short-lived randomly distributed shock-like structures). We will discuss recent work on these intense events including results in shock-turbulence interactions, its amplification of turbulence, as well as the modification of the shock. We argue that intense gradients due to small-scale intermittency and shocks present similar scaling in the high-Reynolds number limit and may explain the so-called broken regimes in shock-turbulence interactions. Very strong shocks can also create thermal non-equilibrium which in turns modifies turbulence. We will review recent progress on this interaction.

Speaker Biography:

Dr. Diego A. Donzis is assistant professor in the Department of Aerospace Engineering at Texas A&M University where he is also co-director of the National Aerothermochemistry Lab and the director of the Turbulence and Advanced Computations Lab (TACL). He received his PhD from the Georgia Institute of Technology and continued his research at the University of Maryland and the International Centre for Theoretical Physics, Italy. His main interests are in high-performance computing at extreme scales, the physics of turbulence and turbulent mixing in incompressible and compressible flows. Among Dr. Donzis major recognitions are an NSF CAREER award, the Francois Frenkiel Award from the American Physical Society, two DOE INCITE awards, and two TEES Young Faculty Award for research.

Oral Presentations

Session 1: Biological and Environmental Fluid Mechanics (8:05 - 9:01 AM)

8:05 AM - David Bark Jr.[1,2], Hamed Vahabi[1], Hieu Bui[1], Brandon L. Moore[1], Arun K. Kota[1,2], Ketul C. Popat[1,2], and Lakshmi P. Dasi[1,2]

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

Hemodynamic and thrombotic properties of a superhydrophobic prosthetic heart valve

Half of current heart valve replacements consist of mechanical valves, which are prone to thrombosis, but are more durable than approved alternatives. Therefore, we developed a prototype superhydrophobic mechanical heart valve (SMHV) to partially alleviate some of the thrombotic risk of a mechanical valve. Superhydrophobic surfaces are characterized by the degree that water beads on the surface and are exemplified by the lotus leaf in nature. To develop a SMHV, we applied a coating consisting of fluorinated polyurethane mixed with silica particles to a St. Jude Medical bileaflet mechanical heart valve. We performed contact angle goniometry to demonstrate that the surface is superhydrophobic. In addition, hemodynamic performance was evaluated for coated and non-coated valves by measuring the effective orifice area in a flow loop and through turbulence calculations for particle imaging velocimetry data. Thrombotic potential was assessed by measuring free hemoglobin after incubating surfaces with whole blood and by evaluating platelet adhesion after incubation with platelet rich plasma. Overall there were slight improvements in hemodynamic performance, limited by the presence of form drag on the valve and there was a dramatic reduction in thrombotic potential based on static assays.

8:19 AM - James Browning[1], Brett E. Fenster[2], Jean R. Hertzberg[1], and Joyce D. Schroeder[3]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]Division of Cardiology, National Jewish Health, Denver, CO
[3]Division of Radiology, National Jewish Health, Denver, CO

Right heart 3-dimensional flow in the normal and RVDD disease state

Heart disease, the leading cause of death in the US, is a complex pathology which may manifest in several ways including morphological and hemodynamic changes in the heart and circulatory system. Recent advances in time resolved cardiac magnetic resonance imaging (4DMRI) have allowed for characterization of blood flow in the right ventricle (RV) and right atrium (RA), including calculation of vorticity and qualitative visual assessment of coherent flow patterns. This presentation presents background on heart disease, specifically right ventricular diastolic dysfunction (RVDD), and 4DMRI tools and techniques used for quantitative and qualitative analysis of cardiac flows in the normal and disease states. Results of an ongoing study of right heart vorticity in subjects with RVDD are presented in addition to visualizations of right heart flow patterns in normal and pathological states.

8:33 AM - Kenny Pratt[1] and John Crimaldi[1]

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

Impacts of non-divergence-free behavior on the coalescence of initially distant buoyant scalars on a turbulent free surface

Buoyant scalars on the free surface of a 3D incompressible turbulent fluid are advected by a 2D compressible velocity field, resulting in scalar behavior different from that seen in a 2D incompressible flow. Our research uses both numerical and experimental approaches to investigate the coalescence of two initially distant reactive scalars to infer the impact of non-divergence-free behavior on buoyant scalar coalescence. Preliminary numerical results, utilizing incompressible and compressible chaotic 2D models, indicate that non-divergence-free behavior increases the likelihood of scalar coalescence and therefore

enhances any reactions between the scalars. Experimentally, we have constructed a 60 X 60 X 60 cm tank that generates three-dimensional turbulence via the random pulsing of 36 jets on the tank bottom. Buoyant fluorescent red and green particles will then be used to quantify coalescence. Experiments for compressible and incompressible cases (by adding a thin surfactant film on the free surface) will be conducted. From these results, we hope to elucidate the role of free-surface flow on the coalescence of initially distant buoyant scalars, and extend these results to oceanic mixing problems, such as the transport of phytoplankton blooms and oil spills.

8:47 AM - Claire Strebinger[1] and Nils Tilton[1]

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

High-accuracy numerical simulations of concentration polarization in reverse osmosis systems

Reverse osmosis (RO) membrane filtration systems have numerous applications in seawater desalination and advanced water treatment. RO systems filter contaminated water by forcing the water through semipermeable membrane sheets at high pressures. The large pressure difference across the membrane forces the clean filtrate through the membrane while dissolved solutes remain in the bulk flow. During this process, the filtered solutes accumulate on the membrane surfaces and produce an osmotic pressure that increases the resistance to flow through the membrane. This coupling between the transmembrane filtrate flow, pressure, and osmotic pressure make RO systems difficult to simulate numerically. Additional complications related to conservation of mass also arise due to the fact that RO systems are long open systems with flow entering or leaving through inlets, outlets, and membrane surfaces. We present our progress towards the development of a high-accuracy numerical method tailored to simulating RO. The method couples the transmembrane filtrate flow, pressure, and osmotic pressure using Darcy's law on the membrane. Using the method, we simulate the accumulation of solutes on membrane surfaces and the resulting formation of concentration boundary layers. We show that the concentration boundary layer depth grows as a power law with downstream distance. We also show that by rescaling the flow fields, the resulting concentration boundary layer likely has a similarity solution.

Session 2: Biofuels and Reacting Flows (9:01 - 9:57 AM)

9:01 AM - Carlos Quiroz Arita[1], David Bark[1], Lakshmi Prasad Dasi[1], Jason C. Quinn[2], and Thomas H. Bradley[1]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO
[2]Department of Mechanical and Aerospace Engineering, Utah State University, Logan, UT

Scalability of photobioreactors based on cyanobacteria by fluid dynamics approaches incorporated in growth kinetic models

A refined growth model based on cyanobacteria, where fluid dynamics is incorporated, was developed for scalability purposes of photosynthetic biorefineries (PBR) systems. Unlike the current state-of-the-art photoautotrophic biomass modeling, where the influence of fluid dynamics is ignored, in our novel approach we coupled mixing information obtained from computational fluid dynamics (CFD) models validated by Particle Image Velocimetry (PIV). CFD outputs, as a result, allow that cyanobacteria tracks small and large light-dark duty cycles in the growth kinetic model. By comparing preliminary results obtained from a growth kinetic model where fluid dynamics is ignored in a well-mixed reactor with our novel approach where mixing from a turbulent photobioreactor is incorporated, the well-mixed reactor depicted unrealistic cyanobacteria biomass, 1.4 times of lab conditions, whereas our novel approach depicted 40% lower cyanobacteria biomass than lab conditions. This significant information obtained from our CFD models will be utilized to scale down the rapid light changes, observed in scaled-up photo bioreactors, at the lab to resemble more realistic conditions and predict reliable growth rates of photoautotrophic microorganisms for scalability purposes.

