Research Expo

2012

HERRICK LABORATORIES
PURDUE UNIVERSITY

EAST WING

Station 1

Station 2

Station 3

Station 4

Station 12

EAST

WING

DEVELOPMENT OF A ROBUST TECHNIQUE TO MITIGATE THE IMPACT OF UNCERTAINTIES IN IMPLEMENTATION OF REAL TIME HYBRID SIMULATION

Research Assistants: Amin Maghareh and Gregory Bunting

Principal Investigators: Professors Shirley J. Dyke and Arun Prakash

Sponsor: National Science Foundation

ABSTRACT

Real time hybrid simulation (RTHS) is a promising cyber-physical technique for the experimental evaluation of civil infrastructure systems. Hybrid simulation allows for simulation of highly complicated civil infrastructure systems in a cost effective manner by partitioning them into numerical and experimental sub-assemblages. The coupling between the two substructures is achieved by enforcing equilibrium and compatibility at the interface between the physical and computational portions. The potential of hybrid simulation has been noticeably extended by executing the simulation at run time. Performing the hybrid simulation at real-time (RTHS), offers the capability to preserve rate-dependence while examining the performance of the physical substructure. In hybrid simulation, response of the computational part is obtained within a time-step using either explicit or implicit numerical integration for a particular ground excitation. At each time step, displacements (and velocities) at the interfaces of the substructures are computed based on the numerical substructure and commanded to the servo-hydraulic actuator(s) and the experimental substructure. The restoring force of the experimental substructure is measured and fed back to the numerical substructure for the next time step calculation. The experimental and numerical substructures, the integration algorithm and the servo-hydraulic actuator(s), all run at real-time, constitute a real time hybrid simulation.

However, systematic and random uncertainties developed in the experimental substructure are inevitable and can have substantial impacts on the quality of the simulation results. Uncertainties are introduced into hybrid simulation through other sources of error such as structural modeling and numerical integration. In hybrid simulation, structural modeling error associated with the numerical substructure includes error due to discretization (stiffness and mass distribution), condensation/negligence of some higher modes, and modeling of damping effects. Error associated with the numerical integration scheme can cause instability issues, period elongation, amplitude variation, alteration of frequency content, and introduction of numerical damping. Among all sources of error in hybrid simulation, the results of hybrid simulation are most sensitive to experimental errors. Hence, to acquire more reliable simulation results in hybrid simulation, it is vitally significant that the propagation of experimental and numerical uncertainties is effectively mitigated.

COMPARATIVE STUDY OF CONTROL ALGORITHMS ON A SDOF SPRING-ACTUATOR SYSTEM FOR REAL TIME HYBRID SIMULATION

Research Assistant: Gaby Ou

Principal Investigator: Professor Shirley Dyke

Sponsor: NSF – CMMI-0927178

ABSTRACT

Real-time hybrid simulation (RTHS), in contrast with traditional pseudo-dynamic hybrid testing approach, is applicable for structures with rate-dependent components. One of the major challenges in RTHT is that it requires both guaranteed execution of each test cycle in a small time step and appropriate compensation for time delays and actuator dynamics. Time delays and inherent dynamics will lead to inaccuracies and even further instabilities while running test in real time.

Many researchers provided different control algorithms to solve aforementioned problems; this poster illustrates one comparative study between common used control algorithm in RTHS, including inverse compensator, FF-FB control and H-infinity control. Criterion of control performance are based on time delay between desired signal and output, RMS error in time domain and further sensitivity of specimen property change during testing.

The baseline comparison is implemented on the band limited white noise displacement tracking of a 1 kip actuator attached one spring with stiffness of 400 lb/in. A parameter based actuator-spring model is used for designing each control algorithm. During the test, springs with different stiffness varying between 0 lb/in to 600 lb/in are used to test the robustness of control algorithm in respect to specimen property change. Corresponding results and conclusions are given in the poster.

CYBER-PHYSICAL CO-DESIGN OF WIRELESS CONTROL AND MONITORING SYSTEM

Research Assistants: Zhuoxiong (Charlie) Sun and Bo Li

Principal Investigator: Professor Shirley Dyke

Sponsor: National Science Foundation

ABSTRACT

Cyber-physical systems (CPS) are smart systems that have cyber components, deeply embedded in and interacting with physical components, and sensing and changing the state of the real world. Cyber-physical systems (CPS) have been given growing attention in the research community and several reports from White House have recommended developments and progresses in CPS areas. In our work, a cyber-physical co-design approach is developed for wireless structural control and monitoring system. Wireless network systems have advantages compared to traditional wired network system such as flexible installation, rapid deployment, low maintenance cost, low power consumption. Rapid developments in wireless sensor software, hardware, middleware make wireless structural control and monitoring a very promising field.

In our approach, an integrated simulator is developed with both cyber components – wireless network system and physical components-structural control system. Cyber components are built in TinyOS network simulator-TOSSIM and physical components are built in MATLAB Simulink. Both components are built with high fidelity. Two sample cases illustrate our approach have been provided within this work, one for benchmark active mass driver (AMD) structural control model, the other for benchmark cable-stayed bridge structural control model. These two sample cases are used to show the scalability and generality of our approach. The structural control performances of these two models can be evaluated with different controller and wireless network systems.

ESTIMATION AND COMPENSATION OF FUEL QUANTITY VARIATION IN HIGH PRESSURE INJECTORS

Research Assistants: Sai Shirsikar and Pranav Bhalerao

Principal Investigator: Professor Peter Meckl

Sponsor: Cummins Fuel Systems

ABSTRACT

The injected fuel quantity does not always follow the desired value. It varies for a commanded on-time at a given rail pressure for an injector. The main focus of the work is on finding the causes for this variation in fuel quantity, estimating the fueling error and compensating for it using an adaptive control strategy in high pressure multiple pulse fuel injectors. The project is sponsored by Cummins Fuel Systems, Cummins Inc. (The injector is a solenoid driven heavy duty high pressure injector). A number of things like pulse to pulse interaction, start and end of injection delays, body pressure variations etc. cause fueling errors in multiple pulse injectors. With the help of a six sigma strategy Critical Parameter Flow Down, two parameters namely body pressure variation and armature over-travel were decided to be the major error causing factors. Rate shapes and body pressure variations were analyzed using a simulation model based in GT Fuel. It was seen that we could relate the under or over fueling of the injector to the rising or falling nature of the body pressure waveform. Thus it was necessary to estimate the body pressure at a given instant and relating it to the fueling quantity, thus making us able to measure the aberration with respect to the commanded fueling. A simple physics based state space model was developed and simulated treating rail pressure as controllable input and body pressure as output. The fueling error mainly depends on the rate shape parameters. These parameters are estimated from the body pressure waveform using regression resulting in the estimation of fueling errors. A scheme was deduced to change the on-time to compensate for the estimated fueling error. For single pulse injection in high pressure injectors, rate shape parameters can be estimated using relationships defined as functions of the commanded on-time and rail pressure. This work extended to multiple pulse case will serve to find out the rate shape parameters accurately for the body pressure model and estimation of the fueling error.

REAL-TIME ON BOARD INDIRECT LIGHT OFF TEMPERATURE ESTIMATION AS A DETECTION TECHNIQUE OF DIESEL OXIDATION CATALYST EFFECTIVENESS LEVEL

Research Assistant: Raymond Sutjiono

Principal Investigator: Professor Peter Meckl

ABSTRACT

The latest US emission regulations require dramatic reductions in Nitrogen Oxide (NOx) emissions. Selective Catalyst Reduction (SCR) is the current technology that achieves NOx reductions of up to 90%. It is typically mounted downstream of the existing aftertreatment system, i.e, after the Diesel Oxidation Catalyst (DOC) and Diesel Particulate Filter (DPF). Accurate prediction of input NO2:NO ratio is needed for the SCR urea injection control algorithm to reduce NOx output and NH3 slippage downstream of the SCR catalyst. Most oxidation of NO to NO2 occurs in the DOC since its main function is to oxidize emission constituents. The DOC thus determines the NO2:NO ratio as feedgas to the SCR catalyst.

The prediction of NO2:NO ratio varies as the catalyst in the DOC ages or deteriorates due to poisoning. Therefore, the prediction cannot be determined by a single model. Instead, the model should take into account the correlation of DOC conversion effectiveness and the aging of the catalyst. This research project is aimed at detecting the aging level in the DOC in real time on board the vehicle by estimating light off temperature. The estimation strategy has been proven quite accurately to measure indirect light off temperature with thermal balance. The work in this paper could suggest an on board catalyst aging detection technique with existing aftertreatment sensors.

ROBUST STATISTICAL STRATEGY TO IDENTIFY ANOMALIES IN DIESEL ENGINE PERFORMANCE

Research Assistants: Ben Warman and Aniket Jayant Vagha

Principal Investigators: Professors Peter Meckl, Galen King, and Kristopher Jennings

Sponsor: Cummins Inc.

ABSTRACT

Tighter emissions and On-Board- Diagnostic (OBD) regulations mandate that engine manufacturers add emissions-reducing systems while also developing diagnostic tools to meet OBD regulations. Failures within the physical systems on the engine are expected to be detected by the OBD system. However, current methods of data analysis may prove to be insufficient in detecting engine anomalies not associated with a healthy system. We propose a diagnostic method based on statistical methods that will help identify the latest working state of a diesel engine as healthy or faulty. Previously, a two class - classification algorithm called Sparse Linear Discriminant Analysis (SLDA) was used by researchers at the Herrick Laboratories to effectively select the most relevant signals for classifying between healthy and faulty conditions. Field test data is used to develop a technique to match the operating conditions (engine speed, torque and environmental factors) of healthy engine data and faulty engine data. It was found that the steady state data makes for a more accurate classification of the working conditions. Hence a full proof algorithm to extract steady data is used. Moving from state diagnostics of an individual truck, we aim to raise the applicability of this research. A robust scheme is therefore proposed which will be capable of handling multiple trucks with similar ratings for power and calibrations. This model will be tested using the OBD failure mode data provided by Cummins Inc. and will be used to develop a tool for Systems engineers to analyze the on-road behavior of the truck in a fast, efficient manner.

CHARACTERIZATION, MODELING, AND CONTROL OF VARIOUS DIESEL ENGINE TECHNOLOGIES UTILIZING A FULLY FLEXIBLE VALVE ACTUATION SYSTEM

Research Assistants: Mark Magee, Chuan Ding, Leighton Roberts, Akash Garg

Principal Investigator: Professor Greg Shaver

Sponsors: Cummins Inc and Eaton

ABSTRACT

To meet consumer demands for improved fuel economy as well as increasingly stringent emission regulations, engine manufacturers have begun to investigate the benefits of variable valve actuation systems. By modulating the intake and exhaust valve events, the combustion and gas exchange processes of the engine are directly affected, and depending on the change, possible improvements on emissions or fuel economy or both. For example, by closing the intake valve late or early in a cycle, the effective compression ratio (ECR) can be decreased which when coupled with a standard exhaust valve event leads to different compression and expansion ratios. Traditionally in an engine, these ratios are equal, but by increasing the ratio of expansion to compression, more work can be extracted leading to improved thermal efficiency. Furthermore, for exhaust after treatments to be most effective, they must be maintained within a certain temperature range. Typically, it is more difficult to maintain the minimum required temperature. One of the most direct actuators to maintain this temperature range is the exhaust valve event. By opening the exhaust valve early, more hot exhaust gases are released into the tail pipe raising the temperature seen at the after treatment system. Lastly, in a standard drive cycle, the engine operates at part load conditions for a significant portion of the cycle. Under these conditions, it is not often required that all cylinders are firing. By deactivating several cylinders, fuel economy benefits can be seen as well as an increase tailpipe temperature improving after treatment performance.

Traditionally, engine valve trains are mechanical cam driven systems; however, with the modern technologies several novel valve actuation systems are becoming viable. Characterization of the valve modulation effects mentioned above will be performed on a 2010 Cummins ISB 6.7L engine with almost exclusively stock hardware (stock pistons, injectors/nozzles, turbocharger, etc.). This engine utilizes cooled exhaust gas recirculation (EGR), variable-geometry turbocharging, charge air cooling, and a common rail injection system. A very unique capability of this multi-cylinder testbed is that it is outfitted with a fully-flexible electro-hydraulic variable valve actuation (VVA) system that enables cylinder-independent and cycle-to-cycle control of the engine's valve events. Currently, only the intake valves are flexibly actuated; however, by spring 2012, both the exhaust and intake valves will be fully flexible furthering increasing the testbed's capability.

CLOSED LOOP COMBUSTION CONTROL OF BIODIESEL BLENDS

Research Assistants: Carrie Hall and Gayatri Adi

Principal Investigator: Professor Greg Shaver

Sponsor: Cummins Inc.

