Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 29-30, 2015

Recent Developments of the SPACE Code

Kyung Doo Kim, S. W. Lee , S. W. Bae, B. J. Kim, J. S. Heo, J. H. Lee,
B.U. Bae and B. D. Chung
Korea Atomic Energy Research Institute (KAERI)
1045 Deadeok-daero, Yuseong Daejeon, 305-353, Korea
**Corresponding author: kdkim@kaeri.re.kr*

1. Introduction

The SPACE (the Safety and Performance Analysis Code for Nuclear Power Plants) code [1] is under development to be used for licensing pressurized water reactor design. The SPACE code adopts advanced physical modeling of two-phase flows, mainly two-phase three-field models which comprise gas, continuous liquid, and droplet fields. But it has been modified to be able to handle the classical two-phase two-field model by user's selection. It has the capability to simulate 3D effects by the use of structured and/or non-structured meshes.

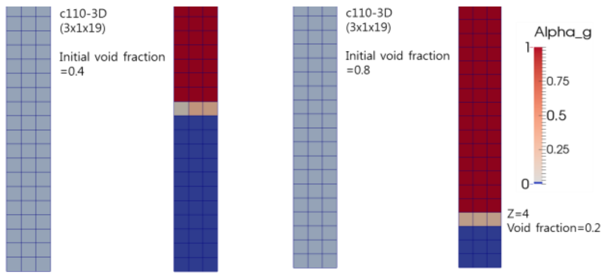
The SPACE code has been developed and validated successfully. We submitted the topical reports to regulatory body for review in 2013. Since then, SPACE code developers and the reviewers are exchanging the questions and answers concerning to the topical reports.

In this paper, recent advances in the SPACE code will be briefly presented. First, the model improvements for multi-dimensional applications are introduced with a few validation results. Second, optional two-phase two-fluid model development activities are described. The comparison of calculation results with two-phase three-fluid model case is also presented. Third, nonphysical phasic velocities for a dispersed field were corrected by improving pressure drop by wall drag and from loss. Fourth, an improved CHF model for pool boiling condition based on instability theory is also introduced. Fifth, a two group interfacial area transport model is incorporated to predict dispersed liquid interfacial area. Finally, uncertainty quantification based on data assimilation technique is demonstrated.

2. Validation for Multi-Dimensional Analysis

SPAEC code can be modeled for 3D in Cartesian and cylindrical coordinates as well as 1D. For 3D modeling, user can model the system by structured and or non-structured mesh systems. We modified the models and correlations developed for 1D system to extend its capabilities to be used in 3D system. Then, it has been validated to illustrate the 3D simulation capability. Several simple conceptual problems are used to verify the 3D component. Among them, phase separation problem are described here.

The phase separation problem is conducted for the 3×1×19 grid multi-D component as shown in Fig. 2. The node size is 0.1 m for all directions. The calculation domain is initially filled with mixture of water and steam. The initial void fraction is set as 0.4. 12th vertical nodes become stratified and void fraction of those is 0.6. Below the 12th vertical node, it is single phase water. As the other case, initial void fraction is set as 0.8. At this case, final stratified node is observed in the 4th vertical region. The void fraction of the 4th vertical nodes is 0.2.



(a) Initial α_g=0.4 (b) Initial α_g=0.8

Fig. 1. Phase separation problem

After simple verification, RPI 2D slab air-water void distribution experiment [2] is used for code validation. RPI 2D slab problem is modeled as the vertical slab geometry. The slab height and width are 0.9144 by 0.9144 m. the depth of slab is 0.0127 m. There are 4 inlet and outlet port slits. Fig. 2 shows the SPACE modeling of RPI 2D slab and the code prediction for void distribution of 3 different elevations.