9:15 AM - Jordan Kennedy[1] and Jonathan J. Stickel[1]

[1]National Bioenergy Center, National Renewable Energy Laboratory, Golden, CO

Rheology of concentrated algal slurries

Microalgae as a feedstock for renewable transportation fuels is currently receiving substantial interest from academic, governmental, and industrial entities. However, several technical challenges must be met before algal biofuels may be cost- competitive with transportation fuels derived from fossil sources. During harvesting, algal slurries are dewatered before further processing, producing thick slurries. In this work, the rheological properties of Chlorella algal slurries were probed as a function of slurry concentration in the range of 28% to 60% total solids. Algal slurries behave as shear thinning fluids with more significant thinning at increasing concentrations. Yield stress behavior was observed for concentrations higher than 45%. The storage modulus was found to be greater than the loss modulus, and the complex viscosity decreased with increasing frequency, corroborating the shear-thinning properties. Initial measurements indicate algal slurries show a non-reversible dependence upon thermal conditions. The steady simple shear curves were fit to the Herschel-Bulkley model and initial calculations for pressure drops in pipe flow will be presented.

9:29 AM - Jack Ziegler[1]

[1]National Renewable Energy Laboratory, Golden, CO

3D multiphase gas/particle flow modeling for reactor-scale biomass conversion and upgrading simulations: heat Transfer, mixing, and basic deactivation modeling in fixed, bubbling, and fluidized bed reactors

An overview of gas/particle flow modeling being conducted at the National Renewable Energy lab (NREL, Golden) is discussed. 3-D simulations using an Eulerian-Eulerian framework treating the solids as a fluid allow simulations of a variety of the relevant reactors at NREL are used for biomass to liquid fuel production, upgrading, and studies of novel catalysts. Current research problems are presented including issues with validation, drag models, deactivation models, and the interactions of heat transfer, kinetics and pyrolysis chemistry.

9:43 AM - Paul Schroeder[1], David Pfotenhauer[1], and Greg Rieker[1]

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

Dual frequency comb spectroscopy of high temperature water vapor absorption: Testing and improving spectral databases for use in coal gasification

Coal gasification is a promising technology for the efficient use of coal resources worldwide. The gasification process takes place at high pressures and temperatures, up to 50 bar at 1700K, so the chemical kinetics are difficult to study and not well known. These kinetics parameters are valuable for the utilization of gasification technology. In situ water vapor measurements provide a window into these processes, but require high fidelity spectral databases for the interpretation of absorption data from unknown environments. We present high resolution (0.003 cm^{-1}), broadband (353 cm^{-1}) measurements of water vapor in a controlled high temperature optical cell up to 1300K using fully resolved dual frequency comb spectroscopy referenced to the NIST atomic frequency standard. The measurements cover 2100 measurable transitions of water vapor. The immediate uses of these measurements are two-fold the evaluation of existing high temperature spectral databases, and the improvement of these databases through direct measurement of individual line parameters. In this talk, we will focus on a comparison of our high resolution data to ideal models generated from the HITEMP2010 and HITRAN2012 databases.

Session 3: Colloidal Systems (10:40 - 11:22 AM)

10:40 AM - Xingfu Yang[1], Fuduo Ma, Hui Zhao[2], and Ning Wu[1]

[1]Department of Chemical and Biological Engineering, Colorado School of Mines, Golden, CO
[2]Department of Mechanical Engineering, University of Nevada, Las Vegas, NV

Propulsion of colloidal dimers by breaking the symmetry in electrohydrodynamic flow

We show that dielectric colloidal dimers with broken symmetry in geometry, composition, or interfacial charges can all propel in directions that are perpendicular to the applied uniform AC electric field. The particle propulsion results from an unbalanced electrohydrodynamic flow on two sides of the dimers. The propulsion direction, speed, and orientation of dimers can be further controlled by frequency or field strength. Quantitative calculations based on a dumbbell model and a scaling law capture all key features observed in experiments. The propulsion mechanism revealed here can be applied to other types of asymmetric particles in general and is important for building colloidal motors.

10:54 AM - Yang Guo[1], Jingwei Huang[2], Xiaolong Yin[2], Keith B. Neeves[1], and Ning Wu[1]

[1]Department of Chemical and Biological Engineering, Colorado School of Mines, Golden, CO
[2]Department of Petroleum Engineering, Colorado School of Mines, Golden, CO

Colloid transport in a microfluidic soil analog: population dynamics and single

Low solubility contaminants such as heavy metals and radionuclides can adsorb onto mobile colloids and travel with underground water in soils over kilometers. However, the detailed flow pathway and rate-limiting transport mechanism for those colloids are largely unknown. To study this phenomenon we developed a microfluidic device to simultaneously measure colloid transport at the microscale (single colloid trajectory) and macroscale (colloid breakthrough statistics). The device consists of a rectangular bead-based soil analog and a T-junction droplet generator. The soil analog consists of pseudo two dimensional packing of 15 μm polystyrene beads with dimension of 900 μm in width, 20 μm in height, and 500 1500 μm in length. The T-junction is placed at the exit of soil analog to encapsulate colloids into dispersed droplets in mineral oil and measure breakthrough statistics. Individual colloidal trajectories of 0.5-3 μm colloids at Peclet numbers of 1600-9600 were measured by fluorescence microscopy. The breakthrough curve shows a saturation of normalized concentration at 1-2 pore volume (PV) and a plateau after 2 PV for a continuous colloids injection. Thus, colloid retention in the soil analog is negligible. The measured colloidal trajectories shows a trend of longer average trajectory length for smaller colloids. The calculated tortuosity from these trajectories shows that larger colloids has a tighter distribution around a peak of 1.10-1.15 compared to smaller colloids. These data show that microscale colloid transport is influenced by the size exclusion effect in the pore space defined by the beads. Macroscale transport as measured by breakthrough is in good agreement of larger scale column experiments. Future studies will focus on using this approach to characterize soil analogs with physical and chemical heterogeneities that mimic those found in natural soils.

11:08 PM - John Mersch IV[1], John Pellegrino[1], Thomas Hauser[2], Sajjid H. Maruf[1], and Yifu Ding[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]Research Computing and Department of Computer Science, University of Colorado, Boulder, CO

Fouling Reduction Through Nano-Imprint Lithography (NIL) on Ultrafiltration (UF) Membranes

Previous work showed promise for colloidal fouling mitigation when ultrafiltration (UF) membranes are patterned with submicron surface topology using the process of nanoimprint lithography (NIL). Specifically, resistance to colloid deposition was studied in both jet-flow and crossflow membrane permeation

cells, and the effects of permeation rate and bulk flow angle-of-attack (versus pattern features), NIL-pattern size, and crossflow velocity were considered. In this work, we have performed continuum fluid dynamics (CFD) analysis on model periodic domains representing these experimental scenarios. These simulations do not include the colloidal particles in the flow, but represent an idealization for comparing the filtration performance metrics and the fluid forces to elucidate the underlying cause(s) that control fouling mitigation. Our working hypothesis is that increased shear induced diffusion from submicron patterning contributes to the reduced fouling. Simulation results conclude that it is not only shear induced diffusion but other phenomena not yet identified. Shear rate's vertical derivative $\left(\frac{\mathrm{d}}{\mathrm{d}z}\left(\frac{\mathrm{d}u_x}{\mathrm{d}z}\right)\right)$ correlates with effects from cross flow velocity and the angle-of-attack. However, within the limits of our model simulations neither shows a significant correlation with the definitive experimental effect of pattern height variation.