ABSTRACT

The use of biodiesel blends has the potential to result in many benefits including decreased reliance on imported petroleum, increased sustainability, decreased net carbon dioxide emissions, and decreased particulate matter emissions. Conventional CI engine combustion modes are mixing-controlled and the main challenges in regard to the use of biodiesel are combating the power losses of the engine as well as increases in NO_x emissions of up to 40% with respect to diesel levels. An accommodation strategy was developed and shown to be capable of virtually eliminating the NO_x increases and power losses typically encountered with biodiesel in mixing-controlled combustion through the use of energy-based fueling and combustible oxygen mass fraction control. The stability of this control technique as well as its robustness to variation in the fatty acid structure of the fuel, such as would result from a variation in biodiesel feedstock, were also demonstrated. In addition, premixed-dominated combustion can also be targeted on fuel-flexible engines through the use of high exhaust gas recirculation fractions and early injection timings. Premixed charge compression ignition (PCCI) can be an efficient and very clean combustion mode; however, the effect of biodiesel in this combustion mode is altered from that in the mixing-controlled case. In PCCI, biodiesel usage results in lower power output, drastic increases in NO_x emissions (over 100% higher than the level of diesel) and earlier combustion timing with respect to diesel fuel. The oxygenation of biodiesel affects the in-cylinder oxygen fraction which in turn alters the combustion timing and NO_x emissions. A control algorithm which utilizes energy-based fueling provides fuel-flexible control of power output, combustion timing, and NO_x emissions simultaneously.

HIGH-SPEED DIODE LASER MEASUREMENTS OF TEMPERATURE AND WATER VAPOR CONCENTRATION IN DIESEL ENGINES

Research Assistant: Gurneesh Jatana

Principal Investigators: Professors Robert Lucht and Gregory Shaver

ABSTRACT

For a better understanding of gas exchange processes in a turbocharged diesel engine, crank-angle-resolved measurements of combustion products and temperature at various engine locations such as intake and exhaust ports, turbocharger inlet and exit, etc. are very important. Traditionally, combustion products are measured by sampling a small amount of gas while temperature is measured by placing thermocouples in the region of interest. Gas sampling usually cannot provide data at the kHz rates required for resolving crank angles at typical engine speeds. Very thin 'bare wire thermocouples' can provide near crank-angle-resolution; however, they do not survive the extreme conditions encountered in an engine, especially on the exhaust side. Therefore, there is a need to develop a sensor system that can provide the necessary resolution for the temperature and gas concentration measurements.

Diode laser based sensor systems utilizing absorption spectroscopy provide capability of performing simultaneous measurements of temperature and gas concentration at rates up to 20 kHz. Absorption spectroscopy is a quantitative, path averaged, and species specific measurement technique. For this study, water vapor was chosen as the target gas species. Water vapor readily absorbs in the near-infrared region and distributed feedback diode lasers as well as optics for near-infrared region are readily available because of their use in telecommunications.

Fresh charge and combustion are the only sources of any water vapor in an engine's gas exchange path. After correcting for the water vapor entering the engine with fresh charge, water vapor concentration can be related to carbon dioxide concentration through stoichiometry. Hence, water vapor concentration at the engine exhaust ports provides a good cylinder-specific measure of the combustion process and water vapor concentration at intake ports may be a useful port-specific measure of the distribution of the recirculated exhaust gas in the intake manifold.

A tunable distributed feedback diode laser is used to scan across multiple absorption transitions of water vapor and output is recorded with the help of high-speed data acquisition system. A spectral fitting code employing differential evolution based least square fitting is then used to calculate the temperature and water vapor concentration from the recorded data. The engine's intake manifold has been modified for optical access by installing connectors that sit flush with the manifold's inner surface. The technique is non-intrusive because the laser radiation does not interfere with the fluid flow. Measurements have been performed in the intake manifold of a diesel engine at various exhaust gas recirculation (EGR) conditions and results have been compared with gas sampling and thermocouple measurements.

CHARACTERIZATION OF THE NEAR-RESONANT VIBRATORY RESPONSE OF PLATES FORMED FROM MOCK ENERGETIC MATERIALS

Research Assistant: Jacob K. Miller

Principal Investigator: Professor Jeffrey Rhoads

ABSTRACT

Today, one of the greatest threats to the Armed Forces of the United States are improvised explosive devices (IEDs), which typically consist of disguised homemade explosive charges, a mechanical structure, and associated electronics. Despite the significant amount of IED-related research that has been conducted in recent years, no unilaterally effective and mass-deployable detection and defeat mechanism has been developed to date. The current research effort seeks to examine the utility of electromagnetic and acoustic excitations in stand-off detection systems. Specifically, the work seeks to develop and evaluate various experimental methodologies for pumping large amounts of energy into shielded energetic materials of arbitrary geometries using carefully-designed acoustic and electromagnetic excitation signals. To this end, this work seeks to characterize the macroscopic vibratory response of plates formed from surrogate energetic materials. The frequency response and operational deflection shapes are recovered from HTPB-based rectangular plates with varying binder ratios, and comparisons are made to similar metrics obtained via a first-principles analytical plate model as well as numerical simulation. In the future, this work will be extended to include the thermal response of the plate, with the aim of exploiting temperature increases within energetic materials for vapor pressure-based detection schemes.

CHARACTERIZATION AND OPTIMIZATION OF ENERGY TRANSFER TO ENERGETIC MATERIALS ACROSS A NONLINEAR INTERFACE

Research Assistant: Christopher Watson

Principal Investigator: Professor Douglas Adams

Sponsor: Office of Naval Research

ABSTRACT

Of utmost importance when attempting to identify and/or degrade energetic materials is the ability to get energy into them from an exterior source. Of particular interest is determining which excitation frequencies will transfer the maximum amount of energy. This can be a challenge, as the containers or casings that contain the energetic materials create physical barriers to energy transfer. Additionally, the interfaces between the material surfaces have imperfect contact profiles which further complicate the ability to understand, and subsequently model, the energy transfer mechanism from one medium to the other.

In order to understand how energy is transferred across the interface, a vibration model using randomly spaced contact patches is derived to predict a range of modal frequencies and shapes. Transmission of energy to the target is then predicted using strain energy of impulse response, and modal frequencies and shapes are identified that produce maximum energy transfer.

A composite buffer/target system is constructed with predefined and prearranged voids at the interface in order to control the experiment. Modal testing helps to validate and refine models. Preliminary results show that the model predicts natural frequencies well for a given % contact area. Predicted and measured maximum strain energy in target also agree in lower frequency range of ~160Hz.

Can the target frequencies be predicted based on the contact area percentage, or does the arrangement of the contact area, within a set % contact, play a significant part? How does varying the contact area % shift the target frequencies? Further research is needed to answer these questions.

OPTIMIZATION OF ACOUSTIC EXCITATION TO INCREASE IDENTIFICATION OF MECHANICAL PROPERTIES OF ENERGETIC MATERIALS

Research Assistant: Jelena Paripovic

Principal Investigator: Professor Patricia Davies

Sponsor: DOD – Semiwave Multi-University Research Initiative

ABSTRACT

Improvised explosive devices (IEDs) are often made to resemble objects that are a natural part of the environment in which the IEDs are placed. This project is part of a group of four Herrick/PCSI projects in which the use of acoustic excitation to distinguish between natural objects and objects that contain explosives is being investigated. This is part of a multi-university initiative; each university focuses on a different aspect of the problem. It is hypothesized that by understanding the mechanical behavior of the explosive material it may be possible to use an object's response to acoustic excitation to distinguish between objects that contain explosive materials and similar looking objects that do not. While the materials will typically be embedded in a system and the system behavior will play a role in the response, the research in this project will be mainly focused on the development of macro-scale models of the mechanical behavior of the materials. Hydroxyl-terminated poly-butadiene (HTPB) based composites are engineering materials utilized for various defense and commercial applications. By embedding inert NH_4Cl crystals within the HTPB matrix a surrogate explosive material is produced. The models, developed from observing the nonlinear response of the material to a variety of excitations, will be analyzed to identify the optimal excitation characteristics that enable robust estimation of the material model's parameters. We are conducting base excitation testing of a material-mass system to characterize the dynamic behavior of the material. The approach is to develop a model of the single-degree-of-freedom system incorporating nonlinear stiffness and damping behavior as needed to match predicted and experimental responses. The initial model incorporates nonlinear stiffness characteristics of the material derived from compression test experiments reported in the literature. This will provide an important step in the development of a model that can relate stress, strain, temperature, and strain-rate for materials used in IEDs. The models developed in this project will be shared with researchers in the other projects where system behavior is being examined.

RATE SHAPING ESTIMATION AND CONTROL OF A PIEZOELECTRIC FUEL INJECTOR

Research Assistants: Dat Le, Jin Shen, Bradley Pietrzak

Principal Investigator: Professor Gregory Shaver

Sponsor: Cummins Fuel Systems Inc.

ABSTRACT

Piezoelectric fuel injectors provide a means to lower emissions, noise and fuel consumption in advanced IC engines by enabling closely spaced, multipulse injection events as well as complex injection rate profiles – referred to a “rate shaping”. For modeling and control of a piezoelectric fuel injector during rate-shaped operating conditions, the project includes 3 steps: developing and experimentally validating a simulation model, simplifying the simulation model to implement a flow rate estimator on an NI CompactRIO system, and controlling the injector. In our group, a dynamic model of a piezoeletric fuel injector was outlined and shown capable of predicting fuel flow rate for single and multiple pulses, in which the needle is completely retracted, and flow rate is saturated. However, rate shaping requires a more precise model in order to capture the flow rate during needle “hovering”. Consideration of the flow rate nonlinearity, piezostack hysteresis, and hydro-mechanical dynamics is required to achieve precise control of rate shaping.

A reduced order model is then introduced by simplifying the physically-based simulation model. The estimation algorithm is derived from model by closing loop of the measurable quantities in the system. The resulting estimator runs at a "loop time" of 6us, is capable to make an online prediction of fuel injection flow rate, and it is also validated against experimental data.

MODELING, SIMULATION AND CONTROLS DEVELOPMENT IN EcoCAR 2

Research Assistants: Ashish Vora, Adam Fogarty, Bilwa Jadhav

Principal Investigators: Professors Greg Shaver and Peter Meckl

Sponsor: DOE, GM, ANL

ABSTRACT

EcoCAR 2 is a three year Advanced Vehicle Technology Competition (AVTC) organized by Argonne National Labs, where student teams from 15 universities are challenged with re-engineering a 2013 Chevrolet Malibu mid-size sedan into a Plug-in Hybrid Electric Vehicle (PHEV) to improve its fuel efficiency and emissions without compromising on performance, safety or consumer acceptability.

In the first year of the competition, the Purdue team had to select an optimal architecture and drivetrain components and plan the mechanical and electrical integration required to implement a hybrid powertrain. This was achieved through modeling and simulation of different hybrid architectures to analyze the vehicle performance on various competition metrics.

The Powertrain Integration team focused on packaging and integration of the additional powertrain components in the given vehicle, using the Siemens NX CAD and FEA package to design various components and demonstrate structural integrity comparable to the stock vehicle.

The ESS (Energy Storage System) team focused on the packaging and integration of the A123 Li-ion battery pack, design of a thermal management system for the batteries and electric motor, and high-voltage system design.

The goal of the Controls development team was to design a functional vehicle control and diagnostics strategy for the supervisory controller, based on the industrial V-diagram process, which includes model development, Software-in-the-loop simulation and validation, and Hardware-in-the-loop simulation and validation.

In Years 2 & 3, the team will work on implementing the proposed changes on the stock 2013 Malibu and achieving a production grade PHEV.

FUNCTIONAL PRINTING WITH INKJET

Research Assistants: Will Boley and Chenchao Shou

Priincipal Investigator: Professor George Chiu

ABSTRACT

With the increase in applications of inkjet functional printing in recent years, the higher throughput associated with multiple nozzle arrays is necessary for large scale production. This provides the motivation for designing print masks that can maximize the quality of inkjet printed devices rather than minimize the number of print artifacts detectable by the human eye. The current study builds upon earlier work by introducing a new drop coalescence cost function to avoid coalescence and by measuring the performance of print masks resulting from the Print Mask Direct Binary Search via image quality metrics including image mottle, raggedness, and fill. Four print masks of differing expected quality are generated and used to print rectangular solid fills. Overall, the results from the printed samples are in agreement with the expected performance of each of the four print masks. These print masks are also used to print a four-probe pattern for electrical characterization. The resulting performance of the resulting sheet resistances is also in agreement with the predicted performance. Additionally, the sheet resistances from avoiding coalescence are compared to that of promoting Coalescence.

IMPROVEMENTS TO MODELS OF FLEXIBLE POLYURETHANE FOAM TO ENHANCE PREDICTIONS OF SEAT-OCCUPANT DYNAMICS

Research Assistant: Yousof Azizi

Principal Investigators: Professors Patricia Davies and Anil Bajaj

ABSTRACT

Many advanced HVAC components, such as chilled beams, rely on vertical temperature gradients for effective operation. Moreover, accurate assessment of environmental-quality must be based on conditions in appropriate occupied zones and not on large-scale averages. Accordingly, it is appropriate to consider indoor environment models that accurately reflect such spatially varying features. However, the computational requirements for utilizing CFD calculations in this type of analysis are large. Furthermore, typical CFD analysis for buildings utilizes fixed boundary conditions and does not consider coupling to a buildinge envelop model. In the present case we develop a practical procedure for coupling a building envelope model to a CFD-based model for the indoor air environment. Reduced order models for the building envelope and indoor environment are generated from detailed models and then coupled to form an overall simulation model. The formulation retains spatially varying features such as water-vapor content and enables calculation of comfort metrics at various occupied locations. The coupled model allows for quantification of the effect of sensor placement on the energy consumption.

