Advances in Fluid Mechanics and Turbulence

Lagrangian and Eulerian Acceleration Properties in Homogeneous Turbulence with Shear and Rotation, **FRANK G. JACOBITZ[1]*, KAI SCHNEIDER[2], WOUTER J.T. BOS[3], and MARIE FARGE[4]** ([1]Mechanical Engineering Department, Shiley-Marcos School of Engineering, University of San Diego, 5998 Alcalá Park, San Diego, CA 92110, USA, jacobitz@sandiego.edu; [2]Laboratoire de Mécanique, Modélisation, et Procédés Propres du Centre National de la Recherche Scientifique, Aix-Marseille Université, 38 rue Joliot-Curie, 13451 Marseille Cedex 20, France, kschneid@cmi.univ-mrs.fr; [3]Laboratoire de Mécanique des Fluides et d'Acoustique du Centre National de la Recherche Scientifique, Ecole Centrale de Lyon, Université de Lyon, 69134 Ecully Cedex, France, wouter.bos@ec-lyon.fr; [4]Laboratoire de Météorologie Dynamique du Centre National de la Recherche Scientifique, Ecole Normale Supérieure, 24 rue Lhomond, 75231 Paris Cedex 5, France, farge@lmd.ens.fr)**.**

The properties of Lagrangian and Eulerian acceleration of turbulence are important for a variety of problems ranging from fundamental theoretical considerations to modeling of dispersion processes. The acceleration statistics of rotating and sheared homogeneous turbulence are studied here using direct numerical simulations. The study focuses in particular on the influence of the Coriolis to shear rate ratio and also on the scale dependence of the statistics. The probability density functions (pdfs) of both Lagrangian and Eulerian acceleration show a strong and similar influence on the rotation ratio. The flatness further quantifies this influence and yields values close to three for strong rotation. For moderate and vanishing rotation, the flatness of the Eulerian acceleration is larger than that of the Lagrangian acceleration, contrary to previous results for isotropic turbulence. A wavelet-based scale-dependent analysis shows that the flatness of both Eulerian and Lagrangian acceleration increases as scale decreases. For strong rotation, the Eulerian acceleration is more intermittent than the Lagrangian acceleration, while the opposite result is obtained for moderate rotation.

Subtle Effects of Air Lubrication, **CAMPBELL DINSMORE*, MARKO PRINCEVAC, RICHARD COPCA, JOSHUA HAUSER, WARNER TSE, PAUL LOU, JIT MALAY, and CHRISTOPHER MORALES** (Department of Mechanical Engineering, University of California, Riverside, 900 University Avenue, Riverside, CA 92521; cdinsmore@engr.ucr.edu).

With the advent of terms like "globalization" and "export driven economy", international trade is clearly an important part of the world's economic landscape – trade that travels, in significant part, across the world's oceans. Even though maritime shipping is efficient, making it more effective is crucial to reducing global transportation costs and the shipping industry's environmental impact. Consequently, numerous studies have investigated the "air lubrication" of the ship's hull in which air bubbles are introduced along a ship's wetted surfaces in an attempt to reduce the drag on the ship. Significant power reductions (about 6% net) have been achieved with some air lubrication systems. Almost completely absent in these studies, however, is the bubbles' impact on the propeller's thrust. Consequently our initial focus has been to design and conduct model tests in a water channel that attempt to capture this effect. During these tests, bubbles were injected near the propeller of a remote controlled boat and the resulting thrust was measured. Combinations of trials with and without bubbles were run and initial results indicate a 4.5% reduction in thrust when bubbles are present. Alongside this experimental work, theoretical investigations into the nature of the bubble/water interaction have also been undertaken. This work has led to the development of creeping flow equations describing flow fields in both very small bubbles and the water that surrounds those bubbles. In this presentation, preliminary experimental results for the propeller tests and theoretical solutions for shear flow around micro-bubbles will be presented.

New Superfog Screening Tool – Development and Validations through Laboratory Experiments, **CHRISTIAN BARTOLOME[1]*, MARKO PRINCEVAC[1], AKULA VENKATRAM[1], DAVID WEISE[2], GARY ACHTEMEIER[3], and SHANKAR MAHALINGAM[4]** ([1]Department of Mechanical Engineering, University of California, Riverside, Riverside, CA 92521; [2]USDA Forest Service Pacific Southwest Station, Riverside, CA 92507; [3]USDA Forest Service Southern Research Station, Athens, GA 30602-2044; [4]College of Engineering, University of Alabama, Huntsville, AL 35805; cbartolo@engr.ucr.edu).

