Transactions of the Korean Nuclear Society Virtual spring Meeting
May 13-14, 2021

Development of MELCOR Analysis Parameters for a TI-SGTR Accident in the OPR1000

Hyung Seok Kang*, Sung Il Kim, and Kwang Soon Ha
Korea Atomic Energy Research Institute, 989-111, Daedeok-daero, Yuseong, Daejeon, 305-353, Republic of Korea

**Corresponding author: hskang3@kaeri.re.kr*

1. Introduction

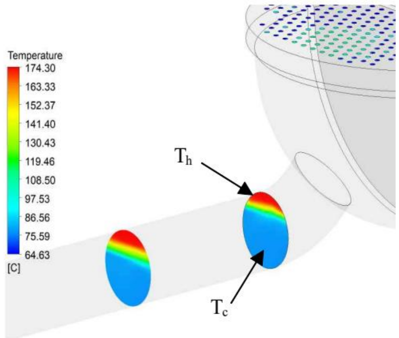
KAERI has been evaluating a release of radioactive materials to an environment from an optimized power reactor 1000 MWe (OPR1000) when a temperature induced steam generator tube rupture accident (TI-SGTR) occurs during a hypothetical severe accident using the MELCOR code [1]. The MELCOR calculation results will be used in considering that the OPR1000 meets regulations related on the severe accident [2]. To perform the MELCOR analysis for the TI-SGTR accident, some parameters representing thermal mixing features of the natural circulation between the reactor and the steam generator should be provided as an input because the MELCOR code is a 1-dimensional parametric analysis code and the natural circulation flow has a 3-dimensional feature. Thus we have performed a 3-dimensional analysis for the natural circulation flow between the hot leg and the SG during the severe accident in the OPR1000 to generate the MELCOR input parameters using a commercial computational dynamic code, ANSYS CFX 19.1 [3].

2. Development of an Analysis Methodology for a Natural Circulation Flow

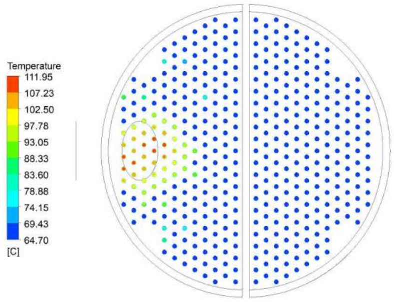
2.1 Analysis Results for the WH 1/7 Scaled-down Test [3]

Westinghouse (WH) performed a series of natural circulation flow experiments between the reactor and steam generators (SGs) during the early stages of severe accidents in a pressured water reactor (PWR) to support the validation of analytical model using a 1/7 scaled-down test facility [4]. To analyze the WH 1/7 test, a 3-dimensional grid model simulating from the hot leg to the SG in the WH 1/7 test facility was developed under assumption of the symmetric flow behavior at the reactor between two steam generators. A total of about 29,025,136 cells with a cell length of 2 - 10 mm were generated in the grid model. The natural convection flow in the SG inlet plenum during the convective flow from the hot leg to SG tubes was solved by applying the mass conservation, momentum conservation with a buoyancy model, energy conservation implemented in the ANSYS CFX 19.1 [3]. A turbulent flow was modeled by the shear stress transport (SST) model with the scalable wall function. A turbulence generation owing to the buoyancy force was included in the turbulence production term of the SST model.

Through this CFD analysis against the test results, we developed a CFD analysis methodology to accurately predict the recirculation ratio, thermal mixing fraction, and discharge coefficient with an error range of approximately ±10% [5]. Though this CFD analysis methodology underestimated approximately 25% for the proposed hot tube number in the SG of the test results, we judged that this discrepancy may not be large when considering uncertainties resulted from 51 measured tube temperatures over all 216 tubes in the test facility.