MODELING AND SYSTEM IDENTIFICATION OF A BEAM INTERACTING WITH NONLINEAR VISCOELASTIC MATERIALS

Research Assistant: Udbhau Bhattiprolu

Principal Investigators: Professors Anil Bajaj and Patricia Davies

Sponsor: Partially sponsored by the National Science Foundation

ABSTRACT

Muscle tissue, skin and several other biomaterials used in the biomedical industry as well as flexible polyurethane foams used for cushioning in the furniture and automotive industries, all exhibit nonlinear viscoelastic behavior. To design systems that incorporate these materials, it is important to be able to model their mechanical behavior and thus be able to predict the dynamic response of the systems. Because of the nonlinear and memory (viscoelastic) properties of the materials, this is a challenging research area. The results and understanding developed here for a pinned-pinned beam on a foam foundation interacting with flexible polyurethane foams should be applicable to more complicated systems incorporating many of the materials mentioned above. The approach to modeling is to start with linear models of the foam, gradually increase the complexity by incorporating, in stages, elastic nonlinearity and then viscoelasticity. Unlike most of the studies reported in the literature, the foam foundation was assumed to react only in compression to encapsulate a wider range of problems seen in practical applications. Thus the model has to be equipped with the ability to predict the changing contact length – this makes the modeling and analysis much more challenging. An experiment is being designed to verify the applicability of the model and initially, harmonic excitation of the system is being studied. The model will then be used to determine the geometry and materials in the experimental system and the excitation requirements. In the near future, a system identification procedure will be developed based on the model structure and the types of excitation that are feasible in the experiment. The procedure will be applied to the experimental system and the range of the applicability of the resulting model will be explored. The system identification approach and simulations of various model structures will be used to extend the capability of the beam-foam system model to predict responses observed in experiments under a wider variety of loading conditions and excitation levels.

MODELLING THE DYNAMIC BEHAVIOUR OF CONFOR AND POLYURETHANE FOAM

Research Assistant: Vaidyanadan Sundaram

Princiapal Investigators: Professors Anil Bajaj and Patricia Davies

Sponsor: Partially sponsored by the National Science Foundation

ABSTRACT

Foams find a wide range of applications from automobile seats, hospital beds etc. used for comfort and for shock isolation in electronic equipments, helmets etc. Optimal use of foam in these areas requires understanding of their behaviour and models that predict the responses of system incorporating them. The goal of this research is to develop models to predict the response of systems incorporating these foams to impulsive and harmonic excitation. The first step was to perform a quasi-static compression test to identify a foam model and then investigate how that model performs in predicting the response of a foam-mass system subjected to dynamic excitation. This approach builds on results from a large body of previous research but there may be fast dynamic behaviour present in the impulsive responses that was not excited in the compression tests, and therefore not modelled. In the next step, a system identification technique is developed using the responses to impulse and harmonic excitation of CONFOR and polyurethane foams around various static settling points of the foam-mass system. The system, a foam block with a mass on top, is modelled as a linear single-degree-of-freedom system and the simplified linear model parameters vary with the static equilibrium position. Stiffness and damping estimates from each static equilibrium position are used to estimate the parameters of a global nonlinear system model that could be then used to predict both the static equilibria and the impulse or harmonic responses about them. The next steps in the project are to complete a series of impulse and harmonic excitation tests on various foam-mass systems and to compare the performance of the models developed from these two different excitations.

ACOUSTICS
WING

METAMATERIALS FOR LOW FREQUENCY BARRIER APPLICATIONS

Research Assistant: Srinivas Varanasi

Principal Investigators: Professors Thomas Siegmund, J. Stuart Bolton and Raymond J. Cipra

ABSTRACT

The design of a sound barrier material that effectively block low frequency noise while minimizing the barrier weight requires an unconventional approach. Research in acoustical metamaterials is primarily directed towards this goal. Locally Resonant Sonic Materials (LRSMs) achieve this objective, although their barrier performance is often limited to narrow frequency bands and in many cases they suffer from a weight penalty since LRSMs commonly require resonating masses to be embedded in a matrix of soft material. As an alternative embodiment, a planar cellular metamaterial which requires a contrast in mass and moduli of the cell wall and the interior of the unit cell has been developed. However, being able to achieve heterogeneity in a material is always a challenge from a manufacturing point-of-view. That concern is addressed by the use of Topologically Interlocked Materials (TIMs) which are materials made by a structured assembly of identical unit elements. These materials offer exciting opportunities to realize the heterogeneity that is primarily due to their segmented nature. That is, heterogeneity can be easily introduced by mixing unit elements having contrasting material properties or by using microstructured unit elements.

In this context, two characteristically different unit elements having two contrasting material phases, namely, a layered tetrahedron having alternating layers of the two material phases, and a chiral tetrahedron formed by the interlocking of two helicoid dissections were studied for their acoustical characteristics. The layered microstructure, in particular, was studied with its layers oriented parallel and perpendicular to the incident sound field. The choice of chiral microstructure was inspired by the interesting twist-stretch properties demonstrated by double helicoid structures in nature such as DNA. To study these systems, a standing wave impedance tube simulation setup was developed using the Abaqus FE code to study the unit elements' acoustical characteristics. Modal analysis was also performed to identify the unit elements' eigenmodes and to relate them to their predicted sound transmission behavior. The predicted characteristics of the microstructured unit elements were compared with those of the homogenized unit element. The homogenized unit element for all the microstructures is the same since the two material phases in all the considered microstructures occupy the same volume proportions. For both element geometries, distinct properties were observed that can be attributed to microstructuring in general, when compared with those of the homogenous unit element. In this study, the influence of the surrounding units or the constraint effect on the unit elements was not accounted for. It was found that all the microstructures have almost the same density of eigenmodes but that not all eigenmodes were excited by the incident sound field. The chiral unit element and the layered unit with its layers oriented in parallel to the incident sound field have similar characteristics. Distinctively, a twist mode was observed in the chiral and the layered element with its layers oriented perpendicular to the incident sound field. But, that mode was excited only in the chiral unit element by the incident sound field. The excitation of the twist mode has important implications in designing novel sound absorbing materials. As a further study, it is proposed to investigate the effect of surrounding elements on the predicted characteristics to enable the design of novel topologically interlocked sound barrier materials.

ACOUSTICAL INTERROGATION OF HIGH-IMPEDANCE MATERIALS

Research Assistant: Andrew Jessop

Principal Investigator: Professor J. Stuart Bolton

Sponsor: Office of Naval Research

ABSTRACT

Vibration of an explosive or chemically unstable material can cause out-gassing that can potentially enhance the detection of dangerous materials by non-contact methods. Characteristic vibration signatures may also reveal the presence of a dangerous material. Acoustical waves in air can be use to initiate vibration propagation in such materials while retaining a standoff distance greater than traditional means of vibration actuation. Therefore, acoustical interrogation of materials would provide a useful component to a vibration-dependent analysis system.

The large discrepancy in density and stiffness between air and most solid materials causes an impedance mismatch that makes it difficult to transmit energy through the fluid-solid interface *via* acoustical waves. However, the formation of an evanescent (i.e., partially non-propagating) pressure distribution on a solid surface has shown the possibility of increased energy transfer to the solid. Acoustical waves generated by multipole sources could generate the required distributions. An advanced model for the propagation of energy from evanescent waves through the fluid-solid interface will yield possible source geometries required to transmit excitation into the material. Experimental measurements of the sound propagation into a solid caused by a monopole-like source (which would generate evanescent waves at small standoff distances) will be performed to verify the theoretical predictions. In future work, acoustical arrays capable of generating these wave patterns on the solid surface will be developed.

Different wave types in the solid have different dispersion properties, and appropriate pairing of these wave types to the correct frequency and desired pattern of vibration excitation in the solid would optimize the transmission of acoustical energy into the solid. A further understanding of the dispersion properties of the solid material in question, including both the simulated explosive materials and any structural materials that may be involved, will allow for a particular type of wave to be targeted for excitation and generate specific vibration patterns in the solid. Predictions of the dispersion properties of coupled systems of materials will reveal excitation avenues in more complicated systems. Experimental quantification of wave types in structural materials is currently being performed; further testing of surrogate explosive materials will be undertaken shortly. Once the individual materials have been tested, measurements of coupled systems will be performed to verify the predictions made from the single-component measurements.

USE OF CFD TO CALCULATE THE IMPEDANCE OF TAPERED HOLE MICROPERFORATED PANELS

Research Assistant: Nicholas Nakjoo Kim

Principal Investigator: Professor J. Stuart Bolton

ABSTRACT

Microperforated panels having various different hole shapes have been developed as sound absorbing materials. However, the classical Maa theory for microperforated materials was formulated only for sharp-edged, cylindrical holes. *Ad hoc* corrections to Maa's theory have been suggested to allow for hole shape variation, but these theories still show discrepancies compared to measurements, particularly at low frequencies. Previously, for sharp-edged and round-edged hole, microperforated panels, it has been shown that the dynamic flow resistance and reactance of a microperforated panel can be calculated using simple CFD models, and that the resistive end correction, in particular, depends on frequency and geometrical parameters (i.e., the aspect ratio of the hole, and the roundness). In this paper, a convenient equation for the dynamic flow resistance end correction has been formulated for microperforated panels with tapered holes by using CFD models to study the effect of various hole geometries. From the results of the CFD computations, it has been possible to formulate a resistive end correction for the tapered hole case that is a function of inlet and outlet diameter and which limits correctly to the cylindrical hole case.

PREDICTION OF OUTPUT SOUND FROM MECHANICAL AND COMBUSTION NOISE SOURCES IN A DIESEL ENGINE

Research Assistants: Michael Hayward and Brandon Sobecki

Principal Investigators: Professors J. Stuart Bolton and Patricia Davies

Sponsor: Cummins Inc.

ABSTRACT

The development of quieter engines is a major driving force in the design and manufacture of competitive engines. Determination of the relative strength and nature of dominant noise sources within an engine is an important step in understanding and reducing undesirable noise characteristics of a particular engine. The investigation presented here is part of an ongoing project to develop a method to accurately determine characteristics of noise sources within a diesel engine. Previous work has included the implementation of a time-domain windowing approach to accurately determine the characteristics of combustion noise emitted, and allow physical measurements of combustion noise to be taken in reverberant testing environments. Currently, work is being undertaken to determine not only the dominance of particular mechanical noise sources (such as the valve train, gear train, etc.), but also the specific characteristics of noise emanating from each component.

In order to determine the nature and dominance of independent mechanical noise sources within the engine, a method to decorrelate the data recorded by input near-field measurements (microphones, accelerometers and cylinder pressure transducers) was required. A method was developed utilizing singular value decomposition (SVD) to decorrelate the data resulting in the calculation of 'virtual sources' – independent noise sources within the engine closely related to mechanical components. By deconstructing the matrix methods used in the SVD procedure, the contribution of each virtual source to the power spectral densities of each near-field microphone and accelerometer could be calculated, and a technique to plot this information was developed. These 'singular value contribution plots' are a visual representation of the contribution and dominance of each virtual source to a particular measured "input": i.e., the accelerometer and near-field microphone signals. These plots display the nature and relative strength of independent information recorded by each input, and combined with knowledge of the locations of these input measurements, inferences can be made about which mechanical component within the engine is emitting the most significant amounts of noise within a particular frequency range.

The relation between the transfer paths that connect the input measurements and the output measurements (far field microphones 1 meter away from the engine) is currently being investigated to determine the nature of noise in the far field. A technique has been developed to estimate the transfer path between any given input and output combination. The effectiveness of this method is currently being studied through the generation of output simulations and coherence calculations between the inputs and outputs. Building upon the methods described here, the desired outcome is to be able to accurately predict the contribution of each of the major noise-emitting mechanical components within the engine to the far field noise, and to be able to simulate and predict far field noise purely on the basis of input data recordings.

FAN NOISE CONTROL

Research Assistant: Seungkyu Lee

Principal Investigator: Professor J. Stuart Bolton

ABSTRACT

The noise of computers, home appliances and various other electronic devices is mainly caused by axial fans used to cool the devices. Considerable amount of the axial fan noise is created by the shearing of fluid in the tip clearance regions, which is a narrow space between the blade tips and the housing of the fan. In gas turbine designs, efforts have been made to reduce the tip clearance noise by directly treating the turbine housings in the tip region to create a finite flow resistance around the fan circumference. The finite level of flow resistance created by the fan housing treatment is expected to reduce turbulence levels in the tip region and so reduce the tip clearance noise. Following this suggestion, experimental research was conducted to reduce the small axial fan noise by focusing on reducing the tip clearance noise. The housing of an 80 mm computer cooling fan was micro perforated to cause the fan scroll to have a finite flow resistance between in the radial direction. The acoustical measurement and performance tests of this micro-perforated fan have been made in the anechoic chamber and the results were compared with the regular fan without micro-perforations. The acoustical measurements are being made using the so-called INCE plenum, which allows the fan to operate anywhere on its performance curve. The fan noise is being quantified on the basis of total sound power (measured using a hemispherical array of microphones), and the contributions of tonal and broadband components are being identified on the basis of cumulative spectra. Tradeoffs between noise reductions and the performance of the fans will be suggested based on the results of this study. Furthermore, it is expected that the results of the work can provide guidelines for design treatments that can be integrated into typical fan designs to create a new generation of reduced-noise cooling fans.

