Book of Abstracts

36th Dayton-Cincinnati Aerospace Sciences Symposium

1 March 2011
Direct Calculation of Unsteady Entropy Generation Rate for Engineering Applications

Doty, John H, University of Dayton, Dayton, OH
Camberos, Jose A, Air Force Research Laboratory/Air Vehicles, WPAFB, OH
Yerkes, Kirk L, Air Force Research Laboratory, WPAFB, OH

A new formulation is presented for calculation of unsteady entropy generation rate for thermophysical systems. This formulation has far-reaching implications for analysis and design of all systems. It has long been considered that direct calculation of entropy generation rate was impossible due to the insufficiency of independent information for simultaneous determination of both unsteady entropy change and entropy generation rate. The formulation presented herein presents a unique approach to solving this dilemma. Recognizing that entropy is a state variable, the entropy is re-cast in terms of other, independent, state properties via the state postulate of thermodynamics. Using the chain rule of multivariable calculus, the rate of change of entropy is represented in terms of other known state derivatives that provide independent closure for direct calculation of entropy generation rate knowing the rate of change of other thermodynamic properties. The formulation is presented for a heat exchanger as a means of demonstrating its applicability for a dynamic, nonequilibrium system. The concept is extensible to other physical systems as long as the state postulate is valid and the state derivatives are known. The application of this formulation allows for the path-dependent entropy generation rate to be directly calculated. Therefore, truer estimates of system performance for modeling and simulation efforts are possible because the 2nd Law of Thermodynamics is explicitly invoked, enforcing feasibility, directionality, and spontaneity of system interactions.

Development of Accurate Nonideal Air Property Relations Using Design of Experiments

Doty, John H, University of Dayton, Dayton, OH
Camberos, Jose A, Air Force Research Laboratory, WPAFB, OH
Yerkes, Kirk L, Air Force Research Laboratory, WPAFB, OH
Wolff, James M, Air Force Research Laboratory, WPAFB, OH

Simple equations of state, such as the ideal gas equation of state, are often used in modeling and simulation applications. The use of these simpler equations introduces unknown and unsystematic errors that vary in magnitude throughout the region of application. As such, when attempting to validate simulation results of models with other models and/or experimental results, there is minimal knowledge of how large the errors are or, perhaps more importantly, where the errors originated from. Additionally, the secondary influence of pressure is often ignored in simpler models, relying solely upon the temperature dependence. In this investigation, a statistical approach is presented for the development of entropy and compressibility relations for nonideal air using both temperature and pressure influences on the properties. Errors in the models are known and quantified as a function of the temperature and pressure. The nonideality of air is quantified over a large range of temperature and pressure using the compressibility factor and visualized by means of a statistical response surface. The nonsystematic errors in the model are presented in order to quantify the behavior of the error as a function of temperature and pressure.

Employing a similar statistically-based methodology, an accurate regression equation is developed for the entropy of nonideal air. The temperature range selected was 200 K to 2000 K and the pressure range used was 10 kPa to 2e6 kPa. The regression equation was developed using a 5th order accurate optimal response surface model. Comparison of the surrogate model for entropy with validated results from Engineering Equation Solver indicate that the
A surrogate model has an overall mean absolute deviation of less than 0.5% over the entire pressure and temperature range studied. The surrogate model accepts both temperature and pressure as inputs and returns the property value of interest as well as the error.

The methodology used to develop the entropy surrogate model has several advantages over standard engineering property evaluations. First, the error in the model is significantly reduced by employing nonideal gas properties for the model source data. Second, the error is quantified as a function of the temperature and pressure, providing knowledge of the nonsystematic errors in the model. Third, the application of the model is valid over an extremely large range of operating conditions, making it valuable for many aerospace applications. Fourth, the model may be used by an inexperienced user in any software application with full knowledge of the validity of the results. Fifth, the model accurately and simultaneously captures the influence of both pressure and temperature on the property of interest. Lastly, the methodology is easily leveraged for development of other surrogate models of relevant properties, such as enthalpy, specific heat, viscosity, etc.

**High-Order Numerical Simulations of a Corrugated Airfoil**

*Barnes, Caleb J., Wright State University, Dayton, OH*  
*Visbal, Miguel R., Computational Aero-Physics/Air Force Research Laboratory, Wright-Patterson AFB, OH*

A number of experimental and numerical investigations have been conducted on a cross-section from a bio-inspired corrugated airfoil taken from the forewing of the dragonfly Aeshna Cyanea. Experimental studies have shown favorable aerodynamics such as delayed stall at high angles of attack and increased lift for high chord Reynolds numbers (10^4 < Re < 10^5). Additionally, the corrugated shape greatly increases the spanwise rigidity of the airfoil. Previous numerical investigations have been limited to low order simulations which were performed at low chord Reynolds numbers (10^2 < Re < 10^3) as more realistic values for micro air vehicle flight. Here, two-dimensional numerical simulations have been performed at various angles of attack using a 6th order compact difference scheme with a high-order filter. Computations were conducted at Re = 1000 with angles of attack ranging from 0° to 40°. Lift and drag coefficients were compared to previous simulations, experiments and those of a flat plate and the vortex structures were studied. The 2D computations have been performed in order to observe differences between the low and high-order solutions and as a first step towards conducting a series of 3D high-fidelity LES investigations.

**Instability and Transition for HIFiRE-5 in a Quiet Hypersonic Wind Tunnel**

*Borg, Matthew P, Booz Allen Hamilton, WPAFB, OH*  
*Kimmel, Roger L., Air Force Research Laboratory, WPAFB, OH*  
*Stanfield, Scott A., Spectral Energies, WPAFB, OH*

Experiments were conducted on the Hypersonic International Flight Research Experimentation-5 (HIFiRE-5) vehicle geometry in a quiet Mach-6 wind tunnel for both low and conventional freestream noise levels. The effect of freestream noise on natural transition on the windward surface and roughness-induced transition on the leading edges was investigated. Kapton tape of varying thicknesses was used as three dimensional isolated roughness elements and as two-dimensional strip roughnesses. Off-centerline crossflow transition was observed under quiet conditions and possible traveling crossflow waves were observed. As expected, elevated levels of freestream noise served to decrease the critical roughness height for quasi two-dimensional and three-dimensional roughness. Surprisingly, preliminary results show no effect of tunnel noise on two-dimensional tape-strip roughness-induced transition.
Two Phase Flow Characterization Experimental Development

Willebrand, Brian J, UES - AFRL/RZPS, Dayton, OH
Kreitzer, Paul J, UES - AFRL/RZPS, Dayton, OH

The abstract is still waiting to pass clearance checks. It will take two weeks to complete these checks. Since it was submitted yesterday, January 12th, it will not be returned until January 26th. Since this date is past both the priority and the final abstract due dates, it was suggested by Ray Kolonay, Dayton Cincinnati Aerospace Science Symposium general chair, that this be submitted in its place. The title of the presentation is Two Phase Flow Characterization Experimental Development. This has been submitted in hope of reserving a spot in the March 1st conference with plans of submitting the final abstract as soon as possible.
Session 2: Combustion I

Chair: Paul Litke, AFRL/RZ

Detonation Propagation Through Ducts in a Pulsed Detonation Engine

Nielsen, Jeff M, Air Force Institute of Technology, Dayton, OH
King, Paul I, Air Force Institute of Technology, Dayton, OH
Stevens, Christopher, Air Force Institute of Technology, Dayton, OH
Schauer, Frederick R, Air Force Research Laboratory, Dayton, OH

A study of configurations to allow a consistent and predictable transition of a detonation from one detonation tube to another is presented. Development of a continuously operating pulsed detonation engine (PDE) without a high energy ignition system or a deflagration-to-detonation transition (DDT) device will increase engine efficiency, reduce cost, improve performance, and reduce weight. The intent of this study was to minimize energy losses of a detonation wave in order to directly initiate another detonation wave. Detonation tube fill fraction, purge fraction, equivalence ratio, cross-over length and cross-over geometry were varied to determine their effect on direct initiation via a cross-over tube. Velocities at or above the upper Chapman-Jouguet (C-J) velocity point are desired and considered successful detonations. It was found that a cross-over tube with a “U” shaped geometry and a width at least 75% of the initiated tube’s width provided the best conditions for direct initiation.

Characterization of Pressure Rise Across a Continuous Detonation Combustor

Russo, Rachel M, AFIT, WPAFB, OH
King, Paul I, AFIT, WPAFB, OH
Schauer, Frederick, AFRL, WPAFB, OH
Thomas, Levi, AFRL, WPAFB, OH

For a continuous detonation engine, propellants enter the combustion chamber from one end and flow axially through the chamber. A detonation is initiated tangential to the flow of propellants at the base of the chamber creating a transverse detonation wave. This experiment is performed in a combustion chamber situated in a test rig on loan from a Pratt and Whitney continuous detonation engine. This particular engine was used in a proof of concept test by Pratt and Whitney to prove that detonation waves will propagate around the annulus of a combustor under the correct conditions. From preliminary, unpublished data, Pratt and Whitney were able to demonstrate the existence of a tangential detonation for an ethylene-oxygen mixture. For the purpose of this paper, the combustor will be run with hydrogen and air as the propellants.

The main subject of this experiment is the characterization of the pressure variation along the length of the combustor. From this, the thermodynamic efficiency of a rotating detonation engine may be obtained. The pressures that will be examined are the feed pressure just prior to the propellants entering the combustor, the pressure of the unburned propellants just after they enter the combustor, the high-pressure region across the detonation front corresponding to the Chapman-Jouguet condition, the pressure just aft of the height of the detonation wave, and the pressure just prior to the exit of the combustor. To accomplish this, several pressure transducers will be placed at key locations along the length of the combustor.

To date, only cold flow tests at the Air Force Research Laboratories have been performed using helium and air as the propellants.
Point-to-plane Pulsed Discharge Initiated Flame Structure Modification in Propane/Air

Schmidt, Jacob, Spectral Energies, LLC., Dayton, OH
Ganguly, Biswa, AFRL/RZPE, Wright-Patterson AFB, OH

The effect of a point-to-plane, pulsed discharge on a propane/air flame has been investigated by phase-locked, simultaneous measurements of the change in gas temperature, OH-planar Laser-Induced-Fluorescence (PLIF), and spatially averaged OH-emission intensity from the pulsed plasma as well as from the flame. Phase-locked simultaneous measurement of gas temperature through spontaneous Raman scattering, chemiluminescence intensity and/or OH PLIF with the variation of pulsed plasma energy and plasma generation location with respect to the flame holder and flame reaction zone have been performed. A fast rise time (15 ns) and slower rise time (150 ns) voltage pulasers are used to produce OH radical densities 50% greater than the ambient flame produced OH radicals in both lean and rich pre-mixed flames. The excess OH radical densities were found to decay with e-folding time constants greater than 100 μs in the burnt gas regions where gas temperatures were greater than 1000 K. The flame perturbation was dependent on the pulse repetition rates and the pulse rise time for similar energy deposition per pulse.

Air Force Independent Research and Development Program Overview

Andrews, Kimberly T, Air Force Research Laboratory, Industrial Partnering Branch, Dayton, OH

This brief will provide an overview of the Air Force (AF) Independent Research and Development (IR&D) Program, in which the mission is to identify industry IR&D technology projects that have the potential to contribute to the technical solution of AF weapon system capability needs and hence could affect out year AF investment strategies.

The basic concepts of the IR&D interaction process consists of three steps. In Step 1, the Air Force provides industry with a coordinated, structured and systematic presentation of AF needs. At this meeting, a series of presentations to industry is given in a plenary session. The AF Product Centers’ Requirements or Plans and Programs organizations present a comprehensive set of needs that are generally rank ordered by priority/need time frame. The Air Force Research Laboratory (AFRL) Subject Matter Experts (SMEs) also brief an overview of the projects they lead that are addressing these needs. They also present the technical gaps that remain. These gaps provide an opportunity for industry to subsequently brief their IR&D projects that may address these gaps.

Step 2 of the IR&D interaction process is where the face to face, Government- to-Industry SME IR&D interaction begins. Step 2 provides Industry the opportunity, based on the message in Step 1, to focus a presentation of their IR&D portfolio, both existing and planned, to meet those needs or bridge the identified technology gaps.

This follow on interaction is the aim of the Step 3 process. As necessary, a workshop may be held to formalize the Step 3 process. This AF-industry SME interaction at the detailed level presents the opportunity for the AF and contractors to adjust their out-year investment strategies to reflect the collaboration that results from the Step 1 and Step 2 IR&D interaction process.
The University of Dayton Advance Rocket Team (UDART) conducted research using powdered food grade green algae as a modifier to a simple solid rocket propellant. UDART uses sugar and potassium nitrate as a simple low cost propellant. Various mixtures of sugar, potassium nitrate, and algae were used to determine how effective powdered green algae are in terms of performance. After several simple experiments, green algae was determined to be a poor substitute for sugar as a propellant fuel.
Session 3: Flight Dynamics

Chair: Haibo Dong, WSU

Insect Wing Dynamic Roll

Zeyghami, Samane, Mechanical and Material Engineering Department, WSU, Fairborn, OH
Dong, Haibo, Mechanical and Material Engineering Department, WSU, Fairborn, OH

In this work we showed that interaction of the insect wings with the sudden changes in flight condition or mode can affect the stability behavior of the insect. The idea comes from considering the fact that insect’s flapping wings have independent degrees of freedom from the body and any change in their dynamic balance can change their motion independently. Despite the fact that wings flapping time scales, mass and moment of inertia are small compare to the total system’s but any rapid change in forces applied to the wing can excite a tradeoff between the balance of total forces on the wing and its motion. So we expect some missing high frequency energy careers that can affect the distribution of the perturbation energy in the system. We used a quasi-steady model for the aerodynamic forces and a damped torsional spring model for the hinge’ moments to show how considering the dynamic behavior of the wing can change the dynamic behavior of the total system.

Flight Performance Characteristics of Self-Inflating Flexible Rogallo-type Wings with Applications to UAVs

Warchol, Matthew Lee, Air Force Research Laboratory, Wright-Patterson AFB, OH
Whitfield, Clifford A., The Ohio State University, Columbus, OH

In 1948 Francis Rogallo, a NASA engineer and his wife Gertrude invented a self-inflating wing they called the parawing, also known as the Rogallo wing. This flexible wing is composed of two partial conical surfaces with both cones pointing in the direction of flight. NASA considered this wing an alternative recovery system for the Gemini space capsule.

The exceptionally light weight and easily packable nature of the Rogallo wing offers advantages in Unmanned Air Vehicles (UAV) operations, specifically in Tier I - small "over-the-hill" and Tier III - high altitude flight above turbulent weather. The packable wing increases ground transport efficiency for both small and large UAVs; as well as a promising method of ground or in-flight launching and deployment for high altitude operations. However, self-inflating wing aerodynamic and stability and controllability performance characteristics for furthering vehicle designs and its applications is very limited.

The following, includes an experimentally systematic approach in determining self-inflating wing performance characteristics with design considerations of varying wing planform, inflating or billowing effect, and wing interior structural layout of a light-weight plastic wing material. Wind tunnel analysis were conducted at the The Ohio State University Aeronautical and Astronautical Research Laboratory 3’ x 5’ subsonic wind tunnel.

Experimental investigations included force measurements using a sting mounted six-component balance for the three translational forces and their corresponding axis moments. Wing lift distributions were determined using static pressure measurements along varying span and chord-wise locations.

Wing flight performance characteristics were gathered and analyzed with varying leading edge sweeps and self-inflating effects were quantified with a defined geometric billowing parameter. Various flight conditions were
investigated and experimentally based applications are discussed for implementing Rogallo-type wings in UAV design and flight operation considerations.

HIFiRE-1 Flight Test Results

Adamczak, David W, AIR FORCE RESEARCH LAB, Wright-Patterson AFB, OH
Kimmel, Roger L, AIR FORCE RESEARCH LAB, Wright-Patterson AFB, OH

The Hypersonic International Flight Research Experimentation (HIFiRE) program is a hypersonic flight test program executed by the Air Force Research Laboratory (AFRL) and the Australian Defence Science and Technology Organization (DSTO). Its purpose is to develop and validate technologies critical to the realization of next generation hypersonic aerospace systems. Candidate technology areas include, but are not limited to, propulsion, propulsion-airframe integration, aerodynamics and aerothermodynamics, high temperature materials and structures, thermal management strategies, guidance, navigation, and control, sensors, and weapon system components such as munitions, submunitions, avionics, and weapon system separation. The HIFiRE program consists of extensive ground tests and computation focused on specific technology areas for hypersonic flight. Each technology program culminates in a flight test.

HIFiRE-1 is the 1st experimental flight with the primary experiment focused on boundary layer transition. The corresponding sub-experiments are, in order of priority, turbulent separated shock boundary layer interaction (SBLI), and optical measurement of mass capture (OMC). HIFiRE 1 was launched at the Woomera Test Range on March 22 at 00:44:00 Zulu time. Telemetry data on the three channels was received for almost the entire flight with only a few small dropouts over the almost 495 seconds of observable flight. The flight does go below the horizon when telemetry is lost and then the vehicle continues into the ground. This effort summarizes the flight 1 launch in March 2010 and the flight trajectory reconstruction effort to determine the best estimated trajectory flown to aid in the interpretation of the experimental results.

Implementation of Fast-Responding Pressure Sensitive Paint to Rotating Machinery Applications

Forlines, Alan, Innovative Scientific Solutions, Inc (ISSI), Dayton, OH
Palluconi, Steve, Innovative Scientific Solutions, Inc. (ISSI), Dayton, OH
Gregory, James, The Ohio State University, Columbus, OH
Watkins, Neal, NASA Langley Research Center, Hampton, VA

The need for improved performance and efficiency of rotating machinery such as helicopter rotors, gas turbine engine compressor blades, and wind turbines requires a better understanding of their aerodynamics. One specific example of rotating machinery is a helicopter blade, which includes unsteady pressure oscillations caused by dynamic stall, blades in forward flight, blade-vortex interaction, and blade warping due to fluid/structure interactions. These unsteady rotorcraft blade aerodynamics must be interrogated experimentally and computationally to understand the underlying physics and improve designs. Experimental measurements of pressure on rotating machinery have always been challenging due to the cost and complexity associated with the installation of pressure taps. Slip ring electrical connections are commonly used, but limit the number of transducers to relatively few points as well as posing other mechanical challenges. Recently, ISSI has applied the technology of fast responding pressure sensitive paints (PSP) to this problem. The team from ISSI, Ohio State, and NASA Langley has developed new paint formulations and detection schemes that allow unsteady pressure measurements on rotating machinery to be acquired optically. The PSP measurements offer high-spatial resolution and bandwidth of up to 20-kHz, thus
allowing unsteady pressure events to be interrogated with high spatial resolution. Pressure data collected on a variety of rotorcraft experimental setups will be presented including data from a 2-meter helicopter model at NASA Langley and propeller data in a wind tunnel.

**Joined-Wing SensorCraft Gust Load Alleviation Testing**

*Martinez, Juan, AFRL, Fairborn, OH*

In completing key wind tunnel tests of a new unmanned air vehicle (UAV) design known as the Joined-Wing SensorCraft, AFRL engineers are a step closer to establishing the novel configuration as a cutting-edge foundation for the next generation of unmanned capabilities. Conducted as part of the Aerodynamic Efficiency Improvement (AEI) program, these tests—which involved a scale model of the craft—are among the many ongoing efforts to develop technologies enabling Alleviation the longer-range, more-fuel-efficient, and lighter-weight UAVs necessary for supporting the high-altitude intelligence, surveillance, and reconnaissance (ISR) operations of tomorrow. The new SensorCraft design features a unique, diamond-shaped wing structure that maximizes both lift and sensor coverage. The setup also incorporates Gust Load Alleviation (GLA), a dynamic controls technology that actively adjusts wing surfaces to reduce turbulence-induced stress. The capacity for such self-adjusting behavior means that aircraft wings, traditionally designed with high rigidity in order to withstand the normal stresses caused by wind gust, can be less inherently stiff. Further, the use of GLA facilitates lighter wing structures and, in turn, reduced fuel consumption and increased range.

The recent test series, performed in the National Aeronautics and Space Administration (NASA) Langley Research Center’s Transonic Dynamic Tunnel (TDT), involved three rounds of testing. In round one, the researchers established the model’s baseline functionality and modulated the system to ensure full control in the subsequent tests. Round two focused on the aerodynamic fitness of the design, with the team subjecting the model to smooth airflow (and observing it to be very effective at maintaining stable flight). For the final test, the model underwent exposure to turbulent airflow in order to validate the GLA wing control system. The results indicate that the GLA technology cut gust load in the wings by 50%, enabling a 25% weight reduction that will translate to a 3,500 lb weight savings in the full-size SensorCraft vehicle to be constructed.
Session 4: Experimental Methods I

Chair: Michael Ol, AFRL/RB

Two-Color Time-Domain Thermoreflectance with an Optical Parametric Oscillator

Gengler, Jamie J., Spectral Energies, LLC, Dayton, OH
Jones, John G., AFRL/RXBT, Wright-Patterson Air Force Base, OH
Roy, Sukesh, Spectral Energies, LLC, Dayton, OH
Gord, James R., AFRL/RZTC, Wright-Patterson Air Force Base, OH

A new experimental setup is described for performing two-color time-domain thermoreflectance (TDTR). The technique is a variation of traditional pump-probe spectroscopy based upon a femtosecond Ti:sapphire oscillator of fixed wavelength and an optical parametric oscillator (OPO) for the purpose of creating independently tunable wavelength. The OPO used is based upon collinear, quasi phase matched interaction in a periodically poled crystal. The input beam is converted into a signal and idler beam with the wavelength relationship 1/input = 1/signal + 1/idler. The input beam is required to be in the range of 775 – 830 nm. The output signal beam (the idler is considered a by-product) is cavity length tunable in the infrared (IR) range of 1000 – 1600 nm. The process is synchronous and jitter free for generating trains of nearly transform limited femtosecond pulses. The OPO used here also has the feature of second harmonic generation for the signal beam with a temperature tuned lithium triborate (LiB3O5, or LBO) crystal. The output in this configuration is in the range 505 – 750 nm. Piezo-actuated stabilization of the cavity length (OPO wavelength output) is achieved through a feedback loop involving an external spectrometer. In this work, the OPO was operated with two different settings: 1) input / output wavelengths of 788 nm / 700 nm, respectively, for use with aluminum thin films, and 2) input / output wavelengths of 780 nm / 600 nm, respectively, for use with copper thin films. Changing between these settings can be attained within 1 hour. The OPO is also slowly purged with dry N2 gas to keep the optics clean, prevent strong IR absorption from water, and to promote the lifetime of the hygroscopic LBO crystal. The two-color TDTR method offers the advantages of 1) spectrally filtering diffusely scattered pump light from reaching the detector, and 2) thermoreflectance signal generation from different metal thin films. Demonstrated results include measuring an aluminum-graphite interfacial conductance of 28 MW m-2 K-1 and two orders of magnitude calibration of thermal conductivity measurements using copper as a thermoreflectance transducer.

"Fuzzy" Fiber Strain Sensor for Structural Health Monitoring

Sebastian, James R, University of Dayton Research Institute, Dayton, OH
Schehl, Norman D., University of Dayton Research Institute, Dayton, OH

Fibers used in composite materials can be coated with carbon nanotubes in a configuration where the nanotubes grow radially away from the fiber surface. In this configuration, the fiber takes on a fuzzy appearance. The network of nanotubes acts as a somewhat conductive coating. This research was a preliminary exploration of fuzzy fiber response to mechanical stimulus, with an eventual goal of incorporation into advanced composite structures, where they would be used to sense strain or crack growth. The resistance change of the fuzzy fibers to applied strain was measured in the following configurations: individual fiber, fiber tow, tow in matrix, and tow in laminated composite. Use as a strain sensor appears promising, where fuzzy fiber tows could be incorporated as a small percentage of a structure. In this arrangement they are non-parasitic, integral, load carrying members of the structure, with the potential to provide wide-area detection of damage.
Improving Safety Planning of Flight Test Maneuvers Through Specific Excess Power Estimation

Usry, Sidney J, Air Force Institute of Technology, Wright-Patterson AFB, OH

This paper explores pre-calculation of aircraft performance in terms of specific excess power (Ps), without a priori knowledge of flight test results. Simply speaking, Ps is the difference between the power required and the power available for a given aircraft state. Ps has been modeled in the past using rules-of-thumb, limited observed performance, and extrapolation. Often in flight tests an aircraft must dive to obtain the test condition. This can heighten risk to the aircraft and aircrew. It is desired to find Ps for an aircraft with only pre-flight data from the wind tunnel or CFD. First principles and governing equations of motion are used to define a method for direct calculation of Ps. Example calculations are compared to flight data for preliminary validation. Recommendations for further validation are presented as well as the integration between Ps and dive angles with time safety margin for risk tradeoff analysis. Ultimately, these techniques stand to provide earlier insight into the risk-return strategy in test program planning.

Manufacturing and Evaluation of a Biologically Inspired Engineered MAV Wing Compared to the Manduca Sexta Wing Under Simulated Flapping Conditions

DeLeón, Nathanial, AFIT, WPAFB, OH
O'Hara, Ryan, AFIT, AL
Murray, Jeremy, AFIT, AL
Palazotto, Anthony, AFIT, AL

In recent years, researchers have expressed a vested interest in the concepts surrounding flapping wing micro air vehicles (FWMAVs) that are capable of both range and complex maneuvering. Most research in this arena has found itself concentrated on topics such as flapping dynamics and the associated fluid–structure interactions inherent in the motion, however there still remains a myriad of questions concerning the structural qualities intrinsic to the wings themselves. Using nature as the template for design, FWMAV wings were constructed using carbon fiber and Kapton and tested under simplified flapping conditions by analyzing ‘frozen’ digital images of the deformed wing by methods of photogrammetry. This flapping motion was achieved via the design and construction of a flapper that emulates several of the kinematic features that can be seen in naturally occurring flyers. The response to this motion was then compared to the inspiring specimen’s wings, the North American Hawkmoth (Manduca Sexta), under the same flapping conditions in order to identify some of the key features that nature has deemed necessary for successful flight.

Wing/Wall Aerodynamic Interactions in Free Flying, Maneuvering MAVs

Geyman, Matthew Kenneth, University of Dayton, Dayton, OH
Altman, Aaron, University of Dayton, Dayton, OH

Particle Image Velocimetry (PIV) was used to study the aerodynamics of the interactions between maneuvering Micro Air Vehicles (MAVs) and building walls. MAVs must be capable of flying in complex terrain which often involves challenging flying conditions. One of the main missions for MAVs involves navigating through an urban environment and avoiding all of the obstructions. While previous research has investigated the effects of rotorcraft flying in close proximity to walls, little research has been devoted to the possibility of the wing/wall aerodynamic interactions for fixed wing vehicles. The research in this abstract focused on the aerodynamic interactions of a fixed wing MAV while flying and maneuvering in close proximity to a wall. The flight testing was conducted in the fully
enclosed Micro Air Vehicle Integration and Application Research Institute (μAVIARI) indoor flight test lab at Wright-Patterson Air Force Base. This lab contained 60 Vicon motion capture cameras used for real-time position and velocity tracking of vehicles. PIV was performed in the flight lab in order to capture data on the wing tip vortices while an off the shelf RC airplane, the Night Vapor, flew both straight and maneuvering flights along the length of an artificial wall structure.

The procedure to analyze the wing/wall aerodynamic interactions of the Night Vapor as it flies in close proximity to a wall involves several steps. First, the Vicon system in the flight lab tracked the Night Vapor as it flew close to the wall. The Vicon system then calculated velocity and acceleration values from position measurements. At the exact same time, PIV derived velocity field data was obtained while the aircraft flew close to the wall. Three different types of flight tests were performed and are listed below.

• First, the Night Vapor was flown straight and level close to the wall for a “baseline”.
• Second, low-speed maneuvering tests are performed close to the wall.
• Third, the aforementioned tests were repeated at relatively high speeds to complete the parametric study.

A full scale proof of concept test was successfully completed. Preliminary results proved that this test was possible and that wing tip vortex wall interactions could be produced and tracked while flying the Night Vapor close to the wall.

Two types of tests were performed during this proof of concept testing. These tests included straight and level flights along the wall as well as banked turning flights close to the wall. The raw PIV images of the wingtip vortices from the PCO 1600 double-shutter camera were passed through the dPIV 2.0 software to produce a velocity vector field. The straight and level flights revealed one vortex forming at the wingtip as the Night Vapor flew along the wall. The core of the vortex was plainly visible and a vector length cutoff filter helped to remove some of the noisy vectors at the vortex’s core. Some of the calculations that can be performed with the vector plots include calculating the vorticity and the velocity of the vortex propagation. The proof of concept testing revealed that the velocity of the vortex propagation grew as the vortex developed and that the vortex gravitated towards the wall and began to climb the wall. The velocity of the vortex propagation was investigated for one of the straight flights and had an initial value of 12.32 cm/s as the wingtip vortex first began to form. The average forward flight velocity of the Night Vapor was 2.55 m/s and was calculated through the Vicon system in the flight lab. When the 0.1232 m/s velocity of vortex propagation is normalized by the 2.55 m/s average flight velocity, a percentage of 4.83% is obtained. When the wingtip vortex was about to exit the field of view the velocity of the vortex propagation had grown to 18.53 cm/s. When this value is normalized by the average flight velocity of the Night Vapor, a percentage of 7.27 % is obtained. It was also shown that the vortex gravitated towards the wall as it developed because the vortex was formed while it was 15.28 cm. away from the wall but by the time the vortex was exiting the field of view, it was only 6.6 cm. away from the wall.

Images were also analyzed to show wing tip vortices forming while the Night Vapor was performing a turning maneuver while flying close to the wall. The cores of two separate vortices could clearly be seen. The first wing tip vortex originated from the wing tip closest to the wall, while the second wingtip vortex originated from the opposite wingtip during the turn. This second vortex propagated at the bottom of the field of view because the wing it originated from was lower than the opposite wing since the aircraft was performing a banked turning maneuver. Both of these vortices have clearly defined cores and lend themselves well to calculating the vorticity and subsequent circulation and vortex force as well as the velocity of the vortex propagation. The proof of concept testing for the turning tests revealed that the velocity of the vortex propagation dwindled as the vortex dissipated within the field of view and that the vortex gravitated towards the wall and began to climb the wall. The velocity of the vortex propagation calculated for one of the turning tests for the first vortex, closest to the wall, had a value of 18.06 cm/s when it was first formed.

The preliminary testing was completed over the course of only four days and led to the conclusion that the Night Vapor produces significant wingtip vortices that can be tracked through PIV. The research into wing/wall
aerodynamic interactions in free flying and maneuvering MAVs is significant because the results of these studies will be used to validate simplified potential flow models. These models will be used later to study the design of real-time closed loop control of maneuvering MAVs. Given the aerodynamic hazards inherent in flying in the urban environment, research is needed to help understand this complicated terrain.
Session 5: Thermal Management

Generic Aircraft Thermal Tip-to-Tail Modeling and Simulation

Roberts, Rory A, Wright State University, Dayton, OH
Eastbourn, Scott M, Wright State University, Dayton, OH
Maser, Adam C, Georgia Institute of Technology, Atlanta, GA

A system-level thermal management aircraft model has been developed in a multidisciplinary modeling and simulation environment. Individual subsystem models developed exclusively in MATLAB/Simulink, representing the vehicle dynamics, the propulsion, electrical power, and thermal systems, and associated controllers, are combined to investigate the thermal management issues of a typical long range strike platform. A thermal “Tip-to-Tail” model allows conceptual design trade studies of various subsystems and can quantify performance gains across the aircraft. The final result is an aircraft that is thermally optimized at the system-level, rather than at the subsystem-level. In addition, the model has been built without the aid of proprietary data, thereby allowing the distribution of the tool to a variety of conceptual design groups and researchers. Special attention has been paid to the development of transient component models within the thermal management systems, including the integrated power package, heat exchangers, fuel and oil pumps, and critical engine and thermal system interactions. As a result, the thermal and power challenges that face modern aircraft can be addressed, potentially increasing the performance capabilities of future tactical aircraft. Preliminary simulation results will be discussed with a specific focus on the thermal challenges encountered during reduced engine power mission segments.

High Capacity Thermal Energy Storage Material for Low Grade Heat

Schmidt, Joel E., Air Force Research Laboratory, AFRL/RXBT, Dayton, OH
Dudis, Douglas S., Air Force Research Laboratory, AFRL/RXBT, Dayton, OH

Phase change materials (PCMs) often have higher specific energy storage capacities at elevated temperatures. Thermal management systems capable of handling high heat fluxes in the temperature range from 20-50°C are necessary but lacking. State of the art PCMs in this temperature range are usually paraffin waxes with energy densities on the order of a few hundred kJ/kg. However, for applications where system weight and size are limited, such as on aircraft, it is necessary to improve this energy density by at least an order of magnitude. The compound ammonium carbamate, NH2COONH4, is a solid formed from the reaction of anhydrous ammonia and carbon dioxide which endothermically decomposes back to CO2 and NH3 in the temperature range 20-50°C with an enthalpy of decomposition of ~1,800 kJ/kg. Various methods to use this material for thermal management of low-grade, high-flux heat have been evaluated including: bare powder, thermally conductive carbon foams, thermally conductive metal foams, hydrocarbon based slurries, and a slurry in ethylene glycol. A slurry in ethylene glycol is a promising system medium for enhancing heat and mass transfer for thermal management. Progress on system property characterization will be presented.
Dynamic modeling of Vapor compression cycles using a finite-volume approach

Miller, Eric S, AFRL, Dayton, OH
Jog, Milind, University of Cincinnati, Cincinnati, OH
Patnaik, Soumya S., AFRL, Dayton, OH

Vapor compression refrigeration cycles (VCS) in aircraft thermal management systems (TMS) are being currently evaluated for increase in fuel efficiency and aircraft performance. Dynamic changes in heat loads are common in aircrafts and variable-speed compressors and electronic expansion valves are being considered to improve the VCS dynamic performance which can lead to highly coupled and non-linear dynamic response. VCSs have traditionally been designed for steady-state operating conditions; a practice that allows for systems to meet the maximum cooling demand, but not necessarily perform efficiently while responding to dynamic cooling demands and a better understanding of the dynamic response of VCS is required for the efficient use of VCS in aircraft TMS. In this talk we present our current effort in developing dynamic models of VCSs with primary focus on modeling evaporator and condenser components using a finite-volume modeling approach. We provide improvement on previous models by calculating spatial variations in fluid properties in evaporator and condenser components. Using Simulink© software, evaporator and condenser models are combined with existing variable-speed compressor and electronic expansion valve models to study the dynamics of the entire vapor cycle system. In future, these models will be used to test vapor cycle control schemes and appropriately size TMS components.

