Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, November 2-3, 2006

A CFD Analysis of Complex Flow Distribution in KSNP Reactor Vessel Lower Structures Based on 3D CAD

Gi-Yong Kim,a Chang-Keun Yang,b Ji-Hwan Jeong,c
a ENESYS Co.,Ltd.,337-2, Jangdae-dong, Yusong-Gu, Daejon, 305-380, Korea, gykim@enesys.co.kr
b Korea Electric Power Research Institute.,103-16 Munji-Dong, Yusong-Gu, Daejon, 305-380, Korea.
c School of Mechanical Engineering Pusan National Univ., Busan.,609-735 Korea.

1. Introduction

During design period and commercial operation of a nuclear power plants (NPP) lots of safety analyses are performed because nuclear regulatory body requires the vendor and utility to report lots of simulation results in order to ensure the safe operation of the NPP. In general, the simulations are carried out using vendor-specific design codes and best-estimate system analysis codes. The thermal-hydraulic system codes have powerful features such as multi-phase flow model, phase-change model and event programming. During the past decade, however, computing power has been dramatically enhanced in terms of speed, capability and expenses. On the other hand, mechanistic computational fluid dynamics(CFD) codes also made a progress during these days. Nowadays, commercial CFD programs are applied to very large and complex systems design such as core design, HVAC design and chemical plant buildings. In spite of the recent progress in computing hardware and software the nuclear industry still uses conventional system codes based on lumped parameter model. It is believed to be beneficial to take advantage of advanced commercial CFD codes in safety analysis and design of NPPs.

During the past decade commercial CFD codes have been applied to simple NPP geometry aiming to examine local thermo-hydraulic phenomena such as the safety injection flow in downcomer, turbulence due to mixing vane, boron mixing and local sub-channel analysis. In spite of the various efforts of commercial CFD code applications to NPP safety issues with classical k-e model, advanced numerical methods such as LES and DNS have not applied to nuclear reactor systems yet. The current status and needs in commercial CFD codes usage for NPPs safety analyses are well described in Yadigaroglu et al.'s paper.

The aforementioned applications of commercial CFD codes to NPP safety analyses have been made with relatively simple or simplified calculation geometry. To the authors' literature survey, there has been no CFD analysis for the flow in lower plenum and upper plenum. In our previous study, analysis which aims to analyze the flow distribution in downcomer and lower plenum of Korean standard nuclear power plants (KSNPs) are performed with quarter core geometry to evaluate steady state flow conditions[1]. Also, in present study, flow transient in core such as steam line break are simulated. If SLB have happened, core flow transient would have half symmetry flow patterns. Thus, the flow symmetry analysis in reactor vessel should have meanings in safety analysis. The results will give a clear figure about flow distribution in reactor vessel, which is one of major safety concerns. This result also can be used in precise estimation of hydraulic head loss factors, k-factors, for 3-dimensional thermal-hydraulic system analysis codes.

2. Numerical Model

Figure 1 shows the 3D CAD drawing of the PWR lower plenum. The geometry of it is very complicated because there are so many reactor internals including flow skirt, lower support structure, flow distributor, and instrument guide tubes. These complicated reactor internals made it almost impossible to build geometry of calculation domain and generate mesh so far. The present work takes advantage of 3D CAD data for the PWR in order to make it possible.

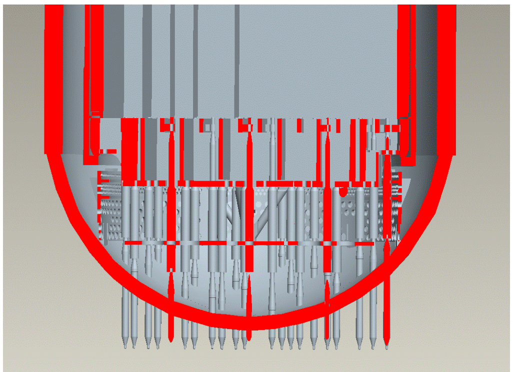

Figure 1. CAD drawing of lower plenum

Resulted unstructured mesh is illustrated in figure 2. Figure 3 illustrates enlarged cross-sectional view of cells around flow distributor just below the lower support structure as well as holes of the flow skirt.

