Transactions of the Korean Nuclear Society Autumn Meeting
PyeongChang, Korea, October 25-26, 2007

3-D Transient CFD Analysis for the Structural Integrity Assessment of a PWR Pressurizer Surge Line Subjected to Thermally Stratified Flow

Dong Gu Kang and Jong Chull Jo*

*Safety Issue Research Dept., Korea Institute of Nuclear Safety, 19 Kusung-dong, Yusung-gu, Daejon 305-338, Korea, *jcjo@kins.re.kr*

1. Introduction

The temperature difference in the fluid region due to the thermal stratification produces undesirable excessive thermal stress at the pipe in axial, circumferential, and radial directions, leading to thermal fatigue damage to the piping systems [1-4]. Several nuclear power plants have so far experienced such serious mechanical damages due to excessive thermal stress as pressurizer surge line movements and its support failures, and cracks in feedwater nozzle, high pressure safety injection lines, and residual heat removal lines. An essential prerequisite for assessing the structural integrity of a piping system subjected to internally thermal stratification is to determine the transient temperature distributions in the pipe wall with accuracy.

Several investigators have made efforts to determine the temperature distributions in the pipe wall by means of laboratory testing of particular geometry, field measurement of temperature, or fully theoretical predictions, or numerical calculations. However, they have addressed only 2-dimensional cases or 3-dimensional simplified cases where the existence of pipe wall thickness is neglected or the piping is assumed to be straight [5,6].

However, most of piping systems at nuclear power plant are designed to have some bent parts and thick wall taking into account the wall thinning effect during the long time operation. When the conducting solid material is included in the solution domain of fluid flow and heat transfer, the problem is usually called the conjugate heat transfer problem.

Recently 2-D and 3-D conjugate heat transfer calculations for a cross-sectional area and full flow and heat transfer domain of a simplified straight pipe model of PWR pressurizer, respectively, have been performed by Jo et al. [5, 6].

In this study, to overcome the limitation in mesh generation for such a complex geometry as 3-dimensionally bent piping, 3-dimensional transient CFD calculations involving the conjugate heat transfer analysis are performed to obtain the transient temperature distributions in the wall of an actual PWR pressurizer surge line subjected to stratified internal flows either during out-surge or in-surge operation using a commercial CFD code called CFX-5.10 [7]. In addition, some key findings derived from the discussion on the CFD calculation results which should be followed in the process of thermal analysis are provided.

2. Mathematical Formulation

2.1 Problems

The configuration of pressurizer surge line of a PWR is depicted in Fig.1. The geometrical data and material properties utilized in the calculations are shown in Table.1. Firstly, considering the situation of out-surge flow, cold fluid at a specified temperature of 51.7 ^{o}C occupies inside the pipe maintaining the steady-state condition initially, and then at a certain time hot water at a specified temperature of 218.3 ^{o}C is considered to begin to flow down into the pipe top nozzle (the inlet for the case of out-surge flow which is connected to the pressurizer) at a velocity of 0.07 $m/\sec$. In case of the in-surge flow, on the contrary to the former case, hot fluid at a specified temperature of 218.3 ^{o}C occupies inside the piping system maintaining the steady-state condition initially, and then at a certain point of time cold water at a specified temperature of 51.7 ^{o}C is considered to begin to flow up into the pipe bottom nozzle (the inlet for the case of in-surge flow which is connected to the reactor coolant system) at a velocity of 0.07 $m/\sec$.

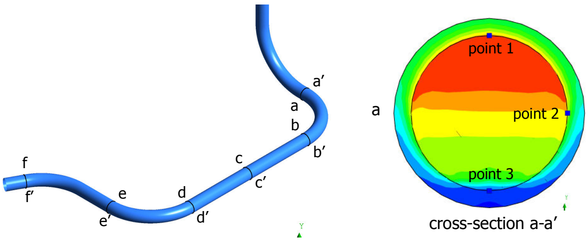


Fig.1 Configuration of pressurizer surgeline.

Table 1 Geometrical data and material properties

Parameters	Values
Material of pipe	ASME SA-312 Type 316
Inner diameter of pipe	0.3048m
Thickness of pipe	33.45mm
Conductivity of pipe, k_s	16.3 W/m-K
Ri [$= \Delta\rho g d_i / (\rho_{avg} u_{in}^2)$]	99.85

2.2 Governing Equations

To closely simulate the flow behavior during the entire discharging phase, the turbulent flow is considered with the shear stress transport (SST) turbulent model. For buoyancy calculations involving variable density, the density difference $\rho - \rho_{ref}$ is evaluated directly [7].