Molecular dynamics simulations of mechanical behavior of thermosetting polymers

Shenogina, Natalia, Wright State University, Dayton, OH
Tsige, Mesfin, University of Akron, Akron, OH
Patnaik, Soumya S., AFRL, Dayton, OH
Mukhopadhyay, Sharmila, Wright State University, Dayton, OH

We use atomistic molecular dynamics (MD) simulations to predict the mechanical and thermal properties of thermosetting polymers. In this talk we discuss the modeling approach to construct realistic and stress free all atom models of densely cross-linked polymer matrices based on amine curing agents and epoxy resins. The main focus of our study is on the properties of DGEBA/DETDA epoxy system. A series of atomistic simulations were carried out to examine and predict the degree of curing dependence of the density, glass transition temperature and coefficient of thermal expansion. We also present our recent results on the variation of the elastic constants of the thermosets on the degree of curing.
Improved Collision Modeling for Direct Simulation Monte Carlo Methods

Summers, Sarah E, Air Force Institute of Technology, Wright-Patterson AFB, OH
Greendyke, Robert, Air Force Institute of Technology, Wright-Patterson AFB, OH

The key to Direct Simulation Monte Carlo (DSMC) is decoupling of particle motion and particle collisions. Particles within each cell are randomly chosen as collision partners, and the collision is then accepted or rejected by comparing a random number to the ratio \( \frac{\sigma_T c_r}{\langle \sigma_T c_r \rangle_{\text{max}}} \) where \( \sigma_T \) is the collision cross-section and \( c_r \) is the relative velocity between the two particles. The denominator is the maximum value for the product within the cell. After the collision is accepted and takes place, the particles move based on their post collision velocities. The distance moved by the particles is a function of the time step used in the program, which is small in order to decouple the motion and collisions. In the Smoothed Accept/Reject (SAR) algorithm, the accept/reject criteria is altered: rather than a binary function of rejection or acceptance, collisions can be partially accepted with a linear weighting between zero and one. The accept/reject criteria is now: \( \frac{\sigma_T c_r}{\langle \sigma_T c_r \rangle_{\text{max}}} \pm \varepsilon/2 \) where \( \varepsilon \) is a user-determined percentage of the ratio \( \frac{\sigma_T c_r}{\langle \sigma_T c_r \rangle_{\text{max}}} \). The weighting is used in sampling the particles in order to calculate the macroscopic flowfield parameters using: \( \bar{Q} = \frac{\sum_i w_i Q_i}{\sum_i w_i} \) where \( Q \) is the cell averaged property, \( w_i \) is the individual particle weighting, and \( Q_i \) is the individual particle property. Previous work included comparisons to experimental data using inverse shock thickness, the results of which implied a relationship for the appropriate value of \( \varepsilon \) of \( \varepsilon = f(M) \), and this is explored in the present work. Additionally, 2-dimensional experimental data is used for comparison between SAR and the original algorithm. Speed distributions of the particles are examined for all algorithms and compared to the speed distribution as derived from the Maxwell Distribution to determine the effect of the SAR algorithm.

Optimal Heavy Fuel Direct Injection Spark Ignition in a Rotary Engine

Benthara Wadumesthrige, Asela A, Wright State University, Dayton, OH
Dong, Haibo, Wright State University, Dayton, OH
Minkiewicz, Gregory R, Wright-Patterson Air Force Base, OH

The main objective of this computational study is to investigate the optimum injection and spark parameters for the direct injection spark ignition (DISI) Wankel rotary engine using diesel fuel. Compared to reciprocating type engines, rotary engines are mechanically simpler, vibrate less, have higher power to weight ratio and achieve better performance at high speeds. Due to the inherent low fuel efficiency of rotary engines and increasing gas prices, application of the rotary engine in conventional automobiles is decreasing. This project seeks to introduce DISI technology to the rotary engine, thus increasing the fuel efficiency and allowing it to be another efficient power source option for aero and automotive applications. During this computational study, less number of parameters was used because engine geometry, injectors and sparks used in the simulation match with the readily available rotary engines, direct injectors and spark plugs, the number of parameters for the optimization process is reasonably small. Full factorial experimental design was used to estimate the sensitivity of different combinations of parameters. The feasibility of multiple injections was also studied by means of their power outputs and fuel efficiencies. Finally from this study, applicability of DISI technology to the rotary engine for getting better performance for different road conditions can be discussed.
Computational study of Flow physics and combustion process in DISI engine

Moda, Suniludayasimha, Wright state university, Fairborn, OH
Dong, Haibo, Wright state university, Fairborn, OH

A computational analysis on a rotary engine is conducted to study the air charge motion in the rotary engine. A Wankel rotary engine model is developed in computational domain for this study. The model is developed in a Solidworks which captures all the important geometry information. This model is cleaned up and meshed using Gambit and the mesh is exported to Fluent. The exported mesh which is a static mesh is combined with user defined function, dynamic mesh options available in Fluent to perform a complete dynamic mesh motion. This mesh motion replicates the real engine motion and provides all the attributes that are necessary in an engine process. It provides a realistic air charge motion inside the computational model coupled with inlet and outlet air charge motion. This realistic environment helps to test the real engine conditions in a computational domain. Using the mesh motion achieved the various simulations are performed. In this study the flow physics of air inside the chamber is visualized and some of the important aspects which aides fuel air mixture formation like air swirl and tumble are studied. The importance of the air charge in formation of fuel air mixture is demonstrated. The important aspects of fuel combustion stages like mixture formation, vaporization and combustion are explained. This study of ignition delay times to incorporate heavy fuel in this engine is studied. The effects of various injection parameters like droplet size, number of orifice and injection time are studied with the help of outputs from the simulations. Droplet distribution study is carried out to study the vaporization time in the chamber and how the droplet size effects the vaporization. Combustion time for the simulations is calculated using the laminar flame speed. Study on unburned hydro carbons, available oxygen in the chamber and carbon-di-oxide produced are considered to study the combustion process.

Shock Initiation of Detonation on a Converging 2D Ramp

Stevens, Christopher A, Air Force Institute of Technology, WPAFB, OH
King, Paul I, Air Force Institute of Technology, WPAFB, OH
Schauer, Fred R., AFRL/RZTC, WPAFB, OH
Hoke, John L, ISSI, Dayton, OH

In the development of pre-detonators for pulse detonation engines, it is necessary to transition a detonation from the small diameter of the pre-detonator tube to the larger diameter of the PDE thrust tube. Detonations propagating in tubes with diameters less than 13 cell widths are known to decouple when encountering sudden expansion. Recent experiments and computational modeling have shown that secondary detonations may be initiated by reflecting the decoupled shock front off a converging ramp. In the model and experiments, a detonation diffracted over a 90° corner into a larger channel. The detonation wave decoupled into a leading shock wave and a trailing combustion front. Subsequently, the leading shock reflected off a 24° converging ramp creating a high pressure and temperature region. This region became the kernel for a secondary detonation front. The secondary detonation kernel resulted in continued detonation of the fuel/air mixture in the channel. In this prior work, the converging ramp extended until the channel height was the same as the initial height, but the detonation kernel formed some distance before the end of the ramp. Using the same ramp angle and initial step height, it is possible to devise a geometry which acts as a “diffuser” for detonations simply by truncating the ramp a short distance after the detonation kernel forms. In this work, the formation of the secondary detonation kernel was investigated. The origin of the kernel was measured using a combination of chemiluminescence and schlieren visualization. The spatial and temporal origin of the secondary detonation were found by extrapolating from the growth of the detonation over 40 μs. The kernel formed 15.4 mm below the edge of the step expansion, and 135 mm downstream. The kernel formed 8.9 μs after the primary detonation wave encountered the step. The measured wave speed of the secondary detonation was 2390 m/s indicating highly over driven propagation.
Simulation of thermal transport in microscale graphite foam for latent heat thermal energy storage applications

Hu, Xin, UES, Inc, Dayton, OH
Patnaik, Soumya S., AFRL, WPAFB, OH

Graphite foams with their high thermal conductivity and porous cellular structure have been gaining interest as thermal conductivity enhancer (TCE) for phase change materials (PCM) in transient thermal energy storage (TES) applications. In paraffin wax saturated graphite foam (paraffin-GF) systems, the high thermal conductivity of the foam provides the PCM (paraffin wax), which has high latent heat but poor thermal conductivity, an increase in the effective thermal conductivity. Compared to macroscale foams, the porous structure of micro-scale foam provides an increase in contact surface area with the PCM, further increasing the effective heat transfer rate and improving the transient response. In the present study, we use numerical simulations to systematically investigate the thermal transport in paraffin-GF systems for a better understanding towards achieving optimum storage capacity and transient response of these TES systems.

Earlier studies of heat transfer of PCM-foam systems have been based on macroscopic volume-average approaches which do not take into account the detail microstructure of the foam. These studies have included either one-temperature (local thermal equilibrium) or two-temperature models (local thermal non-equilibrium). In the present study, we have carried out microstructure based direct simulation of heat transfer in PCM-GF systems. A simplified open-cell foam structure based on a cubic unit cell and spherical pores is considered and simulations are carried out on single arrays with varying number of unit cells. The enthalpy-porosity method is used to solve the phase change problem and conjugated heat transfer is considered at the interface of solid foam and PCM. Two different boundary conditions, i.e., constant heat flux and constant temperature are considered. We also carried out simulations with one-temperature and two-temperature models with the effective thermal conductivity and effective heat transfer coefficient calculated from direct simulation for foam porosities from 0.75 to 0.99. Unlike the 1D temperature profile observed for simulations using the volume-average method, the direct simulations show a more complicated 3D temperature profile with the melt front spreading from the PCM-foam interface. The direct simulations also provided a more accurate estimate of the total volume of the melted PCM and potential for better prediction of the heat storage capacity and transient response.

Coupled Radiation-Gasdynamic Solution Methodologies

Martin, Christopher L, Air Force Institute of Technology, Wright-Patterson AFB, OH
Greendyke, Robert B., Air Force Institute of Technology, Wright-Patterson AFB, OH

In recent years, the United States Air Force has experience a renewed interest in hypersonic and transatmospheric platforms in support of the Operationally Responsive Space and Prompt Global Strike concepts of operation. The resurgence of design and analysis activities promoted by this interest has underscored the continuing need to improve upon existing methods and techniques utilized by the computational aerothermodynamic community. One such area for improvement involves the modeling of the nonequilibrium energy distribution which exists behind the strong bow shocks characteristic of reentry trajectories. It is incumbent upon those conducting analyses in this regime to consider the effects of thermal nonequilibrium on their results. Analyses pertaining to the transport of
Radiative energy are particularly sensitive to such effects, since the emission and absorption of radiation within the flowfield exhibit a complex dependence on both chemical composition and thermodynamic state. The results of the proposed investigation shall be an extension of recent work by the authors to characterize this dependence of the radiative behavior on the chemical composition and thermodynamic state of the flow, particularly as it concerns the selection of an appropriate model of the thermal nonequilibrium. We have sought to advance such a characterization by way of comparison. In this comparison, we have exchanged the popular two-temperature model—which has been integral to practically every major effort to the present to characterize the radiative environments generated by reentry bodies—with a multitemperature model. The calculation of the radiative intensity is made with the use of a high-fidelity radiation model in order to capture as much realism as possible in furthering a detailed characterization of the radiating flowfield. Previous results were obtained by performing the radiation calculation in an uncoupled manner. In the proposed investigation, we intend to perform a similar characterization utilizing a loosely-coupled radiation-gasdynamic solution methodology.

The structured package for radiation analysis (SPRADIAN) will be used to calculate radiation resultant from both of the above temperature models. SPRADIAN accounts for the most significant line and continuum radiation mechanisms for monatomic and diatomic species in the 11-species air model and excels many other radiation packages with respect to its line-by-line capability. In the authors’ previous investigation, obtaining the radiation solution practically amounted to a post-processing activity with respect to the flowfield and was performed only along the stagnation streamline. In order to couple SPRADIAN to the subject flow solver, it will be necessary to account for the various energy exchanges occurring between the radiative and flowfield solutions. The influence of the flowfield solution on the radiative solution occurs explicitly through the thermodynamic variables T, Tvib, s, Te and Ni. However, the reciprocal influence of the radiative solution upon the flowfield solution occurs through various source terms in the total and vibrational energy equations. In order to calculate these source terms, it will be necessary to develop a method whereby to evaluate the radiative transport occurring within the solution domain. The proposed solution method for the radiative transport equations is accomplished via a finite volume method.

The multitemperature model, by addition of the species-specific vibrational energy equations, effectually deconstrains the distribution of internal energy within the diatomic molecules. Further, it has the effect of deconstraining the vibrational and electron energy manifolds from one another. The implications of these two facts are far reaching since they affect important nonequilibrium processes such as chemical reactions and radiation. From the standpoint of the chemical reaction rates, this changes the nature of the important dissociation processes behind the bow shock and the availability of electrons to precipitate electron-impact ionization reactions. From the standpoint of radiation, this affects the electronic state populations of the various atomic species and the electronic-vibrational state populations of the diatomic species. These state populations relate directly to the emission or absorption of radiative energy and the overall transport of this energy within the radiation-gasdynamic solution domain.

In our previous work, we observed that the data obtained from the multitemperature thermal model show that considerable thermal nonequilibrium exists between vibrational energy modes of the constituent species. Also, the electron temperature is distinct from and much lower than vibrational temperatures at their respective peaks, which the two-temperature model assumes to be in thermal equilibrium. In addition, the electron temperature is found to equilibrate with T very slowly relative to the vibrational temperatures. Bound-bound, bound-free, and free-free radiation modes are all highly dependent on these electronic and vibrational temperatures, in a species-specific manner. Therefore, we might expect to see very different estimates of the radiation resulting from these two methods, which is clearly demonstrated in figures 4 and 5. Coupled results utilizing the multi-species, multi-temperature model show that radiative cooling provides a more realistic estimate of the observed in-flight radiative heat loads. In our presentation, we propose to present the results of implementing the finite-volume radiative transport scheme discussed above together with the multi-species, multi-temperature nonequilibrium flow solver and to compare this implementation of the radiative transport with the tangent slab method.
Conjugate Heat Transfer CFD Assessment of Flat Plate Film Cooling Experiments

Johnson, Jamie J, AFIT/ENY, Wright-Patterson AFB, OH
King, Paul I, AFIT/ENY, Wright-Patterson AFB, OH
Clark, John P, AFRL/RZTT, Wright-Patterson AFB, OH

Ongoing work is being conducted to create a greatly-improved pressure-side cooling array design for the high-pressure turbine Research Turbine Vane (RTV) geometry using various optimization techniques to change cooling array variables and value ranges. Flat plate experiments using infrared imaging are being used to evaluate the performance of intermediate designs. A flat plate model is evaluated through the use of novel conjugate heat transfer (CHT) computational fluid dynamics (CFD) at design freestream and film cooling hole blowing conditions. This flat plate model is meant to represent the pressure side cooling array of the RTV. The CHT solver uses high-density unstructured assessment which grids the hot flow, the cooling plenum, as well as each individual cooling hole, and solves the Fourier conduction equation for solid regions which are also meshed. It is a Reynolds-Averaged Navier Stokes (RANS) solver that uses the Wilcox k-ω turbulence model in the fluid domain near the surface of the plate. Computational domains are gridded for the cooling plenum fluid, cooling hole fluid, hot main stream flow, and the plate solid. Predicted surface temperature, overall effectiveness, and midspan heat flux is compared to experimental infrared flat plate data of the same scale.

Novel Heat Exchanger Fin Surface Design for Improved Condensate Management

Yu, Rong, Miami University, Oxford, OH
Sommers, Andrew D, Miami University, Oxford, OH

Novel Heat Exchanger Fin Surface Design for Improved Condensate Management

Rong Yu1 and A.D. Sommers2
1 Dept. of Chemical and Paper Engineering, Miami University, 64 Engineering Building, Oxford, OH 45056
2 Dept. of Mechanical and Manufacturing Engineering, Miami University, 56 Engineering Building, Oxford, OH 45056

Abstract

This project investigated the effectiveness of heat exchangers constructed using anisotropic, micro-patterned aluminum fins to more completely drain the condensate that forms on the heat transfer surface during normal operation with the aim of improving the thermal-hydraulic performance of the heat exchanger. This study presents and critically evaluates the behavior of condensate droplets on micro-grooved fins as well as the efficacy of full-scale heat exchangers constructed from these surfaces by measuring dynamic contact angles, critical inclination angles for sliding, dynamic dip testing data and dry/wet air-side pressure drop. The influences of the underlying microstructure which consisted of parallel grooves tens of microns in width and depth and the alkyl silane coating on droplet behavior were also studied. The results showed that the micro-grooved structure of the fin in combination with the alkyl silane coating increased the hydrophobicity of the surface and reduced the water retention of the heat exchanger by more than 27%. The new fin surface design was also shown to decrease the core pressure drop of the heat exchanger during wet operation from 9.3% to 52.7%. That is to say, this novel fin surface design has shown the ability to improve the condensate drainage behavior of heat exchangers used in air-conditioning applications.
The Efficacy of Transient Heat Transfer Experiments to Evaluate Unsteady Film Cooling

Rutledge, James L., US Air Force, WPAFB, OH
McCall, Jonathan F., US Air Force, WPAFB, OH

The transient heat transfer experiment is a popular way to simultaneously measure adiabatic effectiveness and heat transfer coefficient on a film cooled surface in steady flow. In such legacy transient experiments, a film cooled turbine component model initially at a known uniform temperature is suddenly exposed to flow conditions with a different temperature. Steady adiabatic wall temperature and heat transfer coefficient can then be determined using the surface temperature history and a simple 1-D semi-infinite solid model. Recent interest in the evaluation of unsteady film cooling performance has prompted analysis of the applicability of this experimental technique to unsteady film cooling. When the adiabatic wall temperature and heat transfer coefficient are unsteady, more analysis is required because this simple model no longer directly applies. It will be demonstrated that even average values with any utility cannot be obtained reliably with this legacy technique in unsteady flow.
Session 8: CFD Methods

Chair: Andrew Lofthouse, AFIT

RC-135 Aerodynamic Characterization

Chenery, Michael G, AFIT, Wright-Patterson AFB, OH
Lofthouse, Andrew J, AFIT, Wright-Patterson AFB, OH

Both the RC-135V/W Rivet Joint and the RC-135U Combat Sent aircraft are United State Air Force (USAF) electronics reconnaissance platforms. The Rivet Joint is the USAF’s standard airborne signals intelligence (SIGINT) gathering platform while the Combat Sent is designed to collect technical intelligence on adversary radar emitter systems. Both aircraft are extensively modified C-135’s characterized by protruding “cheeks” along the sides of the fuselage forward of the wings and the addition of numerous antennas along the top and bottom of the fuselage. The major distinguishing difference between the two variants is that the Rivet Joint has an elongated nose while the Combat Sent has a more standard nose with a protruding "chin" modification along the bottom of the nose. The modifications are intended to house on-board sensor suites capable of detection, identification and geolocation of signals throughout the electromagnetic spectrum.

The Rivet Joint variant has recently experienced structural damage to antennas and fuselage skin. Flight testing has revealed unsteady aerodynamic loading on certain antennas which is believed to be caused by vortices shed off of an exhaust louver on the top of the “cheeks”. The exhaust louver is part of a cooling system for the electronics package along the fuselage. The Combat Sent variant has this same modification, but has not experienced the same problems with damage to antennas. I intend to perform an aerodynamic characterization of both variants with a focus on characterizing the difference in air flow between the Rivet Joint and Combat Sent variants. This research is in support of the Air Force Research Lab Aeronautical Systems Center (AFRL/RBAT).

In order to perform this analysis, I will be applying Computational Fluid Dynamics (CFD) using the Kestrel solver. At this time, I will not be presenting any final results, however I will be focusing on presenting the problem, the process through which I will approach the problem as well as some preliminary results.

Chemistry Validation in the Unified Flow Solver

Humphrey, William C, AFIT, Dayton, OH
Lofthouse, Andrew, AFIT, Dayton, OH

Presenter: William Humphrey
Authors: William Humphrey, Maj Andrew Lofthouse
Dayton-Cincinnati Aerospace Sciences Symposium
Briefing Abstract

Most CFD packages out today use a combination of Euler and Navier-Stokes equations, which are also called continuum equations. However, the continuum equations can’t be used when in a rarified flow regime and rarified CFD solvers can be computationally expensive in high density flows. The Unified Flow Solver (UFS) program has been developed to solve for flows in the continuum-rarified transition region and be less expensive than the rarified solvers. UFS works by using both continuum equations with rarified flow equations. For this reason UFS is less expensive because it use the rarified equations only where necessary, which means, it can use the continuum equations everywhere else. UFS also has the ability to add the chemistry of the flow into the solution but as to date
this part of the code has not been fully validated. For this presentation though I will be talking about my thesis research, which will be to validate the chemistry code in UFS. As I have just started my research I will be talking about the process that I plan to use to complete my research. I will also be presenting some preliminary results that I have been able to gather. Finally I will discuss what I have learned about UFS since the start of my research.

CFD Characterization of Blockage with a Wind Tunnel Turret Model

Fievisohn, Robert T, AFRL/RBAT, WPAFB, OH

The Air Force Research Laboratory (AFRL) is interested in testing turret configurations at transonic speeds. Previous testing done in the Tri-sonic Gasdynamic Facility (TGF) found that for one turret configuration, a maximum Mach number of approximately 0.6 was achieved. Increases in power to the tunnel were unable to increase the speed. Numerical simulations using the Air Vehicles Unstructured Solver (AVUS) are used to determine what prevented the tunnel from running faster. Previous simulations only modeled turrets within the test section and were unable to capture the blockage effect that prevented the tunnel from operating at higher Mach numbers. Current efforts are taking into account the entire tunnel from the stagnation chamber to the exit of the diffuser with the goal of correctly capturing this effect. Results are used to characterize the blockage and determine if smaller turrets will let the tunnel run faster and by how much.

Computational Investigation of a Low Speed Cavity Flow

Rinderle, Jessica L, Air Force Research Lab, Dayton, OH

Computational Investigation of a Low Speed Cavity Flow
By Jessica L. Rinderle

CFD simulations of incompressible flow over a 3-dimensional shallow cavity are compared with a correlating wind tunnel experiment. Cavity flow is inherent to the design of structures such as bomb bays and landing gear wells. The experimental and CFD research investigate the effect of various rod configurations placed at the leading edge of the cavity toward the reduction of aero acoustics of the cavity and buffeting effects, which can lead to structural failure. A wind tunnel test of the cavity was conducted at 135 miles per hour (mach 0.176) with a rod diameter of 3mm which was positioned 3mm from the tunnel wall. Pressure taps were placed along the cavity centerline. Pressure measurements are compared to CFD data of the cavity. The pressures match inside the cavity space with about a 0.06-0.1% error, and both experiment and computation show similar trends.

2-D Transient CFD Model of an Isolator Shock Train in a Scramjet Engine

Hoeger, Troy C., AFIT/ENY, Dayton, OH
King, Paul I., AFIT/ENY, Dayton, OH
Donbar, Jeffrey M., AFRL/RZAS, Dayton, OH
Cox-Stouffer, Susan K., AFRL/RZAS, Dayton, OH

A scramjet has four main sections: the inlet, the isolator, the combustor, and the exit nozzle. The isolator’s primary function is to contain the pre-combustion shock train (PCST) and prevent interaction between the combustor and the inlet. If the PCST were to reach the inlet, there would be a loss of air-mass capture, increased pressure and thermal loads, increased drag, and decreased thrust. This phenomenon is known as an unstart and could result in mission and/or vehicle failure. The location of the leading edge of the PCST will tell how close the scramjet is to an unstart, so understanding and preventing an unstart through the detection and control of the shock train in the isolator is
critical to the advancement of scramjets. A useful approach to help solve this problem is Computational Fluid Dynamics (CFD). CFD has already been used to complement the scramjet flowpath design process. It is cost-effective, reduces risks to pilots and vehicles, and enables detailed analysis of the flow field. All known attempts to apply CFD to scramjet isolators have considered steady-state performance, which is much simpler to model than transient phenomena. Though most flow solvers are capable of modeling time-dependent behavior, a transient approach is in several ways more complicated than steady-state analysis. First, many flow solvers are not equipped for time-dependent boundary conditions, without which transient studies are very limited. Researchers pursuing topics in this area may need to obtain flow-solver source code and implement modifications to reflect the intended fluctuating boundary conditions. Second, researchers considering steady-state CFD are interested in the converged solution, not the process path, and for that reason frequently initialize the solution in manners that are not reflective of actual engine processes. These novel initializations greatly simplify simulation requirements in ways that are not available to those intent upon studying transient phenomena. Even a researcher well experienced in steady-state CFD may find himself or herself in unfamiliar territory, forced to learn new, more stringent approaches. Finally, the addition of time as a simulation dimension vastly increases the amount of data to be analyzed and complicates data analysis.

The purpose of the present work is to develop and test boundary conditions, simulation methods, and analysis techniques for the study of transient flows in scramjet isolators. Though the study of actual engines and/or laboratory models will require three-dimensional analysis, the present work considers only two-dimensional simulations, which are simpler and quicker to perform and analyze. Three-dimensional analysis will be addressed in subsequent research.

The simulations for the CFD model will be executed using VULCAN, a cell-centered, structured-grid, finite-volume, multi-block Navier-Stokes flow solver code maintained and distributed by the Hypersonic Air Breathing Propulsion Branch of the NASA Langley Research Center. VULCAN solves equations governing inviscid and viscous flow of a calorically perfect gas or of an arbitrary mixture of thermally perfect gases undergoing nonequilibrium chemical reactions. VULCAN can simulate 2-D, axisymmetric, or 3-D flows. The inviscid fluxes are computed using the MUSCL (Monotone Upstream Centered Scheme for Conservation Laws) scheme, with either Roe's approximate Riemann solver or Edwards' LDFSS (Low Dissipation Flux Splitting Scheme). Viscous fluxes are based on a central difference scheme, with options to include cross-derivative terms for accuracy in highly three-dimensional flows or neglect them for computational efficiency. A variety of implicit and explicit time-integration strategies are available for advancing the solution in time, including a pseudo-time iterative approach for parabolic flows and time-accurate schemes such as a multi-stage Runge-Kutta scheme and several approximate factorization methods. Additionally, convergence can be accelerated using techniques based on the concept of local time-stepping. Various types of one-equation and two-equation turbulence models are available. The code also contains full multi-grid capabilities, allowing rapid convergence for steady-state problems. MPI (Message Passing Interface) routines using an SPMD (Single Program, Multiple Data) paradigm take advantage of the parallelism of modern supercomputers, and arbitrary block-to-block connectivity allows the easy removal or addition of grid-points at zonal interfaces.

In addition, one major advantage of VULCAN is that its source-code is available for inspection and modification by the software users, subject to constraints imposed by the user agreement. Though not every approach for the application of time-dependent boundary conditions requires software modification, the most straight-forward does. In this approach, VULCAN’s constant-backpressure subsonic-outflow boundary condition will be modified to calculate backpressure as a function of iteration or to read the backpressure from a file at each iteration. Another approach would be to model the deflection of the back flaps used in the AFIT wind tunnel to impose the back pressure. This deflection could be modeled internally and smoothly, by modifying VULCAN to adjust the grid as a function of elapsed time or iteration number, or externally and discretely, by creating grids that reflect several distinct degrees of deflection and by manually exchanging them in the simulation. The preferred process is unknown at this time and will be the most significant finding of the present research.

One preliminary phase of the research is complete at this time. In that phase, time-dependent boundary conditions were limited to the sudden application of constant backpressure (switching boundary conditions from extrapolation...
to constant backpressure), and the final position and length of the resultant shock trains were the primary focus of analysis. Lessons learned pertain to the unique grid and procedure requirements of time-dependent simulations and to the limited ability of 2D isolators to mimic 3D behavior. The present research will advance the work of the previous phase by implementing truly time-dependent boundary conditions and by developing procedures for analysis of time-dependent data. Though rigorous comparison with experiment must await three-dimensional simulations, the techniques and procedures applied here will lay the foundation for important developments in unsteady isolator analysis.

Disclaimer: The views expressed in this article are those of the authors and do not reflect the official policy or position of the United States Air Force, Department of Defense, or the U.S. Government.
Session 9: Turbomachinery I

Chair: Waruna Kulatilaka, AFRL/RZ

Time-Resolved Inlet and Exit Measurements of a Turbine Driven by Pulsed Detonations

ROUSER, KURT P, Air Force Institute of Technology, WPAFB, OH
KING, PAUL I, Air Force Institute of Technology, WPAFB, OH
SCHAUER, FREDERICK R, Air Force Research Laboratory, WPAFB, OH
HOKE, JOHN L, Innovative Scientific Solutions, Inc., Dayton, OH

Replacing a Brayton cycle near constant-pressure combustor with a pulsed detonation combustor may exploit the potential performance improvements from low-entropy, pressure-gain heat addition, including reduced specific fuel consumption and increased specific power. Previous experimental work has demonstrated feasibility of such a pulsed detonation driven turbine and shown increased specific power over steady deflagration. Characterizing the performance and efficiency of the unsteady hybrid engine is a major concern in the evolution of these engines. Several formulations for computing average isentropic efficiency for an unsteady turbine have been proposed, one in particular using a numerical analysis to show as much as a ten point difference. Experimental data, however, for unsteady turbine efficiency has yet to be reported for a full-admission, pulsed detonation driven turbine. To that end, several difficult measurements must be made. The operating environment in a pulsed detonation driven turbine is characterized by large, rapid excursions in temperature, pressure, and mass flow. Peak gas pressures, temperatures, and velocities are on the order of 60 atm, 3000 K, and 1000 m/s, respectively. The focus of this work is the presentation of time-resolved measurements of turbine inlet and exit flow fields for a pulsed detonation driven turbine using various instrumentation techniques including, flush wall-mounted static pressure transducers, background oriented Schlieren, optical pyrometry, and particle streak velocimetry. Presented are unsteady results for a Garrett T3-class automotive turbocharger driven by a pulsed detonation combustor. The objective is to evaluate various time-accurate measurements of turbine inlet and exit temperatures, pressures, and velocities which are necessary to calculate turbine efficiency.

Thermal Anemometry Based Flow Meter for High Temperature, Harsh Environment Applications

Gogineni, Sivaram P, Spectral Energies, LLC, Dayton, OH
Kapat, Jayanth, University of Central Florida, Orlando, FL
Ricklick, Mark, University of Central Florida, Orlando, FL

Increased aircraft efficiency and reliability, and reduced cost of ownership may be simultaneously realized through the incorporation of advanced engine control and diagnostic capabilities. While improvements in engine performance modeling will contribute significantly to this effort, there is a strong need for methods of obtaining additional operating variables; leading to an increase in research related to robust, smart sensors. Modern gas turbine engines operate with hot gas temperatures well beyond the melting temperatures of the materials used within the machine, and thus the requirements for sensors located in these regions present significant challenges. As cooling of these components is a crucial aspect of engine performance, and considering the additional purposes bleed flow serves in aerospace applications, in-flight knowledge of the levels of air bled from the compressor serves multiple purposes; including aiding in situational awareness, assessing maximum engine capabilities, as well as improved mission and maintenance planning. Through the development of a novel, polymer derived ceramic (PDC) thermal anemometer based flow meter, Spectral Energies, LLC in collaboration with University of Central Florida (UCF)
plan to provide a solution to this difficult sensor need. This light weight, robust, dynamic sensor will not only provide a smart sensor solution to bleed flow measurement needs, it will also pave the way for other PDC based sensors for harsh environment applications.

**High Altitude Effects of Small UAV Engines**

*Schmick, Peter J, AFIT, WPAFB, OH*
*Polanka, Marc D, AFIT, WPAFB, OH*
*Litke, Paul J, AFRL, WPAFB, OH*
*Hoke, John, ISSI, WPAFB, OH*

The ever increasing use of man portable unmanned aerial vehicles, UAV, by the US military in a wide array of environmental conditions calls for the investigation of engine performance under these conditions. Previous research has focused on individual changes in pressure or temperature conditions of the air stream entering the engine. The need was seen for a facility capable of providing environmental conditions at various simulated altitude conditions. This facility was developed and a representative engine was tested over a range of engine loads and speeds at specific environmental conditions. In addition to testing engine performance as a function of altitude, the impact of carburetor tuning was investigated to see how it would impact brake specific fuel consumption, and engine torque. Lastly the variability of engine performance between identical engines from the same manufacturer was tested in order to quantify average expected performance and coefficient of variation.

**Heat Transfer Experiments on a Pulsed Detonation Driven Radial Turbine Exhaust**

*Longo, Nicholas C, AFIT/ENY, WPAFB, OH*
*King, Paul I, AFIT/ENY, WPAFB, OH*
*Schauser, Fred R, Propulsion Directorate, AFRL/RZTC, WPAFB, OH*
*Hoke, John L, Propulsion Directorate, AFRL/RZTC, WPAFB, OH*

Experimental results are presented for an investigation of exhaust flow downstream of an unsteady radial turbine powered by pulsed detonation combustion (PDC). Recent research has been conducted on the increased performance of pulsed detonation combustors over constant pressure combustors found in conventional gas turbine engines, specifically with regard to thrust and fuel consumption. Previous experimental work at the Air Force Research Lab demonstrated improved specific power output from a Garrett GT2860RS automotive turbocharger driven by PDC versus steady deflagration combustion (SDC). The current objective examines the enthalpy in the exhaust flow of a T3 journal bearing turbo driven by a PDC. Exhaust measurements are made for pulsed detonation combustor with and without a turbocharger. Enthalpy measurements are necessary to provide a means of validation for previously determined high speed measurements and would also aid in evaluating efficiency of turbochargers when integrated with PDCs. This includes a discussion of the relationship between turbine performance and PDC operating parameters.

**Data based model development and application of NMPC for gas turbine.**

*GODBOLE, AMIT A, BELCAN ENGINEERING, CINCINNATI, OH*

In current research, we investigate application of Nonlinear Model Predictive Control (NMPC) for controlling performance of a gas turbine engine. Performance parameter under examination is the thrust of a single spool gas turbine engine. The development of a nonlinear black-box model based on Weiner modeling approach is presented. The model consists of linear dynamic block parameterized by Laguerre filters, orthonormal basis functions,
followed by a static nonlinear block captured in Artificial Neural Network (ANN). This model is used to predict gas turbine performance and a closed-loop controller is realized using NMPC formulation. Model validation in open-loop highlights potential of the identification methodology. Closed-loop simulation results showcase very satisfactory control performance for gas turbine engine thrust tracking control problem and bring forth the applicability of NMPC formulation for highly nonlinear systems with fast dynamics.
Session 10: Micro Air Vehicles I

Chair: Donald Kunz, AFIT

PIV on Simple Mechanical Flapping Wings for Hover-like Kinematics

Hagen, Benjamin J, University of Dayton, Dayton, OH
Altman, Aaron, University of Dayton, Dayton, OH

A Particle Image Velocimetry (PIV) study on aspect ratio effects was performed to highlight the role of three-dimensional aerodynamic contributions relative to inertial contributions to lift and power for a hover-like passive pitch rotation flapping kinematic motion. Interest in the research and development of flapping wing micro air vehicles (MAVs) continues to grow. Despite the large body of work performed recently, an abundance of unanswered questions still exist. The approach taken in this study is to begin by reducing the complexity of the problem through parametric variation of a single geometric parameter, aspect ratio. Three different aspect ratio wings were tested to highlight the role of 3-dimensional aerodynamic contributions relative to inertial contributions to lift and power requirements for a hover-like flapping kinematic motion. The high aspect ratio wing was found to provide the most thrust while being the least efficient from a power standpoint. The mid aspect ratio wing provided almost 23% less thrust, but was the most efficient from a kinematic and overall propulsive efficiency standpoint. The low aspect ratio wing was the best from the power efficiency side, but had by far the worst thrust. It was determined that aspect ratio does play a role in flapping wing aerodynamics and kinematics. The present PIV research has a parallel ongoing force acquisition experiment led by the US Air Force Research Labs (AFRL/RBAL). The next step in this study is to compare the aerodynamically derived forces, via the PIV obtained velocity field, to the pending AFRL/RBAL integrated force measurements. When the AFRL study has been completed, the lift and inertial force results will be used in conjunction with the PIV results to provide more insight into the physics behind the resulting forces.

