

# Computational Fluid Dynamics

## Fluid Dynamics Simulation and Experimentation

With faster processors and better numerical techniques, computational fluid dynamics (CFD) tools have revolutionized engineering design and optimization—limiting expensive experimentation and providing virtual solutions with short turnaround time. Today, CFD is used extensively to analyze applications as diverse as aircraft wing design and sportswear manufacturing.

Because the processes are complex and uncertain, CFD simulations may require experimental support for input data and validation. The Geosciences and Engineering Division (GED) at Southwest Research Institute® (SwRI®) has extensive experience and capabilities in CFD simulations of complex engineered and natural systems and experimentation. Results provide clients with optimized design solutions.

### Capabilities

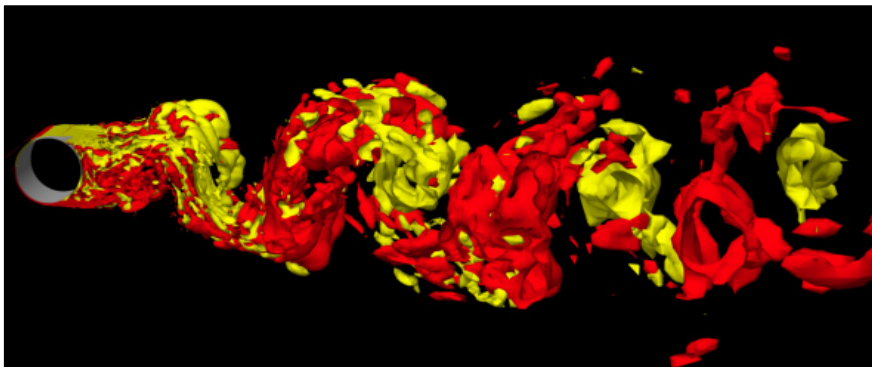
Our integrated multidisciplinary approach uses code customization, analytical model development and applications, and experimental investigation to accurately and effectively solve complex fluid flow and heat transfer problems including:

- Conjugate heat and flow analysis with multimode heat transfer and phase change
- Complex turbulent unsteady flow and acoustic analysis
- Numerical simulation of porous media flow
- Integrated flow and thermal analysis of engineered and natural systems with interface modeling
- Mesh-free, particle-based computing and smoothed particle hydrodynamics
- Particulate and droplet flow simulation
- Free surface flow and fluid-structure interaction
- Fire dynamics simulation

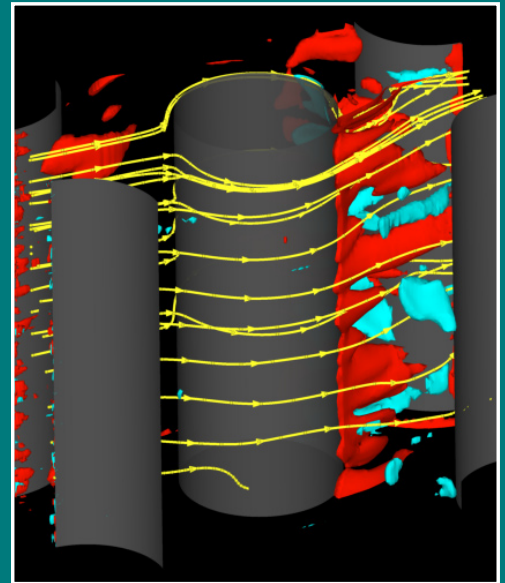
### Applications

GED staff members also have expertise in CFD code modification, algorithm development, and experimental benchmarking to address specific client needs. GED has provided extensive CFD research and technical assistance to a variety of programs. The broad spectrum of staff experience can be applied for:

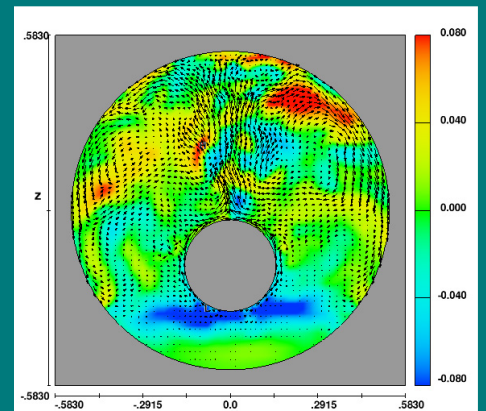
- Fluid flow and transport analysis of pressure vessels
- Chemical process streams
- Environmental systems and natural hazard analysis for tsunamis and volcanic eruptions
- Simulation of subsurface flow in porous fractured rock
- Pipeline flow analysis
- Fire dynamic analysis of onshore and offshore structures
- Flow and thermal study of mixing chambers