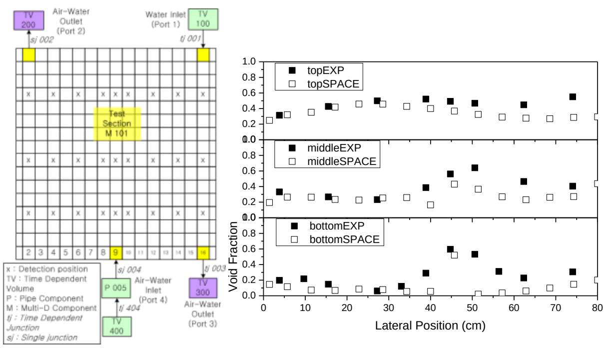


Fig. 2. RPI 2D slab modeling and SPACE prediction

3. Optional Two-Phase Two-Fluid Model

The main objective to incorporate 6-equation model into SPACE is to enhance the calculation speed. As classical two-phase flow model, droplet field calculation processes were totally removed. The 6-equation option can be activated by user. The solver and closure relations are modified to eliminate the droplet effects. For closure relations, the dispersed flow regime is added when the void fraction is large. At that instance, the liquid fraction is forced to be divided into film and imaginary induced droplet.

The calculation speed of 9- and 6-equation systems of SPACE are compared as in the Table 1. The simple 2 phase problem is about vertical pipe flowing water and steam mixture. Pipe is modeled by 20 nodes. The CHF problem consists of 20 node vertical pipe and heat structure that transfers high heat fluxes enabling critical heat flux condition. The reflood problem is added a subcooled liquid flow injected from the bottom of vertical 20 node pipe and heat structure. The calculation results are compatible for both 9- and 6-equation models.

Table 1. Calculation time comparisons between 9- and 6-equation systems

problem	Simple 2 phase	CHF problem	Reflood problem
Problem time	50	50	200
9 equation	5.897	6.645	469.579
6 equation	4.711	5.959	449.470
Time difference	-1.186	-0.686	-20.109

4. Pressure Drop for Wall Drag and Form Loss

Recently, we brought light to the wall drag term for dispersed flows, examining the averaged momentum equations based on the equation of a solid/fluid particle motion [3]. The wall drag term on the bubble phase accounts for the interaction between the stresses of the undisturbed water and the bubble phase. As a result, the total pressure drop by the wall friction of the continuous phase must be apportioned to each phase in proportion to each phase fraction. By doing so, the relative velocity of the dispersed phase against the continuous phase can be correctly predicted in a pipe, contraction, and expansion.

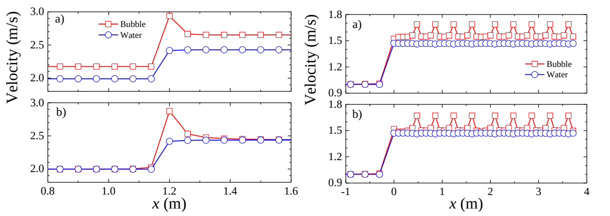


Fig. 3. Horizontal bubbly flow in the contraction and for form loss at various locations: a) no wall drag on the bubble phase and b) new model

In addition, a new form loss model was proposed for dispersed flows. According to the existing form loss model, the bubble is predicted to be faster than water even for a fully-developed flow. To solve this deficiency, the total momentum loss by the continuous phase is first calculated. After that, the total momentum loss is apportioned to each phase in proportion to each phase fraction. This partitioning approach is consistent with the wall drag partitioning. This is not surprising because the form loss is merely a different expression of the wall drag [4].

5. Hydrodyanmic Model for Critical Heat Flux

Interfacial instabilities play an important role in the development of critical heat flux (CHF) models. The Rayleigh-Taylor, Kelvin-Helmholtz, and Plateau-Rayleigh instabilities are used to formulate the critical heat flux models for saturated pool boiling on infinite horizontal surfaces. The most of existing CHF models have been developed with the results of the linear stability analysis of inviscid flows. Therefore, there is no consideration on the effect of fluid viscosities in the existing CHF models. As the pressure increases, the viscosities of vapor and liquid become closer. And thus the effect of fluid viscosities cannot be ignored.

