Argonne NATIONAL LABORATORY

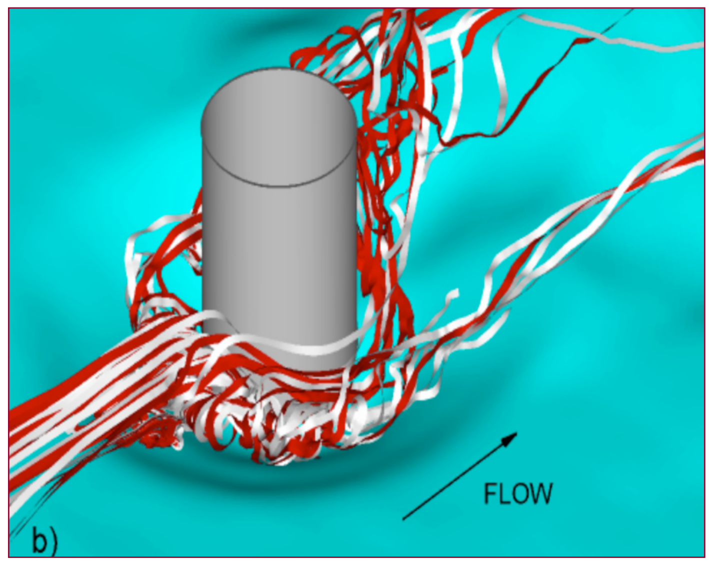
Computational Fluid Dynamics Analysis Applied to Transportation Research

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 to predict the details of fluid flow and its effects on structures in the flow and interactions with boundary surfaces. Examples include determination of drag forces on road signs and other hardware generated by wind or the effects of the flow of water and sediment in rivers on bridges, culverts and other waterway structures. After validation against experimental results, CFD can be an efficient, cost-effective tool for predicting the effects of flow under a broad range of conditions, including storms and floods that are difficult to reproduce in a laboratory.

CFD is widely used in many industries, including transportation, power, bioengineering, weather forecasting, homeland security, and defense. 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 as bridges, signs, traffic signals, and other 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. Knowledge gained using CFD analysis can be used to improve the safety and useful life of transportation infrastructure designs and a variety of other purposes such as adjusting culvert design to allow for easy fish passage.



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 applications, 3-dimensional (3-D) CFD models can provide the most accurate and detailed analysis results for use by engineers and planners. Such 3-D models, commonly used in both steady-state and transient analysis, require substantial computing power. The latest-generation versions of CFD software tools run on parallel and massively parallel computers, such as those at TRACC, including the commercially available CFD software packages FLUENT, STAR-CD, and STAR-CCM+. TRACC's license for CD-adapco's STAR-CCM+ and STAR-CD CFD software allows unlimited use of all the compute processors on the TRACC cluster. TRACC's license for the ANSYS FLUENT CFD software suite allows a total of up to 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

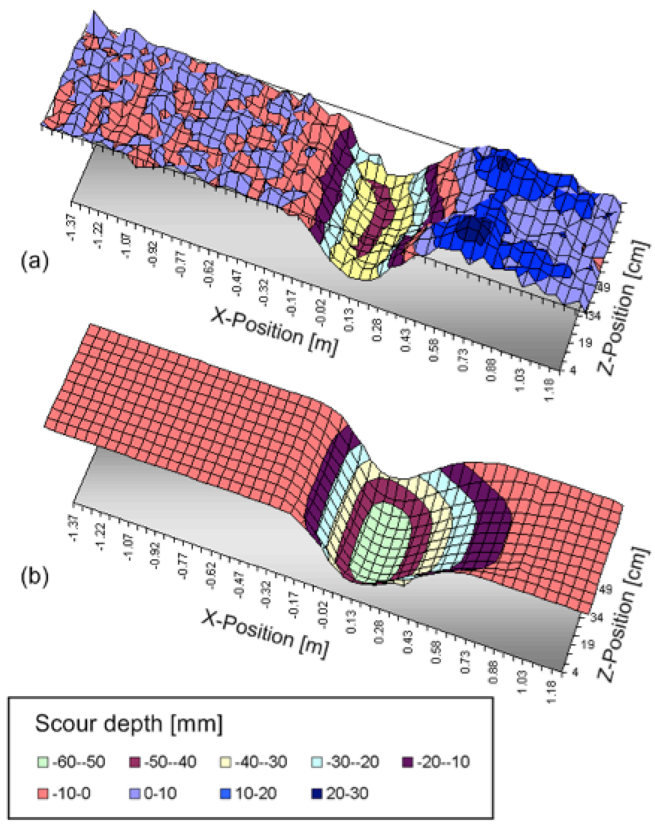
Scripts for CFD software have been developed to partially automate the process of submitting CFD jobs to the TRACC cluster job queue. Detailed information for users on setting up the TRACC cluster environment for use of CFD software is posted on the TRACC wiki (https://wiki.anl.gov/tracc), including instructions for interactive pre and post processing of large data sets using software with a graphical user interface (GUI).

TRACC's expert staff is available to assist in the use of the computing facilities and software analysis tools. TRACC training courses focus on the application of the CFD software to the analysis of problems in hydraulics and transportation infrastructure. Courses are held periodically at TRACC and include participants at remote locations using Internet2 videoconferencing and web meeting software.

Current Projects

Working with the Federal Highway Administration and university collaborators, TRACC analysts are validating computational practices that address the transportation community's CFD analysis needs. CFD simulations are being applied to hydraulics and aerodynamics problems in transportation infrastructure, including the assessment of lift and drag forces on bridge decks during floods; evaluation of scour risk at bridges; assessment of wind damage risk to roadside hardware; and environmental issues such as fish passage through culverts.

Over time or during major flood events, the erosion of riverbed material, or scour, can undermine pier and abutment bridge support structures and cause bridge failure. About half a million bridges in the National Bridge Registry are over waterways and more than 85,000 of these (17%) are considered vulnerable to scour. About 26,000 (5%) of these are classified as "scour critical" based on the Federal Highway Administration's scour analysis guidelines. Scour critical means the bridge is likely to fail in a major flood event. Of more than 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.



3-D scour map for a laboratory scale scour hole beneath a flooded bridge deck (a) Experimental (b) Numerical simulation

For further information, contact
Steven Lottes, PhD
TRACC Simulation, Modeling, and Analysis Leader
630.578.4251
slottes@anl.gov
www.tracc.anl.gov