Repeatable Flapping Wing Micro Air Vehicle Wing Fabrication

Dawson, Bob, Air Force Institute of Technology, Wright Patterson AFB, OH
Cobb, Richard, Air Force Institute of Technology, Wright Patterson AFB, OH
Coutu, Ron, Air Force Institute of Technology, Wright Patterson AFB, OH

There has been great progress in the past decade in the research of propulsion for flapping wing micro air vehicles that range in size from large insect to small bird. These natural flyers all fly in the unsteady-state regime, so it has been proposed that the best form of propulsion for MAV’s that are modeled after them is flapping wings. While complex prototypes have been produced in a laboratory environment that mimic natural flapping wings, they were handmade and not reliably repeatable. In order to determine the importance of various features, we need a wing that is reliably and repeatably manufacturable. In this presentation, a manufacturing method will be described for making wings from 0.005 inch thick titanium foil, which has been calculated to have properties similar to a natural hawk moth wing. This method, using Microelectromechanical Systems fabrication techniques, has been shown to repeatedly manufacture wings with features less than the thickness of the titanium foil. With this repeatable manufacturing process, researchers will easily be able to perform parametric studies on complex flapping wing micro air vehicle wing designs, which has not previously been performed.
Stimulating Flapping Motion of the Manduca using Electromyography signals

Tubbs, Travis B., USAF, Dayton, OH
Palazotto, Anthony, Air Force Institute of Technology, Dayton, OH
Willis, Mark, Case Western Reserve University, Cleveland, OH

Neuromuscular impulses directly related to the flapping motion of Manduca Sexta (hawk moth) are recorded using electromyography signals recorded through intramuscular implants and artificially resupplied to the primary flight muscles (the dorsal longitudinal and dorsal ventral muscles). The process of imputing electrodes into the biological system will be examined as well as some of the signal processing required to clearly identify the electromyographical signals. A comparison between the natural flapping motion and the stimulated one is accomplished with high speed photography in an attempt to characterize the differences between natural wing movement and those induced due to electrical stimulants. This research lends itself to greater understanding of the bioelectrical signals necessary for flight with application to Micro Air Vehicle design and testing.

Flapping Wing Micro Air Vehicle Wing Manufacturing with Aerodynamic and Dynamic Repeatability Testing

Sladek, Nathanael J, AFIT, WPAFB, OH
Anderson, Michael L, AFIT, WPAFB, OH
Cobb, Richard G, AFIT, WPAFB, OH

2011 DCASS Abstract

Flapping Wing Micro Air Vehicle Wing Manufacturing with Aerodynamic and Dynamic Repeatability Testing

Nathanael J. Sladek, Michael L. Anderson, and Richard G. Cobb
Air Force Institute of Technology, WPAFB, OH 45433

Much research has already been pursued by various universities in the development of Flapping Wing Micro Air Vehicle wing manufacturing techniques. Commonly, these methods rely on some carbon fiber structure adhered to a thin, flexible membrane. Minimal attention though is given to repeatability of wing aerodynamics and dynamic response, which is crucial to avoid asymmetric flapping. Thus the focus of this research becomes twofold. First, a method to develop repeatable wing manufacturing techniques to ensure flapping wings have similar aerodynamic and dynamic characteristics. For this purpose, four wing designs are chosen to not only test the aerodynamics of the different designs, but to also validate manufacturing techniques. Three-layer 210 \( \mu \)m cured carbon fiber is used for the wing structure and 7.5 \( \mu \)m Kapton plastic for the wing membrane. Aluminum blocks are also used for wing assembly. The carbon is cured in a temperature and pressure controlled autoclave with both the cured carbon and Kapton cut with a laser to ensure repeatability. Repeatability of the wings themselves is assessed using two methods: through dynamic and aerodynamic data. Dynamic data, specifically the wing’s vibrational response, is measured using a 3D laser vibrometer. From this vibrational data, the wing modes can be determined which should correlate strongly between the various wing designs if the manufacturing techniques are repeatable. Next, using a piezoelectric flapping actuator, the various wing designs are tested. This data is then used to determine how aerodynamically similar the individual wings perform. These two data sets provide a quantitative means to compare the characteristics of a given wing design; whether the manufacturing process itself sufficiently reduces variation in assembly to produce nearly identical wings; and if these manufacturing techniques are reliable enough to warrant optimal design and consequent mass production. The presentation will discuss results obtained to date.
Increasing the Bio-mimicry of a Quad-Winged Flapping Micro Air Vehicle

Maples, Matthew R., Wright State University, Dayton, OH

The Wright Dragonflyer I.0 is a quad-winged flapping micro air vehicle. It was designed to imitate dragonflies’ flight kinematics which were observed using three-dimensional high speed photography. The rotating motors and geared drive systems of today’s powertrains create large and heavy fuselages that require larger wings which make achieving appropriate flapping frequencies difficult. Mimicking dragonflies’ frequencies and ranges of motion places high stress on the wings making materials selection and construction more complicated. These lessons, learned from the original, were incorporated into the design of the second generation Wright Dragonflyer I.1 to increase bio-mimicry while also keeping future testing needs in mind. Modifications to the powertrain, construction methods, and resulting motion of the wings, observed through high speed photography, will be highlighted by comparing both vehicles and their biological inspirations.
Session 11: HiFIRE

Chair: Michael Brown, AFRL/RZ

Optical Sensor for HIFiRE Flight 1 Hypersonic Experiment

Barhorst, Todd, Air Force Research Laboratory, Wright-Patterson AFB, OH
Brown, Michael S, Air Force Research Laboratory, Wright-Patterson AFB, OH

The demonstration of an in-flight Tunable Diode Laser Absorption Spectroscopy system for the measurement of mass capture was recently successfully executed in the Hypersonic International Flight Research Experimentation (HIFiRE) Flight 1. Key to integration into a flight payload was the design and construction of an instrument that met flight requirements for weight, size, power, and demanding environmental conditions. This paper traces the instrument evolution from design considerations and bench-scale hardware to flight integration and operation during flight. Two electronics packages were installed on the flight payload. Both implemented oxygen-based absorption; one employed direct absorption and the other wavelength modulation spectroscopy. Fiber-optic-based hardware enabled two angled lines of sight for both packages across two channels in the vehicle outer surface just downstream of the flair. The angled lines of sight permitted velocity measurements via Doppler shifting of the optical absorption frequencies.

Optical Sensor for HIFiRE Flight 2 Scramjet Ground and Flight Experiments

Brown, Michael S, Air Force Research Laboratory, Wright-Patterson AFB, OH
Barhorst, Todd, Air Force Research Laboratory, Wright-Patterson AFB, OH
Gruber, Mark, Air Force Research Laboratory, Wright-Patterson AFB, OH
Terry, William, Innovative Scientific Solutions, Inc., Dayton, OH

An optical sensor based on tunable diode laser absorption spectroscopy has been designed and tested for application to both ground tests and the eventual flight of the HIFiRE Flight 2 hypersonic engine experiment. The flight package consists of a near-infrared diode laser tuned across three water absorption features along with the photodetectors, transimpedance amplifiers and attendant electronics. The ground testing employs similar hardware with the ability to scan over additional absorption features. For both ground tests and the anticipated flight experiment, the laser output is split along several paths crisscrossing the exit plane of the combustor. Use of multiple lines-of-sight permits spatial reconstruction of the temperature, water concentration, and pressure fields at this axial location. Optical measurements on the ground test engine at NASA Langley permits comparison with wall-mounted sensors, an indirect view of engine dynamics, assessment of combustion efficiency, and feedback to computational fluid dynamics efforts. In addition, the data acquired provides a series of test cases on which to evaluate numerical algorithms for reconstructing the spatio-temporal maps of temperature, water concentration and pressure. Results of bench tests and anticipated sensor performance will be presented along with identified means at tomographic reconstruction.
HIFiRE Flight 2 – A SCRAMJET Flight Experiment

Jackson, Kevin, Air Force Research Laboratory, Wright-Patterson AFB, OH
Gruber, Mark, Air Force Research Laboratory, Wright-Patterson AFB, OH

A collaborative international effort, the Hypersonic International Flight Research Experimentation (HIFiRE) Program aims to study basic hypersonic phenomena through flight experimentation. HIFiRE Flight 2 teams the Australian Defence Science and Technology Organisation (DSTO), the United States Air Force Research Lab (AFRL), and NASA. Flight 2 will develop an alternative test technique for acquiring high enthalpy scramjet flight test data, and will explore in flight Mach 8, hydrocarbon-fueled scramjet performance and dual to-scram mode transition. The generic scramjet flowpath is research quality and the test fuel is a simple surrogate for an endothermically cracked liquid hydrocarbon fuel. HIFiRE Flight 2 will be a first of its kind in contribution to scramjets. It will explore suppressed trajectories of a sounding rocket propelled test article and their utility in studying ramjet-scramjet mode transition and flame extinction limits research.

HIFiRE Flight 2 Ground Test and Analysis Activities

Gruber, Mark, Air Force Research Laboratory, Wright-Patterson AFB, OH
Storch, Andrea, ATK Space Systems Group, NASA Langley Research Center, Hampton, VA
Hass, Neal, NASA Langley Research Center, Hampton, VA
Cabell, Karen, NASA Langley Research Center, Hampton, VA

The initial phase of hydrocarbon-fueled ground tests supporting HIFiRE Flight 2 has been completed in the NASA Langley Arc-Heated Scramjet Test Facility. In this effort, combustion experiments were performed in the HIFiRE Direct-Connect Rig. This test article is a full-scale, heat-sink version of the flowpath that will be flown. The primary objectives of the Phase 1 tests were to verify the operability of the HIFiRE isolator/combustor across the Mach 6.0 – 8.0 flight regime and to establish a fuel distribution schedule to ensure successful mode transition and combustion performance prior to conducting the flight experiment. In addition to the ground test activity, three-dimensional computational fluid dynamics simulations were performed using two Reynolds-Averaged Navier-Stokes solvers using facility measurements to specify inflow conditions and combustor data to serve as benchmarks for code calibration. Modeling parameters (e.g., turbulent Schmidt number and compressibility treatment) were tuned such that the computational results closely matched the experimental results. Combustor performance and operating mode were examined across a range of Mach numbers and were found to meet or exceed the objectives of the HIFiRE Flight 2 experiment. In addition, the calibrated computational tools will be extended to make predictions of combustor operation and performance for the flight configuration and to aid in understanding the impacts of ground and flight uncertainties on combustor operation.

HIFiRE Flight 2 Flight Performance Analysis and Instrumentation Suite

Gruber, Mark R, Air Force Research Laboratory, Wright-Patterson AFB, OH
Liu, Jiwen, Taitech, Inc., Wright-Patterson AFB, OH
Storch, Andrea, ATK Space Systems Group, NASA Langley Research Center, Hampton, VA
Bynum, Michael, ATK Space Systems Group, NASA Langley Research Center, Hampton, VA

Based on results from the HIFiRE Direct-Connect Rig testing and analysis activities, a calibrated three-dimensional computational fluid dynamics tool was used to assess the suitability of the isolator/combustor flowpath and baseline fuel distribution for meeting the science objectives of the HIFiRE Flight 2 experiment. These solutions used simulation results from the forebody/inlet as inflow conditions to the isolator/combustor flowpath. Combustor performance and operating mode were examined across a range of Mach numbers from 6 – 8 and were found to
meet or exceed the objectives of the HIFiRE Flight 2 experiment. In addition to simulation results, the instrumentation suite selected for the flight experiment will be presented. Instrumentation locations were selected based upon the flight computational fluid dynamics results in an attempt to minimize uncertainty in deduced combustor performance and to provide strong correlation to the existing ground test results.
Improved Correlation for Blowout of Bluff-Body Stabilized Flames

Huelskamp, Bethany C., Innovative Scientific Solutions, Inc., Dayton, OH
Kiel, Barry V., AFRL/RZTC, Wright-Patterson AFB, OH
Lynch, Amy C., AFRL/RZTC, Wright-Patterson AFB, OH
Kostka, Stanislav, Spectral Energies, LLC, Dayton, OH

The lean extinction limit is one measure of the stability of combustion systems. Over the past 60 years, many papers have been written on the subject of extinction of bluff-body flame holders. Early in the study of this subject, numerous experiments were conducted over a range of flame holders, pressures, temperatures, and fuels. The authors typically attempted to derive empirical correlations for the lean limit as a function of global conditions that appeared to have arbitrary exponents. In general, these authors concluded that the extinction appeared to be some function of Damköhler number. More recently, with the advent of high-speed diagnostics and computers, new observations concerning the extinction process have been made, with the most general conclusion being that the extinction process is a wake phenomenon, where the flame is highly strained and dominated by large vortices. In the present paper a new correlation for lean extinction is derived using a linear least-squares fit and more than 800 data points from historical and current experiments. Fits of various dimensionless parameters are made, but the best fit is that of a Damköhler number with ignition delay as the chemical time scale, verifying many previous conclusions. Finally, it is concluded that flame-holder size—not shape—is the driving parameter that represents the flame-holder geometry.

Bluff-Body-Stabilized Flame-Shedding Acoustic-Mode Isolation using Proper Orthogonal Decomposition

Kostka, Stanislav, Spectral Energies, LLC., Dayton, OH
Roy, Sukesh, Spectral Energies, LLC, Dayton, OH
Lynch, Amy C., AFRL/RZTC, WPAFB, OH
Kiel, Barry V., AFRL/RZTC, WPAFB, OH
Huelskamp, Bethany C., Innovative Scientific Solutions Inc., Beavercreek, OH

For the past few decades, researchers have been interested in bluff-body flame stabilization for augmenting combustion in gas-turbine engines. This current study focuses on investigating vortex-shedding mechanisms related to stable and acoustically coupled flames with various flame-holder geometries. Two modes of shedding, Kelvin-Helmholtz and Von-Karman, have played a significant role in the stability of flames. However, current state-of-the-art measurements are based on qualitative visualizations and do not provide the quantitative characterization of the vortex-shedding behavior. To fill this gap, proper orthogonal decomposition (POD) was used for quantitative characterization of bluff-body vortex-shedding dynamics in the presence of unsteady heat release. POD is a unique mathematical tool used to identify various instability modes and to characterize their spatiotemporal dynamics. Measurements were performed in an augmentor test rig with a 1.5” v-gutter, flat plate, and cylindrical flame holder placed in an air flow with propane used as a fuel. The fuel flow was used to vary the equivalence ratio from 0.7–1.1 to investigate the transitions between stable and acoustically coupled flame conditions. The use of POD presents itself as a metric for the design of flame holders to prevent or minimize acoustic coupling. POD mode shapes were found to be highly symmetric in the presence of acoustic coupling, while reconstruction of various mode pairs was
able to determine features responsible for acoustic coupling. POD analysis indicated that flame-holder geometry can alter the level of acoustic coupling found in flame-holder environments.

**High-Repetition-Rate Laser-Induced-Fluorescence Detection of a Radical Species via Ultrafast Excitation**

*Stauffer, Hans U., Spectral Energies, LLC, Dayton, OH*
*Kulatilaka, Waruna D., Spectral Energies, LLC, Dayton, OH*
*Gord, James R., Air Force Research Laboratory, Propulsion Directorate, Wright-Patterson AFB, OH*
*Roy, Sakesh, Spectral Energies, LLC, Dayton, OH*

Laser diagnostic measurements have become the standard for non-invasive characterization of complex combusting flows. In particular, laser-induced fluorescence (LIF) provides a straightforward, single-input-beam diagnostic approach toward species number-density determination and environment temperature determination. To date, LIF diagnostics have typically been carried out using nanosecond- or picosecond-duration laser systems, which operate at repetition rates in the 10–50 Hz range. However, the commercial availability of femtosecond laser systems with kHz and greater repetition rates holds great promise for high-bandwidth observation of transient events in combustion environments via the monitoring of number-density and temperature time series. We describe here the development of LIF detection schemes to probe combustion-relevant species using a high-repetition-rate ultrafast laser. In these initial experiments, a femtosecond laser system with a 1-kHz repetition rate is used to induce fluorescence, following two-photon excitation, from hydroxyl (OH) radicals present in a premixed, laminar flame. A range of experiments will be described, including studies of the dependence of observed signal on OH number density and on optical-pulse duration. Advantages and experimental challenges specific to two-photon excitation via a broadband sub-picosecond pulse will be addressed.
Session 13: Fuels
Chair: Rick Wills, AFRL/RZ

Experimental Results for a Parallel Hybrid-Electric Propulsion System for a Small Remotely-Piloted Aircraft

Harmon, Frederick G, AFIT, WPAFB, OH

Small electric-powered remotely-piloted aircraft (RPA) lack the endurance desired by warfighters, while their internal combustion engine (ICE)-driven counterparts generate mission-compromising acoustic and thermal signatures. Parallel hybrid-electric propulsion systems would meet the military’s needs by combining the advantages of hydrocarbon and electric power systems. The paper discusses a two-point conceptual design of a small parallel hybrid-electric RPA that combines an ICE sized for cruise speed with an electric motor sized for endurance speed. The design uses a clutch configuration to disengage the engine from the electric motor and propeller during the Intelligence, Surveillance, and Reconnaissance (ISR) mission segment. The mission requirements dictate whether a charge-depletion strategy or a charge-sustaining strategy is utilized. A hardware prototype of the parallel hybrid-electric system was tested on a dynamometer. Experimental results from the dynamometer testing are compared to the simulation results. The hybrid design provides a fuel savings of approximately 30% compared to a similarly-sized, conventional ICE-powered aircraft with the additional advantage of low acoustic signature during the ISR mission segment.

Energy Efficiency - a Survey of Department of Defense Activities

Somers, Rebecca M., Avetec Inc., Springfield, OH
Halliwell, Ian, Avetec Inc., Springfield, OH

Energy Efficiency – A Survey of Department of Defense Activities

The work presented here is a summary of a very broad survey of Department of Defense Activities in Energy Efficiency, carried out recently by Avetec Inc. under the auspices of the Defense Logistics Agency.

Energy conservation, efficiency and availability are the priorities established in recent executive orders including the 2005 Energy Act, the 2007 Energy Independence and Security Act, and the 2009 American Recovery and Reinvestment Act. The Department of Defense is a national leader on Energy Efficiency issues through planning, directing, and implementing innovative ideas and technologies to establish methods to meet the established goals and objectives. While defense operations will never be compromised, there is a distinct common interest between the military and commercial worlds when it comes to fuel usage and mitigation of its impact on the environment. The video that introduces the presentation is intended to provide a glimpse of the interaction between military operations and the environment. Fuel use and its impact may both be addressed through consideration of energy sources, the application of new technologies and long-term, careful planning.

Several energy sources fall within the directives mentioned above and these include bio-based fuels, renewable energy sources, and fossil fuels. Renewable resources like solar, wind, hydropower, geothermal, waste-to-energy, landfill gas and biomass have all been funded for research and are being incorporated appropriately throughout the nation. Geo-engineering – the human technology invented to manipulate the earth’s climate in order to counteract adverse atmospheric chemistry - is encouraging and carbon capture & sequestration are cited as examples. This review of many current energy sources and approaches illustrates their benefits and limitations.
Each military branch is at the forefront in planning and resource allocation when it comes to energy management. This includes the use of bio-fuel or blended fuels, reduction of consumption, and more efficient engines. Additional plans and requirements are targeting energy intensive facilities. Although not mandated through the directives, research and planning are in place that will strive to decrease energy consumption on the battle front. This relates to energy sources as well as tactical vehicles. The Department of Defense is constantly evolving into a more energy conscious organization, where energy-efficient technologies are identified and incorporated in day-to-day operations through research, development, and planning. A complete DoD strategy that incorporates these technologies, along with alternative energy sources and conservation methods, will enable the mandates required by the executive orders to be achieved. We are all fully aware of the shortage of energy resources and of the adverse effects that burning of carbon-based fuels have on the environment. Solutions to the problem rest with all of us in the technological world. It is hoped that this paper will assist you in establishing your role! We stress that in making this presentation, we do not represent the U.S. Department of Defense or the Defense Logistics Agency

**Characteristics of a Micro Gas Turbine Engine using Blends of Biodiesel Fuel**

Liou, William W, Western Michigan University, Kalamazoo, MI  
Tan, Ing Huang (Edmond), Western Michigan University, Kalamazoo, MI  
Ismail, Syafwan, Western Michigan University, Kalamazoo, MI

The engine performance and fuel consumption of a micro gas turbojet engine running different mixtures of B100 biodiesel fuel and kerosene are reported in this presentation. The work has been developed to evaluate the effect of the biofuel on the operating characteristics of the gas turbine engine that was originally built to run kerosene fuel. The fuel blends include mixture fractions from 0% to 100% B100 biofuel. The engine characteristics are recorded and analyzed in terms of the different fuel mixture contents. The results show that the engine performed consistently well in the wide range of variation of fuel mixtures. For example, it was found that, compared with using the kerosene fuel only (0%), the engine produced higher thrust using mixtures with the B100 biofuel for the same fuel volume flow rate, resulting in a significant decrease of the thrust specific fuel consumption (TSFC) at all engine speeds tested. In the case where the fuel contains only the B100 biofuel, TSFC decreases by more than 60%. Based on our purchasing cost, an estimate projects a saving of 200% of fuel cost using the B100 biofuel, for one thousand hours of operation of the engine.
Quantification of Model Form Uncertainty Using Evidence Theory

Park, Inseok, Wright State University, Dayton, OH
Grandhi, Ramana, Wright State University, Dayton, OH

It is common that two or more models can be created to predict responses of a physical system. Given a set of physical models, response predictions might be significantly influenced by model form uncertainty, which occurs due to the lack of certainty in selecting the true (or at least the best) one from the model set. In this research, a mathematical framework is devised to quantify model form uncertainty using expert evidence within evidence theory which handles imprecise human knowledge more realistically than probability theory. Using the belief structure associated with evidence theory, degrees of belief are numerically specified for subsets of a model set. Response predictions supported by subsets of a model set are integrated into a composite prediction using the disjunctive rule of combination. Then, a nonlinear spring mass system is utilized to demonstrate the process for implementing the proposed methodology.

Quantification of Model Form Uncertainty of Blended Wing Body Configuration

Gannon, Kenneth M., Wright State University, Dayton, OH
Grandhi, Ramana V., Wright State University, Dayton, OH

Simulation based modeling is often utilized in solving engineering problems. However, there are often multiple models available to represent the physical system and environment. For well understood phenomena, the best model is often found through refinement or convergence studies. In multi-physics problems, such as aeroelasticity, the best model is not always certain. Assumptions made during the modeling process vary between models and packages, which often enough produce multiple solutions to a single problem. As a result, there exists uncertainty regarding which model best represents the physical problem. This uncertainty is known as model form uncertainty. One such case in aeroelasticity where assumptions in the modeling process can present variations in the solution is in the modeling of blended wing bodies. Blended wing bodies present a distinctive problem in defining the threshold between the wing and the fuselage. Defining where the wing/fuselage interface is located is a modeling assumption and can impact the response. For instance, ZEUS, an Euler unsteady aerodynamic solver, handles the transpiration boundary condition differently between the wing and fuselage sections. Thus, quantifying the model form uncertainty associated with variations in the models based on the location of the wing/fuselage threshold is vital in developing an accurate representation of the physical system.

Quantification of Parametric, Model-Form, and Predictive Uncertainty in Design

Riley, Matthew E, Wright State University, Dayton, OH
Grandhi, Ramana V, Wright State University, Dayton, OH

Traditional uncertainty quantification in multi-physics design problems involves the propagation of parametric uncertainties in input variables such as structural or aerodynamic properties through a single, or series of models constructed to represent the given physical scenario. These models are inherently imprecise, and thus introduce
additional sources of error to the design problem. In addition, there often exists multiple models to represent the given situation, and complete confidence in selecting the most accurate model among the model set considered is beyond the capability of the user. Thus, quantification of the errors introduced by this modeling process is a necessary step in the complete quantification of the uncertainties in multi-physics design problems. In this work, a modeling uncertainty quantification framework was developed to quantify all three forms of uncertainty concurrently to present a complete representation of the uncertainty involved in design and analysis. The applicability of this framework is demonstrated on the complete quantification of uncertainty in a simple aeroelastic problem.
Session 15: Micro Air Vehicles II

Chair: Mark Reeder, AFIT

Engineering an Insect-Sized Flapping Wing MAV with Independently Actuated Wings

Anderson, Michael L, Air Force Institute of Technology, Dayton, OH
Cobb, Richard G, Air Force Institute of Technology, Dayton, OH

Insect-sized Flapping Wing Micro Air Vehicles (FWMAVs) have been proposed for their ability to operate indoors. Numerous designs have been demonstrated for bird-sized FWMAVs, but few for insect-sized vehicles, which require different design and fabrication approaches. Here, an 8 cm wingspan design will be presented that uses piezoelectric actuators amplified through a four-bar linkage to flap the wings. This actuation system enables the wing flapping amplitude, bias and trajectory to be modulated for vehicle flight control. In addition, each wing is independently actuated so that the wings may be flapped asymmetrically. Independent actuation and control of the wings gives an MAV additional degrees of freedom and makes this prototype the first of its kind. Novel methods for micro-fabrication that enable the repeatable assembly of such small, stiff and lightweight structures will also be presented. Finally, results of prototype testing will be presented including verification of the wing kinematics, wing flapping system dynamics and aerodynamic force measurements in six degrees of freedom.

Dual quaternion representation of rigid-body kinematics

Szalejko, Scott F., Air Force Research Laboratory, Wright-Patterson AFB, OH
Snyder, Richard D., Air Force Research Laboratory, Wright-Patterson AFB, OH

The mathematical representation of rigid-body motions in three-dimensions is a fundamental problem in engineering mechanics. There are a number of such representations. Perhaps the most common method in the field of aerospace engineering is to use rotation matrices based on Euler angles to represent rotations and a vector to represent translations. One alternative is to use dual quaternions of unit length to represent combined rotations and translations. Dual quaternions are not susceptible to gimbal lock, making them an attractive choice for maneuvering bodies that experience large ranges of motion. They also provide a compact representation relative to affine transforms and have some computational advantages. This presentation will describe the mathematics of dual quaternions and will demonstrate their use on maneuvering flapping-wing micro air vehicles.

High Fidelity Structural Dynamic Simulation of a Moth Inspired Flapping Beam Derived from Hamilton’s Weak Principle

Askey, Scott A, AFIT/ENY, Wright Patterson AFB, OH
Kunz, Donald, AFIT/ENY, Wright Patterson AFB, OH

A high fidelity structural dynamic simulation of flapping wing Micro Air Vehicles requires an aeroelastic model allowing for nonlinear flexibility of wings, unsteady fluid flow and aerodynamic forces of the same order as the inertial forces. This effort will focus on the elastodynamic portion of the aeroelastic problem. The elastodynamic model is derived from a mixed formulation of Hamilton’s weak principle applied to a geometrically exact beam theory (based on the Variational Asymptotic Method).
The aerodynamics will be treated as a modular black box allowing the easy use of various fluids models. The aerodynamic forces will be applied to the structural dynamics model as non-conservative generalized forces and moments acting on the wing’s reference axis.

The deformation of insect wings observed in nature grossly exceed the assumptions of linear elasticity. The beam theory based on Variational Asymptotic Method will be used to derive the nonlinear energy functional for the wing in terms of small local strains about the reference axis (flexibility) and arbitrarily large rotations and displacements of the reference axis (rigid body motion). The system of energy functionals, aerodynamic forces and driving forces will determine the action of the system in a Hamiltonian sense. The equations of motion are restructured using Hamilton’s weak principle, yielding a uniformly convergent, stable, finite elements in time scheme.

There will be no need to integrate or even derive the Euler-Lagrange equations of motion. The solution of the space time problem will be achieved without requiring knowledge of how to solve differential equations.

Hamilton’s weak principle, unlike Hamilton’s principle as expressed in most elementary dynamics texts, accounts for virtual work entering and leaving the system.

Virtual work done by the driving and aerodynamic forces can be treated as nonconservative generalized forces acting on the conservative elastodynamic wing. Under Hamilton’s weak principle the variations of the field variables are arbitrary at the end of the interval of integration. The mixed formulation implies that system is developed with the displacements and their conjugate momenta being independent field variables. Hamilton’s weak principle is derived by applying integration by parts to Hamilton’s law of varying action. All the derivatives of the field variables with respect to time are removed. The variation of the action is reduced to terms of the displacement, conjugate momenta, their variations and the variations of their time derivatives. The action is then discretized with the finite element method and integrated. The nonlinear system of equations allowing the variations to be arbitrary provides the system of residuals. The residuals are the basis of a time marching, uniformly convergent and direct solution to motion of the system. The derivation of Euler-Lagrange partial differential equations of motion and the ability to integrate them is not required. This approach is different from the conventional finite element in space-time scheme where second order Euler-Lagrange differential equations of motion are discretized. The numerical challenges associated with the integration of stiff differential algebraic systems, the conventional Multibody Systems Analysis approach (ie MSC Patran), are also avoided.

This effort will accurately predict the structural dynamics of a flapping with mothlike kinematics by treating the wing as a geometrically exact composite beam under the influence of a simplified unsteady or extended quasisteady aerodynamic theory and coupling them within the framework of a Hamilton’s weak principle based direct solution.
Session 16: Facilities

Chair: Matt DeWitt, UDRI

The Assured Aerospace Fuels Research Facility in Operation

Robota, Heinz J., University of Dayton Research Institute, Dayton, OH
Morris, Robert W., U.S. Air Force Research Laboratory, Wright-Patterson Air Force Base, OH
Smith, Russell K., Battelle Memorial Institute, Columbus, OH

The Air Force’s current target for 2016 is to: “Cost competitively acquire 50% of Air Force’s domestic aviation fuel requirement via alternative fuel blends in which the alternative component is derived from domestic sources produced in a manner that is greener than fuels produced from conventional petroleum.” Achieving this target is being enabled through the certification of 50/50 JP-8/SPK blends based on either Fischer-Tropsch or Hydrotreated Renewable Jet synthetic components. In these blends, the petroleum-derived fraction provides molecular components which fulfill a variety of performance requirements that pure SPK cannot. Further, current fuel specifications derive from a long, successful history of using petroleum-derived fuels. In order to move beyond the 50/50 blend to synthetic-rich formulations, a better understanding of how specific molecular components contribute to performance characteristics is valuable. Similarly, distinguishing petroleum-derived empirical specifications from performance driven requirements based on molecular composition in synthetics will be necessary to allow for the potentially wide variety of synthetic molecular compositions. The Assured Aerospace Fuels Research Facility (AAFRF) has been commissioned to deliver a flexible variety of purely synthetic fuels and fuel components in quantities large enough to enable intensive laboratory evaluation and where appropriate, full scale engine testing. AAFRF consists of two components. The first is a laboratory-scale facility in which the synthetic steps enabling the production of targeted fuel compositions are identified and verified. The second is a practical scale sample production reactor system capable of producing 10+ gallons per day of the targeted composition. We present here the results from the commissioning of the practical scale sample plant in which a Fischer-Tropsch wax fraction was converted to a full range SPK blend stock composition. We discuss preliminary lab-scale investigations currently underway for making a requested research fuel composition in order to illustrate the interplay between laboratory and practical scale activities. Finally, we discuss how this facility can be used broadly to enable novel pathways to sustainable synthetic fuels to be more quickly evaluated.

Perimeter Security and Intruder Detection Using Gravity Gradiometry: A Feasibility Study

Tuinstra, Jared D, United States Air Force, Dayton, OH

It is known that changes in the mass distribution around some point on the Earth’s surface induce corresponding changes to the magnitude and direction of the gravity vector at that location. The nine-tensor derivative of the gravity vector, or gravity gradient, is sensitive to very small changes in the gravity vector. With some assumptions, continuous measurement of the gravity gradient using a gravity gradiometer (GGI) can be used to determine the location of a mass change in the local area near the instrument. This investigation sought to determine the effectiveness, operating characteristics, and limitations of a physical perimeter security system that uses an array of GGIs to detect and locate a human intruder. Results were obtained via computer simulations that utilized the closed form solution for calculating a gravity gradient given an object’s size and mass, as well as industry-predicted future GGI performance characteristics.
Design of a Novel Acoustic Liner Test Facility for Augmentor Combustion Applications

Cuppoletti, Daniel R, University of Cincinnati, Cincinnati, OH
Kastner, Jeff, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim J, University of Cincinnati, Cincinnati, OH

The University of Cincinnati Gas Dynamics and Propulsion Lab (GDPL) has designed a unique combustion rig for testing and validating acoustic liners for augmentors. Thermo-acoustic combustion instabilities are a persisting problem that plague high performance gas turbine combustion systems. Augmentors are very high energy flames that operate near stoichiometric conditions and can cause high amplitude pressure oscillations within the augmentor pipe. The Acoustic Liner Rig (ALF) has been developed to conduct a detailed investigation into acoustic liner design for mitigating instabilities. The ALF features a circular test section with a coaxial flow configuration and a pre-burner to preheat the primary air to realistic augmentor inlet conditions up to Mach 0.3 at 1500 F and 4.5 lbm/s airflow. The secondary air can provide up to 1.8 lbm/s for rig and liner cooling. This coaxial flow configuration simulates a low bypass turbofan engines characteristic of modern military tactical jet engines. The test section contains a V-gutter bluff body flameholder and spraybar consistent with a gas turbine augmentor. The rig is outfitted with multiple ports to accommodate various spraybar and flameholder configurations. The rig is fully instrumented with dynamic pressure sensors to detect and identify various thermo-acoustic combustion instabilities common in cylindrical augmentors. The test section downstream of the flameholder has a clamshell design for easy removal to allow replacement of the test section with porous acoustic liners. The goal of the first phase of the project is to characterize the inlet and termination impedances of the rig and identify a lateral or tangential acoustic mode in the test rig. Following full characterization of a tangential instability, acoustic liners will be tested at various pressure ratios across the liners. These tests will facilitate further understanding of augmentor instabilities and provide critical data for improving acoustic liner and flameholder designs. The facility design and preliminary instability results will be presented. Tests are scheduled to begin during the first quarter of 2011. This work is part of a Phase II SBIR funded by the Air Force in conjunction with GE Aviation and ERAC.
Session 17: Heat Transfer II

Chair: Carl Hartsfield, AFIT

Investigation of Casting Defect Identification through Simulation

Gorsky, Daniel A., Wright State University, Dayton, OH
Grandhi, Ramana V., Wright State University, Dayton, OH

In castings, many defects can be detected on their surface, such as defects from improper filling of the mold. Other defects, such as shrinkage or porosity, cannot be as easily detected. Foundries will typically perform radiography to detect large shrinkage, but smaller shrinkage may not be detected by this method. Micro-shrinkage is difficult to track, but in steel, the Niyama criterion was developed to help predict the critical shrinkage regions with the aid of computer simulations.