In this study, we applied the interfacial instabilities of viscous potential fluids including the effect of fluid viscosities on CHF. The viscous potential flow allows a velocity discontinuity at the interface but consider the viscous normal pressure on the interface. These treatments are consistent with the phenomena that the interface waves are induced by pressure, more than by shear force. The circular jet and Kelvin-Helmholtz instabilities of viscous potential flows were applied to the most widely used models: the hydrodynamic theory model. The CHF models were successfully modified to include the effect of fluid viscosities by the interfacial instability analysis of viscous potential flow [5, 6]. We also implemented to SPACE to evaluate the effect in low mass flow condition since SPACE code uses Zuber correlation in this condition.

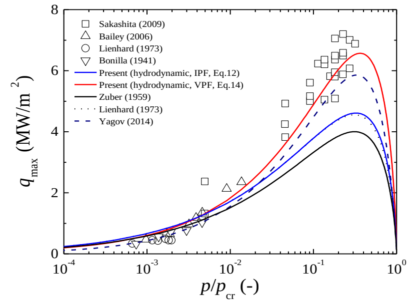


Fig. 4. Comparison of the modified models with experimental data for water

6. Droplet Two-Group Interfacial Area Transport Model

In modeling the droplet field, an interfacial area concentration of the droplet is one of crucial parameters to estimate the interfacial momentum and heat transfer between the droplet and vapor phases. Especially, the droplet breakup at spacer grid increases the interfacial area of the droplet and can affect the quenching behavior and the steam binding at steam generator in LBLOCA condition. Conventional approach has calculated the droplet interfacial area concentration from a droplet size model. This model was based on a non-dimensional Weber number including the effect of the surface tension, which can be valid in a fully-developed flow condition. However, it is not appropriate to consider the dynamic behavior of the interfacial area of the droplet such as the breakup by the spacer grid.

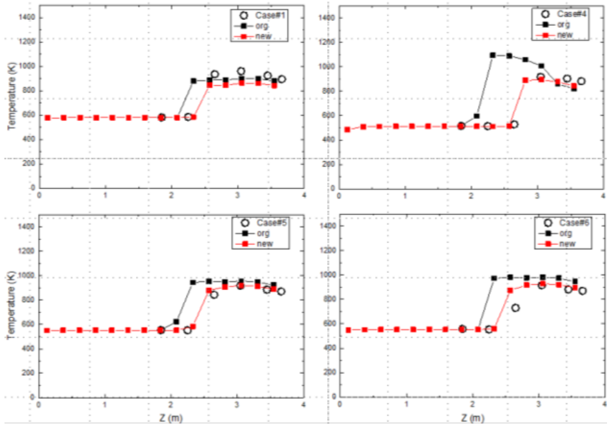
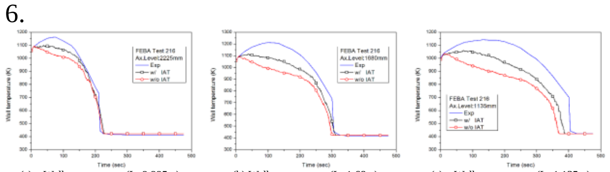


Fig. 5. Comparisons of the calculation results of existing and new models

The SPACE code adopted an IAT (Interfacial Area Transport) model to enhance the prediction capability for the droplet interfacial area. The IAT model for the droplet field estimates the interfacial area concentration by solving the IAT equation. When the droplet collides on the spacer grid surface, a small droplet can be generated by the breakup, which increases the interfacial area and the amount of droplet evaporation. In particular, the breakup on the spacer grid surface can produce small droplets, which needs to be distinguished by large droplets [7]. Considering existence of the large and small droplets, two-group IAT equation model was implemented in the SPACE code.

The FEBA experiments were a reflood test with a 5×5 rod bundle and six spacer grids [8]. The test section was modeled using 38 cells in the vertical direction. As an example, wall temperatures at 3 different elevations for test 218 were compared to the code calculation results with or without the two-group IAT model in Fig. 6.