SOUND FIELD VISUALIZATION AND PREDICTION USING NON-COLLOCATED EQUIVALENT SOURCES METHOD

Research Assistant: Yangfan Liu

Principal Investigator: Professor J. Stuart Bolton

ABSTRACT

The focus of the current work is on improving techniques for acoustical source visualization: i.e. nearfield acoustical holography. In the past, the main work in this area has focused on so-called non-parametric methods in which the sound field is expressed in terms of basis functions that are orthogonal in particular measurement geometries (e.g., planar or cylindrical). More recently, there has been increasing interest in equivalent source methods, in which the actual source is "replaced" by a collection of simple sources. This approach is relatively simple to implement and has shown the possibility of improved performance by comparison with more classical methods. The equivalent sources methods developed to this point can generally be classified into two categories: one in which a relatively large number of lower order sources are fixed in different locations, and one in which a single, high order (i.e., multipole) source is fixed at one location. The present work started with the construction of a model in the latter category, but the individual multipole components were then allowed to be spatially separated, i.e., to be non-collocated, and their locations were determined by using an optimization procedure based on the measured sound field data. When the sources are allowed to move in this way, the source estimation problem becomes nonlinear: however, the problem can be separated into linear and nonlinear components, which simplifies the implementation. To test this approach, experiments were conducted using a small loudspeaker cabinet; measurements were made using a cubical array of microphones around the loudspeaker. It was found that by allowing the multipole components to move, the sound field representation in both the near and far fields was much improved, particularly at high frequencies, when compared to the collocated source case (even when the collocated source was allowed to move as a function of frequency).

THE PREDICTION OF SOUND FIELDS INTO A HARD-BACKED RIGID POROUS GROUND

Research Assistant: Hongdan Tao

Principal Investigator: Professor Kai Ming Li

ABSTRACT

The current study focuses on the problem of predicting sounds fields into a hard-backed rigid porous ground due to a monopole source located above ground. By means of a double saddle point analysis supplemented by the pole subtraction method, an asymptotic solution for the sound penetration into the impedance-backed ground will be derived. Two components are included in the asymptotic solution. They accounts for two acoustical paths, which is the path from the air transmission into the receiver, and the path reflected from the bottom layer to the receiver respectively. The results predicted by asymptotic solutions are in good agreement with those by other numerical schemes like direct numerical integration and fast field program, which give more accurate results but computationally intensively. It is shown that the solution provides means for rapid and accurate computations for sound fields below hard-backed rigid porous ground. Further analysis will be extended to predict the sound fields into a 2-layer sound absorption material, which can be any two layers, and then applied to evaluate the acoustical properties of sound absorption materials.

SOUND FIELD OF A FAST MOVING SOURCE IN A HORIZONTALLY STRATIFIED MEDIUM

Research Assistant: Bao Tong

Principal Investigator: Professor Kai Ming Li

ABSTRACT

A continuous source motion model is developed for computing the sound fields of a uniformly moving monopole point source in a horizontally stratified atmosphere. The numerical model is based on the Lorentz transform and a 2D Fast Field Program (FFP) formulation. The 2D kernel functions are obtained via sampling along the radial direction, then interpolation of the kernel function at Cartesian grid points. This method is applicable when the kernel function is axisymmetric (e.g., in the absence of wind). An inverse transformation is applied to map the spatial Lorentz frame quantities into a sound field time history in the stationary observer frame.

The proposed numerical model can be used to describe an aircraft under cruising conditions. The continuous motion of the aircraft is embedded in the Lorentz transform definition. Atmospheric refraction and ground effects are considered in the model as well. Alternative kernel function sampling techniques involving asymptotic approximations along one dimension of the integral transform is explored. Efficient sampling of the 2D kernel function becomes essential due to the costly matrix inversion operations required in the selected layered atmosphere representation.

LITHIUM-ION BATTERY ELECTRODES MONITORING USING ACOUSTICS AND ULTRASONIC TESTS

Research Assistant: Huan Pham

Principal Investigator: Professor Douglas Adams

Sponsor: Sandia National Laboratories – Critical Skills Master's Program

ABSTRACT

With their superior advantages of high capacity and low percentage of self-discharge, lithium-ion batteries, most commonly used use as power sources for hand-held electronic devices, have become the most popular choice for power storage in electric vehicles. Due to the increased long term use lithium-ion in vehicle applications, manufactures are pursuing methodologies to increase the reliability and reduce the cost of producing their batteries. Methods are now being developed to monitor the health of lithium-ion batteries throughout their life cycle. This research project will be focused on utilizing non-destructive ultrasonic testing methods to monitor changes in the physical properties of the lithium-ion battery electrodes which dictate the states of charge (SOC) and states of health (SOH) of the battery cell. Inside a lithium-ion battery cell, lithium ions travel from cathodes to anodes during charge and reverse during discharge; therefore, the thicknesses of the electrodes inside a cell are different at different SOC. In a previous study, a model was developed to characterize the SOC of a lithium-ion battery cell by measuring the amplitude and phase of the kinematic response as a function of excitation frequency at different SOC of the battery cell and at different times in the life of the cell. Measurements of the thickness of the electrodes can also be obtained by using ultrasonic scanning the cell to measure the time it takes for an acoustic wave to travel through the electrodes materials at different SOC of the cell. The changes in the thickness of the electrodes at different SOC will help to determine the densities, moduli and stiffness of the electrodes which relate to the SOH of the battery. This technology is being patented by Purdue University.

DYNAMIC COUPLING IN LARGE WIND TURBINE GEARBOX/DYNAMOMETER SYSTEM

Research Assistant: Ray Bond

Principal Investigator: Professor Douglas E. Adams

ABSTRACT

The National Renewable Energy Laboratory's National Wind Technology Center hosts a 1.5 Mega-Watt wind turbine dynamometer as part of the Gearbox Reliability Collaborative, a government and industry effort to examine the reliability issues in current wind turbine gearbox and generator systems. The intent of the dynamometer is to test the turbine gearboxes in simulated worst-case real world conditions to predict reliability problems before the turbines are put into the field, where maintenance is very costly. Unfortunately, despite efforts to simulate operational conditions, gearboxes that appear reliable in the dynamometer testing often fail prematurely when placed in the field. This study seeks to evaluate the dynamic coupling between the gearbox and the dynamometer test system, and determine whether this dynamic coupling could be affecting gearbox reliability estimates. Modal impact tests were performed on the gearbox-dynamometer test system, the structural dynamics were evaluated, and significant coupling between the in-nacelle system and the test fixture was found. Frequency response functions for torsional and translational degrees of freedom showed that a lightly damped resonance of the system corresponds to an operational speed of the gearbox, and operating deflection shapes showed that these modes can introduce a dynamic misalignment of the gearbox high speed shaft. Collaboration with other researchers studying this problem have revealed that these types of misalignment conditions can lead to high internal loads within the gearbox, which likely contribute to higher than anticipated wear.

COST BENEFIT OF A WIND TURBINE BLADE MONITORING SYSTEM

Research Assistant: Natalie Barrett

Principal Investigators: Professor Douglas Adams, Professor Karen Marais

Sponsor: Sandia National Laboratories

ABSTRACT

The U. S. Department of Energy is working towards a 20% Wind Scenario where wind energy will provide 20% of the Unites States' electricity needs by 2030 [1]. Offshore wind energy is being investigated since the National Renewable Laboratories 2010 assessment of offshore wind in the US showed that 4150 GW maximum form wind turbines is available [2]. However, offshore wind operations and maintenance cost estimates are high and need to be lowered for offshore wind to be a viable energy alternative. Sandia National Laboratories is investigating the potential cost savings provided by a blade monitoring system for offshore wind turbine blades. As part of this project, Purdue University has developed cost models for operations and maintenance processes for offshore wind plants using Markov chain Monte Carlo methods. A Weibull distribution with no wake effects is used to assess the annual energy production and a Markov chain is used to deteriorate the blade using four blade states that go from new to failed. The model operates using random numbers to represent each of the 8760 hours for each of the 20 years simulated. The defects currently being analyzed are shear web disband, mass imbalance and aerodynamic imbalance. The results of the preliminary cost model indicate that these blade monitoring strategies will produce a cost benefit; however, further analysis is being conducted to determine the magnitude of these savings. Future work will involve refining the cost model by obtaining more accurate operations and maintenance costs, varying the wind profile and incorporating maintenance decisions into the model to optimize costs.

[1] S. Lindenberg, *20% Wind Energy By 2030: Increasing Wind Energyżs Contribution to US Electricity Supply*: DIANE Publishing, 2009.

[2] M. N. Schwartz, *Assessment of Offshore Wind Energy Resources for the United States*: National Renewable Energy Laboratory, 2010.

LITHIUM-ION BATTERY ELECTRODE INSPECTION USING FLASH THERMOGRAPHY

Research Assistant: Nathan Sharp

Principal Investigators: Professors Doug Adams, James Caruthers, Peter O'Regan, Anand David and Mark Suchomel

ABSTRACT

Pulse thermography was used to experimentally evaluate lithium-ion battery electrode quality. The camera data was processed to improve high frequency capabilities. Lab manufactured electrodes with gross defects, thickness variation, and composition variation all were detectable with this method. Thickness variation was shown to have a one to one ratio percent change in thickness to percent change in thermal response and a thickness difference as low as 4 percent was detectable with the method. Lab electrodes were compared with commercial electrodes with comparable results. Both types of electrodes showed a significant thickness oscillation that has not previously been reported regarding lithium-ion battery electrodes. Pulse thermography is shown to be an effective method in determining the homogeneity of battery electrodes and could easily be implemented into a manufacturing environment without significantly delaying the process.

SOUND QUALITY OF SUPERSONIC AIRCRAFT NOISE AS HEARD INDOORS AND IT'S IMPACT ON ANNOYANCE

Research Assistant: Clothilde Giacomoni

Principal Investigator: Professor Patricia Davies

Sponsor: FAA/NASA/Transport Canada PARTNER Center of Excellence on Noise and Emissions Mitigation

ABSTRACT

Manufacturers of business jets have proposed designing and building a new generation of supersonic jets that produce shaped sonic booms of lower peak amplitude than booms created by the previous generation of supersonic aircraft. To determine if these "low" booms are less intrusive and the noise exposure is more acceptable to communities, new testing to evaluate people's responses must occur. Because of the limitations on commercial supersonic flight overland in the US, and the lack of precise control of noise exposure in those settings, these studies must initially be done in a laboratory setting. To guide aircraft design, objective measures that predict human response to modified sonic boom waveforms and other impulsive sounds are needed. In previous research, it was also found that, for outdoor booms, startle and annoyance were highly correlated. Loudness alone did not fully explain annoyance nor startle judgments, and so recent research was focused on startle and annoyance model development. Models that include maximum loudness, rise time and sharpness metrics predict responses well. The next research phase is focused on understanding how people will react to booms when heard inside. House type and the indoor acoustic environment modify the outdoor booms and this must be considered when predicting the sounds and determining the resulting annoyance. Data from NASA's Dryden 2007 low sonic boom study is being analyzed to help in the development of realistic simulations of the sonic booms heard indoors. The importance of binaural (spatial) effects is being explored to determine whether the simulations will need to also include use of head related transfer functions to be realistic. These simulations will be used to generate stimuli for subjective tests that will be conducted to explore how context, reverberation, house insulation/construction, spatial effects and boom-type affect annoyance judgments, and how the response to sonic booms heard indoors differs from that of sonic booms heard outdoors. From this knowledge it may be possible to modify the annoyance model developed in previous research to predict responses to low booms heard both indoors and outdoors.

ASSESSSMENT OF NOISE METRICS FOR APPLICATION TO LARGE CIVIL TILT ROTOR FLIGHT OPERATIONS

Research Assistant: Andrew McMullen

Principal Investigator: Professor Patricia Davies

Sponsor: NASA

ABSTRACT

Metrics that relate noise with the human responses created by that noise are important to the design of mechanical devices and systems. These metrics can be used to assess the noise to determine negative attributes and allow designers to make adjustments accordingly. Noise metrics can also be used together with aircraft noise source and propagation models to predict noise around airports and how that noise affects nearby communities. With these evaluation techniques, airports can restructure aircraft activity so that the noise effect on the surrounding community is decreased. The current metrics used were constructed using fixed wing aircraft noise, and may not be accurate when applied to noise generated by rotorcraft, such as the proposed large civil tilt rotorcraft (LCTR). The sounds of concern are generated during rotorcraft takeoff, level flight, and descent operations. Rotorcraft noise characteristics such as loudness, tonalness, and fluctuation and how they relate to annoyance are of concern. Additionally, the current metrics may under-predict the effect of very low frequency components in aircraft noise. A series of psychoacoustic tests will be performed at Purdue in the Herrick Labs Sound Quality Lab in order to characterize LCTR and other rotorcraft noise, and to understand how attributes of the noise relate to annoyance. Similar tests will be performed at the NASA Langley Exterior Effects Room in order to analyze the effect of low frequency sound on annoyance. The second set of tests being performed at NASA is necessary due to the limitation of the headphones used in the Purdue tests as well as the ability of the loudspeakers used in the NASA tests to produce tactile effects. The outcomes of this research will be used in the further development and implementation of LCTRs.