A steel cast incased in a sand mold is the focus of this work. The Niyama value is evaluated at a local region in a casting when it is almost solidified. This equation is useful in a finite element simulation because the cast region is already discretized into nodes. The Niyama analysis of a casting is beneficial to understanding the shrinkage porosity due to metal feeding and heat conduction. The goal of this work is to understand how critical regions of Niyama can be manipulated by the addition of cast taper. The final goal is to move critical regions of Niyama closer to the original boundaries of a casting so they may be removed by risers.

Partitioning of heat generated by friction between two surfaces

Paek –Spidell, Gracie, AFIT, WPAFB, OH
Baker, William, AFIT, WPAFB, OH
Palazotto, Anthony N, AFIT, WPAFB, OH

Frictional heat generated by two sliding surfaces in contact is partitioned and propagates into both surfaces. The partitioning depends on the variation in the material properties, as well as their thermal diffusivity. A one-dimensional heat transfer model is used to characterize this process. At the interface, a partitioned flux condition drives the thermal propagation into each material. Further, a continuity of temperature condition in considered. The resulting equation is a Fredholm integral equation of the first kind, which captures the partitioning as a function of the frictional flux condition. The resulting solution is approximated in terms of incomplete gamma functions.

Assessment of an active-cooling micro-channel heat sink device, using electro-osmotic flow- A pilot study

Al-Rjoub, Marwan, University of Cincinnati, Cincinnati, OH
Roy, Ajit, Air Force Research Laboratory, Thermal Sciences and Materials Branch, Dayton, OH
Ganguli, Sabyasachi, Air Force Research Laboratory, Thermal Sciences and Materials Branch, Dayton, OH
Banerjee, Rupak Kumar, University of Cincinnati, Cincinnati, OH

Non-uniform heat flux generated by microchips causes “hot spots” in very small areas on the chip surface. These hot spots are generated by the logic blocks in the microchip bay; however, memory blocks generate lower heat flux on contrast. The goal of this research is to design, fabricate, and test an active cooling micro-channel heat sink device that can operate under atmospheric pressure while achieving high-heat dissipation rate with a reduced chip-backside
An experimental setup was assembled and electro-osmotic flow (EOF) was used thus eliminating high pressure pumping system. A flow rate of 82 µL/min was achieved at 400 V of applied EOF voltage. An increase in the cooling fluid (buffer) temperature of 9.6 oC, 29.9 oC, 54.3 oC, and 80.1 oC was achieved for 0.4 W, 1.2 W, 2.1 W, and 4 W of power, respectively. The substrate temperature at the middle of the microchannel was below 80.5 oC for all input power values. The maximum increase in the cooling fluid temperature due to the joule heating was 4.5 oC for 400 V of applied EOF voltage. Heat transfer coefficient (h) for the 4 W case reached a maximum of 292 W/m².K at the channel inlet and decreased to reach 92 W/m².K at the channel outlet. Numerical calculations of temperatures and flow were conducted and the results were compared to experimental data. It was found that using a shorter channel length and an EOF voltage in the range of 400 – 600 V allows application of a heat flux in the order of 104 W/m², applicable to spot cooling. For elevated voltages, the velocity due to EOF increased, leading to an increase in total heat transfer for a fixed duration of time; however, the joule heating also got elevated with increase in voltage.
Computational Study of Carbon Nanotubes under Compressive Loading by Quasi-static Reduced-order General Continuum Method with Barycentric Interpolation

Yang, Yang, Western Michigan University, Kalamazoo, MI
Liou, William W., Western Michigan University, Kalamazoo, MI

Carbon nanotubes (CNTs), since first found as a form of multi-walled CNT tangles, have gained overwhelming attention for their significant and unique properties. Many applications of CNTs have been proposed, such as enhanced composite materials, efficient heat removers, precise drug delivery, and etc. CNTs are slender thin-walled structure that is susceptible to buckling when subjected to compressive loading. Failure induced by mechanical instability has been a major concern in CNT-based devices.

CNTs under axial compression load will ultimately experience some form of morphological change because of the bucklings occurring at either the local or the global level. Buckling in CNTs results in a sudden drop of the total energy and is followed by a weakened structural strength of the CNT. It is of crucial importance to have a better understanding of the phenomenon in order to predict the structural integrity of CNTs. Several reported experiments showed that the CNT exhibited extreme cyclic loading resistance with yielding strain and strength becoming constant after a number of loading cycles.

In this presentation, some results of a numerical simulation of the mechanical behavior of CNTs under compressive loading will be described. The quasi-static reduced-order general continuum method is used to calculate the yielding strain of CNTs. The method uses a widely-accepted atomic potential, the reactive empirical bond order potential (REBO), for hydrocarbon molecules as the base model to calculate the interatomic energy among carbon atoms in CNTs. The REBO potential includes bond order terms that implicitly describe the angular dependence of interatomic forces, so that the computationally expensive operations of registering and tracking many atoms for multibody interactions are avoided. The quasi-static state of CNTs is determined by seeking the minimum configuration of the energy potential. The compressive deformation of CNTs is simulated by using the displacement-controlled method where the displacement boundary conditions are prescribed to each node on both ends of the CNT. Four cases of CNTs that cover the range of possible characteristic configurations of the carbon lattice are simulated and the behavior of the CNTs under the loading-unloading condition is studied. The obtained results are mapped back to the lattice level using barycentric interpolation. A detailed analysis of the observed CNT buckling patterns is provided.

We show that, with different chiral angles, different buckled configurations will be assumed by the CNTs. The zigzag CNT has the most apparent buckling pattern among simulated CNT configurations with larger chiral angles. The buckling strain increases with the increasing chiral angle. The armchair CNT studied exhibits the strongest resistance to the compressive loading. The loading-unloading simulations show that the strength of CNTs is reduced after buckling due to the compressive loading. However, during unloading, the CNTs simulated return to its original state, demonstrating a unique mechanical behavior of the single-walled CNT in response to the compressive loading.
Studies on Porous Silicon Thin Films as the Electrode Materials for Lithium-ion Batteries

Sun, Xida, Department of Electrical Engineering, Wright State University, Dayton, OH
Chu, Kuan-Lun, Department of Electrical Engineering, Wright State University, Dayton, OH

One of the key factors for the storage capacity of a lithium ion battery is the anode material used in the battery. Silicon-based nano-structured materials can incorporate large amount of Li atoms without fracturing and are thus prime candidates for the anode in Li ion batteries. Porous silicon films for lithium-ion battery anode application will be demonstrated and characterized systematically in my research. The Li insertion capability of porous silicon film anodes and carbon-based anodes will be compared. My current experimental results have shown how optimized porous silicon films can be produced in a relatively simple way employing anodic etching of silicon substrates followed by chemical etching. Silicon substrates were etched electrochemically in a solution consisting of Dimethylformamide (DMF) and Hydrofluoric acid (HF) under controlled current densities while etching time was adjusted to obtain a constant charge to generate pores of different sizes. Scanning Electron Microscopy (SEM) results have revealed the properties of the porous layer such as pore distribution, diameter, morphology of the pores, which depend upon the HF concentration, the applied current density during electrochemical etching, and temperature etc.

Keywords: Silicon-based nano-structured materials, Li ion battery, wafer, HF, SEM

Dynamic Characterization and Active Modification of Viscoelastic Materials

Zhao, Sihong, Miami University, Oxford, OH
Singh, Kumar V, Miami University, Oxford, OH

The viscoelastic material models are often frequency dependent hence the dynamic behavior of such materials is characterized by the associated non-linear eigenvalue problems. Traditionally these eigenvalue problems are formulated in state-space form via first order realization of the frequency domain equations to time domain. However, such transformation leads to a large matrix eigenvalue problem, which depends upon number of internal parameters used for approximating the material behavior, and hence they can be computationally expensive. Moreover, the state-space formulation does not provide physical insight of the original co-ordinates which may be used for measurement and control purposes. In this research, it is shown that by solving the associated transcendental eigenvalue problems, these frequency dependent eigenvalue problems can be solved without state-space realization. With few numerical examples, the eigenvalues computed by the proposed method are compared with those obtained in the traditional state-space methods. By preserving the dynamics behavior in the frequency domain, it is also shown that the physical parameters (especially damping and stiffness) of the material can be modified by active means (active control). Such modification utilizes frequency response receptance functions which may be extracted during dynamic experiments. Active modification strategy may be used to alter the material behavior during its operation and may be instrumental in developing more advanced viscoelastic materials.
First Order Conceptual Quick Turn Lighter-than-Air Design Tool

Mitchell, Robert William, WPAFB, Dayton, OH

The United States armed forces have a need for both persistent surveillance and increased mass deployment. One class of vehicles under consideration for both missions is lighter-than-air (LTA) aircraft such as aerostats or airships. To analyze and design LTA vehicles to fulfill Air Force missions, a detailed understanding of these configurations is required. Existing methodologies for the analysis and design of LTA aircraft are scarce and often lacking pedigree due to the age and lack of relevant data. In particular, conceptual design of hybrid airships, which rely upon the generation of aerodynamic lift, is significantly plagued by a lack of correlating data for both structural weight and aerodynamic performance. To aid in the design and analysis of these vehicles, a conceptual quick turn design tool based on historical data and first order principles was created. The tool incorporates consideration for conventional or hybrid configurations, mission and vehicle performance, and sizing of the envelope, tails and components. In addition, the tool allows rapid development of aerodynamic and weight data for the sized vehicle. The tool capability enables LTA configurations to be analyzed and compared to similar aircraft designed at the same detail level for the equivalent mission.

Aircraft Design of an Electric 2 Seater Aircraft

Puttmann, John Paul, University of Dayton, Dayton, OH
Cranston, Brian, University of Dayton, Dayton, OH
Hegadus, Eric, University of Dayton, Dayton, OH

To meet the criteria of the RFP, a two-seat electric powered aircraft was conceptually designed and then experimentally verified. Several design iterations were done with a series of experiments tested to verify the design. The final design presented had a low wing, v-tailed, single boom, tricycle landing gear, puller aircraft with a fuselage length of 30 ft and a wing span of 38 ft. It was tested in a wind tunnel, with a flight simulator, and with CFD. The design was tested at the University of Dayton LSWT, University of Dayton Flight Simulator, and in CFD. The LWST had a 3-D model printed that verified the L/D characteristics. The flight simulator was used to verify the performance characteristics of the design by inputting physical characteristics of the plane and then running the simulator. The CFD was used to verify L/D and to calculation the drag calculations the plane. All of the experiments performed verified the high lift to drag calculations made to the design of the aircraft.

Occupant Protection during Commercial Space Activities

Pelletiere, Joseph A, Federal Aviation Administration, Dayton, OH
Gohmert, Dustin M, National Aeronautics and Space Administration, Houston, TX

The Federal Aviation Administration (FAA) is responsible for regulating and licensing private US companies involved in commercial space transportation. An FAA license is required for any launch or reentry carried out by any US entity or any entity that would operate within the US. To date, the FAA has levied very few requirements for
human space flight. These regulations are detailed in 14 CFR Part 460. The requirements have consisted of informing the crew and occupants of any known hazards and risks, that there are unknown hazards, and that the US government has not certified the vehicle as safe for carrying crew or passengers. The National Aeronautics and Space Administration (NASA) has been updating their requirements for manned space flight. NASA has spent considerable effort in detailing occupant injury requirements to ensure a low probably of injury to its astronauts during spaceflight operations. Much of their effort could be carried over to manned commercial space flights and used as the basis for future safety requirements. While the current regulations are not detailed, it is important for designers and operators to begin thinking about what it takes to protect occupants during commercial space activities. It would be wise to focus efforts into developing common safety requirements to help ensure the success of commercial vendors.
Session 21: Experimental Fluid Dynamics I

Chair: Aaron Altman, UD

PIV in the Vertical Wind Tunnel

Schmit, Ryan F, Air Force Research Laboratory, Wright Patterson AFB, OH

The first attempt at performing Particle Image Velocimetry (PIV) in the Vertical Wind Tunnel (VWT) at Wright-Patterson Air Force Base. Both two component velocity and stereo PIV measurements were performed in an open tunnel. Three sets of lens were tested to determine which lens would provide the largest field of view and result in the highest quality PIV data. Several issues, like seeding amount, camera and laser location, calibration setup and reflection reduction were examined during the test to improve the PIV results. Finally, recommendations are presented so that future testing can be successful when a test article is present.

Single-Shot, Lifetime-Based PSP Measurements on an Oscillating NACA 0021 Airfoil in Compressible Flow

Juliano, Thomas J., The Ohio State University, Columbus, OH
Gregory, James W., The Ohio State University, Columbus, OH
Jensen, Christopher, The Ohio State University, Columbus, OH
Crafton, Jim, Innovative Scientific Solutions, Inc., Dayton, OH

A single-shot lifetime-based pressure-sensitive paint (PSP) measurement technique was used to study compressible dynamic stall on a NACA 0021 airfoil in a 6” × 22” Transonic Wind Tunnel. This instrumentation enables time-resolved measurement of the instantaneous global surface pressure field, with no averaging required. Mach numbers of 0.3, 0.5, and 0.6 and oscillation rates of 0.3, 3, 6, 9, and 11.8 revolutions per second have been tested so far. The angle of attack varied sinusoidally between -8.75 and +12.50 degrees. Preliminary results indicate that the expected trend of delayed separation at higher oscillation rate is detected. Further work will involve refinement of the PSP instrumentation and methods for data quality, and comprehensive investigation of the compressible dynamic stall phenomenon.

Force and flowfield history from experiments on pitching plates

Granlund, Kenneth O, AFRL/RBAL, Wright-Patterson AFB, OH
OL, Michael V, AFRL/RBAL, Wright-Patterson AFB, OH

In aggressive fast-rate maneuvering of Micro Air Vehicles, one is often interested in flowfield development, evolution and shedding of Leading- and Trailing Edge Vortices and how they relate to the generated forces and moments. We discuss the necessary experimental hardware in form of a water tunnel and planar motion mechanism to achieve non-dimensional pitch rates approaching K=1, address motion uncertainty and repeatability, loadcell calibration, static and dynamic loads and how they are affected by different model mounting schemes as well as filtering and postprocessing of recorded data. For qualitative visualization of the flowfield, we use dye injection to visualize vortex cores and a high-speed high-resolution digital camera to record images for individual examination and motion video.
The applied test case is an aspect ratio 2 flat plate wing of which we study two different planforms; a rectangular, as well as an elliptical one, both with rounded edges. Lift and drag data during pitch ramp maneuvers 0-90° for non-dimensional pitch-rates from static to K=0.2 will be compared for both planforms as well as against a nominally 2D case. Difference in lift from different ramp corner acceleration will be investigated, as well as convective time to return to steady-state. The information from the dye visualization is used to explain the differences and similarities in force history during the dynamic motions.

HIFiRE-1 Aerothermodynamic Measurements

Kimmel, Roger L, Air Force Research Laboratory, WPAFB, OH
Adamczak, David W, Air Force Research Laboratory, WPAFB, OH

The Hypersonic International Flight Research Experimentation (HIFiRE) program is a hypersonic flight test program executed by the Air Force Research Laboratory (AFRL) and the Australian Defence Science and Technology Organization (DSTO). Its purpose is to develop and validate technologies critical to next-generation hypersonic aerospace systems. Candidate technology areas include, but are not limited to, propulsion, propulsion-airframe integration, aerodynamics and aerothermodynamics, high temperature materials and structures, thermal management strategies, guidance, navigation, and control, sensors, and system components. The HIFiRE program consists of extensive ground tests and computation focused on specific hypersonic flight technologies. Each technology program is designed to culminate in a flight test. The first technology flight, HIFiRE-1, flew on 22 March 2010. The primary objective of HIFiRE-1 was to measure aerothermal phenomena in hypersonic flight. The primary experiment consisted of boundary-layer transition measurements on a 7-deg half angle cone with a nose bluntness of 2.5 mm radius. The secondary aerothermal experiment was a shock-boundary-layer interaction created by a 33-deg-flare / cylinder configuration. The HIFiRE-1 program created an extensive knowledge base regarding transition and shock-boundary-layer interaction. The current work reports preliminary temperature and pressure measurements from flight one and initial system performance estimates relevant to these aerothermal measurements.

Annular Parachute Aerodynamics: What’s the Big Deal?

Freed, Nicholas, Saint Louis University, St. Louis, MO
McQuilling, Mark, Saint Louis University, St. Louis, MO

The aerodynamics of parachute systems are highly unsteady, including interactions between the payload and canopy, separated flows, vortex shedding, and a very complex interaction between the thin and highly deformable canopy fabric and the resulting unsteady flowfield. Concentric annular parachute geometries potentially offer the highest drag achievable for non-gliding, hemispherical canopies. However, the fluid physics involved with these parachutes are not well understood. This presentation highlights the general characteristics of parachute aerodynamics, including a survey of current geometries in use for airdrop operations and where the concentric annular design resides in the parachute engineer’s design space. Plans for future study of the concentric annular design will also be presented, including the experimental and computational methodologies and expected results.
Experimental and computational characterization of a high-subsonic jet exhausting a conical nozzle with a center-body

Malla, Bhupatindra, University of Cincinnati, Cincinnati, OH
Mihaescu, Mihai, University of Cincinnati, Cincinnati, OH
Kastner, Jeff, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

Experimental and computational characterization of a high-subsonic jet exhausting a conical nozzle with a center-body

Bhupatindra Malla, Mihai Mihaescu, Jeff Kastner, and Ephraim Gutmark
School of Aerospace Systems
University of Cincinnati

The primary objective of this work is to perform a comparative study of the flow field data obtained using Particle Image Velocimetry (PIV) and Computational Fluid Dynamics (CFD). The flow field under consideration is a high-subsonic jet exhaust issued through an axisymmetric converging nozzle with a conic plug. The exhausting jet is hot with a total temperature of 220 degrees Fahrenheit. The jet Mach number is 0.9.

PIV was performed in the Aeroacoustics Test Facility situated in the Gas Dynamics and Propulsion Laboratory at the University of Cincinnati. The PIV data was obtained during the summer of 2009 in the facility. The converging nozzle had an effective diameter of 2.2 inches. Olive oil was used to seed the flow. The PIV system was mounted on a traverse in order to be able to capture the flow field at different downstream positions. Three downstream positions were covered with a total distance of 14 equivalent diameters. In order to verify the repeatability of the process, three sets of 500 PIV images are acquired at each of the downstream positions on three different days.

Computationally, compressible flow simulations using Reynolds-Averaged Navier-Stokes (RANS) turbulence model has been carried out under the same flow conditions as in the experiment. The steady-state RANS with standard k-\(\varepsilon\) turbulence model is used for the preliminary studies of the mean flow and of the Turbulent Kinetic Energy (TKE) levels. The ultimate goal of this study is to capture the flow field unsteadiness associated with this high speed, non-isothermal jet using the unsteady Large Eddy Simulation (LES) approach, which will be used in the future.

A thorough study of the PIV data has already been accomplished using the method of Proper Orthogonal Decomposition (POD), which is very efficient at decomposition random flow field (turbulent flow field) into its fundamental building block (coherent structures). Initially, the study of the results obtained from RANS will help in the validation of conclusions made about the mean flow field and the averaged TKE field through the study of the experimental data. In the future, POD analysis will be performed also on the LES results. That will facilitate detailed comparison between the experimental and computational data. Furthermore, since from PIV only the velocity field can be studied, CFD will be very advantageous due to its ability to calculate other important flow variables such as pressure, density, and temperature.
Aerodynamics of Free-to-Pivot Translating Plates

OL, Michael V., AFRL, Wright-Patterson AFB, OH
Granlund, Kenneth O., AFRL, Wright-Patterson AFB, OH

One approach to mitigating the complexity of flapping-wing aerodynamics is to seek aggressive but hopefully still relevant abstractions. The salient physics include questions of lift/drag/pitch dependency with incidence angle at large values of incidence angle, angle rate of change, and acceleration; the role of forces due to bound vorticity, shed vorticity, and acceleration of the ambient flow (the so-called noncirculatory loads); stability of leading edge vortices and their role in aerodynamic nonlinearities; and the derivation of basic scaling relations, such as those between stroke-averaged forces and normalized stroke amplitude. Here we consider thin flat plates of various aspect ratios (3.4, 5.5 and nominally 2D) executing idealizations of flapping maneuvers, where the flap leading edge follows a prescribed sinusoidal fore-aft trajectory, while the plate is free to rotate within some range of incidence angles, here taken to be ± 45 degrees. We compare a rigid plate (carbon fiber) with a two-element plate hinged along the span at its midchord, as a first approximation to structural flexibility. The experiment is in a water tunnel, with motions obtained by electric linear motors. Diagnostic methods include direct force measurement with submerged load cells, and flow visualization by injection of dye from ports exiting at various locations on the plate leading and trailing edge. Test conditions include “hover”, where the tunnel is turned off and all motion is derived from the linear motors, and “cruise”, where the tunnel is operated together with the linear motors. As often happens in the unsteady aerodynamics of aggressive motions, the dependency of aerodynamic loads on motion history is both approximately quasi-steady and contains subtle nonlinear surprises, some of which have complicated implications for derivation of aerodynamic models and vehicle design, but some of which offer remarkable simplification. Amongst the latter we note that plate aspect ratio has no bearing on either lift coefficient (in the flapping wing sense, thrust) or drag coefficient (force resistive to flapping), as the dimensional forces depend only on plate area. As the ratio of stroke amplitude to plate chord decreases below 1.0, thrust efficiency falls greatly, but at high stroke/chord amplitude the efficiency becomes essentially independent of kinematics. Stroke reversal, where acceleration is maximum, evinces noncirculatory spikes in lift coefficient. Otherwise, lift coefficient is essentially quasi-steady with incidence angle all the way through 45 degrees incidence, with no evidence of hysteresis or unsteady lift build-up in the sense of the Wagner function. A leading edge vortex is relatively stable for the cases of larger stroke to chord amplitude, regardless of the presence of absence of spanwise flow, which does depend on plate aspect ratio. A baffling emerging theme, in this and related experiments, is how it could be possible that flowfields could be different between various geometric or kinematic conditions, while aerodynamic force history is essentially identical.

Instability Process in Synthetic Jets

Oren, Liran, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

Synthetic jets are considered to be a leading solution for active flow control mechanisms. Their current and intended applications varies from future flight control, thermal management, and enhancing of mixing in the flow. The vortex dynamics of the synthetic jet determines the instability process that transitioned them into a continuous jet. Previous studies have shown that the transition in a continuous turbulent jet is governed by the dynamic formation and merging of the Kelvin-Helmholtz vortices that are formed in the shear layer. Ho and Huang showed that for the continuous and excited jets the merging location of the vortices will occur when the center of the vortex rings are aligned and the amplitude of the subharmonics of the fundamental frequency is at its max. The purpose of our study is to discuss how this instability mechanism occurs in the synthetic jets. This can help to further evaluate their effectiveness as flow control mechanism.

Time resolved PIV data was collected at random and phased locked. The results are presented for an axisymmetric synthetic jet and are compared against equivalent excited and conventional turbulent jets. The results show that the
changes in the random and deterministic fluctuations of the flow correspond to the location where vortex merging occurs. These vortex merging, that transition the synthetic jet into a continuous jet, were observed at $x/D = 5.5$ and $x/D = 7$.

**Simulations of 6 and 10 Degree Oblique Shock Generators for Shock Wave Boundary Layer Investigations**

*Galbraith, Daniel, University of Cincinnati, Cincinnati, OH*
*Turner, Mark, University of Cincinnati, Cincinnati, OH*
*orkwis, Paul, University of Cincinnati, Cincinnati, OH*
*Galbraith, Marshall, University of Cincinnati, Cincinnati, OH*

Mixed compression inlets offer a great increase in pressure recovery compared to conventional external compression inlets, and operate by generating a shock train which gradually increases the static pressure and ends in a normal shock. However, these inlets suffer from problems with shock wave boundary layer interactions which cause flow instabilities and severe performance issues. These inherent problems will be investigated experimentally at the University of Michigan in a Mach 2.75 wind tunnel with full length glass side walls, and computationally at the University of Cincinnati. This presentation will discuss the computational results from the experiments using 6 and 10 degree oblique shock generators.

**Evaluation of the open-source CFD code ”FreeCFD” for solving hypersonic aero-thermodynamics problems**

*ZHANG, Huaibao*, Department of Mechanical Engineering, University of Kentucky, Lexington, KY
*MARTIN, Alexandre*, Department of Mechanical Engineering, University of Kentucky, Lexington, KY
*McDONOUGH, James M.*, Department of Mechanical Engineering, University of Kentucky, Lexington, KY

Re-entry vehicles of interest to the Air Force need to travel at hypersonic velocities in order to reach their target in a timely manner. This tremendous speed directly translates into a vast amount of kinetic energy. Although most of this energy dissipates into the surrounding atmosphere, a fraction still reaches the vehicle through convective heating. In order to protect the payload and ensure safe delivery, it is critical to equip the vehicle with an appropriate Thermal Protection System (TPS), and therefore accurately predicts the surrounding aero-heating environment.

This research activity investigates the ability of the CFD code FreeCFD to solve hypersonic aero-thermodynamics problems. FreeCFD is an open-source code that performs massively parallel computations through domain decomposition. The code is fully implicit, using a first-order backward Euler time integration, and is second-order accurate in space. The inviscid fluxes are computed using a variety of approaches, like Roe’s flux difference splitting method and AUSM+up, amongst others. FreeCFD also has many multi-physics options, like turbulence modeling and solid wall conduction.

As a first test-case, a 2-D hypersonic Mach 10 flow over a cylinder is investigated. Because of the limitation of the code as far as physical models are concerned, a simple gas, Argon, is used. Results are compared to the hypersonic CFD code LeMANS, and are in a general good agreement, although the low-dissipation schemes used to represent inviscid fluxes cause irregularities near the shock. To allowed a more direct comparison, the widely used modified Steger-Warming flux vector spitting method will be implemented in the near future.
Computational Study of High Lift LPT Cascade Aerodynamics

Arko, Bryan Matthew, Parks College of Engineering, Aviation, & Technology at Saint Louis University, Saint Louis, MO
McQuilling, Mark, Saint Louis University, Saint Louis, MO

Design trends for the low-pressure turbine (LPT) section of modern gas turbine engines include increasing the loading per airfoil, which promises a lower airfoil count and thus a lower total weight of the engine, or more extracted power from the turbine section. This study uses a commercially available Reynolds-Averaged Navier-Stokes (RANS) finite volume flow solver to simulate the low Reynolds number flowfields around a two-dimensional linear turbine cascade model that matches the wind tunnel test section of the Air Force Research Lab at WPAFB. Three blade profiles have been simulated, including the aft-loaded Pack B which has a nominal Zweifel loading coefficient (Zw) equal to 1.15, the mid-loaded L1M (Zw=1.33), and the front-loaded L2F (Zw=1.59). All three blade profiles are known to be susceptible to varying degrees of laminar flow separation along the suction surface. Turbulence models employed are the SST k-ω and the Abe-Nagano-Kondoh linear low-Re k-ε with Kato-Lauder modification, denoted MPAKN. Steady-state and transient results are compared to experimental data in order to analyze the ability of the turbulence models to appropriately handle the known transitional nature of the flow, where it is shown that RANS simulations with adequate mesh resolution can produce similar separation behavior as seen in experiments. Results also compare the simulation of a single blade passage with periodic boundary conditions to the simulation of the multi-airfoil cascades in order to gain an understanding of the change in flow behavior due to the difference in simulation technique.

Development of Instabilities at Low Mass Flow in a Centrifugal Compressor with Ported Shroud

Gancedo, Matthieu, University of Cincinnati, Cincinnati, OH
Guillou, Erwann, University of Cincinnati, Cincinnati, OH
de Luzan, Charles, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

The use of turbochargers with internal combustion engines requires that their compressor have a large range of mass flow rate to accommodate all the regimes of operation. Dynamic compressors, such as centrifugal compressors, are limited at high flow rates by choke and at low flow rates by surge. The scope of the presented research is to study the effective operational range for a centrifugal vaneless diffuser turbocharger compressor with ported shroud. A turbocharger bench facility was designed and tested in order to define the performances of the compressor under study and to better understand the development and occurrence of surge. Dynamic pressure transducers are implemented on the compressor housing in the inlet, the diffuser and volute as well as the compressor outlet. Dynamic measurements were taken along with the elaboration of the compressor mapping. The data were analyzed using statistical analysis, converted into frequency domain by Fast Fourier Transform and compared temporally to each other. Specific insight has been given at low mass flow rates where unstable phenomena including stall and surge occur.

In this study, the evolution of the pressure distribution in the diffuser and the volute along a speedline was obtained. Three regimes of operation were identified from the combination of the dynamic and performance analysis: the
stable regime, the stall regime and the deep surge regime. Signal features from the different components of the compressor are discussed. Particularly, the large oscillations during deep surge in the compressor housing seem to originate under the tongue of the volute.

**Flow Features at the Inlet of a Turbocharger Centrifugal Compressor with Ported Shroud Using PIV and Dynamic Pressure Transducers**

*Guillou, Erwann, University of Cincinnati, Cincinnati, OH*
*Gancedo, Matthieu, University of Cincinnati, Cincinnati, OH*
*de Luzan, Charles, University of Cincinnati, Cincinnati, OH*
*Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH*

The extent of the mass flow rate range for centrifugal compressors in turbochargers is of primary interest. In order to increase the operational range of centrifugal compressors in turbochargers, Honeywell equipped its products with a ported shroud. This method of passive control has been demonstrated in the literature to extend the range of operability at low mass flow rate with minor negative impact on the compressor efficiency. The extension of the stable operating range is based on the control of surge by release of the stall cells developing downstream into the compressor.

To obtain further insight on the functioning of the device, planar flow measurements were performed in the environs of the ported shroud along and normal to the axis. Particle Image Velocimetry (PIV) – both standard PIV and stereoscopic PIV were employed – was used to identify the aerodynamic changes occurring in the flow field while surge is developing. High speed pressure transducers are also implemented at the inlet upstream of the inducer and in the recirculation channels. Measurements were taken at different operational points along a speedline from choke to deep surge. Hence both steady and unsteady flow phenomena were investigated experimentally. In surge, the use of phase-locked PIV allowed to better understand the evolution of the flow patterns during the cycle. This study presents the flow measurements for different planes in front of the inducer compared to the dynamic pressure measurements taken in the same area and proposes an explanation of the flow patterns occurring in stable regime, in stall, and in deep surge.

**University of Cincinnati Augmentor Design**

*Robillard, Declan Thomas, University of Cincinnati, Cincinnati, OH*
*Cain, Charles Cary, University of Cincinnati, Cincinnati, OH*
*Nieberding, Zachary J, University of Cincinnati, Cincinnati, OH*
*Park, Kevin E, University of Cincinnati, Cincinnati, OH*

Through funding by the Air Force Research Lab (AFRL) at Wright-Patterson Air Force Base, the Aerospace Propulsion Outreach Program challenged students at the University of Cincinnati to modify a small gas turbine engine by adding an augmentor and measuring defined performance parameters of the modified engine. An augmenter or afterburner is an engine modification used to temporarily provide an increase in thrust. This could be for any situation from supersonic flight to short take off’s. By injecting additional fuel into the exhaust of the turbine, it is ignited by the heat produced by the turbine and oxygen left over after combustion in the turbine. Although this dramatically increases thrust it has been observed to be extremely inefficient in terms of fuel consumption. The gas turbine engine that was to be modified was a JetCat P-80SE engine. Through multiple prototypes, the students at the University of Cincinnati were able to demonstrate thrust outputs of 20.9 pounds force. Although unable to power the engine beyond 80%, reliable operations achieved at 75% stand as a success in achieving the maximum augmented throttle at Wright-Patterson Air Force Base with an outputted 15.5 pounds force.
at 75% throttle. With the exponential relationship that thrust has with respect to throttle, it is theoretical that the UC afterburner design could have achieved thrust readings above 150% thrust of the engine running dry.

**Testing and Simulation of Vibration Control Applied to Rotor Blades**

*Lindsey, Bobby L., University of Kentucky, Lexington, KY*
*Renduchintala, Ajay Babu, University of Kentucky, Lexington, KY*
*Baker, John R., University of Kentucky, Paducah, KY*
*Capece, Vincent R., University of Kentucky, Paducah, KY*

Safety and reliability are paramount in aircraft engine design. Engineers strive to improve these factors, and also increase efficiency and service life. Reduction of rotor blade vibration amplitudes during aircraft engine operation can enhance safety and improve performance. To this end, the University of Kentucky Department of Mechanical Engineering is researching the application of active vibration control to rotor blades. Earlier work at the University of Kentucky has included successful proof-of-concept tests in which piezoelectric patches were mounted on stator vanes and on rotor blades and used as actuators to reduce vibration response amplitudes. This earlier work has included laboratory bench testing, as well as some testing during operation in the university’s single-stage axial-flow low speed research compressor (LSRC). In this presentation, some results from tests and finite element simulations on two different rotor blade geometries are discussed. Also, continuing work, aimed at optimizing the control system, is reported. This recent work includes computer simulations, and also tests performed on a simple cantilevered beam.
SIERRA Project: UAV’s as a Training Tool

Charvat, Robert, University of Cincinnati, Cincinnati, OH  
Cohen, Kelly, University of Cincinnati, Cincinnati, OH  
Buhr, Nick, University of Cincinnati, Cincinnati, OH  
Meyer, Ted, University of Cincinnati, Cincinnati, OH  
Bodenschatz, Nate, University of Cincinnati, Cincinnati, OH

The SIERRA (Surveillance for Intelligent Emergency Response Robotic Aircraft) project at the University of Cincinnati, in collaboration with the state of West Virginia, is currently planning a full-scale test this March using an Unmanned Aerial Vehicle (UAV) to assist in wildfire response. As we execute this work, we are continually reviewing additional areas in which UAV’s may provide a benefit.

