Argonne NATIONAL LABORATORY

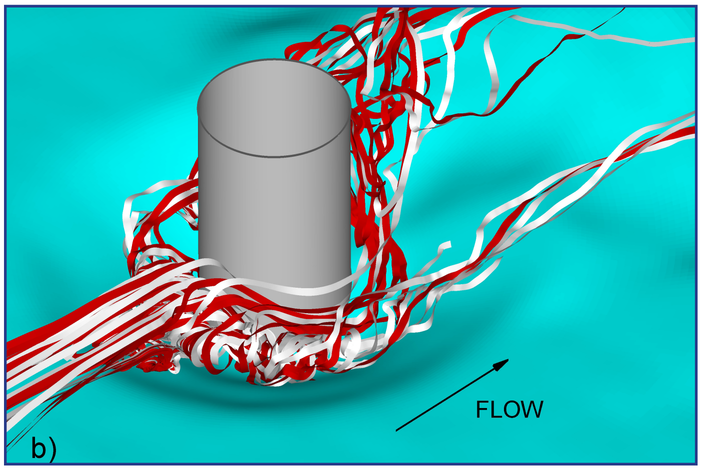
Computational Fluid Dynamics Analysis Applied to Transportation Research

TRACC's use of computational fluid dynamics to improve analysis of the effects of wind and flood forces on bridges and other roadside and waterway hydraulic structures has many benefits. New bridges and other transportation structures can be designed to be more robust, while minimizing cost and maximizing useful life. Maintenance and repair budgets for existing transportation infrastructure can be targeted at structures and projects that will yield the greatest benefits.

Background

Computational fluid dynamics (CFD) research uses mathematical and computational models of flowing fluids to describe and predict fluid response in problems of interest, such as the flow of air around a moving vehicle or the flow of water and sediment in a river. Coupled with appropriate and prototypical experimental results, CFD can be an efficient, cost-effective tool for predicting system response under a broad range of operating conditions.

CFD is widely used in many industries, including transportation, power, bioengineering, weather forecasting, homeland security, defense, etc. In the transportation field, CFD is used in the design and analysis of vehicles (including autos, buses, trucks, trains and aircraft) and transportation system components, such bridges, signs, traffic signals, and other road and roadside structures and waterway hydraulic structures. The response of components of the transportation infrastructure to air and water flow, for example under storm conditions with high winds or floods, is of considerable practical interest.



Stream traces of the horseshoe vortex in the scour hole at the base of a bridge pier from a Large Eddy Simulation (courtesy of Dr. George Constantinescu of the University of Iowa).

TRACC's Software

For many U.S. Department of Transportation CFD applications, 3-dimensional (3-D) CFD models are needed to capture accurately the physical conditions and response of interest. Such 3-D models, commonly used in both steady state and transient applications, require substantial computing power. The code developers recognize this requirement and currently provide latest-generation versions of these software tools that run on parallel and massively parallel computers, such as those at TRACC, including the commercially available codes Fluent and STAR-CD.

TRACC's license for CD-adapco's STAR-CCM+ and STAR-CD CFD software suites allows unlimited use of the 512 compute node cores on the high-performance TRACC cluster. TRACC's license for the Fluent CFD software suite allows up to 4 users and a total of 32 CPU cores to be used simultaneously. Both of these commercial CFD software suites have a large industrial user base, strong technical support, and are under continuous development to improve and extend their modeling capabilities.

For Users

Scripts for this software have been developed to partially automate the process of submitting CFD jobs to the TRACC cluster job queue. For new users, detailed information on setting up their TRACC environment for use of CFD software is posted on the TRACC wiki (**https://wiki.anl.gov/tracc**).

TRACC's expert staff is available to assist in the use of the computing facilities and in the use of the software analysis tools. TRACC training courses focus on the application of the CFD software to the analysis of problems in hydraulics and transportation infrastructure. These are held periodically at TRACC and include remote participation.

Current Project

Over time or during major flood events, the erosion of riverbed material, or scour, can undermine the pier and abutment bridge support structures and cause bridge failure. About half a million bridges in the National Bridge Registry are over waterways, and over 85,000 of these (17 percent) are considered vulnerable to scour. About 7,000 (1.5 percent) of these are classified as "scour critical" based on the Federal Highway Administration scour analysis guidelines. Scour critical means the bridge is likely to fail in a major flood event. Of over 1,000 bridge failures over the past 30 years, about 60% were caused by scour. That's an average of 20 bridge failures per year due to scour, and the number in recent years has been larger due to the aging bridge infrastructure in the U.S.

TRACC analysts are validating computational practices that address the transportation community's research. CFD simulations are being applied to hydraulics and aerodynamics in the transportation infrastructure, including the assessment of lift and drag forces on bridge decks when flooded; optimization of bridge deck shapes to minimize pressure scour; analysis of sediment transport and its influence on scouring; evaluation of active or passive scour countermeasures to mitigate the damage; assessment of wind damage to signs and roadside structures; and environmental issues such as fish passage through culverts.

For futher information, contact

Steven Lottes

TRACC Simulation, Modeling, and Analysis Leader

630.578.4251

slottes@anl.gov