ISSUES IN THE DEVELOPMENT OF A SURVEY SIMULATION TOOL TO EXPLORE ROBUST ESTIMATION OF MODELS OF ANNOYANCE DUE TO AIRCRAFT NOISE

Research Assistant: Kevin Foertsch

Principal Investigator: Professor Patricia Davies

Sponsor: FAA/NASA/Transport Canada PARTNER Center of Excellence

ABSTRACT

Annoyance to aircraft noise is currently predicted by using airport operations data and sound propagation software to predict a noise metric called Day-Night Average Sound Level (DNL). This metric is based on an average A-weighted sound pressure level with a 10 dB penalty for events occurring at night. People argue that other characteristics of the noise exposure are important and should be included in annoyance models. Researchers have investigated the formation of annoyance models incorporating the number of aircraft events occurring over a time period, a variable known as the number-of-events. The models described in the literature are typically linear regression models which contain two predictor variables, a sound level term (e.g. A-weighted maximum sound level or Effective Perceived Noise Level) and a number-of-events term (e.g. total number-of-events or number-of-events exceeding a certain sound level). The relative importance of the predictor variables differs among various proposed models. As the number-of-events heard by people in a community increases, the average level of the noise to which a person is exposed also goes up (the variables co-vary), and so a challenge in the development of these models is to find a variety of types of exposures within populations around airports so that the contribution of each variable can be identified with precision. A question that arises is: can we use data from previously conducted surveys in a meta-analysis to determine if number-of-events explains any more of the response variance than is explained when using measures of average sound level alone? Is there sufficient data currently available that could be used to produce robust estimates of model parameters? Data sets from several noise surveys have been analyzed individually and in combination to determine the quality of parameter estimates that can be derived. A reason for combining multiple datasets is to test a hypothesis that co-variation among the independent variable data can be reduced, improving the estimation of annoyance models. Co-variation among noise survey independent variable data can also be affected through careful control of the sample design of a survey. A survey simulation tool capable of simulating different sample designs has been developed to examine the effect of sampling strategy on annoyance model estimation. These endeavors are focused on the improvement of aircraft noise annoyance models.

WEST
WING

INVERSE BUILDING MODELING TO ENABLE CONTROL AND RETROFIT ANALYSIS

Research Assistant: Jie Cai

Principal Investigator: Professor James Braun

ABSTRACT

There exist several methods for building modeling, two of which are forward modeling and inverse modeling. In forward modeling, one can establish a model based on the physical properties of the building. In order to do this, a detailed description of the building envelope needs to be provided, which usually consumes a considerably big amount of time and efforts. Inverse modeling, however, can be applied to avoid those efforts, especially in the case where a detailed building description is not available. The inverse model we are investigating applies a gray-box method, which has a better performance and needs less training data compared to the pure black-box method. The model is based on a simplified thermal network, where the values of resistances and capacitances are going to be obtained during the training process. This inverse modeling approach has been tested for an existing multi-zone building located at Philadelphia using both simulation data and field data as the training data. In the near future, it will be coupled with DX system model to enable control strategy development and retrofit analysis.

REDUCED-ORDER ENVELOPE AND INDOOR AIR MODELING FOR ASSESSING OPTIMAL SENSOR PLACEMENT, CONTROLS, AND COMFORT CONDITIONS

Research Assistant: Donghun Kim

Principal Investigator: Professor James E. Braun

ABSTRACT

Many advanced HVAC components, such as chilled beams, rely on vertical temperature gradients for effective operation. Moreover, accurate assessment of environmental-quality must be based on conditions in appropriate occupied zones and not on large-scale averages. Accordingly, it is appropriate to consider indoor environment models that accurately reflect such spatially varying features. However, the computational requirements for utilizing CFD calculations in this type of analysis are large. Furthermore, typical CFD analysis for buildings utilizes fixed boundary conditions and does not consider coupling to a buildinge envelop model. In the present case we develop a practical procedure for coupling a building envelope model to a CFD-based model for the indoor air environment. Reduced order models for the building envelope and indoor environment are generated from detailed models and then coupled to form an overall simulation model. The formulation retains spatially varying features such as water-vapor content and enables calculation of comfort metrics at various occupied locations. The coupled model allows for quantification of the effect of sensor placement on the energy consumption.

ASSESSMENT OF ALTERNATIVE APPROACHES FOR MODEL PREDICTIVE CONTROL IN BUILDINGS

Research Assistants: Vamsi Putta and Guangwei Zhu

Principal Investigators: Professors Jim Braun, Jianghai Hu

ABSTRACT

Simulation based performance assessments of a few promising control strategies for building heating, ventilation and air-conditioning (HVAC) system were performed. Using a Purdue Living Lab single zone model as a case study, optimal control strategies were generated using different algorithms. Model predictive based approaches were considered in optimal strategy generation due to the flexibility in handling weather conditions and varying time-of-day prices. A conventional control profile was also generated using a fixed set-point schedule. The resulting trajectories were evaluated in terms of energy costs and occupant thermal discomfort. Results indicate potential savings in both energy costs and lower occupant thermal discomfort are possible using model predictive control strategies when compared to conventional control. Additionally, the benefits and implementation issues of each algorithm considered are highlighted.

DEEP RETROFIT SOLUTIONS FOR BUILDING ENVELOPE AND HVAC INTEGRATION

Research Assistants: Janghyun Kim, Bonggil Jeon, Zhidan Zhao

Principal Investigators: Professors W. Travis Horton, James E. Braun, and Thanos Tzempelikos

Sponsor: DOE – Energy Efficient Buildings Hub

ABSTRACT

The goal of this project is to develop guidelines for highly integrated retrofit solutions that are cost effective and can achieve 20-50% reductions in energy use for buildings relevant to the EEB Hub region. An initial deep retrofit case study is being considered for an existing building at the Navy Ship Yard in Philadelphia, PA. TRNSYS simulation results are being utilized to analyze different retrofit options for the envelope, lighting, ventilation, controls, and HVAC space conditioning systems in order to determine the most cost effective solutions for target energy savings. Specifically, an original baseline model and an improved baseline model have been developed in TRNSYS. The original baseline model was developed based on actual building information. It represents significantly lower performance for the roof, walls, windows, and HVAC compared to current standards. The improved baseline includes improvements that are representative of current practice. The improved baseline will be used for comparison with a range of higher performance and integrated solutions as a means of determining an optimal retrofit scenario.

EXPERIMENTAL MEASUREMENTS OF LIGHTING ENERGY SAVINGS IN PERIMETER OFFICES WITH AUTOMATED SHADING AND LIGHTING CONTROLS

Research Assistants: Hui Shen, Ying-Chieh Chan

Principal Investigator: Professor Thanos Tzempelikos

Sponsors: DOE, Kawneer/Alcoa, Lutron Electronics, Viracon

ABSTRACT

Shading and lighting controls are used in perimeter zones of commercial buildings in order to prevent glare, increase daylight utilization and reduce lighting energy consumption. However, there is no standard or optimal procedure for controlling these devices in order to achieve optimal results. The effectiveness and practicability of different control algorithms under variable weather conditions needs to be further studied. This paper presents experimental results for different shading and lighting control combinations under different sky conditions and shading properties. The experiments were conducted in full-scale office test facilities with flexible shading and lighting control capabilities. Shading was controlled based on either work plane illuminance values or avoiding direct sunlight near the work plane surface. Two lighting systems were used in the experiments: suspended direct-indirect and recessed direct. Both on/off and continuous dimming controls were used for lighting and different sky conditions and shading properties were studied. The results include measured work plane illuminance at several points, light dimming levels, lighting energy usage, and glare calculations. The experiment results provide useful insights for realistic lighting energy savings and can be used to validate the accuracy of lighting simulation models in order to further optimize façade and daylighting design on an annual basis.

MODELING AND ANALYSIS OF OPEN-LOOP UNGLAZED TRANSPIRED COLLECTORS INTEGRATED WITH BIPV/T SYSTEMS

Research Assistant: Siwei Li

Principal Investigator: Professor Panagiota Karava

ABSTRACT

The objective of this study is to develop models for the design and control of open-loop unglazed transpired collectors integrated with Photovoltaic (PV) systems. Open-loop unglazed transpired collectors (UTC) consist of dark porous cladding through which outdoor air is drawn and heated by absorbed solar radiation for use in building HVAC systems (i.e. heat pumps, solar absorption cooling, or as direct supply air) to reduce energy consumption. UTC integrated with PV cells can generate electricity and heat. Recovering heat from the Photovoltaic panel cools the panel thereby improving its electricity generation efficiency. Despite its simple concept and cost effectiveness, the performance of this system is not well-understood, which is attributed to the complexity of the air flow, caused by the perforations and corrugation geometry, and its impact on convective heat transfer and consequently the overall thermal-electrical efficiency.

In this study, a full-scale test facility equipped with a solar simulator was used to measure the air velocity and temperature field for a corrugated transpired collector subjected to a wall jet. High resolution Computational Fluid Dynamics models were developed and validated with the experimental data with the results showing that the Renormalization Normal Group k-ε (RNG kε) and the Shear Stress Transport k-ω (SST k-ω) turbulence closure models have the best overall performance. The study explores innovative methods to integrate actively cooled PV systems into building facades in order to enhance their thermal management, optimize their overall performance and reduce uncertainty and risk. The study considers different cell technologies (e.g., Si, vs CIGS vs GaAs) and PV module materials and parameters such as wind speed, suction velocity, corrugation geometry, perforation dimensions, etc. in a comprehensive analysis and optimization framework. Preliminary results show that PV/T systems integrated with transpired collectors can reach 80% efficiency with existing PV technology.

INTEGRATED SOLAR ABSORPTION COOLING AND HEATING SYSTEM AT PURDUE UNIVERSITY

Research Assistants: Yi-Shu Kung and Donghao Xu

Principal Investigator: Professor Ming Qu

Sponsor: National Science Foundation

ABSTRACT

The traditional building cooling and heating system uses fossil fuel as primary energy. It takes the main responsibility for the carbon emission issues on our planet. In contrast, solar absorption cooling and heating system, which combines solar energy and absorption technology to provide building cooling and heating, becomes one of the promising alternatives. Purdue University at West Lafayette built a test facility to investigate the solar absorption cooling and heating system performance by using XCPC solar collectors to drive a double-effect absorption chiller. This poster will illustrate the two main parts: solar collectors and absorption chiller. The initial experimental data and expected result will also be showed in this poster.

PERFORMANCE ANALYSIS OF CCHP SYSTEM AT DATA CENTERS

Research Assistant: Donghao Xu

Principal Investigator: Professor Ming Qu

ABSTRACT

Currently, typical energy supply systems for buildings are separate cooling, heating and power (SCHP) systems. Because of low efficiency, large amounts of waste energy and negative environmental impacts, more interests have been aroused in combined cooling, heating and power (CCHP) systems. CCHP systems simultaneously generate power and thermal energy on site from a single fuel source. In this way, CCHP systems have a higher efficiency by taking use of waste energy and therefore have fewer impacts on environment and cost less to operate. Currently most of CCHP systems are applied to industrial area. The commercial applications only employ less than 12% CCHP systems in terms of capacity. It is because industrial process always demands relatively constant amount of power and thermal energy, which makes CCHP more applicable and practical while commercial buildings always have an occupancy-dominated and flexible load. Among all types of commercial buildings, data center is a good option for CCHP application for its relatively stable load profile. But there are few studies based on operational data. To give a real picture of CCHP systems' performance and its suitability for data centers, this paper studies a CCHP system at a data center in San Diego, California. The performance is evaluated in terms of energy, economics and environment by means of operational data analysis and modeling. Based on the results, recommendations are given in terms of operation and potential improvement of performance.

MODELING AND PREDICTIVE CONTROL STRATEGIES IN BUILDINGS WITH MIXED-MODE COOLING

Research Assistant: Jianjun Hu

Principle Investigator: Professor Panagiota Karava

ABSTRACT

Mixed-mode cooling refers to a hybrid approach for space conditioning, employing a combination of natural ventilation, driven by wind or thermal buoyancy forces, and mechanical systems. Effective implementation of this technology requires intelligent controls to minimize energy use and maintain occupant thermal comfort. Existing techniques for mixed-mode buildings are heuristic, i.e. based on fixed schedules that are not optimized for a particular building or climate and may lead to increased operating costs or occupant discomfort. The objective of this research is to develop a flexible modeling platform for design, “offline” and “on-line” model predictive (MPC) strategies of multi-zone mixed-mode buildings. The present study investigates offline MPC strategies for mixed-mode cooling in buildings with hybrid ventilation and presents an initial implementation framework for on-line anticipatory controls. A transient thermal model is developed in MATLAB based on thermal network representation for a high performance institutional building and it is validated with experimental data. Within the model, parameters are classified into design and control variables in order to investigate their impact on the effectiveness of mixed-mode cooling. Future cooling load, fan energy consumption, along with the operative temperature are predicted by the model are used to evaluate the overall cost associated with energy consumption and occupant thermal comfort. The GenOpt software is used with the particle swarm optimization (PSO) algorithm as optimizer. The results show that the cooling load on the following day can be significantly reduced or offset depending on the weather conditions, the design characteristics (e.g. thermal mass, window-to-wall ratio, orientation) and the comfort constraints of the studied building using the inputs obtained from the MPC controller. A generic and efficient identification approach for developing a reduced-order model along with on-line MPC strategies, considering the nonlinearity caused by the advection term, associated with the air exchange between zones, is in progress.