One such area is flight training. In addition to the direct benefits UAV’s provide operators, these systems may be able to benefit the aviation community at large through more economical flight training. Flight planning, weather interpretation, Air Traffic Control communication and routing could be taught to prospective manned aircraft pilots using UAV’s at significantly decreased cost, allowing a greater number of people access to flight training and a chance to reverse the negative trend in pilot population that aviation has been experiencing in the past decade. As an educational exercise, the SIERRA Project at the University of Cincinnati is committed to ensuring the future and continued excellence of the United States Airspace System.

The project is funded by the Ohio Space Grant Consortium and NASA.

Application of an Adaptive Tuned Vibration Absorber for Mitigating Cryogenic Cooler Vibrations

Koo, Jeong-Hoi, Miami University, Oxford, OH  
Karnes, Michael, Miami University, Oxford, OH  
Sinko, Robert, Miami University, Oxford, OH

The primary objective of this study is to develop an adaptive tuned vibration absorber based on magneto-rheological elastomers (MREs) that suppresses vibrations of a cryogenic cooler. Miniature cryogenic coolers are widely used to cool down sensitive image sensors in observation and surveillance satellites. The linear actuator in the cooler compressor (the pumping action) produces unwanted vibrations, causing significant image distortion. To address this vibration problem, this work proposes the use of magneto-rheological elastomer (MRE) in a tuned vibration absorber system (TVA). MREs are a new class of smart or multifunctional materials whose elastic modulus or stiffness can be adjusted depending on the magnitude of the applied magnetic field. Hence, they can be used as controllable stiffness elements in engineering systems. In this Using a 2-DOF test setup, the tuning ability of the MRE-based adaptive TVA is investigated in this study. The results show the feasibility of MRE-TVAs for the cryogenic cooler vibration mitigation application.
Experimental Low Reynolds Number Plunge and Pitch Fixture Development

Sparks, Jeremy R., University of Kentucky, Lexington, KY
Smith, Suzanne Weaver, University of Kentucky, Lexington, KY
Bailey, Sean, University of Kentucky, Lexington, KY

The Plunge/Pitch Fixture in development at the University of Kentucky is being constructed in conjunction with the University of Louisville to investigate low Reynolds number flow over a flapping foil. This fixture simplifies flapping wings as a two degree of freedom motion in plunge (translation) and pitch (rotation). Motivation for this fixture is comparing performance of flat and corrugated foils at a Reynolds number below 10,000 and Strouhal number relevant to natural fliers.

Flow visualization over flat and corrugated foils at varying angles of attack will be conducted to observe flow characteristics of a static foil at low Reynolds number before the fixture is operated dynamically. Observations here will be mainly made around the stall angle of different foils.

Verification of the desired motion may be validated with multiple methods: laser displacement sensors or high speed video with photogrammetric measurements. Each method is non-contact so as not to disturb the light structure. With validated motion, our final step is to operate the fixture in a low Reynolds number scenario while recording lift and drag forces.

In this presentation, the design and construction will be described, along with testing completed to date.

Characterization of MEMS-Based Inertial Sensor Errors

Aggarwal, Priyanka, University of Toledo, Toledo, OH
Bhatt, Deepak, University of Toledo, Toledo, OH
Bhattacharya, Prabir, University of Cincinnati, Cincinnati, OH
Devabhaktuni, Vijay, University of Toledo, Toledo, OH

Introduction

Development of real-time Battlespace visualization and navigation methods to obtain the precise knowledge of friendly and enemy forces is crucial for the success of military operations. In order to obtain accurate navigation information about the units, multiple sensor data fusion is necessary for compensating errors of individual sensors such as Inertial Navigation System (INS), Global Positioning System (GPS), and Light Detection And Ranging (LiDAR) system. Micro Electro Mechanical Systems (MEMS) have enabled the sensor technology to evolve from restricted, expensive, and inflexible units to miniaturized, low-cost and low-power silicon-based units. However, being small in size and light in weight, MEMS sensors experiences more errors like run-to-run biases, in-run biases, scale factor drifts, and other environment dependent errors, which are generally smaller or negligible in higher grade inertial sensors. For example, a higher grade IMU with a gyroscope bias of 1 0/h will experience a position error of 1.7 m in a minute, while a MEMS IMU with gyroscope bias of 100 0/h will have 171 m of error. Therefore, for seamless and continuous navigation solution, the characterization of different error sources and their reliable estimation or compensation is of significant importance.

The inertial sensor errors are divided into two parts: random and deterministic (systematic). The random errors include in-run biases or scale factor drifts that correspond to the rate, at which, the errors in inertial sensor accumulate with time. The deterministic error sources include the bias and the scale factor errors, which remain constant during a run and can be removed by specific calibration procedures in a laboratory environment. However, for low-cost sensors such as the MEMS sensors, these errors are quite large and their repeatability is typically poor because of their environmental dependence, especially temperature, which makes frequent calibration a necessity. In this paper, we evaluate and then compare these significant errors with the manufacturer specified values to establish, if time-consuming laboratory characterization is necessary.
Methods
We performed laboratory calibration of all the three gyroscopes and three accelerometers in a low-cost Inertial Measurement Unit (IMU). The calibration procedure for gyroscopes included run-to-run bias and in-run biases drift tests as well as evaluation of random walk parameters. The run-to-run biases were calculated by averaging the static data collected for 1 min, removing the offset bias calculated by six-position method, by switching on the IMU in each run. The in-run biases drifts were calculated using 1 min static data collected at 15 min intervals over 1 hour duration, with the IMU switched on throughout the test. We then collected static data for 24 hours duration, which was processed by Allan variance method to obtain velocity and angular random walk parameters. The performances of the sensors were compared using the laboratory calibrated values and the manufacturer specified values to investigate if these repeated calibrations are necessary or if manufacturer parameters are sufficient for reliable navigation. Further, effects of temperature changes on biases of gyroscope and accelerometer were determined by running the IMU continuously for a period of 24 hours at the ambient temperature, including the sensor warm up time.

Results
Results obtained by the laboratory calibration tests were compared to the manufacturer provided values. Run-to-Run bias for gyroscope was calibrated to be 0.1801 o/sec, which was significantly smaller than the manufacturer specified value of 0.75 o/sec. The calculated in-run bias variation over the duration of an hour was 0.18 o/sec, which was in agreement with the manufacturer specified value of 0.2 o/sec. The angular random walk determined by Allan variance method over the duration of 24 hours was significantly less (1.5 o/√h) than the manufacturer value (3 o/√h). Similarly, the velocity random walk was 0.06 m/s/√h, while the manufacturer value is 0.105 m/s/√h. No significant drift in biases was observed with variations in ambient temperature, mostly because the sensors come with the manufacturer pre-calibration and compensation of temperature-related drifts.

We further intend to compare the effectiveness and validity of the laboratory calibration method and the manufacturer values through both an independent laboratory test and a kinematic test vehicle using MEMS-based IMU and high-quality IMU mounted on it.

Conclusions
The run-to-run bias, in-run bias drift, and Allan variance test calibrated values were within the manufacturer specified values. However, for the precision and accuracy necessary for the battlefield navigation applications, the laboratory calibration procedure should be performed for low-cost MEMS units.

Experimental and Computational Study of an Impingement Cooling Jet Array

Schroder, Andrew, University of Cincinnati, Cincinnati, OH
Ou, Shichuan, Air Force Research Laboratory, Wright Patterson Air Force Base, OH
Ghia, Urmila, University of Cincinnati, Cincinnati, OH

Internal cooling of gas turbine blades is commonly performed through the combination of impingement cooling and serpentine channels. This research is focused on impingement cooling. When large arrays of impingement jets are implemented inside of a gas turbine blade, cross flow effects from spent jet air reduce the cooling effectiveness of downstream jets. A compact test rig has been developed for studying the convective heat transfer coefficient of jets in the presence of cross flow utilizing a steady state infrared thermography technique. The facility features an array of 55 impingement jets and two exit flow configurations (high and low cross flow). Experimental results for jet Reynolds Numbers of 4,000, 8,000, 12,000, and 15,000 will be presented for both exit flow configurations. A numerical study has also been performed for the same flow configurations. The Reynolds Averaged Navier Stokes (RANS) equations have been solved using the Fluent computational fluid dynamics code in conjunction with a k-omega turbulence model. Results from this numerical study will be presented and compared to the experimental results obtained from the test rig.
Acoustic Measurements for a Pulse Detonation Engine Installed In a Long EZ Aircraft

Shaw, Leonard L, AFRL/RBS, Dayton, OH
Harris, Kevin, AFRL/RBS, Dayton, OH
Swanson, Cynthia, AFRL/RBS, Dayton, OH

The pulse detonation engine (PDE) consists of a single, or multiple tubes, which are utilized to detonate the fuel at very high pressures. Since the detonation is occurring at a very high pressure a shock wave is generated in the tube and exits at the end of the tube. The associated noise of the process is of great interest since shock waves can generate very intense noise levels near the exit of the tubes. The purpose of this paper is to present the results of an acoustical survey of a PDE installed in a flight certified LongEZ aircraft. The Air Force Research Laboratory’s Propulsion Directorate (AFRL/PR) developed the engine and installed it in the aircraft. The new PDE was developed in-house with a fairly low level of funding by utilizing existing engine components. The acoustic environment of a pulse detonation engine installed in a Long EZ aircraft was measured. The engine consisted of four detonation tubes which were detonated at 20 Hertz in sequence. Fill fractions and equivalence ratios of 1.2 and 0.6 were tested. The ground run-up measurements were made on a runway near the National Museum of the USAF at Wright Patterson Air force Base. An array of 16 microphones was utilized to measure the environment. The microphones were located from 7.5 feet to 120 feet from the tube exits at a height of 5 feet. Time histories and narrow band spectral analysis were generated. The results indicate that a very high level pulse is generated near the exit of the tubes but tends to decrease in amplitude fairly quickly with distance since the higher frequency energy dissipates rapidly in the atmosphere. The levels varied from 148 dB at 7.5 feet to a low of 114 dB at 120 feet. The internal cockpit level was measured at 126 dB. It was determined that the test aircraft was safe to fly from the standpoint of pilot noise exposure and community noise impact.

An Experimental Study of Microjets on a Tactical C-D Nozzle

Perrino, Michael, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

A convergent-divergent nozzle with a conical geometry with a sharp edged throat and micro jets at the trailing edge has been built and tested. This nozzle was modeled after adjustable geometry nozzles used on high performance tactical aircraft. This nozzle has a conical geometry with a sharp edged throat. The micro jets are used to increase mixing in the shear layer between the core flow and a secondary flow. Near field pressure measurements, far field acoustic measurements, as well as particle imaging velocimetry measurements have been taken at a core flow Mach number of 1.56 and at a secondary flow Mach numbers of 0.0, 0.1, and 0.3. These results are compared with a baseline nozzle of the same geometry to see what affect the micro jets have on the jet noise production of the nozzle. The fluidic injection was shown to decrease the sound pressure at lower frequencies with a minimal high frequency penalty with these effects become more prominent as the secondary flow of Mach number increases. The presence of the micro jets decreases the overall sound pressure level for most cases. The main reason for this decrease in overall sound pressure level is due to a decrease in broadband shock associated noise. The azimuth direction that benefits most changes from forward observation angles to aft observation angles as the secondary flow increases.
Using Stochastic Estimation to Study the Temporal Evolution of the Velocity Field from a High-Speed Jet

Kastner, Jeff, University of Cincinnati, Cincinnati, OH
Cuppoletti, Daniel, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

Analysis of jet noise sources is based on the decomposition of dependent variables into flow related components and acoustic components. The flow governing equations can then be reformulated into an inhomogeneous wave equation. Fundamental understanding of noise generation and the development of noise reduction technology requires the development of tools that can analyze simultaneously the relationship between the turbulent flow field and the near-field pressure. Dynamics of large-scale coherent structures and small-scale turbulence are a critical part of this correlation. Because the coherent structures are generally embedded in a chaotic and random field, the challenge for both experimentalists and numerical analysts is to separate them from the turbulent background and then determine their characteristics and effect on noise production.

This work presents results from simultaneous measurement of flow velocity and near-field pressure in an axisymmetric jet. The near-field pressure is measured by two 16 microphone, linear arrays diametrically opposed, and the flow velocity is measured by streamwise particle image velocimetry (PIV). The simultaneously measured velocity and pressure are analyzed using conventional statistical methods and spectral analysis. Near-field pressure measurements have temporal resolution but limited spatial resolution while PIV measurements have high spatial resolution but no temporal resolution. Thus stochastic estimation and proper orthogonal decomposition (POD) are used to generate the temporal evolution of the PIV data. By generating the temporal evolution of the flow field, it is then possible to relate the flow dynamics to the hydrodynamic pressure field just outside the jet. It is found that the first asymmetric POD mode (m = 1) retains its signature into the near-field pressure much more efficiently compared to the axisymmetric mode (m = 0). This is quantified by higher correlation values between the asymmetric POD modes and the near-field pressure. Also, by having temporal information of the velocity field, it is also possible to generate modal spectra for the POD modes which are beneficial towards identifying the preferred frequency for each POD mode and relate it back to the near-field pressure spectra.

Novel Fluidic Enhancement of Mechanical Chevrons

Heeb, Nick, University of Cincinnati, Cincinnati, OH
Cuppoletti, Daniel, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

An experimental investigation was completed regarding non-azimuthally symmetric fluidic enhancement of mechanical chevrons applied to a supersonic jet. The driving force behind this configuration was twofold, first to introduce an asymmetry into the main jet causing a distribution of the quazi-periodic shock cell structure seen for all operating conditions for the jets in question and second to employ the shielding nature of multiple jets located in close proximity. Injection schemes were tailored to try and take advantage of these ideas, leading to biaxially symmetric and single direction preferential arrangements. Results are focused on the over-expanded regime due to the practical operating range of military style aircraft being modeled. Far-Field acoustic results show comparable noise reduction to the azimuthally symmetric fluidic enhancement geometry whilst reducing the mass flow rate required by approximately half. Cross-Stream Stereo-PIV shows modification of the high speed core of the jet from a circular into an elliptical cross-section. The goal to achieve appreciable noise reduction with a feasible fluidically enhanced chevron arrangement has been shown. Further improvement could be possible through use of converging diverging injection ports, or a higher pressure injection source allowing an increase in the momentum flux ratio at higher nozzle operating conditions.
Electromagnetically driven degassing of molten metallics in an induction furnace

Shi, Yinghui, University of Dayton, Dayton, OH
Petrykowski, John C., University of Dayton, Dayton, OH

In a number of aerospace materials processing applications, including float zone refining of electronic materials, forming of metallic glasses and induction melting of light alloys, time-dependent electromagnetic forces associated with the processing are found to influence surface shape, nucleation of precipitates, evolution of crystal nucleation sites, segregation of alloy components, grain refinement and degassing. For this last action, which finds its prime occurrence in specially designed induction furnaces, the scale up from test stand to prototype is especially sensitive to a detuning that can occur when a common set point is sought for optimizing the concomitant electrical and mechanical performance of the system. This paper outlines a continuum based model that can be used to identify a favorable set of operating conditions so that an effective and efficient, electromagnetically-induced vibrational degassing operation can proceed within the furnace. The optimization metric utilizes a coupled magnetoacoustic system of governing equations, which is subsequently solved to obtain the dynamic response of a molten metallic to an eddy current-type excitation. The solutions display both a transient and steady state response, as well as eigenmode and eigenfrequency characteristics which capture both the spectral signatures of the furnace as well as the optimum operating conditions for degassing. The solutions are obtained with the aid of higher transcendental functions of Bessel type, generated within a MATLAB environment. A set of operating conditions is identified which would promote optimal degassing for light alloys in commercial size induction furnaces. Extensions to other applications are also discussed, including grain refinement during light alloy solidification, and metallic glass forming for which the characteristics of the vibrational field can be utilized to effectively diminish crystal nucleation.
Session 26: Micro Air Vehicles III

Chair: Gregory Parker, AFRL/RB

A Bio-mimicking Flight Control and Flapping Mechanism for MAV Design

Dawson, Jaderic Cleo, Wright State University, Dayton, OH
Huang, George PG, Wright State University, Dayton, OH
Smith, Todd J, Wright State University, Dayton, OH

Many current flapping wing Micro Air Vehicle (MAV) designs use rudder and elevator control surfaces to control the pitch and yaw of the vehicle. This is used in conjunction with adjustment to the flapping wing frequency. However, in nature, insects adjust their center of gravity by deflecting their body to control flight trajectory. To mimic this type of flight control, a mechanism was developed consisting of a worm gear control set used to adjust the center of gravity of the vehicle. With this setup, the motor, control board, and battery can rotate around the flapping mechanism to effectively adjust the center of gravity to enable both hover and forward flights. In addition to this control mechanism a bevel gear set was designed capable of producing a 180 degree flapping angle resembling a ladybug during hovering. As a result of the 180 degree flapping angle a clap-and-fling effect occurs at the end of each wing stroke as demonstrated by the ladybug. To illustrate the ability to create such a flapping and control motion, a prototype model was constructed.

Comparative Benchmarking for Micro Air Vehicle Platforms

Feist, Alexander, Wright State University, Dayton, OH
Lauf, Adrian, Wright State University, Dayton, OH
Huang, George, Wright State University, Dayton, OH

Comparative Benchmarking for Micro Air Vehicle Platforms
Alexander Feist, Adrian Lauf, P.G. Huang
Department of Mechanical and Materials Engineering
Wright State University
3640 Colonel Glenn Hwy.
Dayton, OH 45435-0001
USA

The domain of the Micro Air Vehicle (MAV) provides exciting new possibilities in classic fields of surveillance, reconnaissance, and search-and-rescue, due to ever-decreasing size, increasing durability, and the ability to deploy multiple vehicles within a theater. Particularly suited towards indoor flight and maneuverability, the MAV class of aircraft permits tactical operations where no other aircraft platform has previously been used. By using sensors such as vision, laser range finding, magnetometers and gyroscopes, MAVs can successfully navigate complex indoor environments with ease thanks to their agility and size.

With continuous advances being made in the development of the micro aerial vehicle (MAV) aircraft class, there arises a need to assess the performance of three main competing aircraft configurations: fixed-wing (considered a traditional method of providing lift when thrust is applied to the airframe), rotary-wing (more-commonly known as a helicopter, with a rotating main wing coupled to a gyroscopic stabilization system employing a tail rotor), and flapping-wing (in which both lift and thrust are generated by one or more pairs of moveable wing surfaces.) Each thrust/lift configuration possesses distinct advantages and disadvantages in large-scale aircraft, particularly when
considering the first two categories. For instance, rotary-wing aircraft are inherently unstable in their flight characteristics, requiring that reactionary torque be countermanded by a tail rotor or jet; in addition, sudden changes in environmental air currents can cause unexpected lift events, as the rotary wing travels in all possible directions through the course of its operation, unlike a fixed-wing aircraft. Such disadvantages come with superior maneuverability, and the ability to hover, in contrast to the inherent stability and speed of a fixed-wing aircraft, which is generally unable to hover or operate at low flight speeds.

Flapping-wing MAVs signal a departure from conventional aircraft design by utilizing bio-inspired lift and thrust that is delivered by the actuation of two or more pairs of semi-flexible wing surfaces. The work currently in progress at the Wright State University’s Center for Micro Air Vehicle Studies (CMAVS) continues to advance the development of efficient, agile flapping-wing MAVs through successive iterations of smaller, lighter, faster, and more-efficient designs. Because of the improvements made with each iterative generation, it becomes important to compare the performance characteristics of each new MAV with current, available micro-scale aircraft platforms. In particular, work being done in conjunction with the Wright-Patterson Air Force Base’s Air Force Research Lab (AFRL) seeks to identify individual strengths and weaknesses for different micro-scale UAVs. This leads to an analysis of each aircraft platform with respect to various tactical requirements of different operational theaters.

Understanding MAV flight characteristics becomes even more important when considering group behaviors. In such conditions, aircraft can operate jointly through the use of ad-hoc networking capabilities, such as that included in the 802.15.4 networking standard. Because aircraft operating in a group must be able to maneuver carefully and precisely so as to avoid interference with each other, understanding flight dynamics is key to optimizing flight path, which has a direct effect on battery life and the overall efficiency of the joint operation.

Existing work regarding the design and implementation of a test course generally is focused on one particular MAV platform; this work differs in both intent and approach, which is to create one single benchmark with which to test a variety of micro-scale UAVs, independent of their construction or target flight characteristics. By integrating advanced testing and motion capture methodologies with a thorough understanding of how a proposed course can maximize observations of flight characteristics, we can better understand the individual strengths and weaknesses of each aircraft platform with respect to the others.

**Miniaturization of Micro Air Vehicles – A Case Study**

*Strebig, Travis R, Wright State University, Dayton, OH*

*Huang, George, Wright State University, Dayton, OH*

*Smith, Todd, Wright State University, Dayton, OH*

*Feist, Alexander, Wright State University, Dayton, OH*

Miniaturization of Micro Air Vehicles – A Case Study
Travis Strebig, P.G Huang, Todd Smith, Alexander Feist
Department of Mechanical and Materials Engineering
Wright State University
3640 Colonel Glenn Hwy.
Dayton, OH 45435-0001
USA

In the field of Micro Air Vehicles (MAV), size and weight is an important factor. Currently, at the Wright State Center for Micro Air Vehicles, design changes are being made that will drastically reduce the size and weight of the current flapping-wing MAV. The miniaturization process will create a new maneuverable MAV that will weigh around 5 grams while utilizing a wingspan of 10cm or less; this will be approximately the size of the average human hand.

In the miniaturization process, optimization of the system is required. The first step in the process is to reduce the size of mechanical components, such as the gears and frame, while still retaining the strength and integrity required
to remain reliable. Through this process, the previous drive gear diameter of 19mm has been reduced to half the size. Other components including wings, tail, and rudder are also being reduced to 50% of the original size, resulting in additional weight reductions. Due to the previous stated weight reductions, smaller boards and actuators are also being implemented for additional weight savings. These changes will help to obtain the 5 gram goal. None of these modifications are possible without continuous advances in micro machining and printing. Utilizing cutting edge technology, the miniaturization process of MAV technology is possible.

Conceptual Design Tool to Analyze Electrochemically-Powered Micro Air Vehicles

Pederson, Doug J, AFIT, Dayton, OH

A multi-fidelity conceptual design tool was developed to assess electrochemically-powered micro air vehicles (MAVs). The tool utilized four areas of contributing analyses (CAs) in the areas of aerodynamics, propulsion, power management, and power sources in order to determine the endurance duration of a given mission. The low-fidelity aerodynamic CA consisted of drag polar calculations and the high-level CA used a vortex theory code called Athena Vortex Lattice (AVL). The low-fidelity track of the propulsion CA employed QPROP, a program that used blade-element and vortex formulation, to predict propeller-motor combination performance. Alternatively, the high-level propulsion CA used a MATLAB code that combined data from experimental propeller tests and known motor information to determine motor and propeller performance. The power management CA sought to maximize the performance of the electrochemical power sources by determining the split between power-dense and energy-dense devices. The low-fidelity power management CA was a user-defined split and the high-fidelity CA was an optimization algorithm to maximize endurance duration. The power source CA used specific energy and power calculations for low-fidelity analysis and both polarization curves and Ragone plots for the high-fidelity track. Model Center software allowed for easy integration of each of these CAs into one model. Based upon the state of the art battery and fuel cell technology, the model resulted in an endurance duration of 20.6 minutes for and a total mission time of 40.6 minutes. Looking to the near future in electrochemical technology, the endurance duration was extended to 37.5 minutes for and a total mission duration of 57.5 minutes.

Leading edge serrations on flat plates

Cranston, Brian, University of Dayton, Dayton, OH
Altman, Aaron, University of Dayton, Dayton, OH

Leading edge serrations on a wing can have profound effects on the associated aerodynamics. They are seen in nature, and could possibly have a link to improved acoustic signature that can be an indicator of the overall lift and drag performance of the planform. Our experiment begins with the most basic wing planform and wing profile, a rectangular flat plate, and will quantify the aerodynamic effects of leading edge serrations at low Reynolds numbers. We accomplished this by measuring lift and drag forces for flat plates with different size serrations on the leading edge, at different angles of attack, and at different speeds in a low speed wind tunnel. Regions of interest discovered from the force results used flow visualization captured with a high speed camera to help explain any aerodynamic characteristics resulting from the different serration densities. The data collected created a baseline for further research on more advanced planforms and leading edge features and could contribute to subsequent integration into low speed aircraft designs. In analyzing the initial data collected the results have shown that at low Reynolds number the serrations had little effect on the lift curve slope of the baseline flat plate. The smallest serration size showed an overall increase in lift compared to the other serration sizes and baseline flat plate. However, all of the serrations delayed stall to some extent. From the smallest to the largest serrations the stall angle increased with serration size. The large serrations had the most prominent effect on the delay of stall with however a less steep lift
curve slope. A similar trend can be seen from a drag standpoint in that as the serration size grew the coefficient of drag decreased for the same angles of attack. Overall the smallest serration size showed the greatest benefit.
Simulations of co-axial ducted jets with and without forward flight effects

Mihaescu, Mihai, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim J, University of Cincinnati, Cincinnati, OH

Jet engine nozzle design can significantly change the behavior of the exhausting flow as well as its acoustic signature. Research carried out on jet noise suppression methods led to a variety of nozzle concepts. One of these includes mixing of the cold fan-stream with the hot core-stream within the exhaust external nozzle. Such long duct mixed flow exhaust systems shield with the surrounding long duct (i.e. secondary nozzle) some of the fine-scale turbulent mixing noise generated downstream of the trailing edge of the primary nozzle, within the shear-layer between the primary and secondary streams. Experimental or computational research performed in the past on subsonic long-duct mixed flow exhaust systems considered axisymmetric confluent mixer configurations (i.e. conical primary nozzle) or forced mixer devices (i.e. lobed primary nozzles) without tertiary flow effects.

The present paper shows the obtained numerical results concerning a round co-axial ducted jet with and without tertiary flow effects. The flow is simulated by using the steady-state, compressible, Reynolds-Averaged Navier-Stokes formulation with Realizable $k$-$\varepsilon$ turbulence model. The comparisons between the numerical and the preliminary experimental stereoscopic Particle Image Velocimetry data just downstream of the nozzle for the co-axial ducted jet without tertiary flow show a good agreement. The final paper will contrast and compare the data obtained for the co-axial ducted jet with and without tertiary flow, emphasizing the differences between the two cases in terms of velocity, temperature, and turbulent kinetic energy levels.

An Investigation of Cavity Resonance and its Relationship to Store Force and Moment Loading

Coley, Christopher, Air Force Institute of Technology, Wright-Patterson Air Force Base, OH

Achieving a safe and acceptable release of a store from an aircraft is not a trivial matter. The flowfield around the vicinity of the store influences the release characteristics for the store. This is especially true in the case of a release from an internal bay, where the flow is highly unsteady. The flow characteristics are dominated by the interaction of the shear layer which spans the cavity opening and the cavity itself, creating an acoustic feedback loop which ultimately creates a highly turbulent flow. Pressure transducer information is often used to estimate store loading, although no direct correlation between the observed frequencies from the pressure transducers and the store loading has been developed.

To investigate the relationship between the acoustic modes present in the cavity and the force and moment loading on a store released from a bay, a computational fluid dynamics analysis of cavity flows is conducted. The acoustic modes of the cavity are calculated from the pressure fluctuation histories along the cavity ceiling and walls. Empty cavity solutions are compared to the frequencies predicted by the Rossiter equation and to experimental data from the Weapons Internal Carriage and Separation database. Additionally, the cavity pressure fluctuation spectra are compared to the force and moment loading spectra for a store located in the cavity in both the carriage position and the shear layer.
Numerical Investigation of Transitional Flow over a Rapidly Pitching Plate

Garmann, Daniel J., AFRL/RBAC, Wright-Patterson AFB, OH

A computational study of a plate undergoing high amplitude, pitch, hold, and return motions is presented. An implicit large eddy simulation (ILES) technique was used to capture the laminar-to-turbulent transition process as the plate was pitched to high angles of attack. Simulations were performed for Reynolds numbers between 5,000 and 40,000 with motion profiles of varying accelerations and hold times. The simulations show extremely favorable qualitative flow field comparisons of span-averaged stream-wise velocity and out-of-plane vorticity with available experimental PIV results. The computed flow field structures at specific Reynolds numbers of 5,000 and 40,000 are compared for two motion profiles and shown to be relatively similar despite the rapid transition of the higher Reynolds number cases. Good agreement is found between the experimental and computational aerodynamic loads throughout the motions.

Orographic Cloud Formation and Evolution about Ice Giant Vortices

Warning, Sally W., Saint Louis University, Saint Louis, MO
LeBeau, Raymond P., Saint Louis University, Saint Louis, MO
Palotai, Csaba, University of Central Florida, Orlando, FL

Large geophysical vortices provide a useful and interesting examination into the computational simulations of atmospheric dynamics and meteorological phenomena. Vortex features observed on Uranus and Neptune have exhibited similar changes to that of Earth hurricanes, responding to internal and external conditions by changing strength and location both in shape and in meridional drift. The most notable of these dynamic vortices was the original Great Dark Spot (GDS-89) observed by Voyager II in 1989, which through eight months of observation drifted towards the equator by ten degrees in latitude and oscillated in shape over an eight-day period during the month of closest observation. The Ice Giant Dark Spots are also notable for co-existing with a bright methane ice cloud that moves with the vortex as it travels through the atmosphere. These clouds have been characterized as likely orographic clouds, generated as the atmosphere is pushed over the vortex. Both observations and meteorological simulations have indicated these persistent companion clouds may increase the stability of these vortices, allowing them to persist longer than vortices without companions. Clouds may also mark the existence of a vortex able to maintain a long-term orographic cloud, but does not provide sufficient contrast the planet’s natural hue in order to be visible.

This presentation will examine the link between these clouds and vortices, both behaviors seen on the surface and through vertical layers. A numerical model called the Explicit Planetary Isentropic-Coordinate General Circulation Model (EPIC GCM) is used since observation data is limited. This data provides the basic knowledge needed to initialize the model, such as zonal wind distribution and vertical temperature-pressure profiles. Vortices are also initially set, but are allowed to progress computationally. A methane microphysics model tracks the distribution of this trace gas and allows for the development of cloud features. Results are compared to observations in order to confirm validation of the simulation techniques. The process of orographic cloud formation on Earth and on the Ice Giants is likewise similar, even if the atmospheric chemistry and forcing mechanisms are different. The ultimate goal is the creation of a validated ice giant model that can be used to link data from isolated observation windows into a more continuous evolution of the atmospheres of these planets.
Low Reynolds Number Loss Behavior of Highly Loaded Low Pressure Turbine Airfoils with Forward and Aft Loading

Lyall, M Eric, AFIT, Wright Patterson AFB, OH
King, Paul I, AFIT, Wright Patterson AFB, OH
Sondergaard, Rolf, Turbine Branch, Propulsion Directorate of AFRL, Wright Patterson AFB, OH
Clark, John P, Turbine Branch, Propulsion Directorate of AFRL, Wright Patterson AFB, OH

This talk presents an experimental and computational study of the midspan low Reynolds number performance of two highly loaded low pressure turbine airfoils, designated L2F and L2A, which are forward and aft loaded, respectively. Both airfoils were designed with incompressible Zweifel loading coefficients of 1.59. Computational predictions are provided using two codes, Fluent (with k-kl-ω model) and AFRL’s Turbine Design and Analysis System (TDAAS), each with a different eddy-viscosity RANS based turbulence model with transition capability. Experiments were conducted in a low speed wind tunnel to provide transition models for computational comparisons. The Reynolds number range based on axial chord and inlet velocity was 20,000 < Re < 100,000 with an inlet turbulence intensity of 3.1%. Predictions using TDAAS agreed well with the measured Reynolds lapse rate. Computations using fluent however, were overly conservative, with separation occurring at significantly higher Reynolds numbers as compared to experiment. Based on arguments from rapid distortion theory, the discrepancy is likely due to the turbulence model under-predicting the freestream turbulent kinetic energy, thereby delaying the onset of bypass transition.
**Session 28: Space Applications**

*Chair: Eric Swenson, AFIT*

---

**Design of Attitude Control Actuators for a Simulated Spacecraft**

*McChesney, Christopher G., Air Force Institute of Technology, Wright Patterson AFB, OH*

*Swenson, Eric D., Air Force Institute of Technology, Wright Patterson AFB, OH*

The Air Force Institute of Technology's attitude dynamics simulator, SimSat, is used to for hardware-in-the-loop validation of new satellite control algorithms. In order to provide the capability to test algorithms for control moment gyroscopes, SimSat needed a low cost control moment gyroscope array. The goal of this research was to provide this capability through the design, construction, testing, and validation of a control moment gyroscope array for SimSat. The first step in this process was to establish design requirements necessary for sizing the array. These requirements were based on the size and available space on the vehicle, as well as maneuver requirements. Next, the vehicle and control dynamics were modeled to determine actuator requirements and provide a baseline for validation. Actuators were then built, calibrated, and installed on the vehicle. The actuators were then validated against the dynamics model. Testing shows some deviation from the expected behavior as a result of small misalignments from the theoretical design.

---

**Feasibility Study of Orbital Maneuvering using the Atmosphere and Electric Propulsion**

*Co, Thomas C, Air Force Institute of Technology, Dayton, OH*

*Black, Jonathan T, Air Force Institute of Technology, Dayton, OH*

Traditional space operations are characterized by large, highly-technical, long-standing satellite systems that cost billions of dollars and take decades to develop. Many branches of the US government have recognized the problem of sustaining current space operations and have responded by heavily supporting research and development in a field known as Operationally Responsive Space (ORS). ORS research focuses on hardware, interfaces, rapid launch and deployment with the overall goal of reducing per-mission-cost down to $20 million. Some research is also done in the area of orbit design to maximize the coverage time over specific areas. However, there are no studies on the feasibility of maneuvering to different orbital planes once an asset is launched. If it is possible for the user to task a satellite in a timely and fuel-efficient manner, then its mission can be modified to meet requirements based on world events. The existing paradigm on maneuvering is that it is too cost-prohibitive especially to perform orbital plane changes. This paradigm along with traditional space programs have to change and a transition to small, responsive, low-cost, and rapidly available systems must take place to meet the needs of space users. This paper proposes a phased approach that will help this transition by first using existing technology to perform slow, yet efficient orbit changes to repeatedly maneuver and perform multiple missions. The eventual goal is to achieve more rapid orbital changes by harnessing the potential of orbital and sub-orbital flight of space planes which would enter the atmosphere, perform the desired change and ‘skip’ back to space.

Many operational satellites are maneuverable but they are designed to operate in ‘static’ parking orbits. The technology to maneuver efficiently is available and in use, but the concept of operation (CONOP) needs to change. Low thrust electric engines enable satellites already in orbit to perform slow, precise, and highly efficient station-keeping maneuvers. The current CONOP intends for the spacecraft to arrive at its orbital state and maintain its orbit, almost exclusively, for the life of the vehicle. Most spacecraft are designed in this manner so not much thought is given to powered flight and the potential it has. When necessary, these engines can move large satellites into orbits...
to serve different terrestrial theaters in the case of a geosynchronous system, or change the time a satellite arrives over a target (time over target [TOT]) for a low-earth orbit system. To harvest this potential, the CONOP must be build around the assumption that these spacecraft do not necessarily have to operate within the orbit they were first launched into. This paper demonstrates that existing satellites can maneuver significantly to change its TOT and provides a feasibility study by comparing the use of low-thrust, highly efficient electric propulsion to traditional chemical propulsion.