The Reynolds-Averaged Navier-Stokes equations for conservation of mass, momentum, and turbulent quantities for the present problem in a Cartesian coordinate system can be expressed as in reference [7].

2.3 Initial and Boundary Conditions

In either situation, the cold fluid occupies the lower space of the pipe without mixing well with the hot fluid occupying the upper space due to the difference in density between the two fluids. This results in so-called thermally stratified flow in the pipe. Because the plant piping system is generally insulated to prevent heat loss, the adiabatic condition is specified at the outer wall surface of present pipe model. On the solid wall, the wall function method is applied.

3. Conjugate Heat Transfer

To take into account the thermofluid-structure interaction between the pipe wall and stratified flow inside the pipe, the conjugate heat transfer analysis has been incorporated into the CFD analysis. The solid domain of the pipe wall region, in which the equations for heat transfer are solved, is created. Within the solid domain, the conservation of energy equation is simplified since there is no flow inside a solid, thus conduction is only mode of heat transfer.

4. CFD Analysis

The SST model is employed to simulate the turbulence flow in this study because, as compared with the standard $k-\varepsilon$ turbulence model, it is known to be more conformable to analysis of flow separation and swirl flow and has high accuracy. The solution domain is divided into a finite number of hexahedral control volume cells. The discretization of the governing equations is performed following the finite volume approach. The convection terms are approximated by a higher-order bounded scheme. The calculation was performed with the transient mode and the physical time step set to 0.1sec for the out-surge case and 0.05 sec for the in-surge case with 10 of the maximum number coefficient iterations per time step. Convergence of the iterative computations for each time step is determined when the RMS residual of the major parameters is less than 0.0001. The interesting time range of the present work is from the starting time point of either out-surge or in-surge operation through the elapsed time of 500 sec.

Transactions of the Korean Nuclear Society Autumn Meeting
Pyeong Chang, Korea, October 25-26, 2007

5. Results and Discussion

Sensitivity studies to check both mesh and time step independencies on the calculation results have been performed. The results are shown in Figs. 2 for out-surge case. The optimum number of mesh and time step sizes are 230,000 and 0.1 sec for the out-surge case (0.05 sec for the in-surge case), respectively. It is seen from the Figs. 2 that excessive computation time is required with little changes in the computation results even if the number of nodes is increased above 230,000 or the time step size is reduced below 0.1sec for the out-surge case and 0.05 sec for the in-surge case.

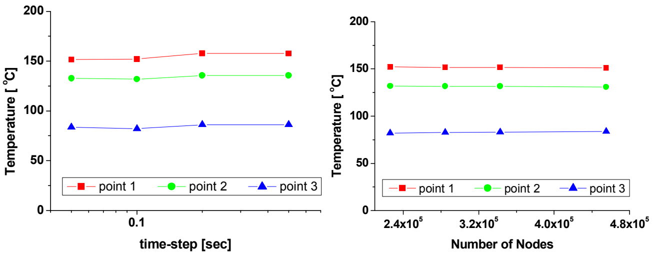
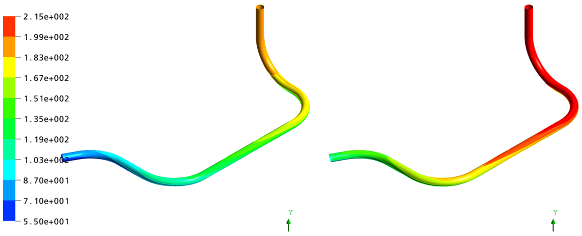


Fig. 2 Mesh and time step dependencies for out-surge flow.



(a) (b)

Fig. 3 Transient temperature distributions at the inner wall surface at t=200sec.

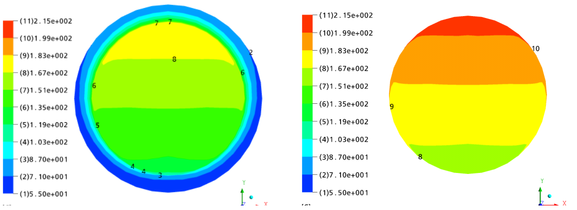


Fig. 4 Transient temperature distributions at the cross-section c-c' (see Fig. 1) at t=200sec.