PARTICLE SWARM OPTIMIZATION ALGORITHM FOR ENERGY-EFFICIENT HOUSING

Research Assistant: Yeon Jin Bae

Principal Investigator: Professor W. Travis Horton

ABSTRACT

As energy consumption in the building sector across the United States continues to increase, energy modeling and simulation software is being incorporated more regularly into the building design process to simulate thermal loads and predict annual energy consumption. The greatest benefit of using numerical models lies in their potential ability to provide design engineers with feedback regarding how best to optimize these very complex systems. Energy simulation programs typically require hundreds of design parameters as input, which means they are not normally utilized at a very early design stage because many of the parameters are still uncertain. Since any given building project can include a large number of design parameters, and each parameter may have a large number of possible values, it is impossible, because of computational time limitations, to do exhaustive search in this design space to find the parameters that optimize the building design subject to various project objectives and constraints. Optimization methods must be employed to find the best combination of design variables to achieve the desired objectives. Past researchers have identified several applicable building energy optimization methods but the long computational processing time remains as a major challenge to the more widespread use of this technique by both researchers and practitioners. The objectives of this study are to perform an optimization of energy efficient residential construction, to utilize a particle swarm optimization algorithm and to use complex building energy simulation models in conjunction with real world cost data(material cost) across as many climates as possible to reach as reliable, and wide a solution space as can be achieved. The challenge and focus of the future work is to improve the performance of the optimization method, possibly through variable screening and algorithm hybridization, so that the user can have greater freedom in selecting variables and optimization goals, extracting more useful information from the simulation results, and acquire the desired results in a time effective manner. Cost modeling will be improved with regards to adding of replacement and maintenance costs as well as labor cost.

FREE COOLING APPLICATION FOR ENERGY SAVINGS AT PURDUE UNIVERSITY

Research Assistants: Bonggil Jeon, Rita Jaramillo

Principal Investigators: Professor Jim Braun, Professor Travis Horton

Sponsor: Purdue Physical Facilities

ABSTRACT

Purdue University Northwest Chiller Plant (NWCP) was originally designed to operate during the warm months of the year in conjunction with Wade Power Plant to meet the chilled water requirements for campus. As campus requirements have evolved there is desire to operate the plant through the entire year to help meet the chilled water demand. Currently, there are over 150 buildings in operation at West Lafayette campus, the largest of them requiring cooling throughout the year due to high heat gains from equipment, lighting, solar glazing and people. In consequence, the average cooling load during the colder months of the year (October to April) is above 5,000 tons. This project examines the possibility of operate NWCP during the colder periods in free cooling mode (cooling without operating the compressor of a centrifugal liquid chiller) to help meet campus chilled water demand while reducing the use of electric or steam-driven chillers at Wade Power Plant with the subsequent economic and environmental benefits. The free cooling method considered in this project is called indirect free cooling, which consists on installing a plate-frame heat exchanger piped in a parallel by-pass circuit with the chiller to allow the cooling tower water to cool the campus chilled water when conditions are appropriate. However, in order to make this feasible, some issues related to cold weather operation such as the potential for a large plume of condensation to descend from the cooling towers coating nearby streets and also the potential for water in the cooling towers to freeze, must be addressed. The results of this project, which represents a partnership between Academics and Physical Facilities at Purdue, include performance modeling, economic analysis and technical considerations to determine the feasibility of implementing free-cooling at NWCP.

INTEGRATED, ON-LINE PERFORMANCE MONITORING AND DIAGNOSTIC SYSTEM BASED ON VIRTUAL SENSORS FOR A PACKAGED ROOFTOP AIR CONDITIONER

Research Assistant: Woohyun Kim

Principal Investigator: Professor James Braun

ABSTRACT

The primary goal is to develop a complete implementation and demonstration of fault detection and diagnostic (FDD) system applied to a packaged rooftop unit (RTU) that incorporates integrated virtual sensors and fault impact evaluation. The specific activities of the project include: 1) develop, implement, and evaluate virtual sensors to be used for fault detection, diagnosis, and fault impact evaluation, 2) demonstrate a complete diagnostic implementation for RTU in the laboratory, 3) develop performance indices that can be used within a decision-support system to assess whether RTU service should be performed, and 4) perform a preliminary cost analysis to estimate the costs of an embedded diagnostic system for RTUs. A virtual sensor uses low-cost measurements and a simple model to estimate a quantity that would be expensive and/or difficult to measure directly. Laboratory tests will be performed to evaluate FDD performance and define reasonable thresholds for FDD system with multiple simultaneous faults to assess whether RTU service should be performed. The tests will also quantify the benefits of this technology with measurements of equipment performance (cooling capacity and power, etc.), and demonstrate implementation with low sensor costs. For the demonstration RTU, virtual sensors will be implemented for the compressor, electronic expansion valve, condenser and evaporator fan/motor combinations, heat exchangers, refrigerant charge, and economizer in order to provide the following diagnostic outputs: 1) loss of compressor performance, 2) low or high refrigerant charge, 3) fouled condenser or evaporator filter, 4) faulty expansion device or liquid-line restriction, and 5) economizer faults. In addition to developing the laboratory demonstration, some virtual sensors will be demonstrated using additional data obtained from other sources including United Technologies Research Center (UTRC) and Building 101 at the Navy Shipyard in Philadelphia, PA.

INVERSE MODELING OF VAPOR COMPRESSION EQUIPMENT TO ENABLE SIMULATION OF FAULT IMPACTS

Research Assistant: Howard Cheung

Principal Investigator: Professor James E. Braun

Sponsor: National Institute of Standards and Technology

ABSTRACT

The importance of faults in terms of their effects on energy efficiency for air conditioners and heat pumps is increasingly recognized. This has led to the development of fault detection and diagnostics (FDD) tools that are now available in the market. However, consumers currently have no way to compare products in terms of their ability to reliability detect and diagnose faults and to evaluate their economic benefits. Therefore, Purdue is developing a methodology for evaluating the performance of FDD algorithms that is based on the use of a comprehensive database of measurements for a range of different system types with different faults over a range of environmental conditions. In order to provide a wider range of possible fault levels and operating conditions, the data is used to train inverse models that can be used to generate a larger database. In order to reliably simulate faults and increase the reliability of the models to extrapolate beyond the tested conditions, semi-empirical (gray-box) component models are developed that are based on some physical principles and trained with component-level data. These models were combined into a cycle model and charge tuning was implemented to conduct simulations at various charge levels. Up to the present, 8 different cooling systems have been modeled and validated with experimental data under different faults such as non-standard charging, condenser and evaporator fouling, compressor valve leakage and liquid line restriction.

INTEGRATED, ON-LINE FAULT DETECTION AND DIAGNOSTICS SYSTEM FOR PACKAGED AIR CONDITIONER ECONOMIZERS

Research Assistant: Andrew Hjortland

Principal Investigator: Professor James Braun

ABSTRACT

When outside air is sufficiently cool and dry, outdoor air economizers can reduce building cooling energy by using outdoor air in place of mechanical cooling. Because of the potential for energy savings as well as improved indoor air quality, many energy codes and standards have incorporated economizers into designs. In order to ensure proper operation of outdoor air economizers, good controls, dampers, and maintenance are needed. This is especially true for packaged air conditioners due to the compactness of their design and high incidence of faults over the life of the system. A fault detection and diagnostics tool has been designed for a 4-ton packaged air conditioner with an integrated outdoor air economizer to provided on-line performance monitoring. Benefits include diagnosing control and damper problems as well as detecting temperature sensor faults.

Utilizing a rule-based fault detection method derived from system energy balances, fault-detection sensitivity and false-alarm rates are optimized using robust temperature sensor corrections. Air-side mixed-air temperature measurements in packaged air conditioners are often unreliable due to the nonuniform temperature and flow distribution in the undersized mixing box. In order to overcome this, a robust mixed-air correction model has been applied utilizing a single-point mixed air temperature measurement and the outdoor air damper control signal. A similar model has been applied to outdoor air temperature measurement due to return-air recirculation in the outdoor-air intake hood. The system also uses a virtual supply fan airflow and power sensor to detect and diagnose fan and fouling faults. Economizer fault diagnostics is performed by actively controlling the outdoor-air damper to provide system redundancy.

EXPERIMENTAL PERFORMANCE EVALUATION OF DUCTLESS HEAT PUMPS

Research Assistants: Simbarashe Nyika and Howard Cheung

Principal Investigator: Professor James Braun

Sponsor: Ecotope Inc.

ABSTRACT

Residential ductless heat pump (DHP) systems are becoming more common in the United States, having been popular in Asia for at least the last two decades. This project aims to characterize the performance of these systems under a wide range of operating conditions. The performance testing involves testing three DHP systems in sequence using the Herrick Laboratories' two ASHRAE standard psychrometric chambers over the course of 24 months. One of the three units has two indoor units and the tests cover both heating and cooling operation modes. Particular attention will be paid to heating mode operation given the extreme cold climates under which the units can be expected to operate in large parts of North America.

Currently, there is not much information on residential DHPs besides manufacturer's claims and mandatory single point rating data at standard conditions. This project will increase the information available to consumers as the data from the experiments will be used to generate performance maps and equipment models for use in building simulation programs. While software is available that can model the equipment performance at the more detailed hardware level, empirical data is important to capture the complex control routines that govern the operation of DHPs. Empirical models are also simpler, execute faster and don't require as much detailed knowledge of the equipment parameters to provide accurate results.

The equipment models generated will be simulated in different residential building types and climates across the United States to assess the energy savings DHPs can provide in both retrofit and new construction applications. The DHPs will be compared against conventional ducted split system heat pumps as well as combinations of ducted air conditioners with electric heating such as baseboards in the winter. Although only three distinct models will be tested, their empirical performance models can be treated as representative of the performance of similar units on the market. The models generated will also be generalized in nature to account for different nominal capacities so that they can be scaled up or down for bigger or smaller units.

MECHANISTIC MODELING OF A DUAL-UNIT DUCTLESS HEAT PUMP SYSTEM

Research Assistants: Howard Cheung and Simbarashe Nyika

Principal Investigator: Professor James E. Braun

Sponsor: Electric Power Research Institute

ABSTRACT

With increasing awareness of energy efficiency of buildings, the popularity of variable speed ductless heat pumps (DHP) with multiple indoor units increases in North America. These systems claim efficiency savings due to better part-load performance and smaller fan power consumption. However, in order to better understand the primary performance benefits and their sources, a mechanistic model is being developed and validated using experiments. The modeling and experiments are being carried out for a dual-unit DHP. Tests are being performed in the psychrometric chambers at Purdue with the data used to tune parameters of the model. To date, heating operation of the dual-unit ductless heat pump system has been modeled and validated with the experimental data. Future work includes extending the simulation to cooling operation and using the model to better understand the benefits of this technology.

ANNUAL PERFORMANCE COMPARISON OF VARIOUS RESIDENTIAL HEAT PUMPS

Research Assistants: Seth Holloway and Simbarashe Nyika

Principal Investigators: Professors James E. Braun and W. Travis Horton

Sponsor: Ingersoll Rand

ABSTRACT

The overall goal of the work was to assess the performance of four heat pump systems for a range of residential buildings in different climates. The heat pumps include single-speed, dual-capacity, and variable-speed compressors and fans. Performance indices include annual energy consumption, comfort characteristics, efficiency gains or losses, and overall performance. Simulation was employed to evaluate and compare the heat pump systems for different types and styles of homes that vary in efficiency, and in different U.S. climates. Building data and simulation models are utilized from [1].

Heat pump models were developed based on data obtained from equipment manufacturers. Ducted heat pump systems were implemented using the ASHRAE secondary toolkit direct expansion model [2] approach. The ductless variable-speed model was based on a recent paper by Cheung and Braun [3]. The paper describes a gray-box model based on experimental evaluation of a commercially available heat pump. The model is quasi-steady and was implemented TRNSYS simulation software and written in the FORTRAN programming language. A program built around TRNSYS was then used to study combinations of home type, location, and equipment type.

The results will demonstrate how the performance benefits of single and variable-speed equipment depend on home type and climate with the ultimate goal of identifying the most appropriate technology for different situations.

References

[1] Holloway, S.O., Nyika S., Horton, W.T., Braun, J.E. (2013). *Prototypical Building and HVAC Models for Assessing the Performance of Ducted and Ductless Residential Heat Pumps*. Proceeding of International Conference Clima 2013 - REHVA World Congress –International Conference on IAQVEC. Prague, Czech Republic.

[2] Brandemuehl, M.J., S. Gabel, and I. Andresen. (1993). *HVAC2 Toolkit: Algorithms and Subroutines for Secondary HVAC System Energy Calculations*. Atlanta, GA: ASHRAE.

[3] Cheung, Howard and Braun, James E., *Performance Characteristics and Mapping for a Variable-Speed Ductless Heat Pump* (2010). Publications of the Ray W. Herrick Laboratories. Paper 6. http://docs.lib.purdue.edu/herrick/6

OPTIMIZING REFRIGERANT DISTRIBUTION IN EVAPORATORS

Research Assistant: Christian K. Bach

Principal Investigators: Professors James E. Braun, Eckhard A. Groll, and W. Travis Horton

Sponsors: California Energy Commission and Department of Energy

ABSTRACT

With growing concerns about environmental impact and peak electrical demand, the expectations and minimum standards for equipment efficiency have been rising. At some point it becomes necessary to incorporate changes to the basic vapor compression cycle in order to achieve higher efficiencies. One such modification involves individual control of refrigerant flow within each circuit of the evaporator.

For a given overall superheat exiting the evaporator of a vapor compression cooling system, maximum capacity and efficiency are achieved when each of the refrigerant circuits within the evaporator has the same superheat. Non-uniform superheat of multi-circuit evaporators is caused by uneven airflow and/or refrigerant flow distribution, air side fouling and uneven air inlet temperature distribution.

It is believed that it is impractical to install conventional expansion valves on each circuit of the evaporator to control superheat because of costs. Therefore a hybrid method of control is proposed, involving fine-tuning of small balancing valves in each circuit along with using a primary expansion device to control the overall superheat.

