Analysis of Flow Distribution in a PWR Reactor Vessel Using CFD Code

Ji Hwan Jeong, Byoung-Sub Han

Dept. of Environmental System, Baekseok Col., Anseo-dong, Cheonan, 330-705 Korea, jhjeong@ccfs.ac.kr
ACT Ltd, #402 1688-5, Sinil-dong, Daedeok-gu, Daejeon, 306-230 Korea, hanbs@actbest.com

INTRODUCTION

Nuclear regulatory bodies require the vendors report lots of simulation results in order to ensure the safe operation of nuclear power plants (NPPs). In general, the simulations are carried out using vendor-specific design codes and best-estimate system analysis codes. Currently, MARS, RELAP, TRAC and CATHARE are used as a best-estimate system analysis code. Most of these system analysis codes were developed based on 1-dimensional lumped parameter models because the computing resources were expensive when these codes were under development. During the past decade, however, computing power has been dramatically enhanced in terms of speed, capability and expenses. Mechanistic 3-dimensional computational fluid dynamics (CFD) codes also made a big progress during this period. In spite of this progress, the codes that are used for safety analysis and design of NPPs are still the same. It is believed to be beneficial to take advantage of advanced commercial CFD codes in safety analysis and design of NPPs.

The present work has been initiated aiming to investigate the flow distribution in downcommer and lower plenum of Korean standard nuclear power plants (KSNPs). The KSNP has two steam generators. Each steam generator is connected to the reactor vessel (RV) through one hot-leg and two cold-legs. Since a RCP is installed in each cold-leg, the KSNP has four RCPs. The real flow distribution in reactor core is one of major safety concerns. In addition, if asymmetric flow field develops due to any reason such as a RCP trip, the information of real flow distribution at the inlet of the reactor core becomes very critical in evaluation of safety margin. In spite of these concerns, however, it is not appropriate for the conventional 1-D system analysis codes to analyze real flow distribution.

STAR-CD, a widely used commercial CFD code, is used in the present work. Flow field is analyzed aiming to figure out the flow distribution and hydraulic head loss in KSNP reactor vessel.

NUMERICAL MODEL OF A KSNP REACTOR

The geometry of lower plenum is very complicated because there are so many reactor internals including flow baffle, lower support structure, flow distributor, and instrument guide tubes. Figure 1 shows a quarter of the KSNP reactor vessel.

These complicated reactor internals makes it almost impossible to build geometry of calculation domain and mesh. The present work takes advantage of 3D CAD data for the KSNP. In the beginning, the geometries of reactor vessel and internals were modeled using 3D CAD package. Next, the calculation domain is produced from the CAD data. Then, the geometry data of the calculation domain is imported into a mesh generation tool. The average size of cells is about 2 inches and minimum 16 edges are made around a circle. These criteria generate 4.5 million tetrahedral cells.

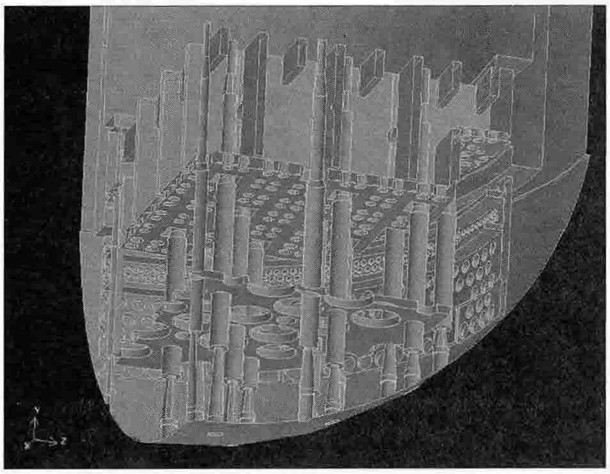

Fig. 1 Geometry of a KSNP lower plenum

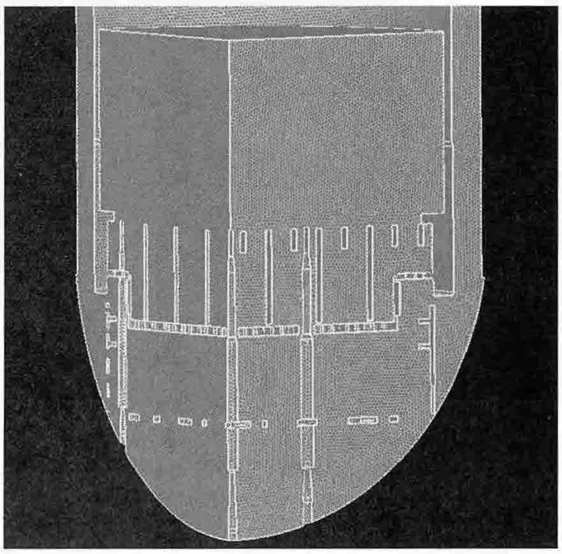

Fig. 2 Schematic of tetrahedral cells

SIMULATION

In order to get figures about fluid motion in the KSNP reactor vessel, numerical simulations are performed using STAR-CD Ver.3.20. In general, this CFD package solves mass conservation, momentum conservation, and energy conservation equations for models. SIMPLE algorithm and second-order upwind schemes for the convection terms are used. The standard k-e model is used in order to consider turbulence which mix transported quantities such as momentum. In general, the near-wall modeling significantly affects the results since the walls are the main source of vorticity and turbulence. In the present simulations, the viscosity-affected inner region is not resolved, instead is bridged by a wall function. Since a quarter of the reactor vessel is modeled, one RCP exit flow rate and pressure are used as the input boundary condition for the cold leg inlet nozzle. As the pre-core condition is simulated, the core region is empty space. The empty core region should be modeled in order to use 'outlet' condition which can be used at fully developed region.

RESULTS

The flow field and pressure distributions in downcomer and lower plenum are examined. The flow field data can be used to estimate flow distribution at the bottom end of Rx core. The head loss data is translated into an average loss coefficient and can be utilized in k-factor evaluation for lower plenum of KSNP.