Session 4: Fundamentals (11:22 AM - 12:18 PM)

11:22 AM - Michelle Maiden[1]

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

Nonlinear wave trains in viscous conduits

Viscous fluid conduits provide an ideal system for the study of dissipation-less, dispersive hydrodynamics. A dense, viscous fluid serves as the background media through which a less dense, less viscous fluid buoyantly rises. If this fluid is continuously injected into the exterior fluid, an interface forms that is similar to a deformable pipe. The conduit equation is a nonlocal, dispersive nonlinear partial differential equation that models the interface between the two fluids. In this talk, experiments, numerics, and asymptotics of the conduit equation will be presented. Structures at multiple length scales are discussed, including dispersive shock waves (DSWs) and periodic waves. In particular, the stability of periodic waves will be explored by deriving the Nonlinear Schrödinger (NLS) Equation in the small-amplitude, weakly non-linear regime. Modulational instability (stability) is identified for sufficiently short (long) periodic waves due to a change in dispersion curvature. These asymptotic results are confirmed by numerical simulations of perturbed nonlinear periodic wave solutions over long time. Also numerically observed are envelope bright and dark solutions that bifurcate from the NLS regime.

11:36 AM - Ian May[1], Prasad Dasi[1], and Karan Venayagamoorthy[2]

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

Universality of dissipation scales in turbulence

The premise of classical universality of small scales of turbulence is that it is locally isotropic and that the shape of the distribution of these small scales is similar across all Reynolds numbers, i.e. a simple scaling can collapse all distributions onto one. We show that the distribution of the dissipative scales can be analytically derived from the probability density function (PDF) of velocity increments. Additionally, a two-point single time correlation function is developed through the use of a directional Taylor microscale. These Analytic predictions show excellent agreement with direct numerical simulation. By exercising the developed model, the notion of classical universality was probed. For any turbulent flow, a universal family of scalings for dissipative scales is presented, defined by Reynolds number, the directional Taylor microscale, and a local inhomogeneity parameter. This work extends the statistical framework of Kolmogorov's theory to finite Reynolds number, inhomogeneous turbulence.

11:50 AM - Colin Towery[1], Alexei Y. Poludnenko[2], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]Laboratories for Computational Physcs and Fluid Dynamics, Naval Research Laboratory, Washington, DC

The dynamics of strongly compressible turbulence

Strongly compressible turbulence, wherein the turbulent velocity fluctuations directly generate compression effects, plays a critical role in many important scientific and engineering problems of interest today, for instance in the processes of stellar formation and also hypersonic vehicle design. This turbulence is very unusual in comparison to "normal", weakly compressible and incompressible turbulence, which is relatively well understood. Strongly compressible turbulence is characterized by large variations in the thermodynamic state of the fluid in space and time, including excited acoustic modes, strong, localized shock and rarefaction structures, and rapid heating due to viscous dissipation. The exact nature of these thermo-fluid dynamics has yet to be discerned, which greatly limits the ability of current computational engineering models to successfully treat these problems. New direct numerical simulation (DNS) results of strongly compressible isotropic turbulence will be presented along with a framework for characterizing and evaluating compressible turbulence dynamics and a connection will be made between the present diagnostic analysis and the validation of engineering turbulence models.

12:04 AM - Benjamin Miquel[1], Keith Julien[1], Philippe Marti[1], Michael Calkins[1], and Edgar Knobloch[2]

[1]Department of Applied Mathematics, University of Colorado, Boulder, CO
[2]Department of Physics, University of California, Berkeley, CA

Multiscale analysis of the equatorial dynamics of protoplanetary disks

Protoplanetary Disks (PPD) are sheet-like, large scale astrophysical vortices that rotate around central massive object such as young stars. These disks are mainly composed of cold gas with a small portion of dust. Though PPD are believed to be the birthplace of planets, no mechanism has been identified so far to explain the aggregation of dust into planets on timescales shorter than the lifespan of the disks themselves, of order of magnitude one million years. The generation of local, small scale vortices in the flow would provide a confining mechanism for dust. For this reason, it is believed to be a valuable candidate to initiate dust aggregation.

In this talk, we present a multiscale model that describes the coupled dynamics of a weak, large scale accretion flow and small scales perturbations on a Keplerian flow background. The potentiality of small scale instability will be discussed.

Session 5: Aerodynamics and Wind Energy (2:00 - 3:24 PM)

2:00 PM - Jennifer Annoni[1,2], Peter Seiler[1], and Mihailo Jovanovic[3]

[1]Department of Aerospace and Mechanics, University of Minnesota, Minneapolis, MN
[2]National Renewable Energy Laboratory, Golden, CO
[3]Department of Electrical and Computer Engineering, University of Minnesota, Minneapolis, MN

Reduced-order modeling for control of complex fluid systems

The objective of this work is to obtain a reduced-order model for complex fluid systems that can be used for control design. The work is motivated by the control of systems that involve fluid and/or structural dynamics. One specific example is wind farm control. The overall performance of a wind farm can be improved through proper coordination of the turbines. High fidelity computational fluid dynamic models have been developed for wind farms. These high fidelity models are accurate but are not suitable for control law design due to their computational complexity. Simplified, control-oriented models are needed. There are a variety of techniques that currently exist, including Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD), that extract the dominant characteristics of the flow. These dominant characteristics can be used to construct a reduced-order model

of the system. This presentation addresses an extension to DMD to handle inputs and outputs. This new method, termed Inputs/Outputs DMD (IODMD), uses direct N4SID subspace identification on a low-dimensional subspace to construct a reduced-order linear model. This proposed IODMD method is demonstrated on a linearized channel flow example and is compared to standard reduced-order modeling techniques.

2:14 PM - Alec Kucala[1], Sedat Biringen[1], Mahmoud Hussein[1], and Osama Bilal[1]

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

Phononic subsurfaces for laminar flow control

Efficient methods of drag reduction in wall-bounded shear flows remains an important, yet elusive problem in fluid mechanics. There are two potential avenues for drag reduction. One involves the delay of laminar-to-turbulent transition in the boundary layer, where in general the skin friction drag is lower for laminar flows. The other involves reducing the skin friction in an already fully-developed turbulent flow. In both cases, finding efficient methods of drag reduction remains a challenging problem. In the present computational study, we utilize a suction/blowing slot to delay transition in a channel flow with three-dimensional instabilities, where the phase and amplitude of the slot is tuned in a specified manner such that a substantial reduction in the amplitude of the primary instability wave is achieved. We extend this active control method to utilizing a passive phononic subsurface, whereby the elastodynamic response of the subsurface is precisely tuned to control the phase relationship between the pressure and velocity at the fluid/structure interface to locally attenuate the growth of perturbation energy in the fluid.

2:28 PM - William Liller[1], Charles F. Wisniewski[2], Aaron R. Byerley[2], William H. Heiser[2], and Kenneth W. Van Treuren[1]

[1]Baylor University, Waco, TX
[2]United States Air Force Academy, Colorado Spring, CO

Designing small propellers for optimum efficiency and low noise footprint

A topic of increasing importance in the Unmanned Aerial System (UAS) community is the design and performance of open propellers used in hand launched, small UASs. The design and testing of these propellers is necessary to accurately predict UAS operation. This paper describes the design methodology used by Baylor University and the USAF Academy to design propeller blades for optimum efficiency and low noise. Propeller blade design theories are discussed as well as an overview of several of the existing design codes. Included is a discussion on geometric angle of attack, the induced angle of attack, and their impact on propeller design. The design program BEARCONTROL was developed which incorporates the programs QMIL and QPROP. Supplemental codes were also developed to work with BEARCONTROL to design a propeller with a constant chord and variable twist. This resulted in the angle of attack for L/Dmax being used from the propeller hub to the tip. BEARCONTROL is a program written in MATLAB that gives a user the ability to quickly design a propeller, predict its performance, and then create a 3D model in SolidWorks. Also incorporated into BEARCONTROL is the program NREL AirFoil Noise (NAFNOISE) developed by the National Renewable Energy Laboratory (NREL). This program predicts the noise of any airfoil shape and provides a comparison for optimizing/minimizing predicted noise for the propeller being designed.