Two parallel projects are being carried out on the topic of the optimization of refrigerant flow in evaporators. The first project, sponsored by the California Energy Commission, will use conventional electronic expansion valves as balancing valves. The performance benefit achievable by employing the hybrid control method was evaluated for the following applications: Residential 5-ton heat pump (HP) and 3-ton large room cooling system (LRCS). Additionally, experiments are conducted using a 4-ton rooftop AC-unit.

The results from both, the LRCS and the HP showed that maldistribution exists in typical applications. This is the case even if there is no visible frost built-up or fouling, due to design limitations and manufacturing imperfections. For those, so termed “clean coil” conditions, an average of 3.7% (2.3%) improvement of COP for the LRCS (HP) was achieved by using the hybrid control scheme compared to an EXV control scheme. For conditions were airside maldistribution was applied to simulate typical levels of frost built-up, a COP improvement of up to 30% (20%) was achieved for the LRCS (HP). Additionally to the described benefits in terms of COP, the system capacity increased in a similar fashion. It was found that, in general, the benefit of the hybrid control scheme increases with increasing initial maldistribution and available system capacity.

The second project, as a subproject of the cold climate heat pump project, sponsored by the Department of Energy, will employ new types of balancing valves. These valves are currently being developed in cooperation with Emerson Climate Technologies. Passive and active concepts were initially investigated. For reasons of cost and reliability, a solenoid type balancing valve was chosen as the final candidate. This valve is currently tested to fine-tune its dimensions for the application in the cold climate heat pump.

MODELING AND EXPERIMENTAL VALIDATION ON TWO LEVEL VAPOR INJECTED SCROLL COMPRESSOR

Research Assistant: Yuanpei Song

Principal Investigators: Professors Eckhard Groll, Jim Braun, W. Travis Horton

Sponsor: Department of Energy

ABSTRACT

Previous research indicated that a cycle using a scroll compressor with multiple vapor injection ports and economizing has the potential to significantly improve the energy efficiency and maintain high heating capacity for vapor compression heat pumps, particularly at low outdoor temperatures. In addition, it was shown that two-phase refrigerant injection performs slightly better than vapor injection with the same number of injection ports. However, considering the economic feasibility and the difficulty to control two-phase flow, only research on vapor injection has been performed as part of this study. To investigate the benefits of injection and the injection parameters including injection location, shape of injection port and its size, a model of the scroll compressor has been developed. This model includes such sub-models as the geometry model, thermodynamic model, leakage model and heat transfer model. Injection will be treated as a part of leakage into the compression chamber. Based on a tradeoff between cost and benefit, a dual-port compressor prototype was chosen for testing to provide the experimental results, which are used to validate the model predictions. A hot gas bypass load stand has been modified to accommodate the testing of the prototype compressor with two intermediate refrigerant injection ports. Refrigerant is injected as saturated vapor at two pressures within the compression process. The model predictions are compared to experimental results at the same conditions as those used during the testing. In addition, the overall performance of the compressor will be investigated.

EXPERIMENTAL ANALYSIS OF OIL FLOODING IN SCROLL COMPRESSORS

Research Assistant: Sugirdhalakshmi (Sugi) Ramaraj

Principal Investigators: Professors Eckhard Groll, Jim Braun and Travis Horton

Sponsor: Department of Energy

ABSTRACT

In the current state of affairs where energy consumption is increasing day by day, reducing power consumption and improving system efficiency has become a matter of concern in air conditioning and heat pump systems. One alternative technology for improving the efficiency of vapor compression refrigeration cycles is liquid flooded compression with regeneration. In this method, a significant amount of oil or non-volatile liquid is mixed into the stream of refrigerant vapor entering the compressor to absorb the heat of compression, which results in a quasi-isothermal compression process. Flooding also improves volumetric efficiency through improved sealing of compressor leakage gaps; and can significantly reduce thermodynamic losses associated with the desuperheating of refrigerant in the condenser. The addition of regeneration to a flooded compression cycle improves the system efficiency by reducing losses due to throttling of a two-phase refrigerant in the expansion device. Flooded compression with regeneration involves the use of scroll compressor technology, which is capable of handling relatively large mass flow rates of the flooding agent. To evaluate the benefits of oil flooding, a dual port R410A scroll compressor is flooded with POE oil on a hot-gas bypass load stand. For a range of different oil mass fractions, the result is a monotonic decrease in compressor discharge temperature as the oil injection rate is increased. When flooded with 50% oil by mass, the discharge temperature reduces to half of its value with no oil flooding. In addition, the refrigerant mass flow rate tends to increase as the oil injection rate is increased. At an evaporating temperature of -30°C and condensing temperature of 43.3°C, the compressor isentropic efficiency increases by 25% when the oil injection mass fraction is 0.5. The predictions of the system model indicate that the COP of a R410A flooded compression with regeneration cycle is 22% larger than non-flooded baseline cycle for climatic conditions such as experienced in Minnesota.

A HYBRID MODEL FOR INVESTIGATING TRANSIENT PARTICLE TRANSPORT IN ENCLOSED ENVIRONMENTS

Research Assistant: Chun Chen

Principal Investigator: Professor Qingyan (Yan) Chen

ABSTRACT

It is important to accurately model person-to-person particle transport in mechanical ventilation spaces to create and maintain a healthy indoor environment. The present study introduces a hybrid DES-Lagrangian and RANS-Eulerian model for simulating transient particle transport in enclosed environments; this hybrid model can ensure the accuracy and reduce the computing cost. Our study estimated two key time constants for the model that are important parameters for reducing the computing costs. The two time constants estimated were verified by airflow data from both an office and an aircraft cabin case. This study also conducted experiments in the first-class cabin of an MD-82 commercial airliner with heated manikins to validate the hybrid model. A pulse particle source was applied at the mouth of an index manikin to simulate a cough. The particle concentrations versus time were measured at the breathing zone of the other manikins. The trend of particle concentrations versus time predicted by the hybrid model agrees with the experimental data. Therefore, the proposed hybrid model can be used for investigating transient particle transport in enclosed environments.

VALIDATION OF FAST FLUID DYNAMICS FOR NATURAL VENTILATION SIMULATION

Research Assistant: Mingang Jin

Principal Investigator: Professor Qingyan (Yan) Chen

Sponsor: Department of Energy

ABSTRACT

Natural ventilation is a sustainable building technology that can provide good indoor environment and save energy. The application of natural ventilation in buildings requires a careful design in the early design phase, and simple and fast design tools are highly needed. As an intermediate approach between computational fluid dynamics (CFD) and multi-zone model, fast fluid dynamics (FFD) can provide informative airflow information with a speed of 15 times faster than the laminar CFD so that it could be a potential design tool for natural ventilation. This study thus evaluated the performance of FFD for simulating natural ventilation. The FFD was validated with two cases representing natural ventilation with different driven forces: (1) wind-driven natural ventilation through a scaled building model; (2) buoyancy-driven single-sided natural ventilation in an environmental chamber with a large opening. FFD was further tested for evaluating its ability of predicting impact of wind direction and surroundings on natural ventilation. From comparing the results predicted by FFD and the experimental data, this study found that the FFD was capable of predicting main air flow feature and ventilation rate with reasonable accuracy for the wind-driven or buoyancy-driven natural ventilation in buildings. FFD simulation can reflect the influence of wind direction and surrounding buildings on natural ventilation.

FACTORS AFFECTING OZONE REMOVAL AND BYPRODUCT FORMATION FROM REACTIONS WITH T-SHIRT

Research Assistant: Aakash C. Rai

Principal Investigator: Professor Qingyan (Yan) Chen

Sponsor: Federal Aviation Administration (FAA)

ABSTRACT

Human health is adversely affected by ozone and the Volatile Organic Compounds (VOCs) produced from its reactions in indoor environment. Hence, it is important to characterize the ozone initiated chemistry in different indoor settings such as homes, airliners, etc. This investigation studied the surface reactions of ozone with T-shirt in a mid scale chamber to quantify the ozone removal and the formation of VOCs under different indoor conditions. The experiments were conducted under varying chamber conditions such as presence or absence of T-shirt, amount of soiling of T-shirt, ozone concentration, humidity level, and ventilation rate. The study used Gas Chromatography-Mass Spectrometry (GC-MS) and High Performance Liquid Chromatography (HPLC) to quantify the VOCs at steady state, and Proton Transfer Reaction – Mass Spectrometry (PTR-MS) to identify the temporal and spatial distributions. The results showed that an increase in the soiling level of T-shirt, ozone concentration, and ventilation rate generally led to increase of ozone removal and VOC production. The increase in humidity also resulted in an increase of the VOC production rate, but did not show any effect on the ozone removal. The experiments also showed that the VOCs were produced immediately after ozone injection and were enhanced near the reaction surface compared to bulk air.

INVERSE MODELING OF HEAT TRANSFER AND AIRFLOW BY USING ADJOINT METHOD

Research Assistant: Wei Liu

Principal Investigator: Professor Qingyan (Yan) Chen

Sponsor: National Basic Research Program of China

ABSTRACT

Flow fields in enclosed environment are crucial for creating a thermally comfortable and healthy indoor environment. Normally, a forward method by solving the Navire-Stokes equations is applied to design the air flow fields. It is a “trial-error” process and very time-consuming. To fast obtain the ideal air flow field, this investigation implemented the adjoint method based on control theory. This method was first implemented on a heat transfer problem. The cost function was defined as the temperature match of local location and the state function is the heat conduction equations, with control by the boundary as a heat flux. For the airflow problem, the cost function was defined as the velocity match of local location and the state function is the Navier-Stokes equations of the flow, with control by the boundary as a velocity inlet. This method could generate the adjoint function and optimal function according to the first variation of the augmented cost function. By solving the state equation and adjoint equation with a guessed control, the variation of the control could be calculated by the optimal function and a new control could be obtained. This process is repeated until an optimum solution is approached. Since each design cycle requires the numerical solution of both the state and the adjoint equations, the computational cost was roughly twice of the flow solutions. It was found the number of design circles to achieve a satisfactory design relies on the discrepancy between the guessed control and the optimum control. In the design of two-dimensional flow, the optimal function needs improvements to achieve faster and better optimal results.

A NEW EMPIRICAL MODEL FOR PREDICTING SINGLE-SIDED, WIND-DRIVEN NATURAL VENTILATION IN BUILDINGS

Research Assistant: Haojie Wang

Principal Investigator: Professor Qingyan (Yan) Chen

Sponsor: U.S. Department of Energy: Energy Efficient Buildings Hub

ABSTRACT

Prediction of single-sided natural ventilation is difficult due to the bi-directional flow at the opening and the complex flow around buildings. A new empirical model was developed that can predict the mean ventilation rate and fluctuating ventilation rate due to the pulsating flow and eddy penetration of single-sided, wind-driven natural ventilation in buildings. The governing equation is based on the non-uniform pressure distribution along the opening height. The new model shows that the ventilation rate and wind velocity are linearly correlated. This investigation studied the eddy penetration effect in the frequency domain based on fast Fourier transform. Large Eddy Simulation (LES) and experimental data were used to validate the new empirical model. The model has also been used to analyze the influence of the opening geometry and elevation on the ventilation rate.

ENERGY PERFORMANCE ANALYSIS OF UNDER-FLOOR AIR DISTRIBUTION (UFAD) SYSTEMS

Research Assistant: Yan Xue

Principal Investigator: Professor Qingyan (Yan) Chen

Sponsor: DOE

ABSTRACT

Under Floor Air Distribution (UFAD) systems can provide potential energy saving as well as indoor air quality improvement by generating partial air stratification. Design guidelines for UFAD are available for predicting the energy use and room air stratification. However, little studies have been conducted for air flow and heat transfer in the plenums that can affect UFAD performance. This project is to study the airflow and heat transfer in the plenums by using empirical and theoretical analysis. Previous studies show that heat loss to floor plenum will account for a large part of cooling load. The building energy simulation program EnergyPlus and airflow simulation program FLUENT have been used and the full-scale experiments have been done to validate models. This study will quantitatively predict the impact of the airflow and heat transfer in floor plenums on the energy and room environment for buildings with UFAD systems. A design guide will be developed for designers.

PERFORMANCE COMPARISON OF TWO PHASE FLUIDS IN HEAT PIPES FOR ELECTRONICS COOLING

Research Assistants: Harshad Inamdar and Dr. Orkan Kurtulus

Principal Investigator: Professor Eckhard Groll

ABSTRACT

The objective of this project was to compare the performance of different fluids which are suitable to be used in a flat heat pipe. In the calculations, the temperature gradient across the heat pipe was and all the physical parameters of the heat pipe were maintained constant. Thus, the best performing working fluid was the one which gave the maximum heat transfer rate. The comparison was based on a simulation model which was developed as a part of this project. The simulation model was first validated with experimental results obtained from published literature and then used to predict the heat transfer rates achievable for different configurations of the heat pipe.

Fluid thermal and transport were obtained from REFPROP v9. Heat pipe material properties were obtained from EES. The simulation model inputs included all the physical dimensions of the heat pipe and the working conditions for the device. The working temperature of the heat pipe had to be a calculated parameter; however, the equations leading to the calculation of the temperature were implicit. Hence, the simulation model required a guess value for the working temperature and then iterated over the calculated value to report the final value for the working temperature along with the heat transfer rate.

