A Simple Microbubble Ladened Turbulent Flow Field Dissipation Model and Current Investigations Into This Model, CAMPBELL DINSMORE1*, AMIRHESSAM AMINFAR2, and MARKO PRINCEVAC2 (1Department of Mechanical Engineering, College of Engineering, California State Polytechnic University at Pomona, 3801 West Temple Avenue, Pomona, California 91768, USA; cadinsmore@cpp.edu; 2Department of Mechanical Engineering, Bourns College of Engineering, University of California at Riverside, 900 University Avenue, Riverside, California, 92521, USA, aamin006@ucr.edu, marko@engr.ucr.edu).

The interaction between particles immersed in a turbulent flow field and the fluid that surrounds them is a subject of ongoing study. Past efforts generally focused on identifying the conditions under which a bubble would breakup and key characteristics of the bubbles in a turbulent flow field, such as the size associated with the largest stable bubble that can be supported by the flow. Even though these studies are continuing, new avenues of investigation are focusing on the shape of bubbles in a turbulent flow field and the dissipation of the turbulent kinetic energy in the vicinity of microbubbles. Recent efforts associated with characterizing the dissipation of turbulent kinetic energy in the vicinity of microbubbles will be reviewed. A simple model will be proposed that applies these results at the local level more broadly to the entire bubble laden turbulent flow field. The limitations associated with the local dissipation model, and how they may impact the larger scale model, will also be reviewed. The work to resolve these limitations is ongoing and the latest investigations associated with these efforts, which are specifically focusing on alternative formulations for vector harmonic functions used in the development of the flow field equations, will be presented. Finally, the impact of these limitations on the local and broader dissipation models, as well as these current efforts to resolve these limitations, will be assessed and their impact on future efforts in this area will be discussed.

Large Eddy Simulation of Fluid Flow in the Stem Cell Stirred Bioreactor, MASOUD GHASEMIAN*, HIDEAKI TSUTSUI, and MARKO PRINCEVAC (Mechanical Engineering Department, Marlan and Rosemary Bourns College of Engineering, University of California, Riverside, 900 University Ave, Riverside, CA 92521, USA; mghas002@ucr.edu, htsutsui@engr.ucr.edu, marko@engr.ucr.edu).

Stirred bioreactors are a common method for stem cell expansion. The mixing within a bioreactor should be enough to fully suspend stem cells in their surrounding medium and both transfer nutrients to and waste-products from these cells. On the other hand, shear stresses induced by stirring can have deleterious effects on the growth of stem cells. Therefore, it is necessary to characterize the hydrodynamic forces and fluid structures within the stirred bioreactor. The current study utilizes a Large Eddy Simulation (LES) technique to study the hydrodynamic characteristics of fluid flow in a Corning™ spinner flask. A Sliding Mesh (SM) methodology is employed to capture the impeller rotation. Different impeller speeds are examined. Flow patterns and shear stress distributions are visualized at different regions in the vessel. Time-averaged Kolmogorov length scales within bioreactor are computed. The results demonstrate that smaller Kolmogorov length scales exist at the higher impeller speeds. Higher shear stresses happen close to the impeller which can damage the cells. A correlation between Kolmogorov length scale and cell expansion size is observed. The outcome of this study can be exploited to propose an improved reactor design.
The effect of surface tension on laser induced cavitation bubbles has been investigated. High-speed photography was used to measure the diameter of the bubbles up to 120,000 fps. The collapse time and the growth rate were obtained by spatial transmittance modulation (STM) method for cavitation bubbles formed both in ethanol and water. These liquids were selected for their similar viscosity but difference in surface tension by more than factor of three. A Nd:YAG, Q-switched laser with wavelength of 1064 nm and pulse duration of 6 ns was used with the appropriate attenuator to alter the energy. The optical breakdown threshold for plasma formation in the two solutions was investigated and only varies by 5% for the two solutions. This result can be attributed to the difference in infrared absorption of the two liquids. The comparative experiments in water and ethanol were performed at and above the threshold energy of the plasma formation. We observe that the cavitation bubbles in ethanol grow larger in size than bubbles in water. The STM results indicate that cavitation bubbles in ethanol have longer lifetimes. Liquids with lower surface tension display lower resistance to deformation, therefore the bubbles grow larger in these liquids. In future studies, the surface plasmonic resonance effect of nanoparticles will be investigated. The result from our current research will allow us to separate the role of plasmonic effect from the influence of surface tension on bubbles formed in the plasmonic nanoparticle solutions.

Optical cavitation is a process in which a laser is focused into a liquid, causing it to ionize and leading to the rapid growth and collapse of a vapor bubble (~200 microseconds, ~1 millimeter diameter) followed by formation of a shockwave. Optical cavitation produces local changes in mechanical, optical and thermal properties which may allow cavitation bubbles to be used for applications in biomedicine and micro-fluidic controls. Here we present the preliminary work involving two areas of cavitation bubbles yet to be explored experimentally: measurement of the refractive index inside of a single cavitation bubble and the thermal profile induced around a single bubble. Cavitation bubbles are generated in deionized water using a 1064nm wavelength, 6ns Nd: YAG pulsed laser. A continuous-wave laser illuminates the cavitation bubbles and forms a diffraction pattern. The diffraction pattern is analyzed with a Fresnel propagation approach and fitted with a model of a circular lens aperture to approximate the refractive index inside the bubble. Once obtained, the refractive index inside the cavitation bubble may provide valuable information on other bubble properties. The second study focuses on measuring the thermal effects of cavitation bubbles on the surrounding liquid. Conventional temperature sensors are vulnerable to shockwave damage and influence the bubble dynamics. Here we describe the development of planar-laser induced fluorescence (PLIF) for non-intrusively measuring the temperature field around a bubble. The PLIF method uses Rhodamine-B, a fluorescent dye whose intensity varies linearly with temperature. The temperature field will be observed for different bubble characteristics.