2:42 PM - Ella Sidor[1]

[1]Department of Aeronautics, United States Air Force Academy, Colorado Spring, CO

RANS of HIFiRE-6 flow path

Hypersonic International Flight Research Experimentation (HIFiRE) is a collaboration between United States Air Force Research Laboratory and Defense Science Technology in Australia that investigates

hypersonic flow and next generation vehicles that operate in that flight regime. Amongst the series is the HIFiRE-6 vehicle which focused on control surface research. The behavior of the internal flow also has an effect on the overall performance of the vehicle and therefore it was studied in support of the HIFiRE mission. Reynolds-averaged Navier-Stokes (RANS) simulations were performed on a simplified version of HIFiRE-6 using OVERFLOW 2.2 in with a focus on the internal flow path. This will aid in development of the large eddy simulation (LES) grid system that is being developed which will better account for highly complex turbulent flow system. Three grid systems consisting of 23 grids were created: a coarse grid with about 55M grid points, a medium grid with 154M, and a fine grid with 460M. Since the coarse grid used the least resources it was utilized for the bulk of the investigation presented here. The results of the coarse grid were compared to the fine grid results that were available to check for differences between the results and ensure grid independence. For all the RANS computations, the single equation Spalart-Allmaras turbulence model was used.

2:56 PM - Ethan Culler[1] and John A. N. Farnsworth[1]

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

Spanwise variation of stall flutter on a flexible NACA 0018 finite span wing

In order to increase aerodynamic performance, designers are utilizing more flexible, high aspect ratio wings. Unfortunately, this wing design sacrifices structural performance, allowing for increased wing oscillation, shorter fatigue life and lower divergence speeds. The NASA Helios prototype, which was destroyed over the Pacific Ocean when it encountered unexpected turbulence, is an example of this structural weakening. In an effort to address the problem motivated above, a novel research program has been undertaken to investigate the viability of utilizing aerodynamic flow control as a method for controlling aeroelastic wing deformations. This work is focused on studying spanwise variation and temporal evolution of the unsteady flow field around a finite span NACA 0018 wing undergoing stall flutter. Wind tunnel experiments were conducted in the United States Air Force Academy's Subsonic Wind Tunnel using a finite span NACA 0018 wing cantilevered from the tunnel floor and controlled through a cyberphysical wing system developed by Fagley et. al. The cyberphysical system allows for software based manipulation of the wing's stiffness, inertia and damping. Stereo particle image velocimetry (SPIV) data was collected phase-locked to the angular position of the model wing tip. Additionally, stereo vision data was collected to track the fluttering wing kinematics. Preliminary analysis of the stereo vision data shows a primarily pitching oscillation with a 5.5 Hz frequency and an amplitude that increases along the wing span. This suggests dominance of the first torsional mode in the structural dynamics. Preliminary analysis of the SPIV data shows significant spanwise variation in the size of the flow field region of separation.

3:10 PM - Srinivas Gunter[1]

[1]National Renewable Energy Laboratory, Golden, CO

Observations in MW scale wind turbine aerodynamic tests

The National Renewable Energy Laboratory houses several MW-scale wind turbines, a few of which are capable of collecting high-quality aerodynamic and other operational data facilitated by state-of-the-art instrumentation embedded within these turbines. The number of different experimental facilities around the world is limited, and data collected at such a facility thus facilitates research activities that are rare and unique. This presentation showcases some interesting observations that were made during the analysis of data collected via the pressure taps and five-hole probes installed on a 2.3MW turbine rotor blades installed on-site. It will be shown how various aerodynamic phenomenon such as rotational augmentation and dynamic stall, or measurement effects due to tubing dispersion effects manifest on a wind turbine response, and their sensitivity to data post-processing methods. Potential uses of such data will be highlighted, such as identification of new fluid dynamic physical effects and validation of existing design codes.

Session 6: Numerical Methods (4:06 - 5:30 PM)

4:06 PM - Patrick French[1] and Neal Frink[2]

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

Reduced-order model development for aircraft dynamics using indicial grid movement

In 2008, the Applied Vehicle Technology panel of NATO's Science and Technology Organization initiated a task group consisting of researchers from nine countries to assess the state-of-the-art stability and control prediction methods for NATO air and sea vehicles. The group developed a generic Unmanned Combat Aerial Vehicle (UCAV) called Stability and Control Configuration (SACCON) as a focus configuration. This model was designed to have intricate, unsteady aerodynamics to better evaluate stability prediction methods.

Researchers at the US Air Force Academy (USAFA) have developed a method for creating reduced-order models (ROM) through CFD simulation of indicial grid motions, which acts as a unit step input while uncoupling the aircraft's pitch angle from its angle of attack. Because of their simplified structure, these models can simulate aircraft dynamics without the high computational cost of a full-order CFD simulation. The faster computational time could support reduced-order modeling as a more accurate method to represent loss of control or stall characteristics in aircraft flight simulators.

NASA is currently working in support of NATO's AVT-201 task group by assessing the US Air Force Academy's ROM methodology for the SACCON configuration. They have coupled the USAFA's grid motion generation and reduced- order modeling programs with NASA's USM3D flow solver. A comparison of step responses to USAFA's results, as well as reduced-order models to NASA's previous CFD investigations with the SACCON model, reveals an encouraging yet unsatisfactory relationship between ROM behavior and CFD solutions. Continuing research seeks to eliminate implementation error in the modeling process, and apply results to a range of aircraft configurations and flight maneuvers.

4:20 PM - Eric Brown-Dymkoski[1] and Oleg V. Vasilyev[1]

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

A hybrid adaptive simulation method bounded turbulent flows

Wavelet-based adaptation is a highly efficient approach for resolving the multitude of scales present in turbulent flows by exploiting spatio-temporal intermittency. The performance, however, rapidly degrades in boundary layers, becoming prohibitive at high Reynolds numbers. Since the adaptive mechanism is isotropic, it is unable to distinguish the highly correlated flow along the surface. The computational cost grows exponentially and over-resolves the stream- and span-wise directions. This breakdown is particularly notable for LES and URANS simulations, where the viscous sublayer is often the smallest length scale present. However, by implementing a curvilinear coordinate transform, the mesh can be stretched and refined in a single direction, while preserving the regular, rectilinear computational space required for the wavelet adaptive algorithm. This hybrid scheme uses two separate mechanisms to optimally resolve the different phenomena present in a single flow. Mesh stretching is applied a priori to target the predictable boundary layers, and the wavelet adaptation captures the intermittent, multiscale turbulent structures. By matching the flow physics with more appropriate numerical methodologies, the prohibitive scaling of turbulent flows can be greatly mitigated.

4:34 PM - Landon Owen[1], Stephen M. J. Guzik[1], and Xinfeng Gao[1]

[1]Computational Fluid Dynamics and Propulsion Laboratory, Colorado State University, Fort Collins, CO

Lid-driven cavity flow simulation using fourth-order, compressible solver with mesh refinement

A fourth-order accurate finite-volume method is presented for solving time-dependent compressible Navier-Stokes equations on a lid-driven cavity flow geometry. Time evolution of the solution is achieved

with a fourth-order Runge-Kutta method. This work in progress investigates different strategies for resolving stability issues on the two-dimensional cavity flow with Re = 100 and Ma = 0.1. Additionally, if time permits, we plan to discuss a multi-species mixing flow for evaluating the species transport equations.

4:48 PM - Eric Peters[1], Luke Engvall[2], and John A. Evans[1]

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

A local kinematic/kinetic coupling algorithm for non-matching isogeometric/finite element and isogeometric /finite volume discretizations

The purpose of this talk is to discuss a new algorithm for coupling non-matching geometries in computational fluid structure interaction. This method was developed with the idea of integrating existing technologies in a way that incorporates the best of both worlds, specifically between Iso-Geometric Analysis (IGA) and Finite Element Methods/Finite Volume Methods (FEM/FVM). IGA, a simulation methodology in which computer aided design (CAD) technologies are directly integrated into finite element analysis, is ideally suited for dealing with structures because the availability of smooth, higher order basis functions allow for accurate solutions with far fewer degrees of freedom. Meanwhile, FEM/FVM methods are better suited for fluids as stable and efficient numerical schemes may be constructed for unstructured tetrahedral meshes. By meshing the local Bezier elements of the structure into triangles, a simple algorithm can be defined that maps the surface tractions of the fluid tetrahedrals to the surface of the structure and structural displacements to the wetted fluid surface. The only requirement is that the triangles must respect the boundaries of the Bezier elements. The resulting algorithm is computationally efficient, higher-order accurate, and easy to implement.