The material of the heat pipe was selected to be copper and it was maintained the same throughout the comparison. A few working fluid-copper combinations might not be compatible. In such a scenario alternative combinations along with their performance have been reported. Also, the purpose of this project was to explore the use of heat pipes in electronics cooling- particularly in the cooling of laptop processors. Thus, there was a limitation on the dimensions of the heat pipes to be studied. The heat source and sink temperatures were also limited by the optimum processor chip temperatures and the possible ambient temperature in the environment of a laptop processor.

One of the main assumptions for the simulation model was the heat pipe configuration which did not include fins or any surface enhancements at the sink end. The sink temperature is thus not the actual sink temperature, instead is the surface temperature of the heat pipe on the outside wall at the condenser end. Similarly, the heat source was left out of the model, and so the source temperature is the surface temperature of the heat pipe on the outside wall at the evaporator end. The performance of the heat pipe is thus purely the performance of the heat pipe alone and does not include the efficiencies of the heat transfer from the source (processor chip) to the device and from the device to the air (fin-fan assembly).

Water, ammonia, methanol, ethanol, acetone were predicted to be the better performing fluids, in same order. The remaining working fluids compared were predicted to give similar levels of performance and thus no clear indication of superiority could be inferred.
Other fluid properties were also collected which are toxicity (which was essential due to the device being used in close contact with humans), GWP (which was important because of the climate change issue), dielectric constant (which was important because the device would have to be installed in a space filled with electrical equipment), the working pressure at which the devices would function (which was important because of obvious design issues with the heat pipe body), freezing point, boiling point, ASHRAE standard 34-2010 safety classification, NFPA fire safety classification and material compatibility issues and the price of the working fluid in dollars per kilogram.

The choice of a particular working fluid and heat pipe body has to be based on multiple parameters and a single recommendation is difficult to make.

PERFORMANCE COMPARISON OF LIQUID-LOOP FLUIDS FOR ELECTRONICS COOLING

Post Doc: Orkan Kurtulus

Principal Investigators: Prof. Eckhard A. Groll, Prof. Suresh V. Garimella

Sponsor: Cooling Technologies Research Center (CTRC)

ABSTRACT

In recent years, various electronic cooling applications have attracted the attention of many researchers. As a result, different cooling technologies have been developed to efficiently remove the heat from the electronic components. During this time, the use of liquid coolants has become more attractive because of their higher heat transfer capability than conventional air-cooling technologies. Several heat transfer fluids could be considered for liquid loops that are used in electronics cooling. However, it seems difficult to make a fluid selection without the appropriate knowledge of their heat transfer and pressure drop characteristics. Therefore, this research focuses on identifying thermal fluids and their thermodynamic and transport properties, and to conduct detailed theoretical analyses that will help to identify the most energy-efficient and cost-effective fluids for a range of applications. As part of the project, the heat transfer and pressure drop characteristics of fifty-four different coolants are being compared to each other for either single-phase or two-phase flow. In addition to their thermal performance, the environmental impact (GWP: Global warming potential), toxicity, flammability, material compatibility and dielectric properties are being tabulated. Furthermore, a simulation tool was developed to predict the performance of the various thermal fluids in single- and two-phase microchannel heat sink application for CPUs cooling. In order to rank the thermal fluid performance for given dimensions, working temperatures, and a constant heat dissipation (Q), the application pressure drop and required pumping power (PP) along the microchannel was calculated. The thermal fluid performance is described as Q/PP.

NUMERICAL STUDY OF SUPERCRITICAL CARBON DIOXIDE CONVECTIVE HEAT TRANSFER FOR CONCENTRATED SOLAR POWER BRAYTON CYCLES

Research Assistant: Scott M. Flueckiger

Principal Investigators: Professor Suresh V. Garimella, Professor Eckhard A. Groll

ABSTRACT

Concentrated solar power (CSP) plants focus incident sunlight into high-grade heat for large-scale power generation. Use of traditional steam Rankine cycles for power generation requires large volumes of ambient water to sustain heat rejection from the cycle to the surroundings. However, CSP plants are typically located in arid regions that are subject to water stress and not amenable to the water consumption needed for wet cooling. Next-generation CSP plant therefore require new working fluids that are compatible with dry cooling with ambient air to improve commercial viability. Carbon dioxide, operating in a supercritical Brayton cycle, is a natural and low-cost alternative to steam that supports dry cooling.

Supercritical fluids exhibit intense non-linear variations in thermal transport properties with pressure and temperature, influencing convective heat transfer performance in the power cycle flow loop. An improved understanding of this duct-flow convection for supercritical carbon dioxide is essential for design of future Brayton cycle heat exchangers. A hot gas bypass compressor load stand is modified with an external electric heat source to measure carbon dioxide convective heat transfer coefficients in the supercritical region. This experimental facility enables measurement at supercritical pressures and temperatures beyond previous studies in the literature. A computational fluid dynamics model of the experiment is also formulated to provide additional validation. Inlet pressure, temperature, and mass flux are varied to assess the respective influences on Nusselt number.

COLD CLIMATE HEAT PUMP FOR RESIDENTIAL APPLICATIONS

Research Assistants: Bin Yang, Christian Bach, Sugirdhalakshmi Ramaraj and Yuanpei Song

Principal Investigators: Professors Eckhard Groll, James Braun and Travis Horton

Sponsor: U.S. Department of Energy

ABSTRACT

The traditional air-source heat pump works with dramatically decreased heat capacity and coefficient of performance (COP) under very low ambient temperature. Some modifications are made on the conventional vapor compression heat pump system to improve the heat capacity and COP under very low outdoor ambient temperature. Three technologies are investigated: oil flooded compression with regeneration, cooled compression with economizing, and individual evaporator circuit refrigerant flow control.

The first technology can significantly improve the system efficiency by reducing the thermal loss of refrigerant desuperheating in condenser and that of throttling of two phase refrigerant in expansion devices. This technology involves a scroll compressor running with oil of relatively large mass flow rate mixed with R410A at the suction region, in order to absorb the heat in compression process. The cooled compression with economizing technology can significantly improve the heat capacity and COP with a low cost. The efficiency improvement by this technology is even greater than that of the oil flooded compression technology. It is very difficult for the evaporators used in conventional heat pump systems to provide uniform exit superheat. By controlling refrigerant flow in the evaporator circuit individually, a uniform superheat at the outlet of evaporator can be achieved. A hybrid control using small valves of fine-tuning along with a primary expansion valve is proposed, instead of using conventional expansion valve on each of the evaporator circuit. The primary expansion valve provides the needed pressure drop and small valve balances the refrigerant flow on individual circuit. Based on work done at Purdue and NIST, this individual circuit control can gain the energy benefit of up to 10% for the system.

COLD CLIMATE FIELD DEMONSTRATION OF A TWO-STAGE AIR-SOURCE HEAT PUMP

Research Assistants: Stephen Caskey, Derek Kultgen

Principal Investigators: Professors Eckhard Groll and William Hutzel

Sponsor: Department of Defense/Environmental Security Technology Certification Program

ABSTRACT

This project is a field demonstration of a new air-source heat pump technology that is optimized for colder climates. The heat pump is designed to provide 65,000 Btu/hr (19 kW) of heat at an ambient temperature of -5^{0}F (-21^{0}C) achieving heating COPs that are approximately 45% of the maximum theoretical Carnot heating COPs. The demonstration site requires the heat pump to operate on single-phase power. Larger system capacities become feasible when three-phase power is available. The major modification from a conventional air-source heat pump is the use of two-stage compression with economizing; the high-stage compressor is a variable speed scroll compressor and the low-stage compressor is a large displacement fixed speed tandem scroll compressor. The industry partners, Ingersoll Rand – The Trane Company, Emerson Climate Technologies – Copeland, Danfoss A/S, and Automatic Logic of Indiana donated off-the-shelf components. The component selection will aid in the commercialization potential for the system. The equipment sizing was determined from a previously developed EES code to simulate the system at different conditions and compressor configurations. Two heat pumps of identical design will be installed in two buildings at Camp Atterbury in Southern Indiana for field testing over the course of a year. Each building has two HVAC systems and one in each will be replaced by a heat pump to provide accurate comparisons to conventional HVAC systems. Six performance objectives will be used to outline the comparisons; primary energy consumption, cost, emissions, installation and operation difficulty, and comfort. Each objective has defined conditions to quantify a successful demonstration and motivate the capability of commercialization.

SECONDARY LOOP HEAT PUMP FOR RESIDENTIAL APPLICATIONS

Research Assistant: Tim Blatchley

Principal Investigators: Professors James Braun, Travis Horton and Eckhard Groll

Sponsors: Carrier Corporation, Emerson Climate Technologies, Grundfos

ABSTRACT

The development of secondary loop systems for residential applications is motivated by a desire to use natural refrigerants which have good thermodynamic and thermophysical properties in the primary vapor compression portion of the system, but cannot be used in converntional heat pumps due to their flammability and/or toxicity. Natural refrigerants are also desirable because of their low global warming potentials which will allow them to escape future refrigerant phase-outs. Secondary loops allow natural refrigerants to be used since they never enter the conditioned space. However, due to the addition of an intermediate heat exchanger the secondary loop system may have greater cost and potentially lower performance than traditional direct expansion (DX) split systems. Another disadvantage of secondary loop systems that utilize hydrocarbons as the primary refrigerant is the current regulations limiting the charge of flammable refrigerants permitted in residential systems. However with the use of double-wall brazed plate heat exchangers the charge can be drastically reduced from of a standard DX system. Despite these disadvantages, modeling has shown that a secondary loop heat pump may outperform a traditional DX system since the primary refrigerant and secondary fluid having more advantageous properties. System level cost and performance optimization are being performed to design a prototype secondary loop system and determine how competitive these systems will be compared to traditional DX systems on the market today.

ORGANIC RANKINE CYCLE WITH SOLUTION CIRCUIT FOR LOW-GRADE HEAT RECOVERY

Research Assistant: Abhinav Krishna

Principal Investigators: Professors Eckhard Groll and Suresh Garimella

ABSTRACT

The increasing cost of energy, coupled with the recent drive for energy security and climate change mitigation have provided the impetus for harnessing renewable energy sources as viable alternatives to conventional fossil fuels. However, several of these renewable energy sources, including geothermal, biomass and solar, intrinsically provide low-grade heat (at temperatures between 60°C – 300°C). Furthermore, thermodynamic considerations mean that a large amount of low-grade heat is discharged from power plants, automobiles and various other industrial processes. Nevertheless, technologies attempting to provide low-grade heat recovery solutions have seen very limited commercialization. This is broadly due to two reasons: lack of historical research and development in the area of waste heat recovery due to technical and cost impediments; and technical challenges associated with scaling the technology from utility scale to commercial scale, particularly with regard to expansion machines (turbines). However, due to rising primary energy costs and the environmental premium being placed on fossil fuels, the conversion from low-grade heat to electrical energy is of increasing interest. In this regard, this project focuses on a power generating Organic Rankine Cycle with Solution Circuit (ORCSC), also known as the Absorption-Rankine cycle, for low-grade heat sources.

The foundation for the ORCSC is the Vapor Compression Cycle with Solution Circuit (VCCSC). Reversing the VCCSC creates a power generating cycle similar in nature to the Organic Rankine Cycle, but with features analogous to an Absorption cycle. The working fluid of the ORCSC is a binary mixture consisting of Carbon Dioxide (CO_2) acting as the volatile component in the vapor phase, and acetone acting as a low-viscosity, high boiling point solution. The use of a binary mixture provides the ability to use the temperature glide in the evaporation and condensation processes to match the temperature profiles of the source and sink fluids, and facilitates intrinsic capacity control. Furthermore, this system is easily adapted for a range of source and sink temperatures by adjusting the concentration of the absorbent used in the working pair. This leads to higher overall system efficiencies when coupled with sources that have varying heat input temperatures or loads. Incorporating Carbon Dioxide as the primary working fluid has a number of advantages, including low Global Warming and Ozone Depletion potentials, non-toxicity, wide availability, low cost as well as the thermodynamic advantage of a large volumetric heat capacity, which enables the use of smaller turbines. Carbon Dioxide is also an excellent heat transfer fluid, which serves to minimize the losses in the heat exchangers. The addition of acetone in the solution circuit results in significantly lower working pressures when compared to a cycle with pure CO_2. Given these advantages, this project explores the design space using both a simulation based approach as well as an experimental test stand for concept validation.

OPTIMAL MATCHING OF A SCROLL-TYPE EXPANDER TO AN ORGANIC RANKINE CYCLE

Research Assistant: Brandon Woodland

Principal Investigators: James E. Braun, Eckhard A. Groll, W. Travis Horton

Sponsor: The Herrick Foundation

ABSTRACT

An organic Rankine cycle (ORC) is a power cycle employing an organic working fluid. The term ORC is also applied generally to any Rankine cycle with a low-grade heat source (80° – 300°C). Because ORC are often employed in small-scale applications, use of positive displacement equipment is often favored over the centrifugal units used in large-scale power plants. A key feature of a positive displacement expander is its built-in volume ratio.

An ORC with a scroll-type expander is studied experimentally. It is shown that the adiabatic efficiency of the expander can be fully characterized by its filling factor and the expansion volume ratio imposed across it. In particular, the peak adiabatic efficiency occurs near a filling factor of unity and an expansion volume ratio near the built-in volume ratio of the expander. A procedure is presented which allows prediction of cycle performance based on knowledge of the expander efficiency versus expansion volume ratio, cycle operating conditions, and working fluid. Use of this procedure suggests that an optimal expander can be chosen for a set of cycle operating conditions based on knowledge of its peak efficiency and built-in volume ratio alone.