CFD Methodology and Validation for Single-Phase Flow in PWR Fuel Assemblies

Nuclear Engineering and Design
Pages:
2088-2095
Volume:
240
Issue:
9
Date:
Wednesday, September 1, 2010
DOI:
http://dx.doi.org/10.1016/j.nucengdes.2009.11.031

This paper presents the CFD modeling methodology and validation for steady-state, normal operation in a PWR fuel assembly. This work is part of a program that is developing a CFD methodology for modeling and predicting single-phase and two-phase flow conditions downstream of structural grids that have mixing devices. The purpose of the mixing devices (mixing vanes in this case) is to increase turbulence and improve heat transfer characteristics of the fuel assembly. The detailed CFD modeling methodology for single-phase flow conditions in PWR fuel assemblies was developed using the STAR-CD CFD code. This methodology includes the details of the computational mesh, the turbulence model used, and the boundary conditions applied to the model. The methodology was developed by benchmarking CFD results versus small-scale experiments. The experiments use PIV to measure the lateral flow field downstream of the grid, and thermal testing to determine the heat transfer characteristics of the rods downstream of the grid. The CFD results and experimental data presented in the paper provide validation of the single-phase flow modeling methodology. Two-phase flow CFD models are being developed to investigate two-phase conditions in PWR fuel assemblies, and these can be presented at a future CFD Workshop.

Author Name:
M. Conner
E. Baglietto
A. Elmahdi
Industries:
Energy[1]
Products:
STAR-CD®[2]

software, support and services. We have over 30 years of experience in delivering industrial strength engineering simulation.