**Satellite Detection and Real-time Orbit Estimation with Commercial Telescopes**

*Briggs, Gregory C, Air Force Institute of Technology, Wright-Patterson Air Force Base, OH*

Initial orbit estimation with commercial telescopes has been demonstrated by previous Air Force Institute of Technology research; however, measurements used were obtained via prior knowledge of the satellite’s orbit, and results showed limitations due to lack of range information from optical measurements. In order to mitigate these obstacles, a method for satellite detection and tracking using commercial telescopes and cameras is being developed in conjunction with real-time orbit estimation. With these capabilities, once a satellite is detected, the telescope could track the satellite across the sky, recording and analyzing imagery data to estimate the orbit in real-time. This orbital estimate would then allow a second telescope (separated by a suitable baseline) to begin tracking the satellite as well. With both telescopes tracking, triangulation would provide the desired range estimation. In this way, one could generate higher fidelity orbit estimates. This presentation will show the techniques for satellite detection and real-time orbital estimation used in this research.

**Augmentation of ground-based satellite tracking telescope system**

*Moran, Gregory, AFIT, Dayton, OH  
Cobb, Richard, AFIT, Dayton, OH*

AFIT is investigating the feasibility of integrating a newly designed ranging sensor into its existing optical satellite tracking system. A modified system engineering process is being used to design, analyze, build, and test the rapidly developed prototype of this new capability. The systems engineering application focuses on rapid acquisition; Fast Inexpensive Simple, and Tiny; Test early, test often; risk management, and systems architecture. Both the system and the associated test plans are being developed in parallel while assessment of the utility of each is continuous throughout. Incremental testing has been conducted and this briefing will focus on the results from those tests. Lessons learned will also be addressed which can be applied to future research in this area.

**Satellite Detection Utilizing Reflected RF Signals**

*Barker, Richard Thomson, AFIT, Dayton, OH*

Increases in technology have made it possible for a smaller entity to design and build a system at low cost to perform actions which would traditionally be accomplished by a larger entity such as the federal government. One such action is satellite detection and tracking. As the cost of equipment goes down and the access to information goes up it is useful to determine what we are capable of building with limited resources. This presentation describes the theory and the equipment required for a system to detect low earth orbiting satellites by utilizing reflected radio frequency signals resulting in an academic exercise giving students hands on experience with orbital predictions and satellite detection.
Session 29: Combustion III

Chair: Jason Slagle, GE

Low-dimensional finite-rate chemistry combustion models based on discrete dynamical systems

Barnes, Jeremiah J., Paul Laurence Dunbar High School, Lexington, KY
McDonough, James M., University of Kentucky, Lexington, KY

As turbulence levels increase, combusting systems deviate farther from easily-computed equilibrium states. The goal of this research is to study sets of elementary reactions representing combustion of diesel fuel (and possibly gasoline) in air for turbulent, non-equilibrium situations. An important part of any such mechanism is that corresponding to H2 and O2 reactions. In the present talk we discuss formulation of a discrete dynamical system modeling turbulent combustion via a seven-species, nine elementary reaction mechanism for this chemistry. We provide results in terms of time series and phase portraits (scatter plots) for velocity, temperature and various chemical species, and we make qualitative comparisons with experimental data. We note that the technique to be presented applies more generally, and in particular could be used in modeling turbulent combustion of aircraft engine fuels.

Flashback characteristics prediction using flame front and velocity field measured by PIV

Perelstein, Yuri, Dept. Aerospace Engineering and Engineering Mechanics, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim J., Dept. Aerospace Engineering and Engineering Mechanics, University of Cincinnati, Cincinnati, OH

A vortex break down mechanism is often used as an aerodynamic flame holder in combustion chambers. As the swirling air enters combustion chamber, strong adverse pressure gradients resulting from area expansion reverse the flow and a central toroidal vortex is set up. This allows holding the flame. Among the processes, which may prevent a proper flame holding, is flashback. This is when the flame moves upstream into the swirl generator. Flashback overheats the components of combustor, and thus, reduces engine life.

Three mechanisms of flashback are known: upstream propagation of the flame if the flame speed is higher than the velocity of reacting mixture; flame propagation in the boundary layer with small enough velocity gradient; strong combustion instabilities leading to flame propagation into the premixing tube.

Forth mechanism was experimentally discovered (Fritz et. al., 2001): Combustion Induced Vortex Breakdown (CIVB). This mechanism exists in premixed combustion systems only. Whereas for isothermal flows (at the exit of any particular swirl generator) the location of recirculation vortex is controlled by the swirl number, addition of heat (due to combustion) results in displacement of the reaction zone towards the swirl generator.

Flashback detection experiments were performed for premixed combustion of swirling mixture of natural gas and air for CIVB prone configurations. Velocity field and flame front location at near-flashback conditions were measured using PIV. As the density of the gas changes due to combustion, so does the amount of the seeding particles per unit volume. Therefore, variation of the average intensity (of the light emitted by the particles) in the emitted light images obtained from PIV measurements was used to determine the flame front contour, and its transient behavior. Using this approach, flame front location and velocity field are measured simultaneously. This eliminates the necessity of using PLIF to detect the flame front contour, and thus, simplifies the experimental setup.

In these experiments, confined flames were created using the Triple Annular Research Swirler (TARS), a lean direct
injection fuel nozzle. The TARS consists of three concentric swirlers and two separate fuel circuits. The PIV measurement system included an Nd-YAG laser and frame straddling camera.

Flashback characteristics, for given a swirler and a set of air flow rates, were found based on comparison of characteristic time scale of combustion to the characteristic time scale of turbulent flow. Characteristic time of the chemical reaction was calculated using different approaches. At the near-flashback conditions, the minimal distance between the flame front and the stagnation point of the recirculation region is attained. The time required for the flame front to pass this critical distance is used as a characteristic combustion time scale. Another approach attempted was to use the time scale of laminar combustion for this purpose. Obtained flashback characteristics are presented and compared.

A Study of Lean Direct-Injection Flames through Simultaneous PIV and OH*
Chemiluminescence

Villalva-Gomez, Rodrigo, University of Cincinnati, Cincinnati, OH
Gutmark, Ephraim J., University of Cincinnati, Cincinnati, OH

Simultaneous particle image velocimetry measurements of the velocity field and OH* chemiluminescence acquired with low exposure times were performed in a swirl-stabilized dump combustor, allowing for instantaneous visualization of both the velocity field and its corresponding OH* distribution. These techniques allow improved understanding of the relationship between the velocity fields and reaction regions for a turbulent swirling flame. Swirl-stabilized combustors rely on vortex breakdown of a swirling flow to provide aerodynamic anchoring of the flame. The sudden expansion of the swirling flow results in the creation of a recirculation bubble which supports the heat and mass transfer processes involved. The swirler configuration, inlet conditions, and equivalence ratio define the velocity field inside the combustor and thus the flame position and structure. The shape and location of the recirculation bubble will greatly influence the flame, and with it phenomena such as instabilities and flashback. The main focus of the presented research is on the recirculation bubble and reaction zone location and their effect on flashback as equivalence ratio increases. The effect of equivalence ratio and expansion ratio at the dump plane are shown, both for averaged values and instantaneous corresponding image pairs. Images of the velocity field overlayed on the corresponding OH* distributions are also included. For the particular swirler configuration studied, increasing equivalence ratio moves the tip of the central recirculation zone downstream while moving the reaction region upstream, until flashback occurs, changing the flow regime and eliminating vortex breakdown inside the combustor. Except for cases near lean blow-out, the edge of the reaction region is located at the shear layer between the swirling jet exiting the nozzle and the central recirculation zone; this applies for both the averaged results and for instantaneous velocity fields and OH* intensity image pairs. Changing the diameter of the test section affected the shape of the recirculation region and the sensitivity of the locations of the recirculation bubble and the flame to changes in equivalence ratio. Statistical analysis was performed on the individual image pairs; the recirculation bubble and flame tip locations were extracted for each image pair and used as source data. Statistical data on the individual images support the averaged trends and yield further insight into the flame/recirculation bubble relationship across a range of equivalence ratios. The correlation between the location of the flame tip and recirculation bubble stagnation point for corresponding image pairs requires additional study to better understand their behavior, but the simultaneous measurements provide the necessary tools to do so.
**Direct Detonation Initiation of a Pulse Detonation Engine from Shockwave Transfer through a Crossover Tube**

Driscoll, Robert, University of Cincinnati, Cincinnati, OH  
Stoddard, William, University of Cincinnati, Cincinnati, OH  
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH

Pulse Detonation Engines are becoming a reoccurring research topic due to their simplicity in design and theoretically improved thermodynamic cycle over modern air breathing combustors. This near-constant volume combustion system is essentially a cyclic device operating with five distinct phases: Fill, Ignition, Deflagration-to-Detonation Transition, Exhaust, and Purge. Generally, the Exhaust and Purge phases occur as a product of the overall cycle design of the system. Recent research has aimed to shorten or even combine the Ignition and deflagration-to-detonation transition phases with the expectation of increasing the overall cycle efficient. Some research has been done using plasma ignition while other research implements devices such as the schelkin spiral to transition to a detonation sooner. Unfortunately, for multi-tube, high frequency PDEs, ignition devices are required to have a firing frequency ranging up to 40 Hz for long intervals. To accomplish this, each detonation tube would need individual energy supplies. In addition, transition devices can create large pressure drops across the detonation tube. A simple solution would be to use one detonation tube (driving tube) to ignite another tube (driven tube). Essentially, use a transferred shockwave to reflect off the inner wall of the driven PDE causing direct initiation of a detonation. With this implementation, no transition device is needed and the only limitation to fire frequency is the overall cycle frequency.

A University of Cincinnati experimental pulse detonation study was carried out to investigate a dual pulse detonation engine crossover system. Cases using crossover tubes of varying lengths and bend angles were tested. Detonation tubes were connected using an air-filled crossover tube. Only the driving detonation tube was ignited. From burning wave speeds, results have shown a transferred shockwave reflecting off the inner wall of the driven detonation tube can achieve direct initiation of a detonation. The results have also yielded cases where the initial shockwave reflection does not directly initiate a detonation in the driven detonation tube but rather causes ignition leading to likely accelerated deflagration to detonation transition. Overall results have shown that for respective crossover length and bend configurations there is a specific length range in which direct detonation initiation is possible. Multi-cycle results show potential for functioning dual pulse detonation engine systems utilizing direct detonation.

**Numerical Study of Shock-Reflection Obstacles on Detonation Initiation in Pulse Detonation Engines**

Stoddard, William A, University of Cincinnati, Cincinnati, OH  
Gutmark, Ephraim J, University of Cincinnati, Cincinnati, OH

The inherent potential for efficiency of the Pulse Detonation Engine (PDE) has made it a quickly developing area of research. The PDE system runs through a cycle that can be divided into 4 phases: Fill, Ignition, Exhaust, and Purge. The fill phase introduces a fuel-oxidizer mixture into a PDE tube. The exhaust happens naturally as the high pressure detonation products exit the tube. Purge is then done with oxidizer or fuel only to prevent ignition of the next cycle prematurely. The ignition and deflagration to detonation transition (DDT) results in a loss of efficiency for the portion of fuel going through deflagration before DDT. A process to directly ignite a second PDE’s detonation wave using a shock carried through a crossover tube from an initial PDE’s detonation wave has been demonstrated in previous studies at the University of Cincinnati. These result in no DDT length requirement, and a more efficient system, as the shock reflects inside a second PDE to directly lead to detonation. An initial experimental look at obstacles at the end of the crossover tube just inside the second PDE has been demonstrated as...
a proof of concept. Multiple successful detonation transfers were seen using a V-shaped obstacle just inside the second PDE over the end of the crossover tube, as the shock was focused and temperature increased due to shock reflection. The need to improve performance in order to expand the range of fuel-oxidizer mixtures for which this ignition method is effective is the current subject of study.

A numerical study using a Reynolds-Averaged Navier-Stokes (RANS) Computational Fluid Dynamics (CFD) program Fluent, by ANSYS is being performed to test potential new obstacles at the end of crossover tubes for igniting PDEs directly. These will test various angled obstacles and tube geometries of the second PDE and the ability to create a detonation will be determined. These designs will then be modified and re-tested. The best designs could be tested experimentally in the future.
Automated Quadrature-free Discontinuous Galerkin Method Applied to Viscous Flows

Galbraith, Marshall C, University of Cincinnati, Cincinnati, OH
Orkwis, Paul D, University of Cincinnati, Cincinnati, OH
Benek, John A, Wright Patterson Air Force Base, Dayton, OH

Abstract The discontinuous Galerkin (DG) formulation has received much attention in recent years as a promising higher order method for solving the Navier-Stokes equations. However, it suffers from high operation count compared to traditional finite volume and finite difference schemes. The high operation count can partially be attributed to the lack of a single set of numerical quadrature points that are capable of accurately integrating both volume and boundary integrals. In order to reduce operation counts, quadrature-free integration is employed and implemented through a pre-processor (PyDG) developed by the authors. The PyDG pre-processor relies on a freely available symbolic manipulation package to perform the analytical integration. The integrand is expressed as a matrix operator where all zero entries are automatically eliminated. This leads to a reduced operation count compared to traditional numerical quadrature implementations. In addition, this formulation allows numerical implementations of partial differential equations to be expressed in a manner similar to the analytical equation. Using the PyDG pre-processor, the authors have also developed a C++ polynomial library (DGPoly++) that uses overloaded operators to allow expressions involving polynomials to be written as if they are scalars. This feature permits rapid numerical formulation of computer programs for solutions of partial differential equations. The quadrature-free DG method is demonstrated here for a set of Euler and Navier-Stokes flows.

Characterization of the dynamical behavior of the compressible `poor man's Navier--Stokes equations"

Strodtbeck, Joshua P, University of Kentucky, Lexington, KY
McDonough, James M, University of Kentucky, Lexington, KY
Hislop, Peter D, University of Kentucky, Lexington, KY

The compressible `poor man's Navier--Stokes equations" (PMNS equations) are a discrete dynamical system derived from a Galerkin expansion of the compressible Navier--Stokes equations. A sketch of the derivation is presented, with attention given to the differences from the original, incompressible case. A thorough numerical investigation of the bifurcation behavior is given in the form of regime maps characterizing the different kinds of dynamical behavior, bifurcation sequences, power spectral density analysis, time series, and phase portraits. As in the case of previously studied incompressible PMNS equations, the full range of dynamical behavior associated with physical turbulence is exhibited by the system of coupled maps. The conclusion is drawn that this system can be viable as a source of temporal fluctuations in synthetic-velocity subgrid-scale models for large-eddy simulation.
Curve fitting 3-D experimental turbulent flows with the poor man's Navier-Stokes equation

Mitchell, Taylor C., Paul Laurence Dunbar High School, Lexington, KY
McDonough, James M., University of Kentucky, Lexington, KY
Bailey, Sean C. C., University of Kentucky, Lexington, KY

The 3-D poor man's Navier-Stokes (PMNS) equation is a discrete dynamical system (DDS) whose solutions retain much of the dynamical behavior of the partial differential equations from which it is derived, and yet is very easily executed---far faster than real time. We briefly outline derivation of this DDS and then discuss a general procedure for curve fitting DDSs to chaotic experimental data. This technique (first introduced by McDonough et al., Appl. Comp. Math. 1998 and later used by Yang et al., AIAA J. 2003 in a 2-D Navier-Stokes setting) employs a least-squares method to generate a global (long-time) fit of chaotic data that produces details of experimental time series in a manner more appropriate for representing fluid turbulence (including sensitivity to initial conditions) than are short-time extrapolation techniques found in, for example, Farmer and Sidirowich, Phys. Rev. Lett. 1987. We apply this least-squares approach to three-component velocity measurements in grid turbulence described by Bailey and Tavoularis, J. Fluid Mech. 2008, and demonstrate that the PMNS equation can reproduce the structure of all three components of the experimental velocity which is nearly isotropic in two spatial directions. We present comparisons of time series, spatial and temporal energy spectra, and various statistics including auto and cross correlations, flatness and skewness. A possible application of such curve fits would be to real-time control of turbulent (or transitioning) physical fluid flows.

Numerical Experiments With Sub-grid Scale Synthetic Velocity Model: Poor Man's Navier-Stokes Equations

Polly, James B., University of Kentucky, Lexington, KY
McDonough, James M., University of Kentucky, Lexington, KY

The Poor Mans Navier-Stokes (PMNS) equations comprise a computationally inexpensive discrete dynamical system (DDS) which permits thorough numerical study of local flow behaviors. Time series, power spectra and regime maps have been studied in much detail for the 2-D PMNS equations permitting precise identification of solution behavior types. Our current work extends the associated analysis techniques to the 3-D PMNS DDS. We begin with a study of 3-D isotropic flow and compare this with anisotropic turbulence, as generated by the PMNS equations, in terms of qualitative features of time series and power spectra. We also investigate agreement with the Kolmogorov K41 theory in the isotropic case. Solution behavior types and related bifurcation sequences will be further characterized via regime maps of both cases.

Distributional DSMC with BGK Collision Modeling

Schrock, Christopher R., Department of Mathematics and Statistics, Air Force Institute of Technology, Wright Patterson AFB, OH
Wood, Aihua W., Department of Mathematics and Statistics, Air Force Institute of Technology, Wright-Patterson AFB, OH

Over the last forty years, the Direct Simulation Monte Carlo (DSMC) method has gained acceptance for practical applications in rarefied gas dynamics. The method relies upon a stochastic simulation of the interactions of a portion of the molecules in the gas to approximate a solution to the Boltzmann equation. Weak L^1 convergence of such methods has been established by Babovsky, Illner, and Wagner. Recently, the authors proved that by a slight modification to the approximation of the velocity distribution function, such methods can achieve pointwise convergence for bounded solutions.
In traditional DSMC methods, simulated particles possess only a single velocity vector and energy state. However, as only a fraction of the total number of actual particles in the gas can be simulated, each simulated particle is assumed to represent $W = N/N_p$ actual particles. In practice $W$ may be quite large ($10^6$ or more), therefore allowing only a single velocity vector per particle is equivalent to assuming that millions of particles all share the exact velocity vector.

The authors believe such methods could greatly benefit through the use of distributed particle velocities. As each simulated particle in fact represents a (potentially large) collection of actual particles, a nonsingular velocity distribution function should provide a more accurate representation of the true behavior of those particles. Collision interactions between simulated particles could then be treated as a relaxation process over the collection of actual particles they represent. We have termed this approach Distributional DSMC (D-DSMC).

In this presentation we present the results of employing the Bhatnagar-Gross-Krook (BGK) equation to evolve particle distribution functions in D-DSMC simulations. This application results in a hybrid stochastic-deterministic method where collision selections are still made utilizing stochastic criteria, but collision modeling is completed in a distributional deterministic framework. Use of the deterministic collision modeling is facilitated by the fact that simulated particles possess not just a velocity vector, but rather an estimate of their velocity distribution function. This method has been applied to the well-known space-homogeneous solution of Bobylev and preliminary results indicate a drastic reduction in the variance associated with such methods is achievable in this framework.
Session 31: CFD Flight Dynamics

Chair: Rolf Sondergaard, AFRL/RZ

A Study of the Effect of a Cabin Door Opening in Flight on the Performance and Handling of a General Aviation Aircraft

Moehle, Robert C, Saint Louis University, St. Louis, MO

Little research has been performed on the effect of a door unexpectedly opening in flight on the performance characteristics of an unpressurized general aviation aircraft near the ground. Manufacturers especially do not perform this kind of testing—in general, the pilot’s operating handbook notes that flight with the doors or hatches open is prohibited.

There have been about 100 reported incidents of doors or windows opening in flight at low altitudes since 1964. Of these, sixteen recovered to land safely, while the rest resulted in an accident (NTSB). The following example was selected for review because the case is currently in litigation:

On September 3, 2008, a Socata TB-20 Trinidad was operating from a flight school in Iowa on a cross-country flight to Wisconsin. The right cabin door opened in flight, and the pilot initiated an emergency diversion while the instructor held the door closed. During the missed approach procedure, the door again flew open, and the aircraft rolled to the right and crashed. The official NTSB report stated the probable cause as “The left seat pilot's failure to maintain control of the airplane during a go-around and the inadvertent in-flight opening of the cabin door for undetermined reasons. Contributing to the accident was the distraction of the door opening” (NTSB Identification: CHI08LA272).

The purpose of this project is to determine the change in the aircraft’s performance caused by the opening of the cabin door. Data is collected using computational and experimental methods, and the results are compared between these two techniques and with known data.

One future application of this research could involve programming the performance derivatives into flight simulation software for a group of pilots to fly the same mission. The results of such trials could point toward whether the aircraft was truly unrecoverable or if distraction played a larger part in the accident. This information could be used in court, and if shown to provide helpful evidence, the approach could be used for investigating many aviation accidents.

Analysis of Dispersal of Small Bodies Using an Implicit Immersed-Boundary Method

Gaston, Zachary, Wright State University, Dayton, OH
Dong, Haibo, Wright State University, Dayton, OH

Currently, little is known about the reaction of engineered MAVs to gusts, and even less is known about how to design MAV systems that are gust tolerant. In this summer work, we intend to develop an implicit immersed-boundary method based DNS tool for simulating freely moving objects at low Reynolds numbers and perform a parametric analysis of trajectories of small bodies dropped in air. The problem is very challenging owing to many factors, including inherent fluid-structure coupling between the falling body and shed vorticity, as well as the chaotic nature of trajectories for certain para-metric selections. A Cartesian grid based sharp interface immersed boundary solver is eventually developed to handle such flows in all their complexity. Simulations are carried out in order to
examine the sensitivities of several key design parameters to fluttering or tumbling bodies. High resolution, high-speed video of fluttering ellipsoids is obtained and the data is processed using an in-house image-processing tool to extract real-world time varied coordinates. From this data, 3D kinematics is calculated and the trajectory of the descent is analyzed. Through this investigation, we hope to better understand the physics of freely moving bodies at low Reynolds numbers and pave the road for future research of statistical influence of gusts and the probabilistic response of bodies in gusty conditions.

Reduced-Order Models for Efficient Stability and Control Analysis

Butler, Jed H, AFIT, WPAFB, OH

It is well known that today there are four main avenues of obtaining aerodynamic data for an aircraft: Analytical analysis, wind-tunnel testing, flight testing, and Computational Fluid Dynamics (CFD). High-fidelity aerodynamic results of “simple” configurations are easy to come by via simple linear methods that exist. Difficulties arise as configurations of maneuvers that are nonlinear in nature are looked at and a better/faster/cheaper way of obtaining this data is an important issue for Acquisition Engineers in the Air Force. High-fidelity CFD simulations are currently able to provide accurate results for these nonlinear regimes, but the problem of hours of computational time is an issue when dozens of different configurations, flight conditions, and maneuvers are considered. This is where the areas of system and parameter identification come into play. These areas have been used to characterize the Stability and Control (S&C) of aircraft by some Reduced Order Model (ROM) in the past but techniques rely on the data available to develop these ROMS. In the past this data is obtained from flight test and/or wind tunnel test. The data obtained from these, although accurate, can be very sparse due to cost and limitations in maneuvers that can be conducted in wind tunnels. This is where CFD can be well utilized to help create a database of S&C data at relatively low cost with very few “holes” that then enables an accurate ROM, or actually, many different ROMS “fused” together to cover the full flight envelope to a desired accuracy. These “fused” ROMS can then be utilized to calculate the aerodynamic parameters required by acquisition engineers in a matter of seconds on a computer.

Future work to be examined for my thesis includes the generation and evaluation of a new training maneuver that will better fill the regressor space. The “regressor space” being the required variations of the model variables (angle of attack, pitch rate, etc.). For my thesis, I will be using a grid of the F-22 and use Kestrel as the solver. Kestrel is a recently released AF code that is based on the AVUS solver. Special attention has been given to implementing grid motion (among other things) for Kestrel. This allows for a specific maneuver to be specified and the CFD simulation is calculated for the entire maneuver and stability data can be extracted directly from this. This data will then be used to fill the regressor space for creating a ROM.

The presentation for DCASS will not display these results as they will not be completed until after this event. Instead, the background information of this topic, why it is important, and the process I will use will be presented. Along with this, preliminary work will be presented to include results of a model being run in Kestrel through some maneuver and data being extracted to fill the regressor space.

Verification of 6 Degree of Freedom Motion Capability in Kestrel

Finster, Kevin M, Air Force Institute of Technology, WPAFB, OH

As computation processing power continues its exponential trend, engineers in the field of computation fluid dynamics (CFD) continue to leverage these new advances in order to provide more accurate and reliable simulation results. One such effort is Computational Research and Engineering Acquisition Tools and Environments – Air Vehicles’ (CREATE-AV) Kestrel Fixed Wing Aircraft Tool. This modular multidisciplinary program allows for simulation of the determining factors of aircraft flight including aerodynamics, structural dynamics, kinematics, and
kinetics. In Version 1 of Kestrel, any desired motion was directly forced upon the model, rather than letting the calculated aerodynamic forces and moments act upon the model. A recent module added to Version 2 of this software package includes the capability of solving 6 degree of freedom (6DoF) motion through the use of the rigid body equations of motion. Initial testing of a Mk-82 munition shows promising results, but more detailed study is needed in order to validate this new capability. This project will consist of a CFD simulation of an unguided air weapon dropped at several varying freestream flight conditions and utilizing the new 6DoF module. The results will be compared to that of analysis done by the Air Force SEEK EAGLE Office (AFSEO). Aerodynamics forces and moments, as well as the resultant motion, will be compared to these experimental results in order to validate the new software capability. Once this new 6DoF capability is validated, it will prove an invaluable resource to aid in the reliable and accurate computational simulation of aircraft flight.

The first step in this process is to get better acquainted with the Kestrel software. Since Version 2 is not yet released at this time, the first CFD runs will be static tests of the weapon at various angles of attack. Then, utilizing the prescribed motion module, a dynamic angle of attack sweep will be conducted. Results will be compared with available data, but the primary purpose is software familiarization.

**Transient Simulation of a Dual-Mode Scramjet Transition**

_Yentsch, Robert John, High-Fidelity Computational Multi-Physics Lab, Department of Mechanical & Aerospace Engineering, The Ohio State University, Columbus, OH_  
_Gaitonde, Datta V., High-Fidelity Computational Multi-Physics Lab, Department of Mechanical & Aerospace Engineering, The Ohio State University, Columbus, OH_

Scramjets are the most attractive form of air-breathing propulsion at Mach numbers higher than about six. However, their operation is complicated by the need to maintain combustion at relatively high velocities, causing low residence time in the combustor and high thermal loads throughout. At startup and under accelerating conditions, fixed geometric designs operate in dual-mode; i.e., at lower speeds a strong normal shock precedes the combustor, while at higher speeds the shock is swallowed to yield scramjet operation with supersonic flow throughout. A systematic high-fidelity unsteady analysis has been initiated to explore the key phenomena that arise during mode transition, with a focus on the configuration and flight path of the HIFiRE 2 Flight Test. The presentation will focus on two thrusts. First, the simulations are verified and validated by comparing steady state ground test results with those obtained at the NASA Langley Arc-Heated Scramjet Test Facility. Numerical issues such as grid resolution effects and combustion model validation are reported. The second thrust examines one of the most important facets of scramjet simulation: mixing of fuel with air abstracted by a jet in supersonic crossflow. Accordingly, numerical results on a helium jet in Mach 8.76 nitrogen are presented. The conditions mimic results obtained at the LENS hypersonic shock tunnels at CUBRC Buffalo with a new diagnostic capability being developed at The Ohio State University by the Nonequilibrium Thermodynamics Laboratory. Initial results in both thrusts are very promising. Strategies to resolve the key dynamics have been developed with overset, hybrid meshes for both thrusts. Initial results on the ground test configuration show reasonable agreement with experimental results, although some calibration of the combustion model will be necessary before the mode-transition study is undertaken later this year. The main features observed in the LENS study, including penetration distance and pressure distributions are also reproduced relatively well.
Three-Dimensional Modeling of Ablation by Control Volume Approach

Khan, Ovais U., Department of Mechanical Engineering, University of Kentucky, Lexington, KY
Martin, Alexandre, Department of Mechanical Engineering, University of Kentucky, Lexington, KY

Atmospheric entry/re-entry of vehicles takes place in the hypersonic flow regime. In this type of flow, fluid properties such as pressure, temperature and density change abruptly as they cross the bow shock, which results in the flow field between the shock and the vehicle surface to become extremely hot. This leads to large temperature gradients and skin friction within the boundary layer, and results in high heat transfer rates to the vehicle surface. Efficient design of the Thermal Protection System (TPS) is necessary in order to reduce the heat conducted into the body surface, optimized the heat shield and avoid structural failure of the vehicle. One of the available techniques is to use ablative material, which has been proven an efficient way to reduce the rate of heat transfer into the vehicle. However, ablation modeling is a challenging task, as it involves simulating complex processes of mass loss and energy absorption through complex chemical reactions. Moreover, in case of decomposing (charring) ablators, inner pyrolysis takes place. This results in the formation of porous material as well as inner gas, further increasing the complexity of the numerical model.

Available literature indicates that most of the past investigations either do not consider the actual physical processes taking place during ablation, or are limited to a one-dimensional model. The present investigation shows the development of a numerical model for simulating the multidimensional heat transfer phenomena that occurred in a typical ablative TPS. The newly developed model is verified using closed form analytical solutions and validated with available data. This effort consists of the first steps of an ongoing project to develop a comprehensive multi-scale, multi-physics and multi-dimensional material response code aimed at modeling charring and surface ablators.

Numerical Investigation of Blunt Body Hypersonic Flow with Applied Magnetic Field Distributions

Khan, Ovais U., Department of Mechanical Engineering, University of Kentucky, Lexington, KY
Martin, Alexandre, Department of Mechanical Engineering, University of Kentucky, Lexington, KY

Blunt objects are subject to severe aerodynamic heating and pressure loads in hypersonic regime. Flow ahead of the blunt region becomes quite complicated: high temperatures can initiate chemical reactions and ionization, and strong curved bow shockwave results in mixed supersonic-subsonic flow in the downstream region. Therefore, flow control becomes a critical task for reducing the drag and heat transfer on the body. Development in superconducting materials and improvements in artificial ionization techniques have resulted in re-consideration of electromagnetic field as an efficient tool for reducing heat transfer, drag, skin friction and modifying the shock wave location for hypersonic flow. In magneto-hydrodynamics (MHD), the interaction of Lorentz force generated after the application of magnetic field with the plasma, primarily affects the flow field. The direction of Lorentz force depends upon the orientation of applied magnetic field and flow direction. For a cruise flight; it is possible to change the direction of Lorentz force by varying the direction of applied magnetic field.

In this work, uniform, dipole and vortex type of applied magnetic field distributions have been implemented to determine an effective orientation of the Lorentz force for producing maximum effects over a hypersonic flow regime. The governing equations are composed of the Navier-Stokes equation modified to include the effect of magnetic field under low magnetic Reynolds number approximation. Flux vector splitting for the convective terms and central differencing for the diffusion terms are utilized. Time-explicit multistage Runge-Kutta scheme for time
integration towards steady state solution is implemented. The flow simulations are compared to the existing solutions.

Reduce Turnaround Time for Large-Scale CFD Simulations by Using Python

Chen, Yung-Yu, Department of Mechanical and Aerospace Engineering, The Ohio State University, Columbus, OH
Yu, Sheng-Tao John, Department of Mechanical and Aerospace Engineering, The Ohio State University, Columbus, OH

Nowadays, high-resolution simulations that use more than 10 million cells are commonplace for CFD. For such simulations, it is not unusual that pre- and post-processing the solution would take more time than calculating the solution. For simulations using more than 50 million cells, the time spent in non-calculation tasks becomes so significant, that routine utilization of CFD for design iterations becomes prohibitively expensive. There is no doubt, however, the trend of CFD is to use larger and larger meshes for fine resolution. The next milestone of high-performance computing for CFD is routine computation by using up to 1 billion cells. To this end, we have to reduce the costs of man-hour and turnaround time resulted from: (i) manual operations for pre- and post-processing, (ii) lack of parallel computing for pre- and post-processing, and (iii) intermediate input and output (I/O) for solutions. All these issues are related to the work flow of CFD simulation tasks. In this talk, we present a redesigned CFD work flow with extensive automation and extended parallelization. The work flow has been fully materialized in our newly developed code, SOLVCON, which is accessible at http://solvcon.net/.

We pursue automation to handle the first issue. In our approach, scripting, i.e., writing computer programs with a scripting language, for CFD codes and related software is the key. By scripting, pre- and post-processing operations can be recorded in the form of dynamic computer programs and replayed at any time. Significant man-hours can be saved because simulations can be reproduced by just changing parameters in text-based scripts. All commercial CFD codes use graphical interface which can successfully shorten users’ learning time. However, using scripts would be much more efficient for recurring tasks such as simulations for parameterized study. In this talk, we will demonstrate the use of Python, a scripting language, to streamline the work flow of typical CFD tasks. Python has been used in various CFD products or projects, e.g., ANSYS, ESI, CREATE-AV, etc. It is well received that Python is becoming the de facto scripting language for high-performance computing. Its outstanding capability to glue multiple programming languages, especially C, FORTRAN, and CUDA, makes Python suitable for high-performance applications. By using Python in SOLVCON, we can finish a simulation using 23 million cells for supersonic jet in cross flow in one day, including pre- and post-processing.

Parallel computing must also be applied to the pre- and post-processing, in order to achieve the short turnaround time. The cooperative approach of parallel computing among calculation, pre- and post-processing is essential to further improve the work flow. The approach relies on scripting by using Python. Pre-split spatial domain can be loaded into a computer cluster through parallel data input. As such, solutions can be output to organized files in a parallel manner based on decomposed sub-domains. It is important to output the solution in the format native to parallel post-processors, e.g., ParaView, so that one can avoid time-consuming operations of non-parallel output and data format conversion. The use of Python facilitates a clear software structure of SOLVCON, which makes the modularization of parallel I/O possible.

Intermediate I/O for post-processing has to be eliminated to further reduce turnaround time. This is critically important for transient simulations using more than 100 million cells. For such meshes, outputting solutions at each time step for animation is simply impossible. In this talk, we will introduce in situ visualization in SOLVCON. In situ visualization analyzes or visualizes the solution while the simulation is still running. Since the analysis is done hand in hand with the calculation, there is no need of post-processing nor the intermediate I/O. Intelligent algorithms to automatically determine regions of interest for visualization are of an active research area in computer visualization. While parallel post-processing enables numerical analysts to efficiently investigate a snapshot of flow
fields in detail, in situ visualization can provide dynamic analysis for large-scale simulations within reasonable turnaround time. Simultaneously using the both techniques can deliver more insightful analysis of flow fields.