5:02 PM - Joshua Christopher[1] and Stephen Guzik[1]

[1]Computational Fluid Dynamics and Propulsion Laboratory, Colorado State University, Fort Collins, CO

Anisotropic patch-based adaptive mesh refinement for finite-volume method

We propose an anisotropic, patch-based adaptive refinement algorithm based on the finite-volume method for solving partial differential equations on Cartesian and mapped grids. Our motivation primarily stems from a need to regularize a moving mapped grid that becomes either highly compressed or highly stretched in one direction, for example, in the case of piston movement in the internal combustion engine. However, in this work, we will first focus on exploring the performance gain from anisotropic refinement of flow discontinuities compared to isotropic refinement; algorithm infrastructure is the same for both cases. For areas with large flow gradients, such as shock waves, boundary layers, and flame fronts, that are aligned with a grid direction, anisotropic refinement can provide a similar reduction in error using much less grid cells compared to isotropic refinement. The savings in cell count are expected to more than offset the additional program logistics. Validation of the anisotropic algorithm is achieved by comparison to the existing isotropic algorithm for an unsteady shock-tube problem and in solving the bow shock for steady-state supersonic flow past a circular cylinder.

5:16 PM - Meredith Purser[1] and Kenneth E. Jansen[1]

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

Coupled level set volume of fluid method for incompressible two-phase flow

Computational modeling of two-phase flow requires an efficient means of tracking the interface between the phases. For incompressible two-phase flows, there are several existing methods for tracking the interface, but no single method is strictly volume conserving, provides direct information about the shape and location of the interface, and works for parallel computing on unstructured meshes. We have developed

a new hybrid method that incorporates the volume conservation of volume of fluid methods and the direct information about the interface location that is provided by level set methods. Additionally, our coupled level set volume of fluid (CLSVOF) method works for unstructured meshes and parallel computing. This presentation will cover the mechanics of level set, volume of fluid, and CLSVOF methods, and the performance of our CLSVOF method, especially as it compares to level set methods.

Poster Presentations

David Bark Jr.[1,2], Brennan Johnson[2], Deborah Garrity[3], Lakshmi P. Dasi[1,2]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO
[2]School of Biomedical Engineering, Colorado State University, Fort Collins, CO
[3]Department of Biology, Colorado State University, Fort Collins, CO

The pressure field and flow environment in the valveless embryonic zebrafish heart

Cardiovascular development is influenced by spatial and temporal variation in its flow-induced stress environment. Here, we describe methods to quantify flow and relative pressure variation in the developing valveless tubular heart of a zebrafish. Flow is quantified by analyzing spatiotemporal plots and through micro-particle imaging velocimetry. The corresponding pressure field is quantified using an algorithm that integrates the momentum equation in the fluid region of the developing embryo. Through these methods, we show that the atrium builds up pressure, releasing blood into the ventricle as a bolus. The endocardial layer contacts itself during ejection (20% of the cardiac cycle), acting as a valve by increasing the resistance prevent flow reversal at the atrial inlet (reverse flow fraction ¡0.05). During atrial diastole, the atrium begins to sequentially expand beginning at the atrial inlet until expansion reaches the ventricle. This expansion creates a suction causing the blood to refill the atrium. These mechanics are distinct from the straight primitive heart tube and the fully developed adult heart. We further show for the first time how pressure varies spatially through an embryonic heart using experimental fluid approaches.

Brandon Blakeley[1] and Neal P. Sullivan[1]

[1]Colorado School of Mines, Golden, CO

Autothermal reforming and catalytic partial oxidation of methane in a novel ceramic microchannel reactor

Microchannel technology offers order-of-magnitude improvements over traditional heat exchangers and chemical reactors through process intensification. Miniaturization of flow passages amplifies heat and mass transfer processes while reducing the overall size and cost of the device. Ceramic materials allow for high temperature operation in harsh chemical environments that may not be tolerable for metal equivalents. This work examines the fabrication, operation, and performance of a novel alumina microchannel reactor developed at the Colorado School of Mines in collaboration with CoorsTek, Inc. Current experiments focus on creating hydrogen through catalytic partial oxidation and autothermal reforming of methane. Additional insight into reactor chemistry can be gained through a 3-D computational fluid dynamics model in ANSYS Fluent that simulates internal fluid mechanics, heat transfer through solid and gas phases, and a 48-step heterogeneous chemical mechanism. Experimental data demonstrates stable reactor performance in both endothermic and exothermic reaction environments. Autothermal reforming achieved $\sim$90% methane conversion at a gas hourly space velocity of 75,000 hr^{-1}. CFD model results show good agreement with experimental data.

Alexander Bulk[1,2], Brennan M. Johnson[2], Deborah M. Garrity[3], Lakshmi P. Dasi[1,2]

[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO
[2]School of Biomedical Engineering, Colorado State University, Fort Collins, CO
[3]Department of Biology, Colorado State University, Fort Collins, CO

Mechanisms influencing retrograde flow through the atrioventricular canal during embryonic cardiogenesis

When flow mechanics are altered during early stages of embryogenesis, severe changes in heart morphology can be induced, resulting in congenital heart defects. At the atrioventricular canal, a retrograde flow occurs for a short duration of each cardiac cycle that has been shown to influence normal valve development, though the mechanisms causing this retrograde flow to occur have not been well understood.

Here, two factors were determined to influence retrograde flow: 1. Pressure due to rates of expansion and contraction of the atrium and ventricle, respectively, and 2. Decrease in resistance due to decreasing length of closure of endothelium along the contracting atrium. This was shown by performing a computational analysis on image sequences of the heart in embryonic zebrafish between 48 and 55 hours post-fertilization, when normal retrograde flow is most prevalent. Retrograde flow was inhibited in some embryos by altering the cardiac preload via centrifugation between 24 and 48 hours. It was found that in embryos with normal retrograde flow, there was significantly less length of closure to resist reverse flow for the majority of the cycle than in embryos without. Also, embryos with retrograde flow had significantly greater difference in pressure associated with rates of expansion and contraction during the period of the cardiac cycle that retrograde flow occurred. These results elucidated the sensitivity in pumping mechanics of the developing heart to maintaining the necessary conditions for normal cardiac development.

Thomas Cooper[1]

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

Evaluation of Kestrel at hypersonic speeds

This project evaluated the flow solver software Kestrel, by Create (AV for Air Vehicles) at hypersonic speeds. The software is normally reserved for unstructured grids at subsonic through supersonic speeds. At hypersonic speeds, air begins to heat so intensely that chemical changes can occur, namely ionization. Also hypersonic vehicles operate at very high altitudes, where air is rarefied inducing error within the continuum assumption, which the Navier Stokes equation normally assumes. These phenomena are not known to be modeled by most CFD solvers, but hopefully there would be enough tools to accurately model hypersonic flow. A small flat faced cylinder was modelled in pointwise and imported into Kestrel. Beginning at Mach 3, flow was passed over the cylinder in integer increments up to Mach 10, then Mach 14, then Mach 18. The bow shock standoff distance was compared to experimental results published in Liepmann and Roshko?s *Elements of Gas Dynamics*, extrapolated out to Mach 18. The results were astoundingly accurate, and the code never crashed, even at such extremely high Mach numbers. This was aided by a grid refinement feature of Kestrel. To conclude, Kestrel may be accurate enough of a code to use for modeling in the hypersonic flight regime.

