**Images should have 3:2 ratio**

**Publications**

Here is a collection of my publications. A pdf copy of my slides from seminars and poster presentations is uploaded to my GitHub. Conference and journal articles are linked to the publisher site, but a copy can be sent upon request.

Thesis Defense: Porous Trailing Edge for Airfoil and Fan Noise Reduction at Low-Speed Stall Conditions

Summary: This numerical study investigates an airfoil profile from an electronic cooling fan using two trailing edges: a solid and a porous trailing edge that is applied from the half chord. A wall-resolving, pore-scale LES predicts the flow at a low Reynolds number of 15,000 and a high angle of attack of 25˚. Then, the Ffowcs-Williams and Hawkings analogy is used to obtain the farfield noise. The results show that the porous trailing edge reduces noise by up to 2 dB at a high angle of attack, and five flow field effects contribute to noise reduction.

Poster Presentation: Porous Trailing Edge for Airfoil and Fan Noise Reduction at Low-Speed Stall Conditions

Presented at the 20th Annual Symposium on Graduate Research and Scholarly Projects (GRASP).

Summary: In this experimental study, fan blades for an electronic cooling fan were designed with a porous trailing edge from the half-chord. The porous media had gyroid unit cells with an average pore diameter of 1mm and 1.25mm. The rotors were 3D-printed using resin, and acoustic measurements were conducted in a semi-anechoic chamber. The study finds that compared to the solid baseline blades, there is a 4.3% overall noise reduction, especially at low to mid frequencies. However, at high frequencies, the increased surface roughness produces a slight increase in noise.

Poster Presentation: Review of Direct Air Capture (DAC) Technology and Challenges to Achieve Paris Agreement Objectives

Presented at the 20th Annual Symposium on Graduate Research and Scholarly Projects (GRASP).

Summary: The Paris Agreement aims to limit global temperature rise to 2°C above pre-industrial levels, but the global temperature has increased by 1.2°C in 2022. One mitigation strategy is to use DAC to absorb CO2 from the atmosphere. Orca is the largest DAC plant, which has a capture capacity of 4,000 tCO2/year and costs $600/tCO2. To achieve the 20Gt/CO2 objective, 2.4 million Orca-sized plants are needed, requiring 1.4 × 1020 J and costing $20 × 1012 annually. The analysis indicates that scaling up to the required GtCO2/year is currently unfeasible; nonetheless, DAC technologies should continue to develop to ease the transition to renewable energy and for future carbon removal.

Poster Presentation: Microstructures for 3D Printed Wicks For Heat Transfer

Presented at the Undergraduate Research and Creative Activity Forum (URCAF).

Summary: This experimental study explores the use of wicks in high heat flux evaporative cooling systems, addressing the issue of premature coolant dryout due to limited capillary flow. Two microstructures for wicks are investigated: a triangular channel with internal iso truss structures and a cubic lattice structure with internal iso truss structures. The samples were 3D-printed with resin, and a rate-of-rise experiment was conducted to measure the capillary rise. The non-uniform pore structures improve the capillary pumping capability without significantly reducing permeability. The triangular channel with iso truss structures improved maximum liquid height by 100% above the baseline structure.

Guest Instructor: Basics of FVM using OpenFOAM with Applications

Presented as a lecture for ME 782 Engineering Applications of Computational Fluid Dynamics and Heat Transfer, Wichita State University.

Dr. Ahmed invited me to provide two hands-on lectures for his graduate CFD course. Some theoretical content, such as discretization, stability, and numerical schemes, was provided, but a large portion of the lecture was hands-on exercises with meshing using snappyHexMesh, solving using simpleFoam and pimpleFoam, and post-processing using ParaView. The students were also introduced to the university's High-Performance Computing (HPC) cluster, and homework assignments were provided to help them practice their Linux skills and better understand the CFD workflow.

