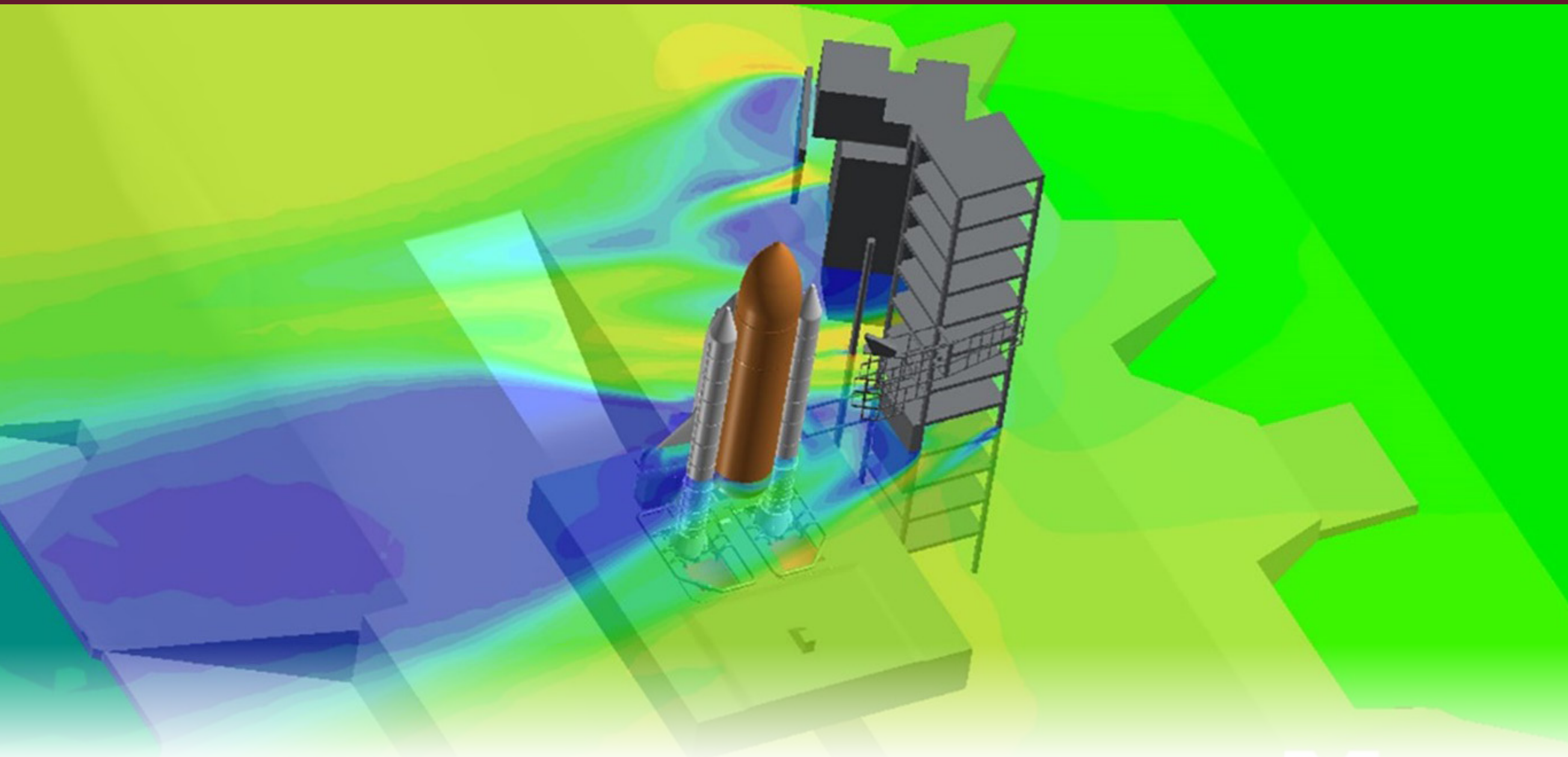


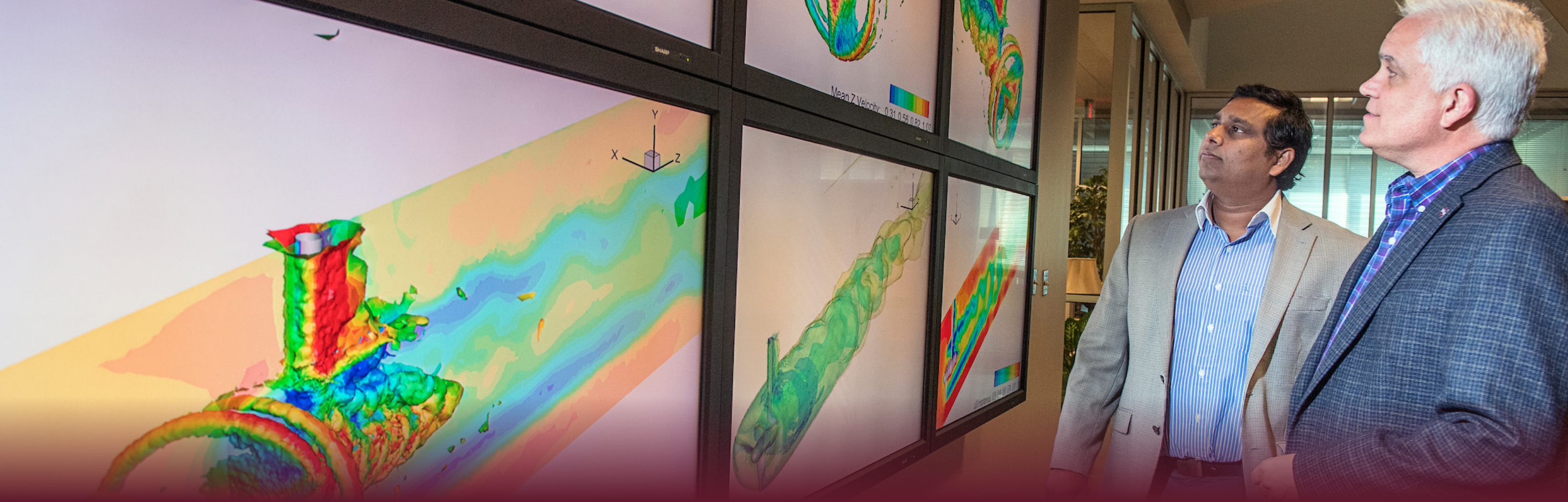
COMPUTATIONAL FLUID DYNAMICS AT CAVS

ABOUT CFD

Computational Fluid Dynamics leverages fundamental laws of mass momentum and energy conservation to predict heat and mass transfer in fluid flows. The advances in high performance computing are allowing solution of these equations for geometrically complex multi-scale/muti-physics problems providing an inexpensive tool for the analysis and design of engineering systems compared to the traditional build-and-test approach.

The Computational Fluid Dynamics (CFD) Modeling and Simulation team at the Center for Advanced Vehicular Systems (CAVS) focuses on research and development of advanced computational modeling, simulation, and design of physical systems to solve real-world problems.

The research team involves faculty members in expertise in scientific computing, applied mathematics, aerospace and mechanical systems, and focus on developing methods and tools for: mesh generation, compressible and incompressible flow solvers, energetics/chemically reacting flows, thermal management, aerodynamics, design optimization, and fluid-structure-thermal interaction, among others.



CAPABILITIES

HIGH PERFORMANCE COMPUTING

CAVS research in computational fluid dynamics is enhanced by Mississippi State University's high performance computing resources, which allows for large-scale modeling and simulation efforts. MSU ranks among the top universities in the U.S. for supercomputing capacity.

SOFTWARE AND CODES

CAVS researchers have developed several open-source or licensed software programs that advance modeling and simulation capabilities. The Loci-CHEM code, developed at MSU, is a foundational computational fluid dynamics software used by NASA and DoD for launch vehicle, propulsion, and missile systems analysis.

FUNDAMENTAL FLOW PHYSICS