Cesar Galan[1]**, Alan Hsieh**[1]**, and Sedat Biringen**[1]

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

Evaluation of turbulence closures for rotating turbulence

Reynolds stress models proposed by Girimaji (1996) and Wallin (2005), as well as heat transfer closures for heat transfer advanced by Nagato (1996) and YWL (2012) were evaluated and tested using direct simulation (DNS) data obtained from simulations with and without rotation. The flow geometry consisted of a plane channel flow rotating about the spanwise axis. In this flow, due to the Coriolis force, there is a mean spanwise secondary flow that presents a challenging complex test case for turbulence models. The evaluations presented in this work display very good predictive capabilities of the closure formulas for nonrotating cases, however for rotating cases significant disagreements with the DNS data are evident. A preliminary optimization study for model constants was also performed considering the realizability of the models.

Torrey R.S. Hayden[1], Paul J. Schroeder[1], and Greg B. Rieker[1]

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

Large amplitude wavelength modulation spectroscopy for sensitive measurements of broad absorbers

We demonstrate wavelength modulation spectroscopy (WMS) of $CO2$ at pressures greater than 30 atm using a fast-scanning MEMS laser. Wavelength modulation spectroscopy is a fast, sensitive technique in which one varies the laser across an absorption feature of interest while applying a high frequency (f) modulation. The resulting harmonic signals, particularly at 2f, are sensitive to the absorbance of light by the molecules, which is a function of gas conditions such as temperature, pressure, and concentration. Traditionally, the technique has been limited to lasers with small tuning ranges; thus, the use of a MEMs VCSEL with a large tuning range enables the measurement of high-pressure gases and large molecules that were difficult to measure before. Finally, a simulation of WMS was created to accommodate the unique properties of the MEMs laser and will enable the extraction of the gas properties from the measurements made. The technique shows promise for sensitive measurements of species with broadband absorption features, such as high-pressure gases and large molecules.

Felix Jimenez[1], James Browning[1], Brett E. Fenster[2], Jean R. Hertzberg[1], and Joyce D. Schroeder[3]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]Division of Cardiology, National Jewish Health, Denver, CO
[3]Division of Radiology, National Jewish Health, Denver, CO

Inter-operator variability in the analysis of time resolved cardiac magnetic resonance imaging

The study of heart disease is critical to the lives of many people, and thus ensuring that the data collected during the process is accurate is very critical. In the process of analyzing data collected by time resolved cardiac magnetic resonance imaging (4DMRI) the agreement between the people carrying out this analysis is something that must be known, and should be considered quantitatively. The presentation gives an introduction into how this inter-operator variability is being taken into account on an on going study on right ventricular diastolic dysfunction (RVDD). The methods being used are presented with an emphasis on how they were implemented.

Ryan N. King[1,2], Werner J. A. Dahm[3], and Peter E. Hamlington[1]

[1]Department of Mechanical Engineering, University of Colorado, Boulder, CO
[2]National Renewable Energy Laboratory, Golden, CO
[3]School for Engineering of Matter, Transport, and Energy, Arizona State University, Tempe, AZ

Autonomic closure for large eddy simulation

We present a new autonomic subgrid-scale closure for large eddy simulations (ALES) that solves a nonlinear, nonparametric system identification problem on the fly. ALES expresses the subgrid stress tensor as the most general nonlinear function of the state variables at all locations and times using a Volterra series. This expansion introduces a large number of coefficients that are determined by solving a regularized optimization problem. ALES captures nonlinear, nonlocal and nonequilibrium turbulence effects by mapping the state variables to a higher dimension space represented by the Volterra terms before solving the optimization. The coefficients are optimized at a test filter scale and then projected to the LES grid scale by invoking inertial range self-similarity. This obviates the need for users to select a predefined turbulence model structure or calibrate models with training datasets. We perform a priori tests of the ALES closure on homogeneous isotropic turbulence and report substantial improvements in capturing the instantaneous, statistical, and energetic properties of the subgrid stresses as compared to the dynamic Smagorinsky model. We also discuss new developments in machine learning that enable

turbulence model discovery using the ALES framework.

Caelan Lapointe[1] and Matthew Churchfield[2]

[1]Union College, Schenectady, NY
[2]National Renewable Energy Laboratory, Golden, CO

An advanced actuator line method for wind energy applications and beyond

The actuator line method has been in use for modeling wind turbines for over a decade. With this method, turbine blades are represented by lines which are then discretized into individual elements. At each of these elements, the local velocity is sampled, and those values are used to look up coefficients of lift and drag from tabulated airfoil data, which gives the blade lift and drag distribution. This method is good for estimating turbine power production, but over predicts tip loading and does not properly capture tip vortices. The current actuator line method simply tri-linearly samples velocity from the flow field to the actuator elements. It also uses a uniform three-dimensional Gaussian function to project the blade force onto the flow field as a volumetric force, resulting in a smooth cylindrical region of body force about the line. We propose an advanced actuator line method that uses a body force projection function that looks more like the real pressure distribution over a blade, which we hypothesize will improve tip predictions. Also, instead of tri-linear velocity sampling, we propose a more robust integral sampling method to obtain a more accurate estimate of local freestream velocity.

Marcus Lehmann[1], C.M Ng[2], J.A Di Paola[2], and K.B. Neeves[1]

[1]Department of Chemical and Biological Engineering, Colorado School of Mines, Golden, CO
[2]Department of Pediatrics, University of Colorado, Denver, CO

Microfluidic flow assays in the diagnosis of Type I Von Willebrand Disease

Type I Von Willebrand Disease (VWD) is a genetic bleeding disorder characterized by low expression in Von Willebrand factor (VWF). VWF and platelet function are shear stress-dependent, but of the available clinical assays, only the PFA-100 incorporates shear stress. Here, we compare the sensitivity to VWF levels (VWF:Ag) of a microfluidic flow assay (MFA) to the PFA-100 and other conventional assays. Whole blood from individuals (n = 19) with suspected type 1 VWD was collected. In the MFA, blood was perfused over a 150 μm strip of type I collagen (500 μg/mL) through microfluidic channels (h=50μm, w=500μm) at wall shear rates of 150, 750, and 1500 s^{-1} for 5 min. Transient platelet accumulation was recorded by relief-contrast imaging and quantified by three metrics; (i) lag time (time to 5% coverage), (ii) velocity (linear growth phase), and (iii) final coverage. Both the PFA-100 and the MFA lag times showed dramatic loss in platelet function for VWF:Ag<30. The lag times and velocity at 750 s^{-1} and 1500 s^{-1}, but not 150 s^{-1} were sensitive to VWF:Ag, demonstrating the effect of shear rate on VWF function. Unlike the PFA-100, the MFA lag time at 750 s^{-1} was sensitive to VWF:Ag<30. The MFA therefore shows promise in filling the void of diagnostic techniques for mild Type I VWD.

Patrick McArdle[1]

[1]Colorado State University, Fort Collins, CO

A numerical model for the determination of biomass ignition from a hotspot

A fourth-order finite-volume method for the determination of biomass ignition from an inert spherical hotspot is presented. The transient ignition-combustion system is modeled by coupled reaction-diffusion equations: one governs the heating characteristics of the biomass and the other governs the mass loss of the biomass. The combustion assumes a one-step, 1st-order Arrhenious reaction. This work is motivated by improving fire danger rating systems on military lands, since the ignition of biomass from military munition fragments is a common problem. Our results show that given the ignition criteria, it is possible to create an ignition probability plot dependent on the physical characteristics of the biomass. In this

talk, I will discuss the numerical model, the physical model and model parameters, and the resulting correlations between the ignition probability and the biomass properties.