(a) Temperature at the SG inlet nozzle



(b) Temperature distribution at the tubesheet entrnace
Fig. 1. CFD Results for the WH 1/7 Test [5]

Table 1: Comparison between Test Data and CFD Results [5]

Parameter	**Test**	**CFD**
Heat loss at tubes [kW]	3.56	3.55
Numb. of hot tubes	75	82
Numb. of cold tubes	141	134
T_h in center of the hot leg [K]	432.4	428.0
T_c in center of the hot leg [K]	359.9	352.3
Mass flow rate in center of the hot leg flowing to SG [kg/s]	0.060	0.053
T_{ht} in tubes at tube sheet [K]	373.9	373.5
T_{ct} in tubes at tube sheet [K]	337.8	337.8
m_t in the SG tubes [kg/s]	0.120	0.106
m_t / m (recirculation ratio)	2.01	2.00
f (mixing fraction)	0.85	0.82

2.2 Analysis Results for the SG Design Data [6]

A 3-D SG model was developed and validated on the basis of the OPR1000 design data [7] because the SG model may play an important role in the natural circulation flow between the reactor and the SG. The number of tubes in the SG model was reduced by a ratio of 1/8 and its diameter was increased 3 times compared to the SG design data of the OPR1000. The diameters of the hot leg and cold leg in the grid model were 1066 mm and 762 mm, respectively. The length of the hot leg and cold leg was approximately 5 m. A total of 53,622,290 cells with cell lengths of approximately 0.05-30 mm were generated in the base grid model. The number of cells generated in the SG tubes was 37,717,620 and its cell type was only hexahedra.

To validate the SG model, the pressure drop and heat transfer that occurred when the reactor coolant flowed through the SG tubes during the normal operation was simulated using the mass conservation, momentum conservation, and energy conservation equations in CFX-19.1 [3]. A turbulent flow was modeled by a two-equation model, the SST model. The pressure loss coefficient through the tubes was used in the momentum equation to induce the pressure drop that occurred during the normal operation. In addition, a heat transfer coefficient of 20,000 W/m^2°C was set at the tube outer wall to simulate the convective heat transfer in the calculation of the energy equation.

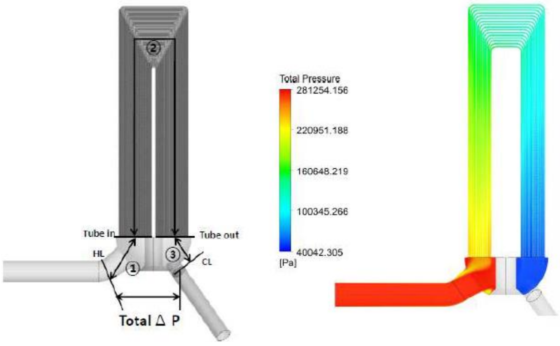


Fig. 2. Pressure Distribution by the CFD analysis using d Standard k-ε Turbulent Model

Table 2: Comparison of Pressure Drop and Heat Transfer between SG Design Data and CFD Results

	SST	**k-ε**	**RSM**
SG total △P [psi]	30.01	31.56	30.66
△P ① (Inlet Plenum)	1.65	3.28	1.51
△P ② (SG Tube)	27.54	27.16	28.41
△P ③ (Outlet Plenum)	0.82	1.12	0.74
Cold Leg Temp. [°F] *Hot Leg Temp. : 621.2 °F	564.3	564.8	564.8

Fig. 2 shows that the pressure distribution and heat transfer takes place when the coolant injected from the inlet condition flows through the hot leg, SG inlet plenum, SG tubes, and SG outlet plenum. Table 2 shows the comparison results of the pressure drop and temperature at the outlet of the cold leg between the CFD results and the design data, which show that the CFD results accurately predict the OPR1000 design data with an error of approximately ±1.5%.