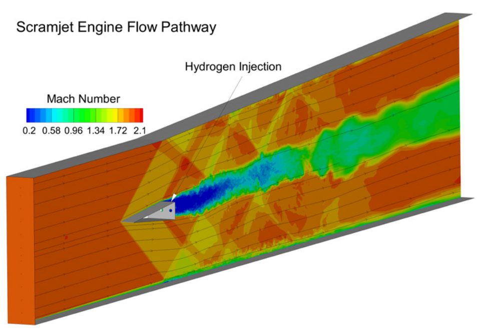
Computational Fluid Dynamics researchers at CAVS are advancing the understanding of fundamental flow physics to help improve complex engineering systems. Using interdisciplinary expertise, our researchers are furthering models related to turbulence, aeroacoustics, flutter, wave breaking and turbulent combustion, among other areas.

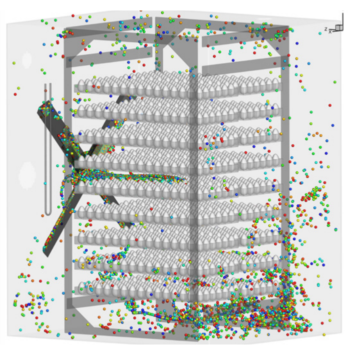
RESEARCH AREAS

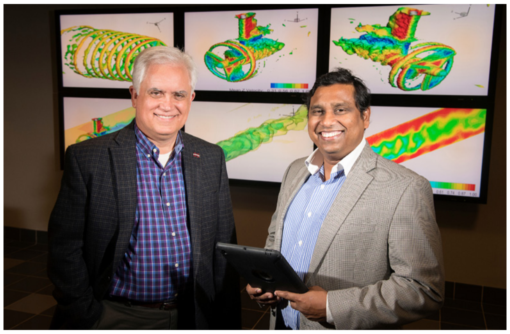
Hypersonics
CAVS researchers are unlocking new possibilities in aerospace with expertise in hypersonic systems modeling and simulation. From rocket systems to scramjet engines, computational fluid dynamics is being used to push the boundaries of hypersonic flight by allowing for the simulation of scenarios that are either cost-prohibitive or logistically challenging to physically test.

Biofluids
Researchers at CAVS are using computational fluid dynamics to simulate and improve novel medical therapies for both adults and children. This research has led to several patents on biomedical devices, including a patented artificial lung device. Projects include modeling blood damage and thrombosis in artificial organs, advancing understanding of key medical issues.

Energy Systems
Computational fluid dynamics research at CAVS is advancing energy systems through the development of novel simulations, such as the environmental impact of hydrokinetic wind farms and using liquid metals to cool nuclear reactors.







WWW.CAVS.MSSTATE.EDU

FOR MORE INFORMATION ON COMPUTATIONAL FLUID DYNAMICS
CONTACT SHANTI BHUSHAN, CAVS CFD LEAD.

bhushan@me.msstate.edu   662.325.5431

Clay Walden
Executive Director • CAVS
walden@cavs.msstate.edu

Shanti Bhushan
Associate Professor, Mechanical Engineering
CFD Technical Lead, CAVS
bhushan@me.msstate.edu

MISSISSIPPI STATE UNIVERSITY™
CENTER FOR ADVANCED VEHICULAR SYSTEMS