Superfog is dense smoke cloud that reduces visibility to less than 10 feet. Major car pile ups, such as ones in Florida on the Interstate Highways 4 and 75 in 2008 and 2012, have resulted due to the formation of a Superfog. Here we will present laboratory modeling efforts that have been conducted to explain the origins of Superfog and that led to the development of a new screening tool.

Thermodynamics of water vapor, droplet size distribution, liquid water content, extinctions coefficients and boundary layer development were explored. Wind tunnel measurements of smoldering smoke and smoke boundary layer growth have been conducted for various environmental and fuel conditions.

In these experiments the favorable conditions for the Superfog formation were determined to be: fog droplet sizes less than 1 micrometer, minimum cloud condensation nuclei concentrations of 10^5 per ccm, liquid water content greater than 2 g kg^{-1}, ambient temperature less than 40^{o}F, ambient humidity greater than 80%, fuel moisture contents greater than 40% by mass, and wind velocities less than 1 m s^{-1} (2.2 mph).

The Superfog Analysis Model (SAM) has been developed to aid land managers to quickly assess situations as favorable or unfavorable to the formation of Superfog. SAM has been validated by laboratory experiments and has been successful in predicting previous Superfog events. The underlying theory and laboratory experiments in the development of SAM will be presented.

A Study of the Flow Field Surrounding Interacting Line Fires, **TREVOR MAYNARD[1]*, MARKO PRINCEVAC[2], and DAVID WEISE[3]** ([1]USDA Forest Service Technology and Development Program, San Dimas, CA 91773, tbmaynard@fs.fed.us; [2]University of California, Riverside, Riverside, CA 92521, marko@engr.ucr.edu; [3]USDA Forest Service Pacific Southwest Research Station, Riverside, CA 92507, dweise@fs.fed.us).

The interaction of converging fires often leads to significant changes in fire behavior, including increased flame length, angle, and intensity. In this presentation, the fluid mechanics of two adjacent line fires is discussed theoretically and experimentally. A simple potential flow model is used to explain the tilting of interacting flames towards each other, which results from a momentum imbalance triggered by fire geometry. The model was validated by measuring the velocity field surrounding stationary alcohol pool fires. The flow field was seeded with high-contrast colored smoke, and the motion of smoke structures was analyzed using a cross-correlation optical flow technique. The measured velocities and flame angles are found to compare reasonably with the predicted values, and an analogy between merging fires and windblown flames is proposed.

Ignition of Live Vegetation by Convection and/or Radiation – Preliminary Results, **DAVID R. WEISE[1]*, THOMAS H. FLETCHER[2], SARA McALLISTER[3], and JONATHAN GALLACHER[2]** ([1]USDA Forest Service, Pacific Southwest Research Station, Riverside, CA 92507, dweise@fs.fed.us; [2]Brigham Young University, Provo, UT 84602, tom_fletcher@byu.edu; [3]USDA Forest Service, Rocky Mountain Research Station, Missoula, MT 59807, smcallister@fs.fed.us).

In wildland fires, the process of ignition is not well-described. Particularly in live fuels (living vegetation), the relative importance of radiation and convection have received little study; however, radiation has been assumed to be the dominant heat transfer mode. We are studying the ability of radiant and convective sources to ignite samples of fresh live vegetation over the annual growth cycle to determine whether the seasonal changes in ignition trends hold between different modes of heating. Convective heating is produced using a flat-flame burner, and tests with both convection and radiation include a radiative panel above the burner. Radiation only tests are performed in a small-scale wind tunnel with radiant panel. Preliminary results from three species - lodgepole pine (*Pinus contorta*), big sagebrush (*Artemisia tridentata*), and chamise (*Adenostoma fasciculatum*) will be presented.

Experiments on Surface Fire Transition to the Elevated Live Fuels, **SUNDAY OMODAN[1]*, CHRISTIAN BARTOLOME[1], RAUL-DELGA DELGADILLO[1], JOEY CHONG[2], GLORIA BURKE[2], MARKO PRINCEVAC[1], and DAVID WEISE[2]** ([1]Department of Mechanical Engineering, Bourns College of Engineering, University of California, Riverside, 900 University Avenue CA 92521, USA, cbartolo@engr.ucr.edu, rdelg002@ucr.edu, marko@engr.ucr.edu; [2]USDA Forest Service, Pacific Southwest Research Station, Riverside, California, USA, jchong@fs.fed.us, gburke@fs.fed.us, dweise@fs.fed.us).

Ecosystem damage by wildland fire is on the increase. An estimated 9,907 wildland fires burned 577,675 acres and an additional 542 prescribed fires were used to treat 48,544 acres by various agencies in California in 2013. Five percent of California wildland is covered by chaparral which can burn vigorously under favorable conditions. Fire in the elevated canopy in chaparral fuels has been described as a crown fire. Fire behavior modeling and measurements lead to tools for decision making in both combating wild fires and validating fire predictions. The majority of crown fire research has been focused on coniferous forests, and limited research has been conducted on chaparral crown fires.

