Transactions of the Korean Nuclear Society Spring Meeting
Chuncheon, Korea, May 25-26 2006

Head Loss Evaluation in PWR Reactor Vessel Real geometry Using CFD Analysis

Jong Pil Park, Ji Hwan Jeong, byoung Sub Han
School of Mechanical Engineering, Pusan National Universit, Jangjeon-dong, Geumjeong-gu, Busan, Email:jihwan@pusan.ac.kr
Enesy, Jangdae-dong, Yusong-gu, Daejeon, Korea, Email:hanbs@enesys.co.kr

1. Introduction

The aforementioned applications of commercial CFD codes to NPP safety analyses have been made with relatively simple or simplified calculation geometry.

The present work aims to analyze the flow distribution in downcomer and lower plenum of Korean standard nuclear power plants (KSNPs). The real geometry is used in the 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 thermal-hydraulic system analysis codes. The STAR-CD, a widely used commercial CFD code, is used in the present work.

2. Numerical Model

As the lower plenum governs the coolant supply to each fuel assembly in the reactor core, it is very important to have a clear picture of flow behaviour inside it with minimum uncertainty, This can be achieved by CFD analyses with non-simplified real geometry. Fig. 1 shows the 3D CAD drawing of the PWR lower plenum. In this work, a quarter of the KSNP reactor vessel and internals from cold-leg inlet nozzle to lower support structure is taken into account. The upper plenum nor the fuel assembly were not considered. An empty space is assumed to be placed above the top-end of the lower support structure just for reliable numerical simulation.

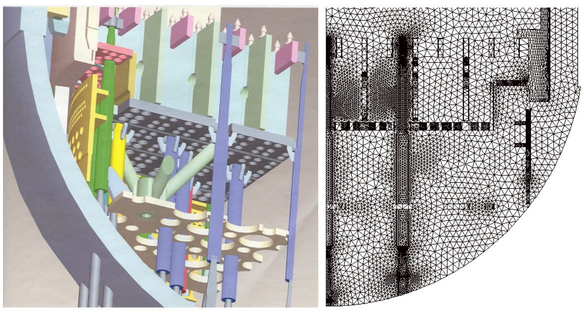

Fig. 1. CAD drawing & Unstructured mesh of lower plenum

2.1 Geometry and Mesh

The average size of cells is about 1 inch and minimum 16 edges are made around a circle. These criteria generated more than 3.3 million unstructured cells. During CFD analysis adaptive cell refinement is performed based on gradient of variables. This process increased the cell number to around 4.5 M cells.

2.2. Turbulence Model

In this work, Craft model (Craft and Launder, 1991) was selected for the term since there are impinging flows against reactor internals. Default values were used for various coefficients for the RSM model and the standard wall function was used to treat wall boundary layer. Based on interim simulation results such as velocity gradient and y+ distribution, cells were refined. Consequently, y+ values were less than 140 when the results described in section 3 are obtained.

2.3 Numerical Simulation

In the present work, a commercial CFD code STAR-CD Version 3.22 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. "Inlet" boundary condition was applied to cold-leg inlet nozzle. The velocity at this location was set based on cold stand-by test condition; mass flow rate = 2250 gpm. The static pressure at this location of 153.0e5 Pa was used as reference value. "Outlet" boundary condition was applied to the imaginary exit that was extruded vertically upward from the top-end of the lower support structure. Both sides of calculation domain that confines the quarter volume of a rector are assigned symmetric boundary condition. Energy equation is not solved so that single-phase flow only is simulated and no buoyancy effect is considered in this simulation.

3. Analysis Results

3.1 Flow Distribution

The flow field and pressure distributions in downcomer and lower plenum have been analyzed. A contour plot for velocity magnitude in the downcomer and lower plenum is illustrated in Fig. 2. Supplied coolant jet impinges onto the inner end of calculation domain (core support barrel) and flows downward. This contour also shows a non-uniform downward coolant flow in the downcomer. A low flow rate region develops below the cold-leg inlet nozzle. Considerable part of coolant appears to flow away from this region.