Monique McClain[1], Clarissa Briner[2], and Peter E. Hamlington[1]

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

Lagrangian coherent structures in high-speed premixed turbulent reacting flows

In order to improve the efficiency of air-breathing combustion engines, it is imperative to understand the coupled effects of turbulence and combustion for reactive fuel-air mixtures. Turbulent fluids can be modeled using Lagrangian Coherent Structures (LCSs), which are skeletal patterns of flow about which fluid pathlines are structured. The purpose of this research is to characterize LCSs for premixed combustion of H_2 and air. This is accomplished by calculating Finite Time Lyapunov Exponents (FTLEs) of neighboring particle trajectories. The FTLEs are determined by calculating the exponential rate at which two adjacent fluid trajectories separate from each other. The ridges of the FTLE field yield LCSs. In addition to characterizing LCSs for a turbulent reacting flow, the effect of temperature on fluid trajectories is analyzed in order to understand the interaction between turbulence and combustion. In particular, characteristics of trajectories that pass through the flame surface are compared to properties of trajectories that do not pass through the flame surface. Ultimately, this research will improve understanding of coupled turbulence and combustion processes, which will, in turn, inform the development of computational models for simulations of real-world combustion and propulsion problems.

Eric Miller[1,2], Gene Titov[1], and Jason Lustbader[1]

[1]Vehicle Systems Group, National Renewable Energy Laboratory, Golden, CO
[2]Department of Mechanical, Aerospace, and Nuclear Engineering, Rensselaer Polytechnic Institute, Troy, NY

Semi-implicit methods for solving the 1D momentum equation at finite volume junctions

Vehicle thermal management systems are becoming more complex in the interest of efficiency. By adding liquid coolant loops to cars and trucks, designers are able to provide climate control for passengers, heat or cool batteries, and capture waste heat from the motor and other components. Vehicle mileage can be improved by optimizing distribution of fluid, requiring extensive simulations or testing. In simulations, finite volumes are typically used to capture liquid temperature as it passes through various heat exchanger components in a closed loop. To simulate fluid flow in such networks, a form of the momentum conservation equation must be solved. Using the electric circuit analogy yields steady-state results, but requires a fully implicit system of equations, making it impractical for refrigeration models, where modularity of components is highly valued. Conversely, a fully explicit solution method requires small simulation steps or numerical damping. The proposed semi-implicit method allows for modular components and improved stability by solving each component implicitly using the previous time step solutions generated by neighboring components as boundary conditions. The method is derived from the first principles, characterized, and used to study a sample flow network of practical complexity.

David Nieves[1] and Keith Julien[1]

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

Asymptotically reduced equations for rapidly rotating and strongly stratified flow

Recent observations by van Haren & Millot (2005) of the Western Mediterranean Sea find vertical fluid motions to be as large as horizontal motions. In an attempt to explain this phenomena, reduced equations and numerical simulations describing flow constrained by rapid rotation and strong stable stratification are presented. The equations presented are an asymptotic reduction of the incompressible Boussinesq equations and describe non- hydrostatic and geostrophically balanced flow valid in the Ro $\ll 1$, Fr $\ll$

1 limit. In particular, $\varepsilon = $ Ro is taken to be a small parameter and requires Fr $\sim$ Ro1/2 in order for stratification effects to be important. Two models arise in this scenario. The first is similar to studies by Smith & Waleffe (2002) and Pouquet et al. (2014) where the imposed background stratification is unchanged by fluid motions. The second model allows fluid motions to modify the background stratification profile. The latter case is similar to Rayleigh-Bénard convection in the sense that the mean buoyancy profile is modified by vertical buoyancy flux. For the sake of comparison to recent works we focus on the former model. An overview of the model development is provided followed by preliminary results of forced triply-periodic numerical simulations.

Ryan W. Oba[1], David L. Bark[1], Ketul C. Popat[1,2], Lakshmi P. Dasi[1,2]
[1]Department of Mechanical Engineering, Colorado State University, Fort Collins, CO
[2]School of Biomedical Engineering, Colorado State University, Fort Collins, CO

In Vitro pulsatile flow loop using human blood to mimic physiological flow conditions

This in vitro flow loop was designed to simultaneously subject two identical prosthetic heart valves to tunable physiological aortic flow conditions, using human blood as the working fluid. The underlying hypothesis of this work is that this flow loop system can provide hemodynamic and hemo-compatibility data that can help to accurately predict in vivo replacement heart valve performance. Validation of the flow loop hemodynamic environment itself was performed by measuring flow and pressure within the chamber, using a blood viscosity-matching analog. Positive values of pressure and flow corresponded to opening and closing stages of the valves, whereas the flat sections corresponded to the closed state of the valves. Minor negative values in the flow illustrated that the valves used had small levels of regurgitation, or reverse flow during testing. Additional validation was performed using particle image velocimetry (PIV), on two St. Jude pediatric mechanical replacement heart valves. PIV results, depict vortex formation originating from the tips of the mechanical heart valve leaflets which continue to form downstream from the valves; agreeing well with results seen from other testing systems. Validation of the ability to control clot formation within the flow loop was performed by using anticoagulated whole human blood (3.2% citrate). Calcium addition to the blood neutralized the anticoagulation from the citrate and large scale clot formation on mechanical heart valves was observed.

Meredith Plumley[1], Keith Julien[1], and Philippe Marti[1]
[1]Department of Applied Mathematics, Colorado State University, Fort Collins, CO

Simulations of rotating Rayleigh-Bérnard convection with Ekman pumping

Rotating Rayleigh-Bérnard convection is of interest in many geoscience applications, with examples such as deep ocean convection or dynamics in our planet's interior all occurring in the regime where convectively driven motions are dominated by the effects of rotation. To better understand these large physical systems, several techniques are used including asymptotic methods, DNS and experiments. While these three methods have seen good agreement in results for free-slip boundary conditions, the case of rigid no-slip boundaries presents an interesting difference. Along the rigid boundaries, Ekman layers form and Ekman pumping occurs. Due to how thin these layers are, it has been thought that they are negligible for small Ekman number. However, new DNS of the 3D Boussinesq equations have provided evidence that this is not the case. A new asymptotic model has been developed to include these boundary layers and verify the impact of the Ekman boundaries on the flow. Results from simulations of this new model will be compared with DNS and experimental results. The results support the findings of increased global heat transport due to the presence of Ekman pumping.

Eliot Quon[1]

[1]National Renewable Energy Laboratory, Golden, CO

CFD investigation of wind turbine tower fairing geometries

Current trends in horizontal-axis wind turbine design tend toward increasing rotor size to increase energy capture and reduce the cost of energy. Upwind rotor blade designs are typically constrained by tower clearance requirements; for all operating conditions, the blade must not strike the tower. As a result, blade stiffness and consequently system mass can increase to the point that benefits from the increased rotor are outweighed by increases in cost. This motivates investigation into downwind configurations in which the rotor operates in the wake of the wind turbine tower. To minimize the effects of the vortical wake including a mean velocity deficit and unsteady impulsive loads, various tower fairing geometries have been considered. Recent PIV studies at the University of Virginia (UVa) have investigated the applicability of the E863 airfoils series as tower fairings and quantified the resultant tower shadow loads in comparison with a standard cylindrical geometry. This talk will discuss recent work at NREL in collaboration with UVa to study the same configurations using the OpenFOAM toolset. The goal of these investigations is to validate NREL computational models and to evaluate the performance of additional fairing geometries under more diverse operating conditions.

Clayton Reid[1] **and James Clifton**[1]

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

Suitability of CFD in place of wind tunnel data for use in ATLAS simulation

The purpose of this project was to investigate the suitability of CFD data in the place of wind tunnel data and flight test data for use in Lockheed Martin's ATLAS (Aircraft Trim Linearization and Simulation) software. This experiment was conducted at the Air Force SEEK EAGLE Office at Eglin AFB, Florida, with the help of computing resources located at Wright-Patterson AFB, Ohio. The data compared was CFD data from an F-16 with a certain store configuration modeled in ATLAS and a pre-existing ATLAS model of an F-16 with a similar store configuration developed from wind tunnel data. Several maneuvers were run in ATLAS using both models and the accuracy of the CFD model was compared to the pre-existing model. Overall, this experiment shows to some extent that CFD can be a suitable method to generate simulation models for use in ATLAS, depending on the quality of the training maneuver used in CFD. This training maneuver covered the longitudinal regressors well, but not so much the lateral-directional regressors. Adding a yaw oscillation to the training maneuver could increase the quality of the ATLAS model obtained using these methods.