3. CFD Analysis

3.1 Grid Model and Boundary Conditions [6]

A 3-dimensional grid model simulating from the reactor to the SG in the OPR1000 was developed based on the validated SG model (Fig. 3) to analyze the natural circulation flow of the mixture of steam-H_2 in the hot leg and the SG inlet plenum. The end of the cold leg nozzle of the SG was blocked to simulate loop-seal phenomenon during the severe accident. A total of about 63,065,389 cells with tetrahedral, pyramids, wedge, and hexahedra elements were generated in the grid model.

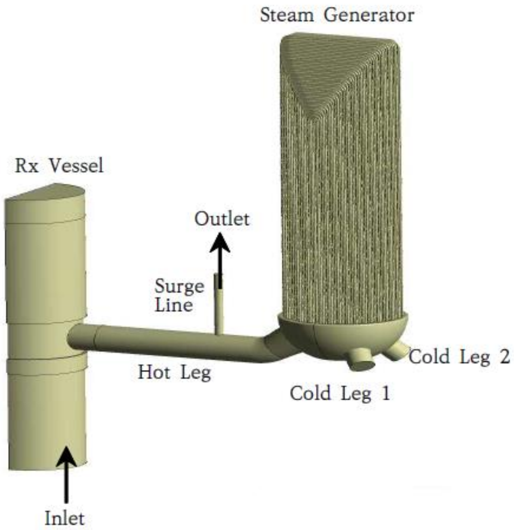


Fig. 3. Grid Model for Simulating Natural Circulation Flow in the Hot Leg and SG Inlet Plenum of OPR1000 [6]

The boundary conditions used for this natural circulation flow are shown in Table 3. These data were obtained from the MELCOR analysis results for the TI-SGTR of OPR1000 [1]. The decay heat generation in the core was not simulated because the purpose of this calculation was only to determine the MELCOR input parameters. To simulate the mixture gas flowing to the pressurizer from the hot leg, the outlet condition was given at the upper region of the surge line. The inlet condition was set at the core inlet to induce the stabilized flow field of the mixture gas in the upper plenum of the reactor vessel. The natural circulation flow field in the hot leg and SG inlet plenum was solved by governing equations, models and methodology applied for the WH 1/7 scaled-down test and the SG model development.

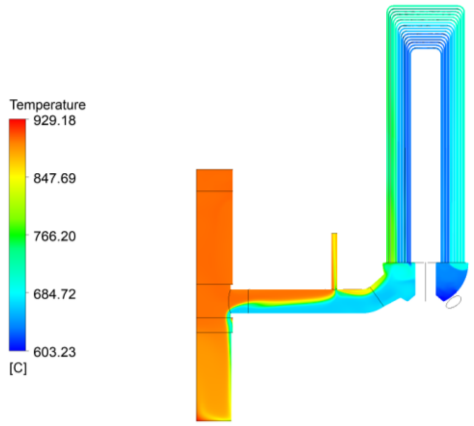
Table 3: Boundary Conditions for Natural Circulation Flow [6]

Inlet	Steam-H_2 mixture gas : 13.24 kg/s, 929.18 °C
Outlet	Zero reference pressure
Wall at SG Tubes	Heat transfer coeff. : 20.37 W/m^2 °C Ambient temp. : 613.75 °C

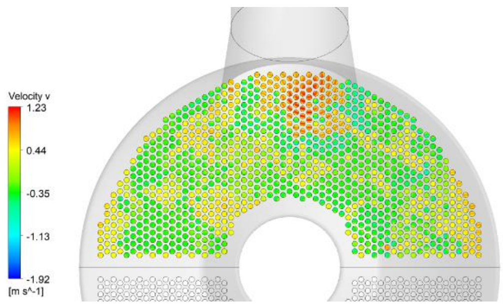
3.2 Discussion on the CFD Results

A steady state calculation using the transient calculation results as the initial condition was performed to obtain the converged solutions through approximately

3000 iterations. The calculation results of the velocity profile and temperature distribution are shown in Fig. 4. Finally, we proposed a mixing fraction of 0.84, recirculation ratio of 1.77, hot tube fraction of 0.44, and discharge coefficient of 0.12 for the MELCOR analysis through this CFD analysis (Table 4). These values are located in the range between parameters of WH SG and Combustion Engineering (CE) SG (Table 5).



