Transactions of the Korean Nuclear Society Spring Meeting
Taebaek, Korea, May 26-27, 2011

CFD analysis for the variation of flow pattern between the real model with holes and simplified model with porous in Steam Generator

Y.G. KIM [a*], Y.M.JEON [a], Y.M. BAE [a], Y.I. KIM [a], C.T. PARK[a], S. CHOI [a]
[a]Korea Atomic Energy Research Institute. P.O.Box 105, Yuseong, Daejeon, 305-600, KOREA
[*]Corresponding author: kimyeonggil@kaeri.re.kr

1. Introduction

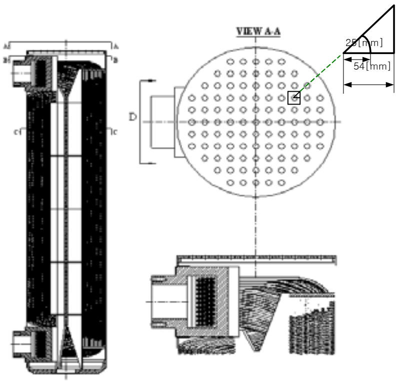
The mass flow distribution to Steam Generators (SG) being developed is one of the great concerns of Reactor Coolant System (RCS) design in SMART. Cover with approximately a hundred of holes is installed upper the each SG and SGs are constituted with many helical tubes in SMART. As computing resources are restricted in many CFD simulations, simplifications for boundary condition or shapes are needed within an acceptable range to simplify problems.

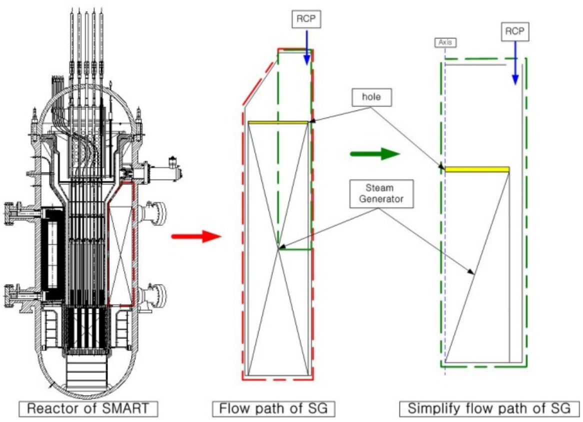
The overall mass flow distribution to SGs will be simulated using porous model [2] in SMART, as the phenomenon is deeply associated with the flow resistances. Before the calculation, we investigate the variation of flow pattern between the real model with holes and simplified model with porous in SG upper region. For a CFD code validation, CFD analysis results are compared to empirical correlations similarly to orifices. Commercial software, FLUENT 12.0 code, is adopted for this numerical analysis.

2. Approaches

2.1 Geometry

Figure.1, 2 and 3 show the geometry of the flow path in SG and 1/8 of reactor from RCP entrance to SG exit.





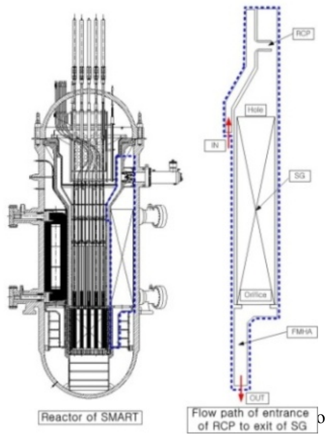


Fig.3 Flo o SG exit

2.2 Empirical correlation

Figure.4 shows the geometry of orifice in diagram 4-15 in reference [4] similar to the flow path of holes in SG.

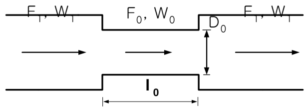


Fig4. Thick-edge orifice in a straight channel

The loss coefficient in the empirical correlation regarding Figure.4 is calculated using following equations (1).

$$\zeta = \frac{\Delta P}{\frac{\rho W_1^2}{2}} = \left[0.5\left(1-\frac{F_0}{F_1}\right)^{0.75} + \tau\left(1-\frac{F_0}{F_1}\right)^{1.375} + \left(1-\frac{F_0}{F_1}\right)^2 + \lambda\frac{l_0}{D_h}\right]\left(\frac{F_1}{F_0}\right)^2 \quad (1)$$

$$\tau = (2.4-\bar{l}) \times 10^{-\phi(\bar{l})}, \phi(\bar{l}) = 0.25 + \frac{0.535\bar{l}^8}{0.05+\bar{l}^8}, \bar{l} = l_0/D_h$$

$$\lambda = \frac{1}{(1.8 \log Re_0 - 1.64)^2}, Re_0 = \frac{w_0 D_0}{\nu}$$

In table 1, the resistance coefficient, shapes, material properties, and operation conditions used in the code validation are summarized.

Table.1 shape and operation condition used in code validation

$D_0(m)$	0.05	$W_0(m/s)$	2.05	$\zeta_1(-)$	70.30
$\ell_0(m)$	0.02	$W_1(m/s)$	0.34	$\zeta_0(-)$	1.99
$F_0(m^2)$	1.96×10^{-3}	$Re_0(-)$	869,000	$\Delta P(kPa)$	2.79
$F_1(m^2)$	1.17×10^{-2}	$Re_1(-)$	357,000	$\dot{m}(kg/s)$	2090
$\rho(kg/m^3)$	670.33	$\lambda(-)$	0.0122	$\mu(kg/m \cdot s)$	7.89×10^{-5}

2.3 Numerical analysis Method

In this numerical analysis, it is assumed that the flow is in steady state and is incompressible. And gravity is not considered and material properties (density, viscosity) are constant.

