A CFD SIMULATION PROCESS FOR FAST REACTOR FUEL ASSEMBLIES

Kurt D. Hamman, Ray A. Berry

Idaho National Laboratory
P.O. Box 1625
Idaho Falls, Idaho, 83415-3840, USA

Extended Abstract

Recent advances in technology and the need for high fidelity computational data have resulted in research efforts focused on the integration of multi-physics phenomena and the development of new and improved numerical solvers and physical models. These contributions are expected to significantly reduce the uncertainty and increase the accuracy of modeling and simulation tools, allowing scientists and engineers to better predict physical phenomena such as critical heat flux (CHF) and reduce the uncertainty associated with hot channel factors. Consequently, simulation size will increase requiring more computer memory, larger data storage, faster data processing and transfer, and improved visualization tools. Unfortunately in the United States, minimal emphasis has been placed on the simulation of large production-type problems using state-of-the-art commercial software and hardware. Large-scale simulations have the potential of providing insight into the problems and challenges that can be expected upon implementing advanced modeling and simulation tools in production codes and running these large-scale simulations on advanced computing architectures. The purpose of this study was to develop and evaluate a streamlined computational fluid dynamic (CFD) simulation process for large problems using an arbitrary fuel assembly design as a benchmark problem. Three dimensional flow distributions of sodium for several fast reactor fuel assembly pin spacing configurations were simulated using commercial CFD software and high performance computers (HPCs). This study focused on a 19-pin fuel assembly, similar in design to the Advanced Burner Test Reactor (ABTR), with wire-wrap spacers and one helical pitch. Several two-equation turbulence models including the k-ε and SST (Menter) k-ω were evaluated. Considerable effort was taken to resolve the momentum boundary layer, so as to eliminate the need for wall functions and reduce computational uncertainty. High performance computers were required to generate the hybrid meshes needed to predict secondary flows created by the wire-wrap spacers; computational meshes ranging from 65 to 100 million elements were common. A general validation methodology was followed, including mesh refinement and comparison of numerical results with dated empirical correlations provided in the literature. Predictions for velocity, pressure distribution, and secondary flows are shown. The inherent uncertainty of numerical models, importance of high fidelity experimental data, and the challenges associated with simulating and validating large production-type problems are discussed.