CFD analysis of PWR core top and reactor vessel upper plenum internal subdomain models

Nuclear Engineering and Design
Pages:
4181-4193
Volume:
241
Issue:
10
Date:
Saturday, October 1, 2011
DOI:
http://dx.doi.org/10.1016/j.nucengdes.2011.08.007

One aspect of the Westinghouse AP1000?[1] [1] reactor design is the reduction in the number of major components and simplification in manufacturing. One design change relative to current Westinghouse reactors of similar size is that AP1000 reactor vessel has two nozzles/hot legs instead of three. With regard to fuel performance, this design difference creates a different flow field in the reactor vessel upper plenum. The flow exiting from the core and entering the upper plenum must turn toward one of the two outlet nozzles and flow laterally around numerous control rod guide tubes and support columns. Also, below the upper plenum are the upper core plate and the top core region of the 157 fuel assemblies and 69 guidetube assemblies.

To determine how the lateral flow in the top of the core and upper plenum compares to the current reactors a CFD model of the flow in the upper portion of the AP1000 reactor vessel was created.

Before detailed CFD simulations of the flow in the entire upper plenum and top core regions were performed, conducting local simulations for smaller sections of the domain provided crucial and detailed physical aspects of the flow. These sub-domain models were used to perform mesh sensitivities and to assess what geometrical details may be eliminated from the larger model in order to reduce mesh size and computational requirements. In this paper, CFD analysis is presented for two subdomain models: the top core region and control rod guide tube region. These models are chosen for simulation because guide tube and top core region (including top grid, top nozzle, and hold-down device) are the major components of upper plenum effecting the flow patterns and pressure distribution.

The top core region, corresponding to ¼ of fuel assembly, includes components as upper part of the fuel assemblies (top grid, fuel rods, top nozzle), core component hold-down devices, and upper core plates. These components distribute the core flow to different sections of guidetube regions. Mesh sensitivity studies have been conducted for each individual part in

order to determine the proper geometrical simplifications. Pressure drop measurement data are compared with the predicted CFD results and act as a guideline for the mesh selection.

The guidetube region includes control rod guidetubes themselves, adjacent support columns and open regions. In this study, two models of subdomains are analyzed: (1) a ¼ section of one control rod guide tube by itself and (2) a representative unit cell containing two ¼ sections of adjacent control rod guide tubes and one ¼ section of a neighboring support column.

Predicted flow rates at each of the outflow locations in conjunction with results from the mesh sensitivity studies provide guidance on (1) what geometry to preserve or remove, (2) what geometry can be simplified to reduce the required mesh, and (3) an estimate of the total mesh required to model the entire upper plenum and top fuel domain.

The commercial CFD code STAR-CCM+ is employed to generate the computational mesh, to solve the Reynolds-averaged Navier?Stokes equations for incompressible flow with a Realizable *k*?? turbulence model, and to post-process the results.

Author Name:
Min-Tsung Kao
Chung-Yun Wu
Ching-Chang Chieng
Yiban Xu,Kun Yuan
Milorad Dzodzo
Michael Conner
Steven Beltz
Sumit Ray
Teresa Bissett
Industries:
Energy[2]
Products:
STAR-CCM+®[3]

Links:
[1] http://www.sciencedirect.com/science/article/pii/S0029549311006327#ref_fn0005
[2] http://www.cd-adapco.com/industries/energy
[3] http://www.cd-adapco.com/products/star-ccm%C2%AE