COMPUTATIONAL FLUID DYNAMICS (CFD) MODELS: CASE STUDIES IN WATER RESOURCES MANAGEMENT

Erin Lincoln, Rene Camacho, and Brian Watson

AFFILIATION: Tetra Tech, Inc., Atlanta, GA 30339
REFERENCE: *Proceedings of the 2017 Georgia Water Resources Conference*, held April 19-20, 2007, at the University of Georgia

Computational fluid dynamics (CFD) models are powerful tools that solve complex fluid flow problems described by Navier-Stokes equations. The computational-intensive models provide accurate and detailed hydraulic information similar to results obtained from physical models, and can be used to predict and optimize design performance. CFD models are used by Tetra Tech for a wide variety of hydrology and hydraulic studies, including restoration designs, fish bypasses, contaminant plumes, settling tanks, and dam failures. Three case studies will be presented. Stream Restoration Design: The City of San Antonio is restoring a concrete channel back to natural conditions at Lackland Airforce Base. A CFD model was developed to evaluate flow velocities and shear stresses under different flow conditions to support and improve the design of a stable channel configuration. Fish Passage Design: The U.S. Army Corps of Engineers (USACE), Savannah District, designed a fish passage at the New Savannah Bluff Lock and Dam in support of the Savannah Harbor Expansion Project (SHEP). A CFD model was developed to ensure that the fish passage design met USACE's required criteria for minimum depths and maximum velocities to allow for movement of a wide variety of fish species. Using CFD model results, the fish passage channel was optimized to ensure proper velocities for a range of flows. Mixing Zone Analysis: As part of SHEP, the Savannah River navigational channel is being deepened, which is expected to decrease dissolved oxygen (DO) in the system. In order to maintain current DO levels, DO injection systems will inject high-oxygenated water back into the river at three locations using Speece Cones. CFD models were developed to investigate the evolution of the solute plumes, both in location and concentration, in the river under various flow and tidal conditions. Preliminary CFD model results will be presented.

Program reference: 6.6.2