To investigate the effect of the existence of pipe wall thickness as is in the actual situation on the prediction of transient temperature distributions in the pipe wall, two different cases where the pipe wall exists or not have been numerically simulated using the CFX-5.10 code. Figure 3(a) and 3(b) show the transient temperature distributions at the inner wall surface at the elapsed time of 200 sec after start of out-surge for the two different cases mentioned above, respectively.

Figure 4(a) and 4(b) display the transient temperature distributions at the cross-section c-c' (see Fig. 1) at the elapsed time of 200 sec after start of out-surge for the two cases, respectively. As can be seen from Figs. 3 and 4, the temperature distributions in the pipe wall predicted with the simple heat transfer model which takes no account of the existence of wall thickness are much different from those for the case where the existence of wall is considered as in the real situation. This is because convective heat transfer between the fluid and pipe wall as well as heat conduction in the pipe wall occur as the hot water flows into the pressurizer pipe which is filled with cold water initially.

The transient evolutions of temperatures both at the top and bottom inner wall surfaces of cross-section c-c' indicated as the point 1 and point 3 in Fig. 1 are plotted for the two different cases in Fig. 5. As is seen from Fig. 5, the temperature difference between the top and bottom inner wall surfaces for the real pipe model is much larger than for the pipe model neglecting wall thickness, generally.

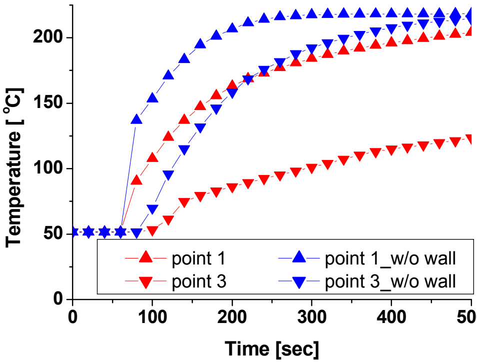


Fig. 5 Transient evolutions of temperatures both at the top and bottom locations of the cross-section c-c'.

It is seen from previous results that the existence of pipe wall thickness is very important and makes computation results different. Therefore the conjugate heat transfer analysis should be performed in order to predict the thermal behavior of a piping system subjected to thermally stratified flow realistically and obtain accurate computational results of transient temperature distributions in the pipe wall.

6. Conclusions

In this work, a detailed numerical analysis of unsteady conjugate heat transfer was performed for an actual PWR pressurizer surge line pipe subjected to internally thermal stratification caused either by out-surge or in-surge flow. Main emphasis of the study was placed on the investigation of the effects of conjugate heat transfer analysis on the determinations of the transient temperature and thermal stress distributions in the pipe wall. A conjugate heat transfer analysis should be performed in order to predict the thermal behavior of a piping system subjected to thermally stratified flow realistically and obtain accurate computational results of transient temperature distributions in the pipe wall.

REFERENCES

[1] NRC Bulletin 79-13, 1979, “Cracking in Feedwater System Piping,” US NRC.

[2] NRC Bulletin 88-08, 1988, “Thermal Stresses in Piping Connected to Reactor Coolant Systems,” US NRC.

[3] NRC Bulletin 88-11, 1988, “Pressurizer Surge Line Thermal Stratification,” US NRC.

[4] NRC NUREG/CR-6456, 1989, Review of Industry Efforts to Manage Pressurized Water Reactor Feedwater Nozzle, Piping, and Feedering Cracking and Wall Thinning, US NRC.

[5] Jo, J. C., Kim, Y. I., and Choi, S. K., 2001, “Numerical Analysis of Thermal Stratification in a Circular Pipe,” ASME J. PVT, Vol. 123, pp 517-524.

[6] Jo, J.C., Choi, Y.H. and Choi, S. K., November 2003, "Numerical Analysis of Unsteady Conjugate Heat Transfer and Thermal Stress for a PWR Pressurizer Surge Line Pipe Subjected to Thermal Stratification, "ASME Transaction J. of PVT, Vol. 125, pp. 467-474..

[7] ANSYS System, 2005, CFX-5.10 Manual.