Conference Paper: Reduction of Input Features from Machine Learning Datasets for Water Quality Analysis

Presented at the IEEE International Conference on Artificial Intelligence, Control, Data Sciences and Applications (ACDSA).

Summary: Traditional water quality testing methods require a high number of distinct tests; however, machine learning can minimize the required number of input features. This study uses a Kaggle dataset with nine features and 2011 data points. The most significant features are first identified using recursive feature elimination with cross-validation (RFECV), permutation importance (PI), and random forest (RF) approaches. Second, the water's potability is predicted using an artificial neural network (ANN) and a support vector machine (SVM). Using the machine learning model, the dataset with five features had 3% more errors compared to the dataset with nine features, yet the reduction in the required test would reduce testing costs by about 65%.

Seminar: Computational Modelling of Surface Tension Driven Flow and Experimental Measurements of Metallic Wicks

Presented as a Graduate Seminar to the Department of Mechanical Engineering, Wichita State University.

Summary: In a high heat flux thermal management system, wicking of the working fluid can prevent surface dryness to maintain high heat transfer. However, modeling two-phase systems is challenging, as are experimental measurements at the micro-scale. In this seminar, recent improvements to numerical algorithms for surface-tension driven flow are presented and the accuracies are discussed. Additionally, a new rate-of-rise experimental setup using an infrared camera is presented, which can accurately measure the working fluid height for opaque metallic samples.

Poster Presentation: Micro-Xray tomography based pore-scale simulation of additively manufactured wicks

Presented at the 19th Annual Symposium on Graduate Research and Scholarly Projects (GRASP).

Summary: This study presents a CFD methodology using micro-Xray computed tomography (μCT) data to accurately simulate additively manufactured (AM) wicks. To validate the workflow, AM wicks were manufactured using laser powder bed fusion and their porosity, permeability and bubble point radius were measured. 2,000 tomographic images with a voxel size of 0.6 μm were generated. Image segmentation was performed in ImageJ, and this study suggests utilizing a bilateral noise reduction filter and Ostu's method of thresholding. The mesh was generated using snappyHexMesh, and the simulations were conducted in OpenFOAM. The predicted permeability of 1.23×10-12 m2 agrees with the Carman-Kozeny relation and experimental measurements.

Seminar: Meshing and Pore-Scale Simulation of Additively Manufactured Wicks in OpenFOAM

Presented as a Graduate Seminar to the Department of Mechanical Engineering, Wichita State University.

Summary: Fundamental understandings of capillary flow through additively manufactured (AM) wicks with highly non-uniform pore structures are essential to design high heat flux two-phase thermal management systems. Traditional volume-average study is challenging to accurately predict key characteristics of the wicks, e.g., permeability, while experimental characterizations are challenging and expensive. A micro-Xray computed tomography (μCT) can generate highly detailed volume meshes for accurate prediction of pore-scale heat and mass transfer of wicks using computational fluid dynamics (CFD). This presentation presents an optimal workflow from μCT data to pore-scale simulation using open-source tools.

**CFD Research**

Here is a collection of my research projects done during my graduate studies. Most of the research projects are funded; however, some projects are done as course projects.

Porous Trailing Edge for Airfoil and Fan Noise Reduction at Low-Speed Stall Conditions

This study served as my thesis for the Master of Science degree where I was interested in reducing the noise produced by airfoils for electronic cooling fans. OpenFOAM was used to study the aeroacoustic impact of a porous trailing edge at low speeds and high angles of attack. The flow field around the airfoil was predicted using LES, while the acoustic properties were done using the Ffowcs Williams and Hawkings Equation. I was interested in the noise reduction mechanism caused by the flow field impacts of the porous media, so I employed a pore-scale approach instead of a volume-averaged approach to obtain an accurate depiction of the flow field around the porous media and the shear layer. The thesis concludes that the Kelvin-Helmholtz rolls are responsible for separation/stall noise, and five flow field effects contributed to the noise reduction. Find out more in my publications!

