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

Head Loss Coefficient Evaluation Based on CFD Analysis for PWR Downcomer and Lower Plenum

Ji Hwan Jeong[1*], Jong Pil Park[1], and Byoung Sub Han [2]

[1] *School of Mechanical Engineering, Pusan National University, Jangjeon-dong, Geumjeong-gu, Busan, Korea*
Tel:+82-51-510-3050 , Fax:+82-51-512-5236 , Email:jihwan@pusan.ac.kr
[2] *Enesys, Jangdae-dong, Yusong-gu, Daejeon, Korea*
hanbs@enesys.co.kr

1. INTRODUCTION

To the authors' literature survey, there has been no CFD analysis for the flow in lower plenum and upper plenum without geometric simplification. 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. 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.

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 5 M cells.

2.2 Turbulence Model

Commercial CFD codes usually solve the Reynolds averaged Navier-Stoke's equations (RANS models) for turbulence simulation and they provide users with various turbulence models ranging from 0-equation model to large eddy simulation model (LES). This work focused on RANS models including the standard k-ε model, quadratic and cubic k-ε models, the renormalization-group (RNG) variant, and RSM model. 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 III 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. The simulations were performed on a Linux cluster. This cluster computer consists of one master node and two slave nodes.

3. ANALYSYS RESULTS

Before start having a look at analysis results and discussion, mesh sensitivity needs to be checked out. Fig. 1 shows the velocity magnitude profiles along a horizontal line that pass through between lower support structure and flow plate. We can see there is a little bit difference between the results from 3.3M cells and 4.2M cells. This difference seems to be negligible when it comes to 4.2M cells and 5.0M cells.

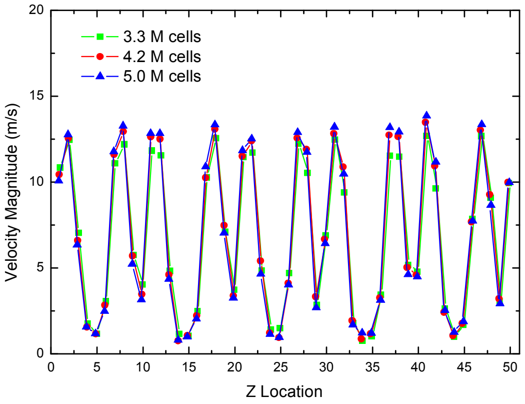

Fig. 1. Mesh Sensibility

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.

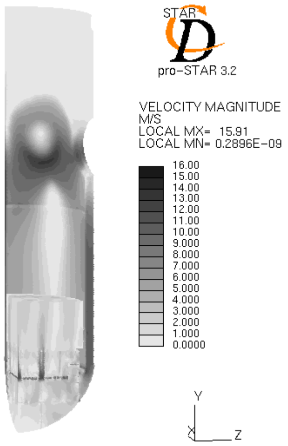

Fig. 2. Velocity magnitude contour in downcomer and lower plenum of a quarter reactor vessel

3.2 Pressure Loss

This work evaluates the total pressure drop between cold-leg nozzle throat and the top of lower support structure as 19.5 psi. The pressure drop across the same station was estimated as 18 psi in engineering calculation note.

3.3 Head Loss Coefficient

It is believed that the present CFD analysis results can be used in k-factor generation. Fig. 8 shows a part of nodalization of the KSNP for an event analysis using RELAP5/MOD3, which corresponds to the present CFD calculation domain. The present 3-dimensional calculation domain was split into sub-domains such that each domain corresponds to a node shown in Fig. 3. Volume-average pressure of each sub-domain, area and area-average velocity at each interface between sub-domains was calculated. Based on these information k-factors were evaluated for each junction in Fig. 8 and listed up in Table 1.

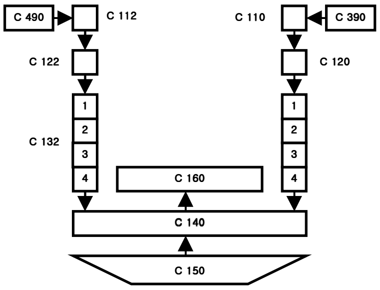

Fig. 3. RELAP5 Nodalization of KSNP

TABLE 1 Calculated k - factor

station	k – factor
490 – 110	0.58
110 – 120	0.69
120 – 130	0.29
130 – 140	2.27
140 - 160	1.38

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).

[4] W. K. In, D. S. Oh and T. H. Chun, “Flow Analysis for Optimum Design of Mixing Vane in a PWR Fuel Assembly,” *J. of the Korean Nuclear Society* 33, 327-338 (2001).