Transition and Flame Spread Characteristics in Chaparral Fires, JEANETTE COBIA INÍGUEZ*, AMIRHESSAM AMINFAR1, ALEJANDRO GONZALEZ1, IVAN HERRERA1, JOEY CHONG2, GLORIA BURKE2, DAVID WEISE2 and MARKO PRINCEVAC1 (1Department of Mechanical Engineering, University of California, Riverside, CA 92521, USA, jcoibi002@ucr.edu, aamin006@ucr.edu, agonz093@ucr.edu, iherr010@ucr.edu, marko@engr.ucr.edu; 2Pacific Southwest Research Station, USDA Forest Service, 4955 Canyon Crest Drive, Riverside, CA 92507, USA, jchong@fs.fed.us, gburke@fs.fed.us, dweise@fs.fed.us).

Southern California wildfires are primarily fueled by chaparral shrubs which are considered crown fires. Crown fires are composed of an elevated live fuel layer, known as crown layer, and a layer of dead fuel located above ground, called the surface fuel layer. A critical process in crown fire behavior is flame transition from the fuel layer to the crown layer. This is because fire spread is typically most severe once it reaches the live crown fuels. Thus, our work aims to characterize flame transition from the surface fuel layer to the crown fuel layer as well as surface and crown fire spread. We designed a series of six wind tunnel scale experiments which focus on the effect of wind speed and surface-crown distance on flame transition and spread. Quantities measured were fuel mass consumption, temperature and flame geometry. Flame geometry parameters characterized included flame height, flame tilt and flame base length for both surface and crown fuel layers. Time to flame transition and surface to crown flame distance were also measured. Flame tilt angles between 30 and 60 degrees were observed for experiments where wind was added. Flame transition from surface fuel layer to crown fuel layer occurred within the first minute after ignition for most experiments. We partially attribute wind effects on flame transition and spread to changes in flame geometry which enhance radiative heat transfer to unburned fuel and thermal energy supply via convective heat transfer.
Application of Computer Vision in Multi-Scale Flow Visualization Using Granular Light Patterns, AMIRHESSAM AMINFAR1*, NAMI DAVOODZADEH1, DAVID RWEISE2, GUILLERMO AGUILAR1 and MARKO PRINCEVAC11 (1Department of Mechanical Engineering, University of California Riverside, Riverside, CA 92521, aamin006@ucr.edu; ndavo001@ucr.edu; gaguil@hotmail.com; marko@engr.ucr.edu; 2 USDA Forest Service, Pacific Southwest Research Station, Forest Fire Laboratory, Riverside, CA 92521, USA, dweise@fs.fed.us)

Flow imagery is one of the primary methods for understanding flow behavior. The classical approach involves adding foreign matter to visualize the flow. Flow can be visualized by processing the raw images using computer algorithms with no addition of particles or smoke. In this project, computer-based flow visualization approach was used to delineate different scales such as the flow of convective hot air around a flame and the blood flow inside a mouse’s brain. Granular patterns of dark and bright spots were generated and imposed on the flow fields. For the fire experiments, the granular pattern was generated by illuminating a printed noise pattern using white non-coherent light. The hot air around the flame caused fluctuations in air density affecting the air’s refractive index, distorting the background image. By comparing distorted and undistorted images using optical flow algorithms, the flow field was visualized. To visualize the flow inside a mouse’s brain, the granular pattern was generated by using a laser beam. This granular pattern, produced by random interference effects in laser light, creates a pattern consisting of dark and bright spots. When the speckle pattern is generated on blood vessels, the flow of blood causes fluctuations in this pattern leading to a blurriness associated with the blood vessel. This blurriness is later processed using computer algorithms which can be correlated to the motion of the red blood cells. This presentation will describe flow visualization of the wind-blown flame and blood in vessels under a mouse’s cranial bone.

Multiple Solutions for Laminar Flow through a Channel Having Porous Walls with Different Permeability, HONGXIA GUO1, CHANGFENG GUI*, PING LIN1,3 and MINGFENG ZHAO4 (1Department of Applied Mathematics, University of Science and Technology, Beijing, 100083, China, g1214619441@163.com; 2Department of Mathematics, University of Texas at San Antonio, San Antonio, TX 78249, USA, changfeng.gui@utsa.edu; 3Division of Mathematics, University of Dundee, Dundee, DD1 4HN, United Kingdom, plin@maths.dundee.ac.uk; 4Center for Applied Mathematics, Tianjin University, Tianjin, 300072, China, mingfeng.zhao@tju.edu.cn).

The existence and multiplicity of similarity solutions for the steady, incompressible and fully developed laminar flow in a uniformly porous channel with two permeable walls that the upper wall is with injection and the lower with suction is investigated. It is shown that there are three solutions identified as type I, type II and type III for the channel flow which agree well with the numerical results obtained by the collocation method. The numerical results suggest that a single solution is found for the Reynolds number $0 \leq R < 14.10$ and two additional solutions appear for $R > 14.10$. The corresponding asymptotic solution for each of the multiple solutions is constructed by the method of boundary layer correction or matched asymptotic expansion for the most difficult high Reynolds number case. Asymptotic solutions are all verified by their corresponding numerical solutions.