Pore-scale Simulation of the Capillary Rise in a Lattice Column

In my research laboratory, we were interested in the accurate modeling of capillary rise in uniform and non-uniform wick geometry. A simple validation case for a body-centered cubic (BCC) lattice column was created using a 3D-printed resin lattice and numerical results from the literature. Two working fluids were considered: water and methanol, and the numerical results from the literature align well with OpenFOAM’s prediction. Compared to experimental values, the results tended to underpredict the rate of rise and equilibrium height, and this was likely due to imperfections during printing.

Capillary Rise in Glass Capillary Tube

As a graduate research assistant, I introduced the twoPhaseFlow library to the laboratory, but validation was required to ensure accurate predictions. Rate-of-rise experiments were carried out in a glass capillary tube with a diameter of 1.1mm, using alcohol as the working fluid, and an analytical solution was obtained using ODE 45. The results demonstrate that OpenFOAM can accurately predict the steady-state solution, but it tends to overpredict the rate of rise. For the initial conditions, we find that submerging the capillary tube deeper into the fluid tends to improve accuracy. Overall, we concluded that the steady state solution is accurate (<5% error); however, the rate of rise prediction can have greater errors (~10%).

LES Study of the Effects of Wind Turbine Placement on Power Generation

This project was conducted as a term project in my Rotor Aerodynamics course, and it looked at how an upstream wind turbine's wake affected downstream turbines. The turbine blades were modeled as actuator lines using OpenFOAM, and the turbulence in the wake was predicted using LES. Three configurations of three wind turbines were considered: aligned axially, aligned diagonally, and aligned in a staggered pattern. The study concludes that the wake has a significant impact on the power production of downstream turbines due to the velocity deficit, and the severity is determined by the spacing and configuration. Most wind farm state ordinances in the U.S. suggest turbines that are too close to each other, and to increase a turbine's lifetime power production, care must be taken on wind turbine placement.

Micro-Xray Tomography Based Pore-scale Simulation of Additively Manufactured Wicks

This investigation was carried out as part of my role as a graduate research assistant, and we were interested in accurate numerical predictions of existing additively conducted evaporator wicks. Colleagues at NASA JPL manufactured metallic porous media and generated high-quality micro-Xray computed tomography (μCT) data. After image segmentation and model reconstruction, a crucial smoothing step is introduced to remove artificial roughness due to the discrete nature of images. The developed open-source μCT to CFD pipeline was used to compare porosity and permeability with experimental data. Results show that the pore-scale simulations accurately predicted the measured permeability of the AM wick within 5.6% error and negligible errors in porosity.

Influence of RANS Turbulence Model in CFD modeling of packed beds

This is a validation study for a term project in my Transport in Porous Media course, which investigated the accuracy of modeling a packed spherical bed using RANS. Packed beds are ideal for simple heat exchangers because of their configurable porosity and permeability; however, accurate turbulent modeling in complex pore geometries remains a challenge. Two turbulence models are considered, Spalart-Allmaras and RNG k-ε and the default coefficients are used. The results show that RNG k-ε outperforms Spalart-Allmaras in terms of predicted pressure drop across the porous media, while Spalart-Allmaras is more accurate in predicting the turbulent viscosity field. The project reminded me that all models are wrong, but some are useful!

**Fun CFD Simulations**

Here is a collection of some fun CFD simulations done during my free time to better understand fluid dynamics and OpenFOAM. Due to limited compute resources, the meshes are fairly coarse; however, the simulations still provide some valuable insights.

Adjoint optimization of car side mirror

Aerodynamic penalty of opening my car window

After a long day of research in the lab, the worst thing is to sit in my hot car as I begin my commute back home. It’s well known that opening the window durin the first couple of minutes is beneficial for cooling the car, however I;ve always been interested in how much this affects the aerodynamic of the vehicle. In this simulation, I modified the DrivAer car model and created a basic interior to simulate the effects of opening the front windows of the car. Surpinsly, after normalizing with the frontal area, the drag penalty is fairly small XXX.