Next-Generation CFD Codes for Heterogeneous Computer Architecture

Yu, Sheng-Tao John, Department of Mechanical and Aerospace Engineering, The Ohio State University, Columbus, OH
Chen, Yung-Yu, Department of Mechanical and Aerospace Engineering, The Ohio State University, Columbus, OH

High-resolution simulations with large meshes are becoming increasingly important for CFD. The driving force in the coming decade will be the use of the emerging, advanced, heterogeneous computer architecture based on many-core technologies, e.g., General-Purpose Graphic Processing Unit (GPGPU) computing. The next-generation CFD codes must meet the demands for: (i) highly accurate, robust, and efficient numerical algorithms, (ii) agile adaption to evolving many-core hardwares, and (iii) short turnaround time for computational tasks. To meet these needs, a new CFD software framework, named SOLVCON, has been developed at Ohio State.

SOLVCON uses the space-time Conservation Element and Solution Element (CESE) method, originally developed by S.-C. Chang at NASA Glenn Research Center. The code uses unstructured meshes which could be composed of mixed elements, including tetrahedra, hexahedra, pyramids, and prisms for three-dimensional problems. Contrast to upwind methods, the CESE method does not use an approximate Riemann solver or a reconstruction step as the building blocks of the numerical algorithm. The logic of the CESE method is simpler and the operation count is more efficient. Nevertheless, the CESE method is highly accurate with superb shock-capturing capabilities. The CESE method is a genuine multi-dimensional method which is inherently suitable for unstructured meshes. The quality of the numerical results obtained by using the CESE method is relatively insensitive to the mesh quality. The CESE method is an explicit time marching method with the simplest and the most compact space-time stencil. Therefore, the method is inherently suitable for massively parallel computing. The advanced implementation of the CESE method in SOLVCON allows pluggable physical models. That is, changing the physical model only involves coding for Jacobian matrices and boundary-condition treatments. SOLVCON is a multi-physics code with several pluggable solver kernels, including the Euler solver for compressible flow, velocity-stress solver for waves in complex elastic solids, etc.

Incorporation of the emerging many-core technologies or GPGPU computing is critical for next-generation CFD codes. Nowadays, running CUDA code on Nvidia GPUs is the de facto approach for GPGPU computing. Extension to use GPU clusters is much harder, because it requires hybrid parallelism, which simultaneously uses distributed-memory parallel computing for clusters and shared-memory parallel computing for many-core. Moreover, many advanced many-core hardwares are in the pipeline and the industry is in the middle of fast evolution. There are two significant trends in the multi-core technologies: (i) More cores in a CPU: by the end of this year, a commodity CPU chip will have 16 cores. (ii) Merging CPU and GPU: Intel is now selling Sandy Bridge chip, which has 8 CPU and 12 GPU cores. Further development of supercomputing will be driven by the evolving heterogeneous hardware. For high-performance computing, CFD codes have to use hybrid parallelism. Due to the complexity of hybrid parallelism, new CFD codes must be well organized. The code development must be done by practicing software engineering. SOLVCON is written by using Python, an object-oriented programming language, to provide a platform for advanced techniques of software engineering. SOLVCON enables the segregated solver kernels for CFD. High-performance technologies, such as pthread programming, OpenMP, GPGPU computing by CUDA or OpenCL, can be exclusively used in solver kernels only to maximize the performance of hardware, while all other code for supportive functionalities does not change.

Last but not the least, SOLVCON aims at significantly reducing the turnaround time of CFD tasks by automation. Conducting high-resolution CFD tasks involves labor-intensive human efforts in pre- and post-processing the solution. SOLVCON provides an environment by using Python scripts to integrate the following tasks: (i)
Conversion of mesh data from mesh generator. (ii) Domain decomposition by graph partitioning software, e.g., METIS. (iii) Parallel calculation by MPI. (iv) Memory management for GPGPU computing. (v) Output and conversion of solution to the format of post-processor. (vi) Communication with supercomputer batch systems. These tasks, along with parallel input and output and in situ visualization, are automatically performed by various Python modules implemented in SOLVCON. The automation shortens the turnaround time by streamlining the workflow of CFD simulations.

**Flow & Particle Simulations in a Realistic Human Nasal Airway Model**

*Mylavarapu, Goutham, University of Cincinnati, Cincinnati, OH  
Mihaescu, Mihai, University of Cincinnati, Cincinnati, OH  
Gutmark, Ephraim, University of Cincinnati, Cincinnati, OH*

**BACKGROUND:**

Nose is a primary defense system of our respiratory system. It acts as an interface between human airway and ambient environment. Different physiological functions of nose such as odor perception, filtration, and humidification of inhaled air are highly dependent on nasal airflow. Also medically, intranasal route is considered as a promising alternate drug delivery approach to the more popular oral consumption of drugs. Large mucosal area in nasal cavity, rapid onset of action in such drugs and an opportunity for targeted delivery by controlling few parameters are some advantages of nasal drug delivery. An effective drug delivery with a precise deposition pattern may improve the response to the drug and potentially reduce drug dosage.

However, patient specificity, complex anatomy and physiology have proven to be significant hindrances for invasive clinical measurements or in gaining a complete understanding of nasal airflow dynamics, treatment of nasal disorders and nasal drug delivery. With advancement of computers in last few years, Computational Fluid Dynamics (CFD) is used extensively to study biomedical flows. There are certain advantages such as non-invasive application, less patient interaction, greater flexibility, low cost, ease of use over traditional clinical studies. The objective of this study is to simulate flow and transport phenomenon in nasal airway model and to study the impact of different parameters such as flow rate of inhaled air stream, particle size, location in the nose where drug is releases on local and total particle deposition efficiency.

**METHODS:**

Anatomically accurate, three dimensional, nasal airway model is reconstructed from 640 pre-acquired axial Computer Tomography (CT) images of a patient using MIMICS. Airway volume is discretized into an unstructured tetrahedral/hexagonal hybrid mesh using GAMBIT. Three different mesh resolutions were generated for grid sensitivity study. Flow and particle tracking numerical simulations were carried using commercial CFD package, FLUENT 6.3. A peak inspiratory phase of breathing cycle is simulated for different cases in this study. An incompressible flow field distribution is predicted using a steady RANS k-w SST turbulence closure model. Second order finite volume upwind schemes were chosen to solve discretized momentum and turbulence transport equations. The coupling between the velocity and pressure fields is realized with SIMPLE algorithm. Volumetric flow rates of 7.5, 15, 30, 45 and 60 lpm spanning range of rest to gasping in human breathing is considered for this study. A velocity boundary condition (BC) at nostril inlet, outflow BC at outlet and no-slip airway wall are considered in the computational model. After the flow-field is solved, an analysis of fluid-particle flow is carried using discrete phase modeling (DPM) in FLUENT. 1-way coupling is assumed between flow and particle flow fields. Mucosal layer on nasal airway wall traps and absorbs drug particles that come into contact with it. Hence particles that reach the airway wall were set to ‘trap’, at inlet faces as ‘reflect’ and at outlet face as ‘escape’ boundary conditions in DPM model. Particles are assumed to be inert, mono-disperse, spherical, uniform diameter. Different particle diameters with a range of 0.5 to 30 microns are considered for this study. Three particle densities of 500, 1000, 1500 kg/m3 are studied. Similarly three different turbulent intensities of 5, 10, 15% is studied.
RESULTS & DISCUSSION:

Computed pressure drop between right and left nasal cavities suggested asymmetry in this patient. Pressure drop increased linearly at lower flow rates and non-linearly at higher flow rate as in a typical pipe flow. Flow distribution shows recirculation region at anatomical location which is sensitive to odor perception. This substantiates claim why we try to sniff hard when trying to smell sometimes. Particle deposition efficiency is defined as the ratio of number of particles trapped in nasal cavity to the total number of particles released. It is observed that increase in flow rates leads to higher deposition efficiencies for same particle diameter. Similarly for same flow rate, increase in particle diameter also increases deposition. However, beyond a certain particle diameter size, irrespective of flow rate, deposition efficiency is large as 95%. The influence of particle density and turbulence intensities were observed to be insignificant. Surface injection at nostril plane had higher depositions for smaller diameters than spray injection. Thus this study shows how CFD can be used to better understand the drug delivery mechanism in human beings and also enable researchers to develop efficient devices for drug delivery.
Control of Flow Separation Using Stagnation Fed Vortex Generating Jets

Bonilla, Carlos H, The Ohio State University, Columbus, OH
Bons, Jeffrey P, The Ohio State University, Columbus, OH

Economic and environmental benefits are driving the science and engineering community to find viable active flow control systems to implement in aerodynamic systems. The methodology to reduce separation, increase lift, and reduce drag via flow control has been developed and proven successful with many different methods. However, most of the current systems being tested require energy, which reduces the benefits gained by the control system. To circumvent such problem it is possible to use the energy in the flow to power the control device. This methodology is used to power stagnation fed vortex generating jets, SFVGJs. The system is composed of an inlet, placed at the stagnation plenum in the flow, connected to the jet outlet. The pressure difference between the two ends forces the flow and creates a jet. To confirm the effectiveness of the system, it was initially tested on a faired circular cylinder in a low speed, open loop wind tunnel at a Reynolds number of 38,000 and 78,000. Different parameters such as pressure coefficient distribution, blowing ratio, and head loss were measured to characterize the efficiency and effect of the jets in delaying separation. Particle image velocimetry was used to better visualize the flow. Additional minor studies were conducted on the inlet geometry of the jets and the state of the boundary layer, laminar vs turbulent, on the cylinder. It was found that the passive jets were able to provide a minor delay on the separation point as well as force immediate transition of the shear layer of the wake.

SFVJGs were also installed and tested on an L1M blade in an LPT linear cascade. The system was tested in a Reynolds number range between 20,000 and 100,000 with different free stream turbulence. The turbulence was varied by using a turbulence grid upstream of the test section. In order to characterize any improvement in performance pressure distribution and wake loss measurements were acquired. The blowing ratio achieved was also measured. The control was able to improve the performance of the blade by keeping the flow attached longer.

References

Mixed Compression Inlet Simulations with Boundary Layer Bleed

Apyan, Alexander Cannon, University of Cincinnati, Cincinnati, OH
Orkwis, Paul D, University of Cincinnati, Cincinnati, OH
Turner, Mark R, University of Cincinnati, Cincinnati, OH
Galbraith, Daniel S, University of Cincinnati, Cincinnati, OH

The University of Cincinnati’s CFD team, in conjunction with the Air Force Research Lab (AFRL) at Wright Patterson Air Force Base, aims to analyze shock wave boundary layer interactions in a realistic mixed compression inlet. Mixed compression inlets utilize a series of oblique shocks followed by a final normal shock aft of the inlet throat in order to compress a flow. After the normal shock, the flow proceeds into a subsonic diffuser and then reaches the aerodynamic interface plane (AIP). As the oblique and normal shocks reflect off of the boundary layer attached to the walls they form the shock wave boundary layer interactions. These interactions have the ability to unstart an inlet thus choking the flow into the engine. This paper shows the effects of bleed on the starting properties.
of a mixed compression inlet. Two different geometrical inlet configurations have been utilized. The first configuration was able to start without using any form of bleed or flow control device. From here, the aim was to start the nominal configuration desired by the team at AFRL. This second configuration required a large amount of bleed around the area of the first shock reflection in order to successfully start and run to convergence. These achievements will allow for examination into different bleed configurations as well as more inlet geometry variances in future boundary layer interaction experiments.

Exploration of MHD Flow Control for a Hypersonic Blunt Elliptic Cone with an Impregnated Ablator

Bishek, Nicholas J, Air Force Research Laboratory, WPAFB, OH
Poggie, Jonathan, Air Force Research Laboratory, WPAFB, OH

A three-dimensional fluid code was loosely coupled to a low magnetic Reynolds number magnetohydrodynamics module to compute hypersonic air flow around a blunt elliptic cone with a fin protruding from the top centerline. A dipole magnet was positioned within the protrusion to reduce local heating on the fin, but had minimal effect because of the low electrical conductivity of the air. Two approaches were explored to increase the effectiveness of the magnet for local heat flux mitigation: increasing the flight Mach number, and using an impregnated ablator to introduce seed particles into the flow. Both approaches increased the electrical conductivity near the fin, making the magnet more effective at heat flux mitigation. However, the seeding approach does not require modification to the trajectory nor does it increase the baseline heat load on the fin, making it a more viable configuration for a magnetohydrodynamic flight experiment.

Application of Active Flow Control to Reduce Endwall Losses on a Highly-Loaded Low Pressure Turbine Blade

Benton, Stuart I., The Ohio State University, Columbus, OH
Bons, Jeffrey P, The Ohio State University, Columbus, OH
Sondergaard, Rolf, Air Force Research Laboratory Propulsion Directorate, Dayton, OH

The purpose of this study is to apply two methods of active flow control to the endwall of a low pressure turbine to reduce total pressure losses due to the secondary flow structure. The two low pressure turbine blade profiles used in this study are the L2F and L2A blade profiles. These blade profiles have been designed as a second generation of airfoils designed with an increased incompressible loading coefficient. This family of research airfoils has been developed from the widely used Pack-B research profile developed by Pratt and Whitney. The first generation of research airfoils designed with an incompressible Zweifel loading coefficient of Zw = 1.34, a 17% increase over the Pack-B (Zw = 1.15). These two profiles were designated the L1M and L1A, to indicate the mid and aft loading distribution respectively [1]. The second generation of airfoils has been designed with a Zweifel loading coefficient of Zw = 1.59, 38% higher than the Pack-B. The two profiles, the L2F and L2A have forward and aft loaded pressure distributions respectively. The L2F has been shown to perform exceptionally well at mid-span at low Reynolds numbers [2].

As part of ongoing research to reduce blade count and improve low Reynolds number performance, there is a large motivation to increase blade loading and employ front-loaded designs. However, it has been shown that increased loading, front loading, and low Reynolds numbers all contribute to increased losses due to a stronger secondary flow structure near the endwalls[3]. For these reasons, further investigation into the secondary flow structure of high lift front-loaded profiles is necessary. Additionally the use of active flow control in the form of slot and jet blowing will be explored as a means of reducing losses caused by secondary flows. This presentation will introduce the initial characterization of the secondary losses for the L2A and L2F in the form of total pressure loss contours taken at
75% of axial chord, downstream of the blade row. The data for the L2A was taken at the Air Force Research Laboratory’s low-speed 6-passage cascade, with the use of a splitter plate to condition the incoming endwall boundary-layer. The data on the L2F was gathered at Ohio State University’s low-speed 3-passage cascade with a similar splitter plate. Data was taken with a free stream turbulence intensity of $Tu = 3\%$ at Reynolds numbers of 60,000, 80,000, and 100,000 (based upon inlet velocity and axial chord).

Additionally, the presentation will outline ongoing plans to investigate the use of active flow control methods to reduce endwall losses. The first method will involve a skewed jet located on the blade surface near the endwall. This was shown to reduce losses through a preliminary experiment performed on the L1A blade profile. The second method has been previously demonstrated on compressor blades in a study out of the Technical University of Braunschweig [4]. This method places a small slot on the endwall. The slot extends out from the suction surface, normal to the tangent vector, at the suction peak of the blade profile. The added momentum in the boundary layer is used to keep the passage vortex from interacting with the suction surface of the blade.


Session 34: Imaging & Diagnostics

Chair: Anil Patnaik, AFRL/RZ

Single-Beam fs-CARS for Selective Excitation and Measurements of Group-Velocity Dispersion

Gord, James R., Air Force Research Laboratory, Propulsion Directorate, Wright-Patterson AFB, OH
Wrzesinski, Paul J., Department of Chemistry, Michigan State University, East Lansing, MI
Pestov, Dmitry, Biophotonic Solutions, Inc., East Lansing, MI
Dantus, Marcos, Department of Chemistry, Michigan State University, East Lansing, MI
Roy, Sukesh, Spectral Energies, LLC, Dayton, OH

Developments in the optical diagnostics of combustion and reacting-flow environments have been closely linked to advances in ultrafast-laser technology. Historically, this field of study has been dominated by the use of nanosecond ($10^{-9}$ s) and picosecond ($10^{-12}$ s) sources. However, femtosecond ($10^{-15}$ s)-laser sources have recently been successfully applied for gas thermometry, pressure studies, and imaging of turbulent gas flows. Although these experiments reinforce the numerous advantages of broadband-laser sources and optical diagnostics on the femtosecond time scale, they also bring attention to an inherent nuance of ultrashort (spectrally broadband)-pulse propagation—femtosecond pulses are much more susceptible to temporal broadening due to group-velocity dispersion (GVD) than picosecond and nanosecond pulses. The use of femtosecond-laser sources for diagnostics of combustion and reacting-flow environments requires detailed knowledge of the optical dispersive properties of the medium interacting with the laser beams. Here the second- and third-order dispersion values for various gases within the 700–900 nm range are reported. The pressure dependence of the chromatic dispersion under pressure conditions that are relevant to combustion is also examined.

Temperature Imaging of the Exhaust Stream of a J-85 Turbojet Engine using High-Speed, Hyperspectral Absorption Tomography

Caswell, Andrew W., Spectral Energies, LLC, Dayton, OH
Roy, Sukesh, Spectral Energies, LLC, Dayton, OH
Ma, Lin, Clemson University, Clemson, SC
Sanders, Scott T., University of Wisconsin-Madison, Madison, WI
Gord, James R., Air Force Research Laboratory, Wright-Patterson AFB, OH

Combustion instability represents a key issue that significantly affects the performance of practical combustors and afterburners. Such instability exhibits two modes: the low-frequency "rumble" and high-frequency "screech." Resolving these instability modes requires diagnostics that are capable of providing continuous measurements with sufficient spatial and temporal resolution. This work describes a tomography sensor based on hyperspectral absorption spectroscopy to address this critical diagnostic need. This sensor enables continuous tomographic imaging of temperature and chemical species with a bandwidth of 50 kHz. Based on the projection measurements, a tomographic reconstruction was conducted to retrieve the temperature and concentration of H2O in an atmospheric-pressure near-adiabatic H2–air flame as well as in the exhaust stream of a J-85 engine for various conditions. Measurements in the J-85 engine were performed for ground idle, full military, and full afterburner conditions. The significance of this measurement capability for retrieving the 2D temperature field spanning 225 grid points at 50-kHz rate in an augmentor will be discussed as well.
Deformable surface reconstruction of human coronary trees from CTA imagery

Huang, Qian, Wright State University, Dayton, OH  
Wischgoll, Thomas, Wright State University, Dayton, OH

Volumetric datasets captured by current 64 slice scanners are being popularly applied in clinical environments. The volume information extracted from these makes them a valuable tool for cardiologists to diagnose various types of diseases. In this work, algorithms are developed for segmenting the coronary trees started from a boundary point cloud of a human heart at more numerically precise subvoxel level with the seed positions identified by the user and a 3D surface constrained Delaunay triangle mesh to visualize the obtained vessel trees. The experimental results show that these proposed algorithms are capable of extracting boundary points of the coronary vessel trees. The meshed surface can visualize the vessel surfaces more smoothly with its subvoxel precision than the marching tetrahedra algorithm. The meshed surface can derive surface quantitative data such as to estimate principal curvatures of vessel surfaces, the minimum principle curvature can accurately visualize the local convex or concave shape on the vessels’ surfaces, a geometrical property that many vessel researchers are studying its influence on the plaque development inside the vessels.

Time Resolved Filtered Rayleigh Scattering Measurement Of Centrifugally Loaded Buoyant Jet

Benhassen, Firas, AFIT, WPAFB, OH  
Polanka, Marc D, AFIT, WPAFB, OH  
Reeder, Mark F, AFIT, WPAFB, OH

The combustion process within the Ultra-Compact Combustor (UCC) occurs in the circumferential direction. The presence of variable flow density within the circumferential cavity introduces significant buoyancy issues. On the other hand, g loading, via the generation of centrifugal forces, ensures the circulation of the flow in the circumferential cavity and enhances the completion of the combustion process (before allowing the exit of the hot gases back to the main flow). The coupling between buoyancy and high g loading is what highlights the behavior of the flow within the UCC. Using a horizontal and a curved section two experiments were run to characterize and measure the effects of both g loading and buoyancy on the overall behavior of a carbon dioxide jet in a co-flow of air. Measurements were made with a Filtered Rayleigh Scattering (FRS) set up using a continuous wave laser and a high speed camera showing adequate signal to noise ratio of up to 400 Hz. Collected time resolved images allowed for the investigation of the effects of g-loading and buoyancy on the mixing properties and trajectory of the jet.
The Influence of Accelerated Flight Velocities on Flutter

Merrett, Craig G., University of Illinois at Urbana-Champaign, Urbana, IL
Hilton, Harry H., University of Illinois at Urbana-Champaign, Urbana, IL

Submitted for oral presentation to
Thirty-Sixth AIAA Dayton-Cincinnati Aerospace Sciences Symposium 2011

The Influence of Accelerated Flight Velocities on Flutter

Craig G. Merrett and Harry H. Hilton
University of Illinois at Urbana-Champaign (UIUC)

In flutter analyses, it is conventional to investigate the so-called V − g dia-grams, with representing a virtual structural damping coefficient and the flight velocity. It can be shown that when reaches the value of zero at , the lifting surface displacements reach a state of simple harmonic motion (SHM) and is defined as the flutter velocity [1 – 5].

In self-excited elastic lifting surface systems, second order differential equations in time model the interactions between inertia, internal forces (stresses) and aerodynamic pressures (lift). It can be readily demonstrated that in such linear systems, the displacements are all proportional to . The alternate protocol developed in the current paper focuses on the evolution of values as increases to and beyond at (Fig. 1). This concept produces diagrams [6]. Based on aerodynamic theories that use SHM or Theodorsen expressions [3], preliminary results for both indicate that the usual sole condition for SHM, i.e., may be insufficient to establish flutter boundaries. Additional constraints of are thus required. Furthermore, bifurcations in values at steady-state were also encountered (Fig. 2). Minimum total energy considerations dictate the selection of the upper dashed green curve, which leads SHM and eventual aeroelastic instability as the flight velocity increases.

When accelerated flight is considered, flutter velocities in the steady-state regions after transients have died out, exhibit significantly higher or lower values compared to pure steady-state flutter conditions.

Fig. 1 – Exponent d as a function of flight velocity

Fig. 2 – Bifurcation of exponent d

Effect of Control Surface on Active Control for Flutter Suppression

McDonough, Laura Anne, Miami University, Oxford, OH
Singh, Kumar V., Miami University, Oxford, OH

Aeroservoelastic analysis is essential in developing control strategies to suppress flutter instabilities originating from the mutual interaction between the aerodynamic forces, structural deformations, and vibrations of the aeroelastic systems. Control methods to suppress flutter are often designed from traditional state space models. However, these models are often based on reduced order approximation of associated Finite Element models. To avoid such approximation and modeling errors, recently a receptance based control was developed for aeroelastic systems in which control is based solely on receptance frequency response functions, which may be obtained from sensor/actuator measurements and circumvent the modeling and approximation errors. It has been demonstrated that in theory the pole assignment for flutter boundary extension for an aeroelastic system by state feedback control force can be realized by receptance based control. In order to mature such a control strategy, its effectiveness needs to be tested on several wing geometries and aerodynamic flow conditions by using Finite Element aeroelastic software. In this work, numerical frequency response functions are computed by selecting the degrees of freedom associated with the locations of sensors and actuators (control surfaces). The control gains are computed from numerical receptance transfer functions corresponding to varying location of the control surface. Examples associated with BAH wings with single control surface (with varying location along the wing span) and multiple control surfaces are considered. The control gains as a function of number of control surface and its locations is studied in order to select the controller with least control effort.

CH-47 Rotating System Fault Sensing for Condition-Based Maintenance

Dickerson, Stephen M., Air Force Institute of Technology, Wright-Patterson AFB, OH
Kunz, Donald L., Air Force Institute of Technology, Wright-Patterson AFB, OH

Rotary wing aircraft are complex vehicles that incorporate a large number of rotating components. In particular, the rotor system that provides lift, propulsion, and control for the aircraft is a complex rotating system. Historically, rotorcraft manufacturers, operators, and maintainers have based their maintenance procedures on time-based metrics. That is, the decision on whether a part needs to be serviced or replaced is based on testing and/or analysis to predict the failure of the part, along with an appropriate safety factor. This approach has three possible outcomes: the part is replaced as it nears the end of its useful life, the part is replaced in spite of the fact that it could have served significantly longer, or the part fails before it can be replaced. The first of these outcomes is optimal, while the second increases operating costs and the third is potentially catastrophic. Increasing the safety factor reduced the probability of the third outcome and virtually assures the second.

The Department of Defense has mandated that the armed services transition from maintenance procedures driven by estimated service lives to procedures bases on the condition of the components. This is known as condition-based maintenance, and has the potential to reduce operating costs and increasing readiness, while maintaining adequate safety standards. One of the principal challenges of implementing condition-based maintenance is sensing the condition of components. This is particularly true for rotating components on rotary wing aircraft, because of the high-G environment and the difficulty of reliably sending signals between rotating and nonrotating components in harsh climate conditions. Therefore, the purpose of this investigation is to analytically examine the relationship between known faults in the rotating components and the response of sensor embedded in the nonrotating structure of the aircraft.

An analytical model of the CH-47D helicopter that can emulate faults due to pitch link adjustment, additional blade mass, and trim tab adjustment has been developed using the Rotorcraft Comprehensive Analysis System (RCAS). The model is based on a previous model of the CH-47D, and also includes an elastic fuselage. Vibratory response
due to the faults listed above has been calculated and compared to existing vibration data for the aircraft. This comparison demonstrates the validity of the model and its potential for identifying rotor system faults based on fuselage sensors.
Session 36: Structures

Chair: Marina Ruggles-Wrenn, AFIT

Initial Design of a CubeLab Module Frame to Facilitate Rapid Turnaround Scientific Research aboard the International Space Station

Clements, Twyman S, Space Systems Lab -Kentucky Space, Lexington, KY
Baker, John R, University of Kentucky, Paducah, KY
Lumpp, James E, University of Kentucky, Lexington, KY

This presentation investigates the design of a CubeLab Module structure that facilitates rapid turnaround, plug-and-play scientific research involving biological, medical, and pharmaceutical payloads aboard the International Space Station. The NanoRacks Platform and CubeLab Standard will be introduced as a new system for developing small payload research in a microgravity environment. The existing need for a plug-and-play payload system to facilitate research, and the potential benefits from such a system, will be overviewed as well as the NASA requirements considered during the design phase. Results from finite element analysis on a system model showing the sealed system will maintain structural integrity if subjected to maximum pressure changes expected in service will be overviewed, in addition to functional testing of a prototype. The presentation will also include suggested improvements for a flight model design. Technology and techniques developed during this research, along with suggested future work, will also be presented.

Testing Bed for Structural Health Monitoring

Vehorn, Keith A, Wright State University and AFRL/RBSI, Dayton, OH
Shaw, Brad W, Wright State University, Dayton, OH
Slater, Joseph C, Wright State University, Dayton, OH

Structural health monitoring (SHM) techniques are important for detecting the possible damage that would compromise the functional integrity of an aerospace structure. SHM detecting systems result in: (1) damage detection; (2) damage location; (3) damage evaluation; and, (4) future life prediction [1]. To accommodate the development of SHM systems the utilization of structural test beds for each of the fore mentioned systems procedures would be a valuable tool to the testing and validation of the developed SHM systems. Development of such a test bed would allow for varying excitation sources, damage detection methods, and ease in transportation as to accommodate the varying and diverse needs of SHM system developers. The focus of this project was to design and manufacture a test bed that would accommodate afore mentioned needs for SHM system development.

The intent of the test bed design was to incorporate many aspects of actual aerospace structures as to provide a valid testing platform that would allow developed SHM systems to be realistically implemented from laboratory to field use. Such aspects include structure material, natural frequency, and stiffness. To insure adequate mobility of the test bed and validity in varying environments the size of the test bed was designed as to not compromise the design aspects of an actual aerospace structure. The core of the test bed includes a mobile base and un-skinned wing that attaches to the mobile base.

With the designed and manufactured test bed it was necessary to ensure that frequency testing and data analysis could be performed on the test bed. This functionality validation was accomplished using arbitrary wing excitation simulating operational loading [1] and simulated wing fastener failure. This experimental testing returned
distinguishable data attributes between damaged and non-damaged frequency responses which validated the test beds ability to utilize SHM systems for damage detection and localization.

Citations:


**Modified Symmetry Cell Approach for Simulation of Laser Peening Over Large Scale Structures**

*Spradlin, Thomas J., Wright State University, Dayton, OH*
*Grandhi, Ramana V., Wright State University, Dayton, OH*
*Langer, Kristina, AFRL/RBSM, Wright-Patterson Air Force Base, OH*

Laser Peening is a surface enhancement technique that is used to extend the fatigue life of structural components by inducing compressive surface residual stresses. The laser peening residual stresses are imparted into the material using high intensity short duration lasers that initiate plastic shockwaves in the work piece exposed. Because laser peening is a relatively new commercial technology, the computational design tools needed to increase processing efficiency are still being developed. Currently, the largest hurdle facing laser peening finite element simulation is the computational time of the process; a single shot on a modern workstation with a relatively coarse element mesh can take nearly a day. For a process that depends on several hundred to several thousand shots to effectively treat most applications, the long simulation time renders computational design time prohibitive. However, by using a symmetry cell approach, the laser peening simulations are reduced to the absolute essential shots necessary in order to create the fundamental building block of a repeating pattern. Using this building block, the larger pattern can then be reproduced across any size component with no further computational cost. This work further develops the symmetry cell approach to include accurate boundary condition placement for large components, geometric novelties such as corners and fillets, and sequence effects on symmetry cell residual stresses.

**Filtering Winding Effects from Control Systems in Cold Rolling Operations**

*Hinton, Jantzen L, Wright State University, Dayton, OH*
*Malik, Arif S., Saint Louis University, St. Louis, MO*
*Grandhi, Ramana V., Wright State University, Dayton, OH*

The increased demand for high quality thin gauge metals has prompted the improvement of quality control systems during the cold rolling process. The presented work discusses stress growth in a strip during the rolling/winding process which has a parabolic, asymmetric cross-section (wedge profile). During manufacturing, current control systems with flatness meters (sensors) are unable to differentiate between “rolled-in” stresses caused by the mill and stresses caused by the winding process. The accumulating affects of winding a strip with a wedge profile increases the potential of the flatness meters to measure a false stress profile which may lead to incorrect mill adjustments. To aide in accurate control system adjustments, a preliminary method is developed to separate the winding stress contribution from the overall stresses that are measured indirectly by a flatness control system. The in-plane stresses, and corresponding flatness, due to winding a strip containing wedge are computed using a conventional method as well as a 4th order polynomial Airy function based on mandrel wrap number and planar spatial position on the strip. With this approach, only a single function is needed to rapidly predict the entire in-plane stress field. Prediction of the stress field throughout the winding allows for the filtering-out of elastic winding stress affects from the flatness control system. This enables more appropriate control corrections for the plastic flatness defects that are due to the distribution of thickness reduction.
Motion Capture and Photogrammetry System Hybridization for Dynamic In-Flight Tracking and Measurement

Allen, Christopher T, AFIT, WPAFB, OH
Black, Jonathan T, AFIT, WPAFB, OH
Jennings, Alan L, AFIT, WPAFB, OH

Many micro aircraft employ deformable structures, and therefore placing sensors onboard can be prohibitive in terms of both weight and power. Regardless, non-contact measurements during normal operation allows for a higher degree of accuracy in measurements of the natural behavior of the craft. By using a real-time videogrammetry system, a set of cameras is coordinated to track large scale motion and produce high speed, high quality images for surface reconstruction. Texture-based photogrammetry, provides high resolution surface profiles for non-contact measurement of the target. Traditionally, features are projected onto the surface for automatic target recognition between photos. New techniques use the local texture of a point to match random speckle between photos. This hybrid non-contact sensor system will provide more capability than each individual system enabling dynamic, in-flight tracking and measurement of structural dynamics. Future work, involving motion tracking of flapping micro-air vehicles, is also presented.

Tape Spring Deployment Analysis and Prediction for Space Structures

Jennings, Alan L, Air Force Institute of Technology, WPAFB, OH
Black, Jonathan, AFIT, WPAFB, OH
Allen, Chris, AFIT, WPAFB, OH
Magree, Daniel, AFIT, WPAFB, OH

Mass and power requirement are two high priority cost components to satellite design. The minimization of the satellite's mass would reduce the overall cost required for launch and increase the satellite's life cycle term. Solar panels and antenna often require lengths that are a challenge for small satellites, such as CubeSat's. Self-deploying rigidizable structures address this need and have been the subject of much research. Tape springs have shown to be efficient solutions for many 2-D systems. They are self actuating, simple and foldable in various configurations for pre-deployment storage. This presentation examines the behavior of tape springs hinges between various lengths of boom during mock deployments. The equipment and methods of analysis are set forth in this paper. Results show that deployment envelopes are predictable for 3-D folds. The envelopes bound motion allowing for verification of clearances.

An auto-Tracking Technology for Character Points in High-Speed Videos

Ren, Yan, Wright State University, Dayton, OH

High-speed photogrammetry with 1000 FPS is used for trajectory study of flapping flights. Some character points (e.g. wing tips) are of particular informative in terms of outlining the motion trajectories, and thus usually be user concerned. The tracking of these character points are often manually performed, which is quite time consuming.
Therefore, an auto tracking technique with reasonable accuracy is preferred, and will greatly improve the image processing efficiency.

In this work, we will develop a point auto-tracking methodology using Gaussian smoothing, and Scale-invariant feature transform (SIFT) techniques. Gaussian smoothing is used to reduce image noise level and remove redundant image information. SIFT is applied to capture image features in a user-defined processing window around the interested points.