Laboratory experiments conducted in a wind tunnel facility are focused on understanding chaparral crown fire behaviors, particularly the ignition, mechanisms of flame propagation, spreading, burning dynamics of the flame front and fuel consumption rates. Impacts of surface fire, relative position of the crown in either the continuous and intermittent flame zone, presence of wind, environmental temperature and humidity, and fuel moisture content are variables being examined.

Experiments conducted under zero wind speed conditions produced near vertical surface flames and lower flame heights which resulted in some cases of crown ignition failure. Introduction of wind speed of 1 m/s led up to 3 times higher surface flames when compared to the zero wind speed conditions. Video footages taken during experiments and data on fire behavior characteristics will be presented.

Characterizing Wicking for Development of Paper-based Analytical Devices, **CARLOS CASTRO*, CINDY ROSILLO, and HIDEAKI TSUTSUI** (Department of Mechanical Engineering, University of California, Riverside, 900 University Avenue Riverside, CA, 92521; htsutsui@engr.ucr.edu).

Paper-based analytical devices are emerging technologies with transformational potential in revolutionizing point-of-care diagnostics. Because paper is a low-cost, ubiquitous, and self-wicking material, it is an ideal substrate for developing simple-to-use, portable, and disposable devices for fluid specimen. Existing devices are typically simple lateral-flow tests whose functions are limited to low-level qualitative detection of analytes. To significantly expand functionality of paper-based analytical devices, it is necessary to better understand fluid transport within the channel networks defined on the paper substrate. In this study, we investigated fluid wicking in various one- or two-dimensional geometries that were created by printing and melting a wax-based ink on a sheet of filter paper. Wet-out flows through porous media are often described using the Lucas-Washburn equation, which models the porous medium as a bundle of rigid capillaries. However, this classical equation is insufficient in describing effects of swelling, tortuosity, channel boundary, and evaporation, all of which are evident in operations of most paper-based devices, resulting in retardation of fluid flow. We experimentally investigated effects of these parameters and developed a modified version of the Lucas-Washburn equation which now provides a more accurate prediction of fluid wicking through paper channels under non-ideal conditions. In addition, we analyzed the wet-out flow in a variety of channel components that can be used to construct more complex fluidic circuitries. Collectively, these results are expected to help develop more functional paper-based analytical devices.

Recent Advances in Optical Thermocavitation, **DARREN BANKS*, MOLLY DANIELS, and GUILLERMO AGUILAR** (Department of Mechanical Engineering, University of California, Riverside, 900 University Ave., Riverside, CA, 92521; dbank005@ucr.edu).

While optical thermocavitation has been observed by photonics and fluids researchers for over 50 years, recently a surge of research interest has led to novel applications and developments in this field. Notably, the use of relatively low-power and readily available continuous-wave lasers to induce optical thermocavitation has opened up a range of possibilities, from using high-velocity liquid jets produced by cavitation bubbles as optical and acoustic waveguides, to drug delivery by micro-perforation in of the stratum corneum. At the same time, new techniques for characterizing cavitation, such as the Spatial Transmission Modulation method, are nascent. We present an overview of modern developments in the field of optical thermocavitation, including advances in measurement and characterization technology, insight into the thermocavitation process, and emerging applications in biomedicine and other fields of thermocavitation.

Single Droplet and Train Impingement Pool Cooling, **DARREN BANKS*, CYNTHIA AJAWARA, JIE LIU, and GUILLERMO AGUILAR** (Department of Mechanical Engineering, University of California, Riverside, 900 University Ave., Riverside, CA, 92521; dbank005@ucr.edu).

The thermal interaction of an impacting, cooling water droplet or train of droplets and a heated (T = 60°C) solid substrate is explored for various pool conditions on the substrate. The cooling effects are related to impact velocity (0-3 m/s), pool depth (0-20 mm), and droplet diameter (1-5 mm); for a train of droplets the influence of impact frequency (0.5-30 Hz) is added. Using high speed video, the droplet diameter and impact velocity are measured, as well as the dynamics of the cavity formed within the liquid pool after impact. Embedded at the substrate surface is a fast response RTD which provides temperature history. Using a finite-time step integration of Duhamel's theorem, the instantaneous heat flux and net heat extraction at the surface are computed.