(a) Wall temperature (L=2.225m) (b) Wall temperature (L=1.68m) (c) Wall temperature (L=1.135m)

Fig. 6. Calculation result for FEBA test 218

7. PAPIRUS, A Parallel Computing Framework for Sensitivity Analysis, Uncertainty Propagation, and Estimation of Parameter Distribution

A statistical data analysis toolkit, PAPIRUS is developed to perform the model calibration including uncertainty band determination, uncertainty propagation, Chi-square linearity test, and sensitivity analysis for both linear and nonlinear problems [9]. The PAPIRUS is multiple packages of methodologies, and building an interface between an engineering simulation code and the statistical analysis algorithms. A parallel computing framework is implemented in the PAPIRUS with multiple computing resources and proper communications between the server and the clients of each processor. One of the nice features of PAPIRUS is to estimate uncertainty bands of the physical model based upon the statistical approach rather than expert judgment.

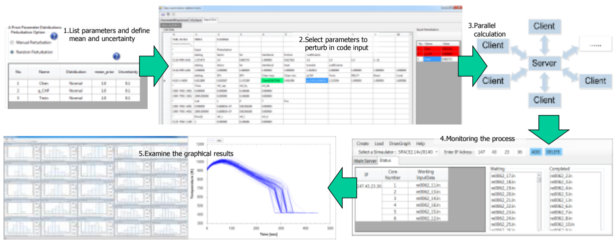


Fig. 7. Schematic diagram of PAPIRUS

3. Conclusions

SPACE code has been developed to be used for a safety analysis of PWR design. To extend application areas and enhance the calculation accuracy, new features have been implemented which includes multi-dimensional model development and V&V, optional two-phase two-fluid model for fast execution, improved wall drag and from loss treatments to correct nonphysical phasic velocities for a dispersed fields, an improved CHF model for pool boiling condition based on instability theory, a two group interfacial area transport model for droplet, and uncertainty quantification based on data assimilation technique.

ACKNOWLEDGMENTS

This work was supported by the Nuclear Power Technology Development Program in the form a Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Ministry of Trade, Industry and Energy (MOTIE).

REFERENCES

[1] S. J. Ha, C. E. Park, K. D. Kim, C. H. Ban, Development of the SPACE Code for Nuclear Power Plant, Nuclear Engineering and Technology, Vol. 43 (1), p. 45, 2011.

[2] K.M. Bukhar and R.T. Lahey Jr., An experimental study of 2-D phase separation phenomena, Int. J. Multiphase Flow, 13(3), pp. 387-402, 1987.

[3] B.J. Kim, J.W. Kim, and K.D. Kim, On the wall drag term in the averaged momentum equation for dispersed flow, Nuclear Science and Engineering 178, p. 225, 2014.

[4] B.J. Kim, S.W. Lee, and K.D. Kim, New wall drag and form loss models for one-dimensional dispersed two-phase flow, Nuclear Science and Technology, accepted for publication, 2015.

[5] B.J. Kim, J.H. Kim, and K.D. Kim, Rayleigh-Taylor instability for thin viscous gas films: Application to critical heat flux and minimum film boiling, International Journal of Heat and Mass Transfer 80, p.150, 2015.

[6] J. H. Lee, B. J. Kim, and K. D. Kim, Modifications of critical heat flux models on horizontal surfaces in pool boiling using interfacial instabilities of viscous potential flows, NURETH-16, Chicago, USA, 2015.

[7] EPRI, Analysis of FLECHT-SEASET 163-Rod Blocked Bundle Data Using COBRA-TF, NUREG/CR-4166, 1985.

[8] Ihle P., Rust K.: FEBA – Flooding Experiments with Blocked Arrays, Evaluation Report, Kfk Karlsruhe, Rep. KfK 3657, 1984.

[9] J. Heo, K. D. Kim, PAPIRUS, a parallel computing framework for sensitivity analysis,uncertainty propagation, and estimation of parameter distribution, Nucl. Eng. Design, Vol. 292, p. 237, 2015.