**EO-FMCW Radar Fusion For Vehicle Tracking**

Gale, Nicholas C, Wright State University, Dayton, OH
Hong, Lang, Wright State University, Dayton, OH
Roy, Arunesh, Wright State, Dayton, OH

The identification and tracking of multiple vehicles has many applications from surveillance and security to traffic monitoring and automated speed enforcement. Many times a single sensor, or a single type of sensor, cannot accurately provide the necessary information required to produce accurate state information for those vehicles. Low cost frequency modulated continuous wave (FMCW) radar have the ability to provide accurate range and range rate information, but provide poor angular information. Conversely, an electro-optical (EO) sensor can provide accurate angular information, but inherently lacks the ability to provide range information. By fusing the information from these sensors in an intelligent way, a more accurate vehicle state can be generated.
Session 38: Fatigue and Structural Analysis

Chair: Ramana Grandhi, WSU

Examination of Lamb Wave Interaction with Cracks in Aluminum Plates via Laser Doppler Vibrometry

Owens, C. Todd, Air Force Institute of Technology, Wright Patterson AFB, OH
Swenson, Eric D., Air Force Institute of Technology, Wright Patterson AFB, OH

The average age of the US Air Force aircraft fleet is now over 24 years. Budget pressures related to the cost of operations in Iraq and Afghanistan, recent economic difficulties and the ever-increasing cost of acquiring new weapon systems mean that number is likely to go up significantly before it reaches a peak. As one example, the newest B-52H in the Air Force inventory was delivered over 47 years ago, yet the service currently plans to keep these aircraft in operation another 15 to 30 years. Some of the KC-135 fleet are even older, yet it will likely be another 8 to 10 years before a replacement tanker becomes operational. Given these facts, there is great interest in developing automated systems to monitor the structural health of aging aircraft. One of the more promising technologies for structural health monitoring involves the use of piezoelectric sensor-actuators to induce and measure ultrasonic elastic (Lamb and Rayleigh) waves in aircraft structures. These waves can travel a significant distance in plate-like structures and their behavior is affected by structural discontinuities such as cracks. Laser Doppler vibrometry can be used to measure these waves in great detail. AFIT is at the leading edge of these capabilities, possessing both three-dimensional laser Doppler vibrometers (3D-LDV) as well as a new ultra-high-frequency (one-dimensional) vibrometer (UHFV). The 3D-LDV provides measurements of velocity both in-plane and out-of-plane, while the UHFV provides out of plane displacement measurements at significantly greater temporal and spatial resolution than the 3D-LDV. This presentation will cover the initial results of an experimental investigation of Lamb wave interactions with cracks in aluminum plates using both 3D and UHF laser Doppler vibrometry.

The Effect of a Graded Layer on the Plastic Dissipation of General Bimaterial Interfaces Under Mixed-Mode Fatigue Loading

Baudendistel, Craig M, Wright State University, Dayton, OH
Klingbeil, Nathan, Wright State University, Dayton, OH

Recent work has developed a technique with which to define the mode-mix along a bimaterial interface based on a dissipated energy approach. In general, this theory correlates steady-state fatigue crack growth rates and monotonic fracture toughnesses of ductile metals using the total plastic energy dissipated ahead of a crack. This is especially useful in predicting the fatigue behavior in layered material applications such as welding, soldering, brazing and laser-based manufacturing. The current study seeks to validate the previous work by modeling the interface between two dissimilar materials as a graded layer instead of a perfect crack interface where there exists a step change in elastic and plastic properties. Not only will this provide a non-ambiguous definition of the mode-mix for general bimaterial interface problems, it will more accurately model the material property change that is present in all layered material applications. In addition, results for the effect of a graded layer on the plastic dissipation will be provided for general bimaterial interfaces. Overall, this research will aid in the rapid evaluation of new-age materials by predicting fatigue crack growth rates based on a substantially reduced test matrix of material properties.
Analysis of Plasma Based Control on a Supersonic Hot Jet

Speth, Rachelle L, The Ohio State University, Columbus, OH
Gaitonde, Datta V, The Ohio State University, Columbus, OH

Noise generated by aircraft is a significant problem in urban areas as well as on aircraft carriers. The noise is largely generated from the engine exhaust mixing with the ambient air. One possible way to reduce noise is to use Localized Arc Filament Plasma Actuators (LAFPA) around the exit of the exhaust to change the structure of the exiting jet air. In this research, we explore the effect of these actuators on hot jets to augment findings reported earlier on cold jets. To this end, numerical methods are employed to complement experiments on the effect of eight LAFPA mounted around the exit of an axisymmetric ideally-expanded Mach 1.3 hot jet. The hot jet has a jet reservoir to ambient temperature ratio of 1.5. The hot jet is analyzed by examining the coherent structures in the instantaneous, time-averaged and the phase averaged flow fields. The effects of the axisymmetric (m=0) and the first (m=1) modes are introduced at frequencies corresponding to the preferred mode and its first harmonic. The results are being analyzed qualitatively (for coherent structures) as well as quantitatively (near field turbulence and pressure). The phase averaged results exhibit many of the features observed for cold jets, including rollers and helices connected by braids. Hairpin-like substructures arise which appear to evolve into braids connecting rollers (m=0) or successive rungs of the helix (m=1). In depictions of the flow, pressure waves emanate from the curves of the helix. These pressure waves travel close to the speed of sound downstream and in the radial direction and are anticipated to have significant impact on components of the far-field sound distribution. A preliminary assessment of the effect of excitation on Reynolds stresses and evolution of turbulent kinetic energy will be presented.

Simulation of a Nanosecond Plasma Actuator in a Quiescent Environment

McCrink, Matthew H, Ohio State University, Columbus, OH
Gaitonde, Datta V., Ohio State University, Columbus, OH

Plasma actuators are a demonstrated mechanism for affecting the flow field around an aerodynamic body. To date, the effectiveness and efficiency of the actuators have been strongly correlated not only with the physics of the actuator, but also the Mach and Reynolds number of the surrounding flow field. Current generation actuators are typically limited to a very narrow operating range, making them unattractive for vehicles which function over a wide speed range. For example, AC Dielectric Barrier Discharge devices (DBDs) induce a bulk motion of the flow field and are therefore typically constrained to the low speed regime. Arc type actuators produce thermal disturbances in the flow field which have been employed mainly in high speed flows. In this work, we considered a relatively new type of actuator based on a Nanosecond pulse discharge (NSPD) plasma source, which have been experimentally validated in the high and low speed regimes.

The coupling mechanism by which the NSPD actuators affect the flow field is through the generation of a thermal pulse, similar to the arc type actuators. However, the low duty cycle, high frequency pulses of this device allow it to operate more efficiently than a comparable arc type actuator. The NSPD actuator has proven effective in the low speed range and preliminary experiments suggest promising potential at higher speeds (Mach 5). They thus offer significant range and efficiency advantages over other actuators.
In this work, we employ a high-fidelity, massively parallel, high-order code to explore the flow field coupling mechanisms of the NSPD based devices. The heat release mechanism is determined heuristically by comparison with experiment. It is shown that a non uniform surface distribution is required for proper flow evolution. The case yields a simple model that can then be employed for the implicit LES of the flow past a NACA 0015 airfoil. A firmer understanding of this mechanism will allow for strategic experimental studies to be performed which best exploit the efficiency advantages of the NSPD devices. Initial results for an actuator in a quiescent environment will be presented and compared to existing experimental data. This case will provide the model calibration and verification parameters to be used on future investigations of separated low speed airfoil flows, and high speed blunt body shock control.

Three Dimensional Effect of Bleed in Supersonic Flow
Manavasi, Seshagopalan, University of Cincinnati, Cincinnati, OH
Morell, Albert, University of Cincinnati, Cincinnati, OH
Hamed, Awatef, University of Cincinnati, Cincinnati, OH

Computational analysis of bleed in supersonic flow was conducted for bleed through normal and slanted-hole rows using the WIND-US Navier-Stokes solver. Results are presented for the computed three-dimensional flow field within the holes and downstream of the bleed zone for 90 degree and 20 degree bleed-hole rows. The effect of bleed on the external flow is demonstrated in the expansion and shock waves across the bleed openings, in the distorted static pressure and axial velocity contours, and in the secondary velocities (normal to free-stream) contours downstream of the bleed zone. The predicted three-dimensional flow fields were different for normal and slanted bleed both within the holes and downstream of the bleed zone. Higher secondary flow velocities were predicted in the case of slanted bleed along with higher Qsonic. The results are compared for two sets of simulations. In the first, the three dimensional flow was resolved throughout a solution domain that extends inside the bleed holes and include the plenum in addition to the external supersonic flow. In the second, the empirical porous bleed model available in WIND-US was applied in a segmented fashion over the bleed hole opening area in the bleed surface.

Computational Study of Thermal Barrier Coated Turbine Blades Erosion
Shin, Dongyun, University of Cincinnati, Cincinnati, OH
Hamed, Awatef, School of Aerospace System, University of Cincinnati, Cincinnati, OH

Erosion of turbine surface by particles impact increases tip clearances and blade surface roughness. A combined experimental and numerical study was conducted to characterize the effect of Alumina particle impacts on thermal barrier coated (TBC) turbine stator and rotor surfaces. Experimentally-measured coating erosion characteristic and particle surface restitution models were used to derive models that were used in the simulations to determine particle rebound conditions after each surface impact. The coated surface impact statistics were computed for numerical simulations based on the three-dimensional flow field and particle trajectories through a stage axial gas turbine of automotive application and modern rotorcraft engine. Predicted intensity and pattern of TBC erosion over the stator and rotor blade surfaces and the variation in the overall blade surfaces with ingestion velocity are discussed.
High-Order Simulations of Low Reynolds Number Membrane Airfoils under Prescribed Motion

Jaworski, Justin W, Air Force Research Laboratory (AFRL/RBAC), Wright-Patterson AFB, OH
Gordnier, Raymond E, Air Force Research Laboratory (AFRL/RBAC), Wright-Patterson AFB, OH

The aerodynamics and aeroelastic response of a membrane wing under prescribed motion are investigated using a high-order, two-dimensional Navier-Stokes solver coupled to a geometrically-nonlinear membrane model. The impact of increasing Reynolds number on the vortex dynamics and unsteady aerodynamic loads is examined for moderate-amplitude plunge and combined pitch-plunge motions at low frequency. Simulation results are compared with classical thin airfoil theory and highlight the differences between rigid and flexible membrane airfoils undergoing small and moderate amplitude motions. Most notably, the present study demonstrates the ability of lifting membrane surface flexibility to enhance thrust production, which may inform the design of flapping wing membrane fliers.

Numerical study of high velocity wear

Palazotto, Anthony N, AFIT, Wpafb, OH
Meador, Stephan, AFIT, OH

The evaluation of the phenomenon of wear at the Holloman High Speed Test track in New Mexico has been considered. Velocities exceeding 1k/sec are normal for the ground tests and therefore designs most consider the interaction of the test fixtures as they speed down the track over a distance of up to three miles. The research was related to the development of a computer model reflecting the physics of high energy impact. Thus, dynamic relationships had to be considered incorporating viscoplasticity and equations of state. Comparisons have been made to an actual experimental run carried out in February, 2008. The use of a hydrocode CTH was incorporated in the analysis and experimentation carried out on the actual recovered specimens led to features of the numerical constitutive equations and initial model. A shock wave developed which was traced at velocities of 600m/s and 1250 m/s as the slipper collided with an asperity.

Space-Time Finite Element Formulation of Membrane Dynamics

Kolst, Kyle F, Air Force Institute of Technology, Wright-Patterson AFB, OH
Kunz, Donald L, Air Force Institute of Technology, Wright-Patterson AFB, OH

The wings of micro air vehicles are commonly designed with pre-tensioned membrane airfoils. The membrane deformations determine the wing’s aerodynamic performance and therefore play a vital role in the ultimate success of the vehicle. When a finite element prediction of the membrane deformation is sought, it is typically performed using a second-order displacement formulation with a time-marching algorithm; that is, the spatial domain is solved using initial conditions, then the initial conditions for the next time step are predicted from known trajectories. This research seeks an alternative approach in two aspects. First, Hamilton’s Law of Varying Action is employed with appropriate Lagrange multipliers to reduce the order of the governing equations to one. The resulting mixed
formulation permits the use of simpler elements with linear shape functions. Second, traditional three-dimensional solid elements are adopted for use in a space-time mesh. The mesh is created by extending any given two-dimensional spatial mesh through time such that each spatial node is duplicated at every time slice. Conventional finite element algorithms then produce a sparse linear system that can be solved for all degrees of freedom in the discrete space-time domain simultaneously. Thus the intrinsic time integration in the finite element algorithm effectively provides the stability benefits of an implicit technique. For this study, membrane deformations are assumed to be small, permitting a linearly elastic analysis with only transverse displacements. Numerical results demonstrate the suitability of this method given these constraints. Static and dynamic test cases are used to validate the model using isoparametric quadrilateral- and triangle-based elements (trilinear hexahedrons and triangular prisms) in structured and unstructured grids. Accuracy, energy conservation, and relative computational expense are addressed.

**Numerical Investigation of Plasma-Based Control for Low-Reynolds Number Airfoil Flows**

*Rizzetta, Donald P, AFRL, Wright-Patterson Air Force Base, OH
Visbal, Miguel R, AFRL, Wright-Patterson Air Force Base, OH*

Large-eddy simulations (LESs) were carried to investigate the use of plasma-based actuation for the control of flows over a wing section at low Reynolds number. The sectional geometry corresponded to the SD7003 airfoil, which is representative of those employed for micro air vehicle applications. Dielectric-barrier-discharge plasma actuators were utilized to modify the transitional flow, and improve aerodynamic performance. Solutions were obtained to the Navier-Stokes equations, that were augmented by source terms used to represent plasma-induced body forces imparted by the actuators on the fluid. Simple phenomenological models provided the body forces generated by the electric field of the plasma surrounding the actuators. The numerical method was based upon a high-fidelity time-implicit scheme, an implicit LES approach, and domain decomposition in order to perform calculations on a parallel computing platform. The computational flowfield was described by 40 million mesh points, and a grid resolution study was performed to confirm numerical accuracy. Flow at a chord-based Reynolds number of 40,000 was considered in the investigation, which is characterized by laminar separation on the suction surface of the wing at low angles of attack. This separation then promotes transition to a more complex state, that can be modified by the use of plasma actuation. Several aspects of control were examined, including different actuator configurations, alternative plasma-force models, both continuous and pulsed modes of operation, and the magnitude of plasma force required for control.

It was found that the pulsed mode of operation was far superior to that of continuous operation. Even at low force magnitudes, plasma control was able to completely eliminate separation on the wing upper surface. This resulted in a 20 per cent decrease in drag and an increase of 18 per cent in the lift to drag ratio. Alternate actuator configurations worked equally well, although the mechanisms for control differed. There was little sensitivity of the results to varying plasma-force models.
Analysis of Flow Migration in an Ultra-Compact Combustor

Bohan, Brian T, AFIT/ENY, WPAFB, OH
Polanka, Marc D, AFIT/ENY, WPAFB, OH

Current research at the Air Force Research Laboratory (AFRL) focuses on developing an Ultra Compact Combustor (UCC) system for integration of a compressor exit vane with a circumferential combustor and a high pressure turbine vane. There is a current push to investigate a larger, fighter sized UCC within AFRL. AFIT has led this push by performing Computational Fluid Dynamic (CFD) simulations to scale the UCC to a fighter size. The Ultra-Compact Combustor attempts to shorten the overall combustion length by performing the combustion in the circumferential direction along the outside diameter of the core flowpath. This presents several design challenges including how to control the fluid velocity in the circumferential cavity and how to turn the centrifugal combustion flow back to the axial direction into the high pressure turbine rotor while presenting a uniform temperature across the turbine blades. The turning of the core flow is accomplished by designing a single airfoil row of 20 vanes that maintain the swirl from the last compressor rotor to the inlet of the high pressure turbine rotor. One of the major factors of this design has been to establish a high g field in the circumferential cavity. Several parameter studies have been conducted to establish relationships to predict tangential velocity based on cavity inlet conditions. Through this control, a UCC section has been configured with a g-loading around 3500 gs at an outside diameter of roughly 0.75m. Furthermore, the UCC airfoil has been designed to optimize the temperature distribution leaving the UCC section.

Design of a Fighter Sized UCC Section

Wilson, Jacob D, AFIT/ENY, WPAFB, OH
Bohan, Brian T, AFIT/ENY, WPAFB, OH
LeBay, Kenneth D, AFIT/ENY, WPAFB, OH
Polanka, Marc D, AFIT/ENY, WPAFB, OH

Current research at the Air Force Research Laboratory (AFRL) focuses on developing an Ultra Compact Combustor (UCC) system for integration of a compressor exit vane with a circumferential combustor and a high pressure turbine vane. To date this work has focused around missile sized combustors. The Air Force Institute of Technology (AFIT) has been complimenting this effort through experimental work using a small scale sectional UCC model. Many insights into the combustion process and fluid mechanics have been gleaned over the years. There is a current push to investigate a larger, fighter sized UCC within AFRL. AFIT has led this push by performing Computational Fluid Dynamic (CFD) simulations to scale the UCC to a fighter size. With sizing of essential characteristics (cavity size, vane height, etc.) known from the CFD results, a large scale UCC section was designed to meet the needs of these future experiments. Of particular interest is the study of flow migration from the UCC cavity section into the main core flow. This rig was designed to allow non-intrusive optical access for laser diagnostics such as planar laser induced fluorescence (PLIF), particle image velocimetry (PIV), and Rayleigh scattering, as well as high-speed imaging. The model consists of a one hundred degree circumferential UCC section with three circumferential optical access points, and a two-vane core flow section with optical access from the top and bottom. A wide range of operating conditions and airfoil designs will be tested over the rig’s life and an accurate, detailed analysis of the
flow will be obtained. This research will be the initial step toward the development of a full scale fighter size engine to utilize the UCC concept.

Investigation of the V-Shaped Ambient Flame from a Sectional Ultra Compact Combustor

LeBay, Kenneth D, AFIT/ENY, WPAFB, OH
Polanka, Marc D, AFIT/ENY, WPAFB, OH

Initial investigation of the sectional Ultra Compact Combustor (UCC) rig at the Air Force Institute of Technology exhibited a V-shaped flame at the combustor's ambient exit. This shape was only initially observed in the straight section which lead to the hypothesis of the centrifugal loading in the curved section increasing mixing to overcoming the V-shape. This hypothesis was investigated using Planar Laser Induced Fluorescence (PLIF) of the OH radical over multiple centrifugal loads and UCC to core flow velocity ratios. Reduction of the PLIF data indicated virtually no dependence on centrifugal load with respect to presence of the V-shape. However, the PLIF data clearly indicated that the presence of the V-shape was determined to be almost entirely a function of the UCC to core flow velocity ratio.
Compressive Residual Stress Optimization in Laser Peening of a Curved Geometry

Vasu, Anoop, Wright State University, Dayton, OH
Grandhi, Ramana V, Wright State University, Dayton, OH

Laser Peening (LP) is a surface enhancement technique which can delay crack nucleation by inducing compressive residual stresses. High stress concentration along the curved portion of a structure is a common cause for crack initiation which leads to fatigue failure. Most fatigue cracks nucleate near the surface and propagate. A 3D finite element model is created using ABAQUS for simulating the dynamic response of a sequential 2 shot LP impact of a component’s surface. Compressive residual stresses along the edge where the component is laser peened is the major prediction of this simulation and serves as the objective function to be maximized. Peak pressure pulse amplitude, mid-span duration, curvature are the design variables considered. Tensile stress constraint at critical locations complete the problem formulation. The computational cost of LP simulation is high due to incremental time analyses. Hence, surrogate simulation models are used in reducing the number of finite element analyses required for optimization. These surrogate models, referred to as function approximations, are based on gradient information and are employed to reach an optimum design more quickly. The work focuses on the design optimization of LP parameters for fatigue life extension of curved geometries.

Structural Optimization of Aircraft Engine Exhaust-Washed Structures

Deaton, Joshua D, Wright State University, Dayton, OH
Grandhi, Ramana V, Wright State University, Dayton, OH

To meet the growing demands for improved mission capability, combat survivability, and versatility of aerospace systems, future military aircraft will continue to rely on low-observable technology. One method of reducing the observability of an aircraft is to embed its engines inside the airframe, which allows for a smooth outer mold line, reduced exhaust noise, and cooler exhaust gases. This decreases radar, acoustic, and infrared detectability while preventing direct line-of-sign into hot engine components. While embedded engine configurations afford tremendous tactical capability, they require that hot exhaust be passed to the rear of the aircraft. This creates a region located aft of the embedded engines that experiences an extreme multidisciplinary loading environment that includes thermal effects in addition to conventional mechanical loading. Structural components in this environment are called engine exhaust-washed structures (EEWS). The structural design of these components is complicated by non-intuitive, configuration specific structural responses to elevated temperatures, thermal stresses resulting from thermal expansion, and combined thermal-structural loading. In fact, prior work has demonstration that conventional room temperature design techniques may not apply in a thermal-structural design environment like that experienced by engine exhaust-washed structures. For example, due to the design dependency of thermal loading, adding material to an EEWS in an effort to reinforce it or lower stresses may actually increase the magnitude of stresses in that component and increase the load transferred to surrounding structures.

To date an analysis framework based on sequentially coupled thermal-structural finite element analysis has been developed to model the EEWS in the extreme thermal-structural environment. For various configurations involving both metallic and composite materials, sensitivity analysis has demonstrated the variability within the complex design space and identified basic design challenges. It has been shown that in a built-up structure subjected to
combined thermal-structural loads that modifying dimensional parameters to address a critical response in one area of the structure may inadvertently create a critical response in another area or even worsen the response in the area of interest. This occurs because the thermal loading that drives the structural response is dependent upon the design configuration at many levels ranging from the effect of temperature on materials properties, the method of determining and representing thermal loads in the structural analysis, and most fundamentally, the fact that the amount of structural material governs the amount of thermal loading.

In addition, the application of sizing optimization to several exhaust-washed structures has shown that without greater design freedom it is difficult to develop an effective, reliable, and weight efficient design. This has provided the impetus for the use of structural topology optimization to develop new structural layouts that are better able to accommodate the loading environment. Early results indicate that topology optimization is able to control and tailor the direction and magnitude of thermal expansion within the structures to prevent excessive thermal stresses and load transferred to surrounding structures. The presentation will highlight the major design challenges related to engine exhaust-washed structures and the results of the application of size and topology optimization in the design of these components.

**Optimal Determination Of The Elastic Modulus Of The Vein Structure From The Forewing Of Manduca Sexta Using Analytical And Experimental Techniques**

_O'Hara, Ryan Patrick, AFIT, WPAFB, OH_
_Palazotto, Anthony, AFIT, WPAFB, OH_

The dynamic forced response of the vein structure of the Manduca Sexta tested under simple cantilever beam conditions is used to predict the elastic modulus of the structure. The experimental forced response of the vein structure is tested using laser vibrometry and modal analysis. Through the use of investigative pathology the volumetric structure of the wing is understood and analytically approximated using simple finite element analysis techniques. Using an unconstrained optimization, the unknown elastic modulus of the vein structure can be easily approximated by tuning the analytical model to match the experimental results. This work is a continuation of efforts to better model and simulates the biological system to garner a better understanding of the critical structural features of the wing for using in future bio-inspired designs.

**Multidisciplinary Optimization of a Hovering Wing with a Service-Oriented Framework and Experimental Model Validation**

_Bryson, Dean E., Air Force Research Laboratory, Air Vehicles Directorate, WPAFB, OH_
_Stanford, Bret K., Air Force Research Laboratory, Air Vehicles Directorate, WPAFB, OH_
_McClung, Aaron M., Air Force Research Laboratory, Air Vehicles Directorate, WPAFB, OH_
_Sims, Travis W., Air Force Research Laboratory, Air Vehicles Directorate, WPAFB, OH_
_Miranda, Jose L., Air Force Research Laboratory, Air Vehicles Directorate, WPAFB, OH_

An aeroelastic flapping analysis is integrated with a commercial, gradient-based optimizer within a computational framework. The aeroelastic analysis couples a geometrically-nonlinear beam formulation with a quasi-steady blade element aerodynamics tool and trigonometric flapping kinematics. Analytic gradient information is produced for peak power required, cycle-averaged lift, and maximum von Mises stress with respect to element chord and thickness and nine kinematic parameters, alleviating the burden of finite-difference gradients. The chord and thickness distributions and kinematics were simultaneously optimized to provide a wing requiring minimum flapping power under constraints on lift and stress. Three optimized designs are presented, yielding more than 70% reduction in peak power requirement from baseline designs, and 28% reduction from a design produced by another
optimization method. This work concludes with an experimental validation of the aeroelastic tool through the comparison of various static, dynamic, and flapping metrics.
Session 43: Nonlinear Dynamics and Control
Chair: Richard Cobb, AFIT

Maneuverability Region Computation for Nonlinear Dynamical Systems: Max-Plus Methods

Polete, Sean P, ENY/US Navy, Dayton, OH

In the study of dynamical systems, the concept of reachable and controllable sets is intimately connected with the solution space to optimal control problems. When considering questions on the feasibility of attaining a system state (or the set of achievable system states) subject to constraints on the controls, these sets offer insight into the existence of time optimal control solution. Feasibility of achieving a system state is captured in the notion of the maneuverability set, which combines the ideas of reachability and controllability providing further insight into system state sets that can be attained and returned from. Additionally, a characterization of the maneuverable set, provides an invariant measures of system performance with application to actuator design, controller synthesis, performance analysis, and time optimal path planning. In this paper the reachability, controllability, and maneuverability regions for general nonlinear dynamical systems are defined and a characterization and description of their set properties are given. Additionally, an algorithm based on level sets and Hamilton-Jacobi type equations is outlined and examples of maneuverability set computation and applications are illustrated. While these notions hold for other dynamical systems, the examples presented are specific to our work with mechanical dynamical systems. Finally, a discussion of further advances for this analysis technique are presented on our work into Max-plus algebraic methods for high dimensional systems.

Near-Real-Time Stochastic Optimal Control: A Pseudospectral Approach for Trajectory Optimization to an Uncertain Target with a Bearing-only Sensor

Ross, Steven M, Air Force Institute of Technology, Wright-Patterson AFB, OH
Cobb, Richard G, Air Force Institute of Technology, Wright-Patterson AFB, OH

A Gaussian Pseudospectral Collocation Method is applied in an effort to simultaneously solve the optimal control and the optimal estimation problem to an uncertain target using bearing-only information. The result is a min-time profile to a desired offset point from the unknown target position, while achieving a minimum specified certainty in relative position as a result of the geometry of the path. This technology can be applied to any system which relies on relative motion about a target and uses only a bearing only sensor, providing first launch opportunity for submarines using passive tracking, maximum kill potential for High-Speed Radiation Missiles, or optimal UAV path planning.

For experimental demonstration, a quadrotor UAV is directed to an approach point and landing on a wire. Related research has demonstrated the capability to recharge batteries using induction on medium-size power lines, providing the potential for greatly extended range and loiter duration, provided power lines can be landed on with cheap navigation sensors, uncertain wire locations, and a bearing-only ranging system using the optical capabilities of the UAV. A simple web-cam style camera provides slow, noisy bearing measurements and is used with an extended Kalman filter for target location estimation. The target estimate is used with a trajectory planning system that determines the optimal path for the UAV to fly to a point offset from the target’s assumed location with a geometry that will ensure enough observability to achieve a level of target location certainty sufficient to begin the landing approach. Requiring a minimum certainty will allow fixed-wing UAVs sufficient safety margin to begin...
stall maneuvers for the wire landing. The optimal control problem is solved recursively with the best target estimate at the time, via collocation to a non-linear programming problem to increase solution speed.

**Development and Verification of a Pseudospectral Forward Model for Collisional Plasma Diagnostics**

*Stults, Joshua A., AFIT, WPAFB, OH*

*Huffman, Richard E., AFIT, WPAFB, OH*

Plasma diagnostics such as microwave interferometry require a forward model to relate the measurement (phase and amplitude shift) to plasma properties (electron density and effective electron-neutral collision frequency). In the present work a two-way wave equation for the electric field of an interferometer beam is discretized by a pseudospectral method of lines approach, and stabilized with Lanczos-factor smoothing. A Chebyshev pseudospectral method is employed that is suitable for non-periodic material properties. This numerical solution of the forward model also allows arbitrary material property gradients to be accurately treated in contrast to simpler analytical approaches (e.g. Appleton-Hartree). The phase and amplitude shift of the interferometer beam is assumed to be caused only by electron currents and electron-neutral collisions in a Lorentz model of the electron motion. The numerical implementation is formally verified using the method of manufactured solutions (MMS). The capabilities of a computer algebra system (CAS) are leveraged throughout for code generation and verification. The model is applied to diagnosing a dielectric-barrier discharge in air.
Investigation of Transverse Jet Injections in a Supersonic Crossflow Using Fast Responding Pressure-Sensitive Paint

Crafton, Jim, Innovative Scientific Solutions, Inc., Dayton, OH
Fonov, Sergey, Innovative Scientific Solutions, Inc, Dayton, OH
Forlines, Robert, Innovative Scientific Solutions, Inc, Dayton, OH
Palluconi, Steve, Innovative Scientific Solutions, Inc., Dayton, OH

The measurement of luminescence from Pressure-Sensitive Paint (PSP) allows for the derivation of distributions of surface pressure. Traditional PSP systems can provide data with high spatial resolution; however, the bandwidth is limited to a few Hz by the response time of the paint. Fast responding paints have been developed using anodized aluminum and porous polymer formulations and these PSPs have demonstrated response times of up to 100 kHz. These fast PSPs have been used with traditional pulsed illumination sources and digital cameras to acquire instantaneous pressure distributions in unsteady flows. There is significant interest in combining fast responding PSP with high-frame-rate cameras to produce data that has both high spatial resolution and high temporal bandwidth. Unfortunately, older PSP illumination sources and fast framing cameras do not provide sufficient signal-to-noise ratio to acquire quantitative pressure data. Ultra-bright LEDs and fast framing cameras—that are now available—combined with porous polymer PSP, can be used to produce a system capable of both high spatial resolution and high bandwidth. Measurements of mean and unsteady pressure have been acquired on an experimental setup composed of a Mach-2 channel flow with transverse jet injection. Mean pressure distributions were acquired using a binary PSP system. The fast PSP system consisted of a porous polymer PSP illuminated with a diode array and imaged with a CMOS camera. Full frame images (for a 1k × 1k CMOS chip) were acquired at 7 kHz; reduced frame images were acquired at up to 25 kHz. The time averaged pressure data from the fast PSP system compared favorably to PSP data acquired using the binary FIB system. The unsteady pressure data clearly resolves structures not present in the mean pressure data. These structures include multiple lambda shocks upstream of a strong bow shock and high frequency perturbations in the location of these shocks. Significant deformations of the bow shock structure are detected as turbulent structures in the flow convect through the shocks and into the jet. Finally, a time series of data can be extracted at each image pixel and the spectral content and phase relationship of the flow can be investigated spatially. This type of spectral map can be created using arrays of fast pressure transducers; here, however, we present data representing an array of over 10,000 fast pressure transducers.

Aeroelastic Jet Impingement Using Fast-Responding Binary Pressure Sensitive Paint

Palluconi, Stephen J, Innovative Scientific Solutions, Inc., Dayton, OH
Derbeneva, Anna, Innovative Scientific Solutions, Inc., Dayton, OH
Fonov, Sergey, Innovative Scientific Solutions, Inc., Dayton, OH
Gregory, James W, Ohio State University, Columbus, OH

Aircraft designs that incorporate long slender wings and morphing wing control surfaces often encounter aerodynamics events such as dynamic stall and flutter, events that include significant unsteady pressure oscillations as well as structural deformation. This phenomenon must be interrogated experimentally and computationally to understand the underlying physics and improve designs. Experimental data with high spatial resolution such as pressure and surface geometry are needed to understand the fluid physics and validate numerical models. It is
essential that the pressure and geometry data be acquired simultaneously since this is by definition a fluid/structure problem. Simultaneous measurements of geometry and pressure allow the computational fluid dynamics grid to be modified to match the experimental data. The computations can then be run on the actual experimental configuration and then direct comparison between the experimental and computational data can be achieved.

Recently, Innovative Scientific Solutions, Inc. has developed a system capable of acquiring unsteady surface pressure and geometry. This system is based on a fast responding binary FIB pressure sensitive paint. The paint exhibits kHz frequency response and compensates for temperature variations using the reference channel. The pressure sensitive paint data is acquired using a single color camera, and therefore, the signal and reference data is acquired simultaneously. Finally, the location of markers or pressure taps can be tracked in the pressure sensitive paint images, facilitating photogrammetric reconstruction of the surface. The systems capability is demonstrated using a simple jet impingement experiment. The impingement surface is mounted on a cantilevered beam, allowing the impingement surface to deflect under the jet loading. The jet is positioned at a shallow impingement angle and the system is allowed to oscillate resulting in a fluid-structure system. The pressure field on the impingement surface and the surface geometry are investigated at several operating conditions.

Development Of High Resolution Eddy Current Imaging Using Electro-Mechanical Sensor

Cherry, Matthew R, University of Dayton Research Institute, Dayton, OH Sathish, Shamachary, University of Dayton Research Institute, Dayton, OH Reibel, Richard, University of Dayton Research Institute, Dayton, OH Blodgett, Mark, WPAFB, Air Force Research Labs, Metals Cermaics and NDE Division, Dayton, OH Welter, John, WPAFB, Air Force Research Labs, Metals Cermaics and NDE Division, Dayton, OH

An electromagnetic coil is the most often used sensor for eddy current based NDE. When the coil is excited by an oscillating current and placed near a metallic material, the periodic magnetic field induces eddy currents in the metal. In turn, the eddy currents produce an opposing magnetic field which affects the impedance of the electromagnetic coil. In NDE applications, particularly detection of surface and sub-surface flaws, the coil is run over a part and the impedance characteristics of the coil are mapped. The spatial resolution in the images is limited by the diameter of the coil and the spread of the magnetic field. To obtain high resolution eddy current images, several approaches have been used such as the use of coils of finer dimensions, Hall Effect sensors and Giant Magneto Resistance (GMR) sensors. While great progress has been made in improving the resolution, several limitations have been encountered in each of these methods. This paper presents an electro-mechanical method of detection of eddy currents that has potential to provide resolution from in the range of microns to millimeters. The electro-mechanical eddy current sensor consists of an air core coil and a sharp magnetic needle held at the center with a membrane (Fig.1). The membrane consists of an electret film and is held near a conductive back plate. When the coil is excited, the magnetic needle vibrates at the same frequency of excitation. This motion is detected by monitoring the voltage between the membrane and back plate. When the sensor is placed in the vicinity of a metallic material, eddy currents are generated by not only the coil, but also the moving magnetic field from the tip vibration. It is shown that while a change in the exciting magnetic field is caused by the eddy currents under the coil, the damping forces experienced from the local eddy currents at the tip have a stronger effect on the tip motion. Since these effects are local to the tip of the needle, the sensor is expected to have higher spatial resolution than the coil alone, and this is indeed shown through experimentation and modeling.
A Computational Study of the Fluid-Structural Interaction on a Morphed Airfoil

Collins, Eric R, University of Kentucky, Lexington, KY
Capece, Vincent R, University of Kentucky, Paducah, KY

Active aerodynamic flow control, such as morphing the geometry of an airfoil during operation, is the subject of considerable research interest. When attempting to actively deform an airfoil, aerodynamic loads must be considered in addition to the structural loads producing the deflection. In this study, a coupled fluid-structural analysis is performed computationally on a morphed wind turbine airfoil to illustrate the importance of aerodynamic loads. The National Renewable Energy Laboratory (NREL) S809 airfoil, composed of an elastic material, is deformed by means of internal pressure channels. Aerodynamically, the airfoil is oriented at an angle of attack of 5.13 degrees in a flow of air at a chordal Reynolds number $Re_c=2$ Million. The influence of the aerodynamic loads on the deflection at the trailing edge of the airfoil is evaluated.