For a single droplet impact, heat transfer appears to be maximized within an intermediate range of impact Weber number. Higher and lower Weber numbers than this range dramatically decreases heat flux. The optimum Weber number is increased by larger pool thicknesses. This range is characterized by the cavity approaching the substrate but not making contact; this brings cool droplet fluid into direct contact with the substrate. Decreasing the Weber number prevents droplet fluid from reaching the substrate; increasing it appears to push liquid away from the measurement point, reducing heat flux at that point. Trains of droplets demonstrated an initial drop in temperature and then fluctuations around a quasi-steady 'cooled' temperature point; the magnitude and period of these fluctuations was dependent on the impact frequency.

Magnification of Concentrations in Cities, **NICO SHULTE*, SI TAN, and AKULA VENKATRAM** (Mechanical Engineering, University of California, 3401 Watkins Drive, Bourns Hall A342, Riverside, CA 92507; nschu003@ucr.edu).

A transit oriented development (TOD) is typically a compact community of homes, offices, and shops built within walking distance of a transit station, such as a light rail or a bus station. TODs can change the dispersive ability of the atmosphere relative to that in open terrain. We are conducting a field study of dispersion within TODs in which we measure particle concentrations and meteorology within different built environments within Los Angeles, and simultaneously measure meteorology at a location located upwind of the urban area. Our goal is to develop a model that allows us to understand the effect of the built environment parameters such as the building height, plan area fraction, and frontal area fraction, on the dispersion within a TOD. The effect of buildings is quantified in terms of the magnification, the ratio of urban to rural concentration. We evaluated several alternative dispersion models for the urban environment using observations obtained in our field study, and found that dispersion in urban areas is governed by the mean wind speed and turbulence at street level. We then use the dispersion models and the micro-meteorological measurements at urban and rural sites to evaluate the magnification. The magnification is usually greater than 1 because the wind speed at street level is reduced relative to the rural area. The magnification is also largest during the early morning, when traffic emissions are also usually large.

Understanding the Impact of Built Environment on Air Quality in Transit Oriented Developments, **SI TAN*, NICO SCHULTE, and AKULA VENKATRAM** (Mechanical Engineering, University of California, 3401 Watkins Drive, Bourns Hall A342, Riverside, CA 92507; stan004@ucr.edu).

Transit Oriented Developments (TODs) are designed to promote walking, cycling, and public transportation to reduce motor vehicle emissions by increasing the density of people through the use of multi-story buildings. In principle, TODs should improve local air quality, but the effect of these multi-story buildings on dispersion within the urban canopy is not yet known. Thus, we are conducting a field study to investigate the impact of building morphology on local air quality in TODs. To understand dispersion in urban areas, we need to develop a model that allows us to estimate micrometeorological parameters at urban street level using values measured at upwind rural locations. The micrometeorological parameters can be used as surrogates for estimating magnification factors, which is defined as the ratio of rural to urban concentrations, to evaluate the effects of buildings. We made micrometeorology measurements simultaneously at an upwind rural site, at rooftop, and at street level to evaluate the evolution of turbulence and wind speed from rural to urban area. The measurements were made in two cities in California, with average building heights of 6m and 36m. The wind speed at rooftop is slightly lower than the corresponding upwind rural value, and is further reduced at street level. Vertical turbulent velocity fluctuation increases from upwind to rooftop, and decreases from rooftop to street level. As a result, the vertical turbulent velocity fluctuation at street level is similar to the upwind value, and is relatively insensitive to the building height.

Near Road Impact of Sound Walls on Air Quality Mitigation, **MARKO PRINCEVAC*, SAM POURNAZERI, and BRANDN GAZZOLO** (Department of Mechanical Engineering, University of California Riverside, Riverside, CA 92521; marko@engr.ucr.edu).

Vehicular emissions are one of the major sources of air pollution in urban areas and several epidemiological studies have shown that long term exposure to vehicle related pollutants increases the risk of respiratory diseases, birth defects, premature mortality, cardiovascular disease, and cancer. Since mid-20th century major highways are commonly accompanied with roadside structures known as sound walls. These structures are mainly designed to protect residential areas close to the highway by damping the roadway noise. The incorporation of these roadside structures brought up the question: how do these structures impact the air quality in residential areas located in the vicinity of highways?

A thorough understanding on the direct impact of sound walls on flow and dispersion can significantly help air quality modelers in development of dispersion models that can accurately predict the human exposure in the areas close to highways. These models can be further incorporated into regulatory approved dispersion models such as AERMOD (Cimorelli et al., 2005) which are used in project-level conformity and hot spot analysis.

The presented research addresses the effects of sound walls through systematic water channel simulations accompanied with numerical modeling, Quick Urban and Industrial Complex (QUIC) model, and traditional simple Gaussian-type dispersion models. In our laboratory setup we visualized the dye spread to observe the average behavior of traffic related pollutants released into the atmosphere. Such visualizations can provide us with great details on the processes involved with the dispersion of such pollutants.