Mixing Studies with Tracer of a Water Treatment Storage Tank Using Computational Fluid Dynamics

Student Researcher: Pete Williams, Kennesaw State University
Research Adviser: Dr. Tien Yee, Ph.D., Kennesaw State University

In numerous water treatment systems, storage tanks installed with the express purpose of maintain the quality of the water placed inside of them, more often than not, will deteriorate over time. One source of the deterioration of water quality is bacteria growth. To combat this, disinfecting chemicals, such as chlorine, are added to the water. Thus, good mixing between water and chlorine is important to ensure that the chlorine spreads uniformly within the storage tank. The numerical simulation technique known as *computational fluid dynamics* (CFD) is a tool that can help simulate and assess the mixing conditions inside of a storage tank. For this research project, a custom CFD code was applied to model the flow and concentration of a tracer within a three-dimensional circular water storage tank with jet nozzles as its mixing mechanism.

The CFD code employs the Reynolds Averaged Navier-Stokes (RANS) equations with the k-ε closure model. The tracer concentration, modeled using the passive scalar transport equation, was used to quantify the mixing that occurs between the chlorine and water inside of the tank. For each design configuration, the velocity magnitudes, and coefficient of variance of the tracer concentration were calculated and compared. The preliminary results of the different design jet configurations will be presented at the conference.

Keywords: CFD, water storage tank, mixing, tracer studies