(a) Temp. distribution in reactor vessel, hot leg and SG



(b) Velocity distribution at the entrance of SG tubes

Fig. 4. CFD Results of Natural Circulation Flow in the Hot Leg and SG Inlet Plenum of the OPR1000

Table 4: MELCOR Input Parameters from CFD Results

Parameter	**Value**
Recirculation ratio (r) $*r = m_t / m_h$	1.77
Mixing fraction (f) $*f = 1-r(T_{ht}-T_m)/(T_h-T_m)$	0.84
Hot tube fraction (a) *based on the areas of hot tube & cold tube	44.4%
Discharge coefficient (C_d)	0.12
T_h : gas temp. flowing to SG inlet plenum	811.9 ℃
T_{ht} : gas temp. flowing to upper region of SG tubes	707.0 ℃
T_{ct} : gas temp. returned from SG tubes	629.1 ℃
T_m : temp. of the mixing zone	696.9 ℃
m_h : gas flow to SG inlet plenum	6.22 kg/s
m_t : gas flow to upper region of SG tubes	11.0 kg/s

Table 5: Comparison of MELCOR Input Parameters between WH SG, CE SG, and OPR1000 SG

Parameter	WH SG	CE SG	OPR1000 SG
Recirculation ratio (r)	2.4	1.05	1.77
Mixing fraction (f)	0.96	0.65	0.84
Hot tube fraction (a)	41%	22%	44.4%
Discharge coeff. (C_d)	0.12	0.13-0.14	0.12

4. Conclusions and Further Work

KAERI performed a 3-dimensional CFD analysis for a natural circulation flow in the hot leg and SG inlet plenum during a severe accident in an OPR1000 using a commercial code, ANSYS CFX 19.1, to produce the MELCOR input parameters. An analysis methodology for the natural circulation flow of the OPR1000 was developed from the validation against the WH 1/7 scaled-down test and the SG design data of the OPR1000. The generated MELCOR input parameters will be used to evaluate the release of the radioactive materials from the containment when the TI-SGTR accident occurs during the severe accident analysis.

ACKNOWLEDGMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (Ministry of Science, ICT, and Future Planning) (No. 2017M2A8A4015280)

REFERENCES

[1] S. I. Kim, et al., Evaluation of Steam Generator Tube Rupture Accident Sequence Analysis, KAERI/TR-7852/2019, Technical Report, KAERI, 2019.

[2] Nuclear Safety and Security Commission, Amendment of Regulation and Notification for Technical Standards about a Nuclear Facility, 2016.

[3] Ansys Inc., ANSYS CFX-19 Manual, 2018.

[4] W. A. Stewart, A. T. Pieczynski, and V. Srinivas, Natural Circulation Experiment for PWR High-Pressures Accident, Technical Report, TR-102815, EPRI, 1993.

[5] H. S. Kang, S. I. Kim, and K. S. Ha, Development of a CFD Analysis Methodology for a Natural Convective Flow in the Steam Generator Inlet Plenum at the Westinghouse 1/7 Test Facility, J. Comput. Fluids Eng., Vol.25, No.3, pp.34-43, 2020.

[6] H. S. Kang, E. H. Ryu, S. I. Kim, and K, S, Ha, Preliminary CFD Analysis for a Natural Circulation Flow between the Hot and the Steam Generator in OPR1000, Proc. Of KNS Spring Meeting, Jeju, Republic of Korea, 2020.

[7] G. C. Lee, Fluid System Engineering Design Data for Plant Safety Performance and Containment Analysis of YGN 3&4, 10487-FS-DD012, Rev.05, KAERI, 1994.