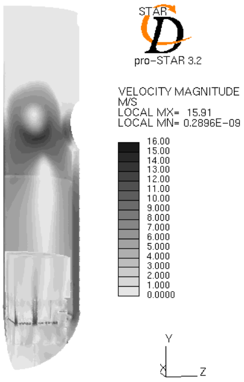

Fig. 2. Velocity magnitude contour in downcomer and lower plenum of a quarter reactor vessel

3.2. Pressure Loss

The total pressure, static pressure plus dynamic pressure, at the surface of calculation domain is plotted in fig. 3. This plot shows that the total pressure of the coolant decreases as it goes to downstream. Furthermore, the pressure drop across flow skirt and flow plate just below the support beams appears to be significant. The pressure drops between successive two stations are summarized in Table 1. 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 19.5 psi.

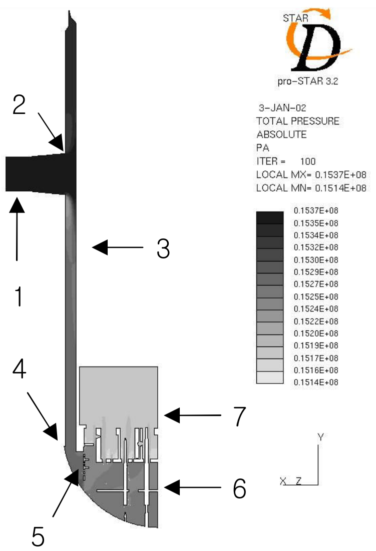

Fig. 3. Total pressure contour

TABLE 1 Pressure drop (piezometric) through RV

station	Δ P (psi)
1 – 4	5.65
4 – 6	2.75
6 – 7	11.10
1 – 7	19.5

3.3 Pressure Loss Coefficient

In order to evaluate k-factor, the head loss due to form loss needs to be estimated. Fig. 4 shows the dynamic head of a representative stream line and the accumulated head loss. The location in the reactor vessel is presented by numbers above this plot which correspond to the numbers given in fig. 3.

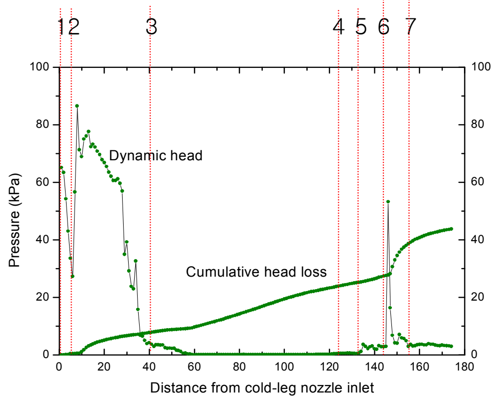

Fig. 4. Cumulative head loss

When a section is selected the cumulative head loss across the section is estimated from the fig. 4. This head loss value and cross-sectional flow area of the section should be put in the above equation to get k-factor.

4. Concluding Remarks

The present results show that it is practically possible to perform CFD analysis with real geometry of nuclear reactor using small computer resources. This approach will be a help to estimate hydraulic head loss factors, k-factors, for system analysis codes, flow distribution at the bottom end of reactor core, and effects of asymmetric operation of RCPs.

REFERENCES

[1] Z. Zhai, Q. Chen and P. W. Scanlon, “Design of Ventilation System for an Indoor Auto Racing Complex,” Proc. of the ASHRAE Winter Meeting, ASHRAE, Atlantic City, U.S.A. (2002).
[2] A. J. Baker, K. L. Wong and N. S. Winowich, “Design and Assessment of a Very Large Scale CFD Industrial Ventilation Flowfield Simulation,” Proc. of the ASHRAE Winter Meeting, ASHRAE, Atlantic City, U.S.A. (2002).
[3] T. Kwon, C. Choi and C. Song, “Three-dimensional analysis of flow characteristics on the reactor vessel downcomer during the late reflood phase of a postulated LBLOCA,” Nuclear Engineering and Design **226**, 255-265 (2003).