Jennifer Rinker[1,2]**, Henri Gavin**[1]**, and Andy Clifton**[2]

[1]Duke University, Durham, NC
[2]National Renewable Energy Laboratory, Golden, CO

Nonstationary stochastic turbulence simulator via temporal coherence

One of the stochastic turbulence simulation techniques currently used in wind turbine design, colloquially called the Veers method, produces stochastic fields that are stationary (i.e., whose statistics do not vary with time). Wind is very rarely stationary, so expanding this simulation technique to produce nonstationary output is important. We have been investigating the utilization of temporal coherence, which creates a correlation between the Fourier phases at different frequencies, to produce nonstationary output. Temporal coherence is easily implemented through the use of non-uniform probability distributions on the difference in phase between the Fourier components at different frequencies. This paper will overview the theory behind temporal coherence, how it is implemented in NREL's stochastic simulator TurbSim, and an analysis of the temporal coherence that was found in turbulent wind data recorded at NREL's National Wind Technology Center just south of Boulder.

Trevor Roberts[1], Nick Wimer[1], and Peter E. Hamlington[1]

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

Effect of offshore wind turbines on wave height

One of the most important aspects of an offshore wind farm is the height of waves around these turbines. When wind interacts with a turbine the velocity of the wind will decrease behind the rotor but speed up around the turbine blades. This effect causes large, turbulent wakes. For conventional, inland turbine farms this effect on the surrounding environment is of minimal interest. However, for offshore turbine farms these wakes can have dramatic effects on the power generation of the farm as a whole. Any and all waves can have an adverse effect on the pitch and orientation of floating turbines. To be able to design an effective offshore wind farm it is necessary to understand and account for the effects waves will have on floating turbines. This research will analyze the effects floating turbines have on the local wind field as well as the effects the wind field has on wave height.

The simulation of atmospheric boundary layer wind speed data will be conducted using OpenFOAM more specifically utilizing NREL's SOWFA package. To execute the appropriate simulations involving wind and wave fields around turbines, a three-dimensional mesh grid will be constructed of size of 3 by 3 by 1 km. Refinements will be made of areas within this domain to enhance the accuracy of data in significant locations. Two such areas are the ocean boundary surface and the areas surrounding the individual turbines. A logarithmic scaling of grid cells from the ocean's surface and upward will insure that the appropriate turbulent boundary layer profile is captured. Once precursor data has been compiled the addition of physical turbines can begin. Using the same mesh grid and domain size, an arrangement of turbines will be implemented. Simulations of the same complexity will be run and analyzed in order to observe the effects the wind turbines have on downstream wakes. More importantly, it will be necessary to observe and understand how the wakes of multiple turbines interfere constructively or destructively. Improving the computational accuracy around the turbines is then the next step.

From there, the wind data obtained from the wind turbines simulations will be fed into a wave solver. Previous projects of this type have used WaveWatchIII, a spectral wave energy solver. WaveWatchIII will be coupled with the wind data previously developed in OpenFOAM to simulate the difference that the turbines make on the ocean surface.

Ryan W. Skinner[1] and Kenneth E. Jansen[1]

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

Active flow control in aggressive subsonic diffuser

High-speed flow through curved ducts and diffusers are prone to flow separation, mean-flow asymmetry, and loss of usable energy downstream. In this computational research, we evaluate active flow control strategies for delaying or eliminating separation in an aggressive subsonic diffuser. Specifically, we use side-wall suction to symmetrize the flow, in conjunction with unsteady 2D tangential blowing to control separation. Simulations are carried out in the compressible version of PHASTA, a massively-parallel, adaptive, finite-element CFD code. The primary turbulence model is delayed detached-eddy simulation (DDES). Results from unsteady Reynolds-averaged (URANS) runs are also presented, but are found to under-resolve key physical features of the control mechanism. Runs are conducted at the Argonne Leadership Computing Facility (ALCF) using up to 32k processors. Simulated performance of both baseline and flow control scenarios are validated against experimental data from collaborators at Rensselaer Polytechnic Institute (RPI).

Aaron True[1] and John Crimaldi[1]

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

The fluid mechanics of siphon flows

Siphon and suction flows are ubiquitous in engineered and natural systems. Recently, numerical modeling of biological siphons with a boundary condition of constant volumetric outflow (as opposed to a

uniform inlet velocity profile) has revealed nontrivial departures from idealized approach (point sink potential flow) and pipe flows (the pipe entry problem), particularly at low Reynolds number. These findings, and our own in-house modeling studies, have inspired us to build an index of refraction-matched flume (mineral oil, borosilicate glass tubing) in which we are employing particle image velocimetry (PIV) to ultimately quantify transient and steady state flow fields within and approaching the siphon tube. Varying siphon diameter, flow rate, and extraction height allows us to evaluate the effects of Reynolds number and siphon geometry on local hydrodynamics. Reduced entrance lengths, larger radial inflows, and modifications to fluid capture zones seen numerically at low Re will be tested experimentally and may have important implications for both biological and engineered siphons. From pipettes, water samplers, and home brewer's siphons to filter feeding bivalves and predation by fishes, there is much to be gained from a detailed understanding of siphon flow hydrodynamics.

Natalie Wisniewski[1], Charles F. Wisniewski[2], Aaron R. Byerley[2], William H. Heiser[2], Kenneth W. Van Treuren[3], and William R. Liller[3]

[1]Rose-Hulman Institute of Technology, Terre Haute, IN
[2]]United States Air Force Academy, Colorado Spring, CO
[3]Baylor University, Waco, TX

The influence of airfoil shape, tip geometry, Reynolds number and chord length on small propeller performance and noise

An extensive experimental investigation to determine the overall efficiency and near field noise signature of propellers utilized by small hand launched UASs has been conducted. This investigation has included wind tunnel performance comparisons of both off-the-shelf and custom designed propellers at realistic thrust and freestream velocities. A propeller design program has been developed that gives a user the ability to quickly design a propeller, predict its performance, and then create a 3D SolidWorks model for fabrication using an SLA printer. This computer code was used to design propellers for a parametric study of airfoil cross-section, chord length and tip geometry which led to an optimized design configuration that greatly out performs available off-the-shelf propellers. The results of this design approach are high pitch propellers with low aspect ratios. The increased chord lengths create large surface areas that lower the rotational speed required to achieve the desired thrust for a given freestream velocity flight condition. These low aspect ratio propeller designs place emphasis on tip geometry to increase aerodynamic efficiency and reduce noise generating vortex strength. The result is a 5 bladed oval tipped propeller configuration that is 12 dB quieter than the commercial propeller and 6% percent more aerodynamically efficient. This represents an elimination of 70% of the baseline propeller near-field noise signature with the potential of increasing the aircraft endurance by 6%.

Kyle Woolwine[1,2]

[1]University of Colorado, Boulder, CO
[2]NASA Glenn Research Center, Cleveland, OH

Reduced order modeling of an external compression supersonic engine inlet

The Supersonics Project under NASA's Fundamental Aeronautics Program is currently conducting research concerning Aero-Propulso-Servo-Elastic (APSE) effects inherent in supersonic flight. Researchers at the NASA Glenn Research Center are constructing the engine portion of the model using Simulink that will be integrated into an overall APSE model. The research contained in this presentation is attempting to model the external compression axisymmetric inlet portion of the engine in a way that will capture the effects of a full 3-D CFD model but maintain the quickness of a lower dimensional quasi 1-D model. The goal is to obtain an efficient, reduced order model that is able to properly simulate the effects of atmospheric perturbations and changes in angle of attack. This is accomplished by creating high fidelity quasi 2-D and 3-D models with the CFD code PHASTA. These models are then used to both validate and drive the creation of a quasi 1D model in the Simulink/MATLAB environment. Results from PHASTA models as well as preliminary results from reduced order model will be presented.