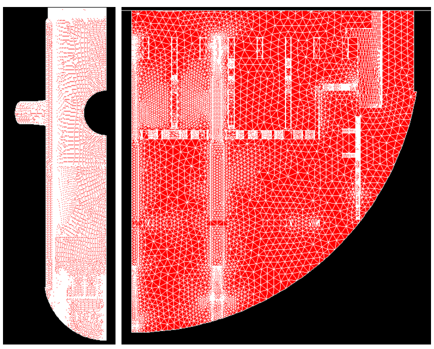

Figure 2. Unstructured mesh

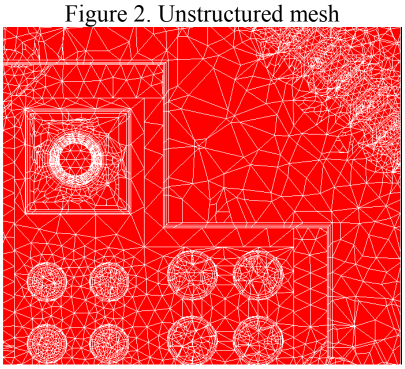

Figure 3. Cells in detail

3. Numerical Simulation

In the present work, a commercial CFD code STAR-CD was used. This is a 3D multi-physics code based on unstructured mesh. Second-order upwind differencing scheme for the convection terms are used. Analyses were performed with SIMPLE algorithm and steady state assumption. The quarter and half geometry calculation domain are modeled to simulate 4 RCP operating steady state and single hot-leg failure condition. Table 1 shows the boundary conditions and initial conditions of each simulation.

Table.1. Boundary conditions and initial conditions

Case domain	Inlet	Outlet	Outlet location
quarter	82500 gpm	2250 psia	Below fuel assembly
half	82500 x 1.3gpm - 82500 x 0.35gpm	2250 psia	Above fuel assembly

Energy equation is not solved so that single-phase flow only is simulated and no buoyancy effect is considered in this simulation.

4. Results

The flow field and pressure distributions in downcomer and lower plenum have been analyzed. Figure 4 show velocity vector seen from the symmetric surface of calculation domain. Figure 5 shows asymmetry velocity magnitude contour in half geometry simulation.

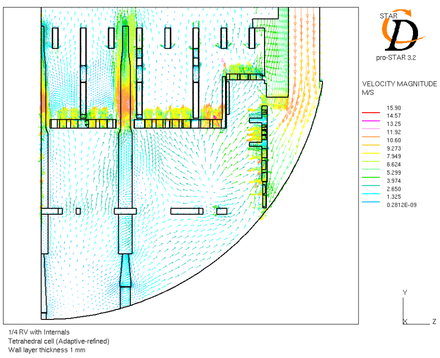

Figure 4. Velocity vector plot in lower plenum

(Full core symmetry with 4 RCP operating)

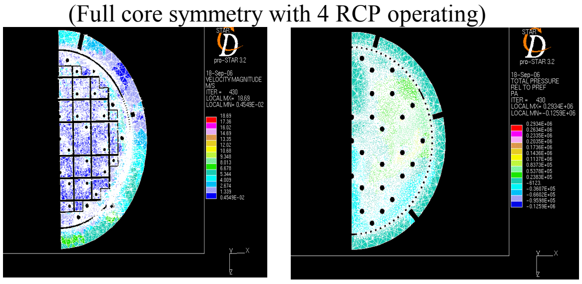

Figure 5. Velocity magnitude contour in half geometry.
(Half core symmetry with 2 RCP operating)

The total pressure of the coolant decreases as it goes to downstream. In order to extract useful information from this result, average piezometric pressure over cross-sections at several levels were calculated. The pressure drops between successive two stations are summarized in Table 2. This table shows that large pressure drop occurs across lower support structure. This work evaluates the total pressure drop between cold-leg nozzle throat and the top of lower support structure as 19.52 psi.

Table 2. Pressure drop (piezometric) through RV

station	ΔP (psi)
1 – 2	5.66
2 – 3	2.76
3 – 4	11.1
1 – 4	19.52

1 : cold-leg nozzle throat
2 : downcomer bottom end
3 : 2” above the flow distributor in the lower plenum
4 : 2” above the lower support structure

5. Conclusion

There has been no CFD analysis for the flow in nuclear reactor’s lower plenum and upper plenum without geometric simplification because the geometries of them are very complicated. The present work took advantage of 3D CAD data for the PWR in order to build geometry of calculation domain for lower plenum and downcomer of PWR. The real geometry of the KSNP was used in the analysis.

A commercial CFD code, STAR-CD, was used to analyze the flow and pressure distribution of single phase coolant in the reactor vessel. The results give a clear figure about flow distribution in reactor vessel, which is one of major safety concerns. The pressure distribution information may be used in precise estimation of hydraulic head loss factors, k-factors, for 3-dimensional thermal-hydraulic system analysis codes.

REFERENCES

[1] Ji-Hwan Jeong and Byoung-Sub Han, proceeding of ICAPP ’05 Seoul, KOREA, May 15-19, 2005. paper 5624.