The Fluent 12.0 code is applied to analyze incompressible Navier-Stokes equation as following formula (2), (3).

-Continuity equation

$$\frac{\partial(u_j)}{\partial x_j} = 0 \quad (2)$$

-Momentum equation

$$\rho\frac{\partial(u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{\partial p}{\partial x_i} \quad (3)$$

Turbulence models utilized in this simulation such as the Realizable *k-ε* (RKE), Renormalization Group *k-ε* (RNG), and Shear Stress Transport *k-ω* (SST) are well explained in reference [5]. The CFD analysis is performed using the standard wall function, second order upwind scheme for momentum and turbulence, standard method for pressure, SIMPLE algorithm as pressure-velocity coupling method, and double precision solver.

2.3 Grid independence

Grid independence tests for the orifice are carried out for all turbulence models; RNG, RKE, and SST. With increase of the numbers of grid, the loss coefficients are converged to constant values as shown in Figure.5. The difference with turbulence models is ignorable in grid tests. So, grid independence test is only conducted for RKE in the SG and 1/8 of reactor simulation.

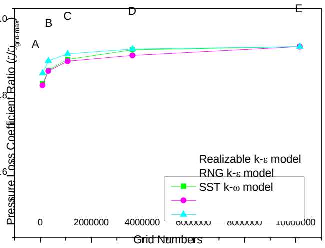


Fig.5 Grid sensitivity

3. Result and Discussion

In comparison between results from the CFD analysis and the empirical correlation, the loss coefficients are very similar to the empirical correlation. The deviation is within 4.5% in D for all turbulence models in SMART normal operation condition.

Table.2 Comparison Empirical Correlation to CFD analysis

	Turbulence model	Grid Number (million)	Value of calculation		Deviation (%)
			$\zeta_1(-)$	ΔP[kPa]	
Empirical Correlation	-	-	70.3	2.79	-
CFD analysis (hole)	RNG k-ε	3.6	67.5	2.69	-3.9
	Realizable k-ε		67.1	2.67	-4.5
	SST k-ω		67.0	2.67	-4.5

Simulation for SG and 1/8 of reactor are performed. Table.3 and 4 indicate that it is possible to substitute the real holes with the porous model for SG cover.

In comparison between results of the real holes and the porous model for SG cover, total pressure loss deviation is within -0.4% in one SG simulation (Figure.7) and the deviation is within 1.0% in the 1/8 of reactor simulation (Figure.8).

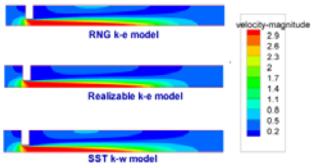
Table.3 Simulation for one SG

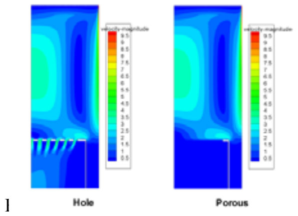
	Section	Value of calculation ΔP[kPa]		Deviation (%)
		Real holes (Grid :26.35million)	Porous (Grid : 13.22million)	
Steam Generator	Hole	5.52	5.50	-0.4

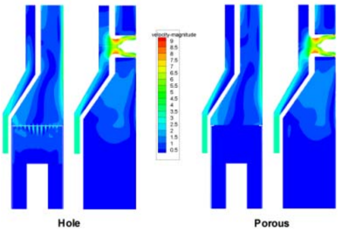
Table.4 Simulation for 1/8 of Reactor

	Section	Value of calculation ΔP[kPa]		Deviation (%)
		Real holes (Grid :98.74million)	Porous (Grid :26.27million)	
1/8 of Reactor	RCP	31.47	31.85	1.2
	Hole	5.26	5.31	1.0
	SG	26.72	26.54	-0.7
	Orifice	21.28	21.23	-0.3
	Total	85.6	86.80	1.3

Figure.6, 7 and 8 show the contours of velocity magnitude for the orifices, one SG and 1/8 reactor including the porous mode simulation. We are able to recognize that the contours for the porous models are very similarly to those for the real holes.







4. Conclusions

Numerical analysis using Fluent 12.0 is conducted to investigate that it is possible to simply SG upper regions using porous model. The loss coefficients for orifices are very close to the empirical correlation. And also, the porous model is proved to be applicable for the SG simulation. Refer to these results, we able to apply the method of porous for the simulation of the mass flow distribution to steam generators in various operating conditions.

Acknowledgement

This study has been performed under a contract with the Korean Ministry of Educational Science and Technology.

Reference

[1] Y.G. Kim, J.K. Kong, Y.M. Kim, Y.J. Jeong, C.T. Park, and S. CHOI, CFD analysis for an axial annular radial diffuser, Transactions of the Korean Nuclear Society autumn meeting, P05D09, 2010.
[2] X.B.Zhang, L.M.Qiu, Z.H.Gan, Y.L.He, CFD Study of a simple orifice pulse tube cooler, Science Direct, Cryogenics 47 (2007) 315-321.
[3] Guohui Gan, Saffa B. Riffat, Pressure Loss Characteristics of Orifice and Perforated Plates, Experimental Thermal and Fluid Science 1997; 14:160-165
[4] I.E. Idelckik, Handbook of Hydraulic Resistance, 3rd, Begell house, 2000, Index 4-15
[5] ANASYS, Inc., Fluent 12.0 Manual, 2009.