Compartment Fire

It's still a work in progress, but I've always thought fire was fascinating. However, it's tough to find guides and textbooks that focus on the computational component without delving too deep into chemistry. This is the first simulation of heptane combustion from a ground-level intake. As the gas burns, it generates heat and raises the room's temperature. The image also shows CO2 contours, and it's interesting to observe how the gas spreads throughout the compartment. I'll provide further updates as I learn more about combustion and flame modeling.

Wind Around Kuala Lumpur

A fun CFD project in which I simulated the wind around my capital city, Kuala Lumpur. A RANS solver is utilized, and the geometry considered was about 1km by 1km. The Petronas Twins Towers are visualized, and the results indicate that on some roads, there are some high wind loads to pedestrians, as evident by streamlines with high velocity magnitudes. The mesh is fairly coarse, and the boundary layers are not well resolved due to the differences in building size; however, the simulation provides a high-level overview for pedistrian comfort!

External Aerodynamics of a Cessna 172 aircraft

A fun CFD project in which the flow field around the world's most produced aircraft is simulated using a RANS solver. The k-ε model is utilized, and the y+ is between 100 and 500. The aircraft is cruising at 122 knots, with an ambient air temperature of 20 degrees Celsius. The mesh is fairly coarse; however, the lift and drag forces are reasonable.

Stirring my morning coffee

A fun CFD simulation where OpenFOAM's sixDoFRigidBodyMotion solver is used to stir some coffee and milk together. To simulate the two-phase flow, interMixingFoam is utilized. The coffee and milk are miscible, while the air interface is immiscible.

**Other Cool Projects**

Here are some cool projects done during college or as a side project.

sUAV Capable of Ground-to-Air Medical Specimen Retrieval

At Wichita State University, aerospace engineering students are required to design and build an RC airplane from scratch and compete at the Boeing Bronze Propeller Competition. For the 2022 cohort, the aircraft must be able to pick up objects from the ground on-the-fly. For my team, we decided to pick up COVID-19 test samples and design our airplane with an external hook. The features of the final design include a high wing, elevated wings, large flaps, a detachable wing, and high thrust. As the aerodynamic lead, I designed the aerodynamic surfaces (wing and flaps, empennage) and validated them during wind tunnel tests, which showed better than predicted performance. Unfortunately, we were unable to pick up specimens due to the high speed of flight; however, the design process was nonetheless an unpleasant experience.

Development of Low Reynold’s Number 2D Flow Analysis

As part of my Experimental Methods in Aerospace Engineering course, my team was chosen to conduct research in collaboration with the National Institute of Aviation Research (NIAR) and Trek Bicycle Corporation. Our project focused on the development of an accurate and economical 2D wind tunnel testing technique for airfoils at very low Reynolds numbers. We designed and manufactured an integrated wake rake with surface pressure taps to capture pressure data on the surface and in the wake. I was responsible for creating the mathematical models, calculating engineering estimates for wind tunnel testing, and validating loads with force balance measurements. Unfortunately, due to the low Reynolds number, errors and uncertainty in the wind tunnel made the integrating wake rake unreliable, and we concluded that a traditional force balance is more suitable if validation of the wake profile is not required.

Ball Collision Simulation in MATLAB

All aerospace engineering students at Wichita State University have completed the dreaded ball collision simulation project as part of the Engineering Digital Computations course. Unfortunately, because I was a tutor for aerospace engineering, I have done variations of this project at least six times! The premise is simple: balls move in a certain direction, and based on collision physics, they should change direction if they hit the wall, having a head-on or an offset collision. The code is written in MATLAB, and it is certainly a fun yet challenging project. If you're a student working on this now, my advice is to start the project early! And remember, everyone before you has done this too, so we know how difficult this is!