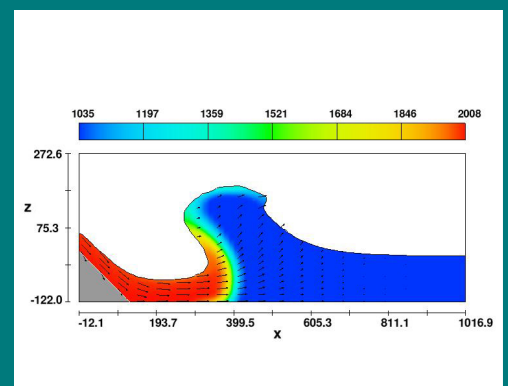
GED performed three-dimensional simulations for high-speed flow over a circular cylinder to validate multiscale hybrid turbulence models.



GED engineers have simulated flow across staggered cylinders encountered in heat exchangers and reactors. The figure shows the streamlines and vorticity surfaces in the domain.



Using CFD simulations, GED predicted the thermal and fluid flow processes that could take place in a high-level waste repository. These simulations establish confidence in full-scale drift modeling results under expected repository performance conditions.



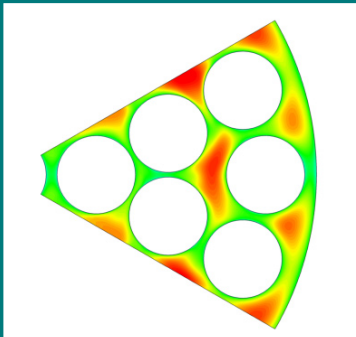
Using the volume-of-fluid (VOF) technique in the solver FLOW-3D®, the motion of a sliding wedge is simulated to predict the wave run-up height.

DO-16732



GED simulated a major fire that occurred in a highway tunnel to evaluate the thermal conditions during the fire.

DO-16735



Simulations show the time-averaged velocity distribution in a rod bundle.

**We welcome your inquiries.  
For additional information,  
please contact:**

**Kaushik Das, Ph.D.**

Research Engineer  
(210) 522-4269  
International: 001-210-522-4269  
kdas@swri.org

**Debashis Basu, Ph.D.**

Research Engineer  
(210) 522-8333  
International: 001-210-522-8333  
dbasu@swri.org

**Geosciences and Engineering  
Division**

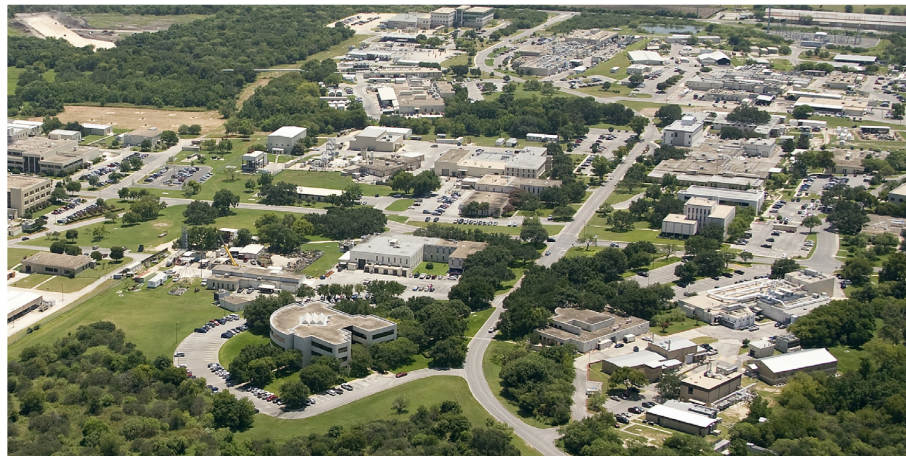
Southwest Research Institute  
6220 Culebra Road (78238-5166)  
P.O. Drawer 28510 (78228-0510)  
San Antonio, Texas USA

**www.swri.org**

**Resources**

Using a tailored selection from among commercial, open source, and in-house CFD software, GED addresses diverse client requirements. Dedicated pre- and post-processing tools for mesh generation and visualization enhance problem-solving and communication of results to clients.

Software	Developer/Source
FLUENT	ANSYS-FLUENT
FLOW-3D	FLOW-Science
SPH-Tsunami	SwRI
MFIX	NETL
MULTIFLO	SwRI
FDS	NIST



Southwest Research Institute is an independent, nonprofit, applied engineering and physical sciences research and development organization using multidisciplinary approaches to problem solving. The Institute occupies 1,200 acres in San Antonio, Texas, and provides more than 2 million square feet of laboratories, test facilities, workshops and offices for more than 3,300 employees who perform contract work for industry and government clients.



*Benefiting government, industry  
and the public through innovative  
science and technology*

Equal Opportunity Employer M/F/D/V  
Committed to Diversity in the Workplace