A Comparison of the Computation Times of Thermal Equilibrium and Non-equilibrium Models for A Two-Fluid Three-Field Model

I. K. Park*, H. K. Cho, J. Kim, H. Y. Yoon, J. J. Jeong
Korea Atomic Energy Research Institute,1045 Daedeok-daero, Yuseong-gu, Daejon, 305-353,Korea
**E-Mail: gosu@kaeri.re.kr*

1. Introduction

A pilot code for a transient three-dimensional two-phase flow analysis has been developed [1], which is based on a two-fluid, three-field model and an unstructured finite volume method (FVM). The three fields in the model represent a continuous liquid, an entrained liquid, and a vapor field. The two liquid fields in the code may have different velocities, however they are assumed to be in a thermally equilibrium state. It would not be desirable in some circumstance[2], and a non-equilibrium model for droplets and liquid might be needed.

This paper presents a comparison of computational time between a thermally equilibrium and a thermal non-equilibrium model for a transient, three-dimensional, three-field model.

2. Mathematical Model

The governing equations of the two-fluid, three-field model are similar to those of the time-averaged two-fluid model derived by Ishii and Hibiki[3]. The continuity, momentum, and energy equations for the k-phase are given by

$$\frac{\partial}{\partial t}(\alpha_k\rho_k)+\nabla\cdot(\alpha_k\rho_k\underline{u}_k)=\Gamma_k \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_k\rho_k\underline{u}_k)+\nabla\cdot(\alpha_k\rho_k\underline{u}_k\underline{u}_k)=-\alpha_k\nabla P+\nabla\cdot[\alpha_k\tau_k] + \alpha_k\rho_k\underline{g}+P\nabla\alpha_k+M_k^{mass}+M_k^{drag}+M_k^{VM} \tag{2}$$

$$\frac{\partial}{\partial t}[\alpha_k\rho_k e_k]+\nabla\cdot(\alpha_k\rho_k e_k\underline{u}_k)=-\nabla\cdot(\alpha_k q_k) + \nabla\alpha_k\tau_k:\nabla\underline{u}_k-P\frac{\partial}{\partial t}\alpha_k-P\nabla\cdot(\alpha_k\underline{u}_k)+I_k+Q'''_k \tag{3}$$

where α_k, ρ_k, $\underline{u}_k$, P_k, and Γ_k are the k-phase volume fraction, density, velocity, pressure, and an interface mass transfer rate, respectively. M_k represents interfacial momentum transfer due to a mass exchange, a drag, and a virtual mass. Further detailed two-phase flow mathematical descriptions are given in Ref. [3]. For a closure of the system of equations, constitutive relations and the equations of states are included.

The thermally equilibrated droplet and liquid energy equations are given at Eq. (4), in which droplets do not have their own temperature field. The droplet and liquid energy conservations at the thermally non-equilibrium state are given at Eq.(5) and Eq. (6), in which droplets have their own temperature and the thermal model can be adapted for droplet phase.

$$\frac{\partial}{\partial t}[(\alpha_l+\alpha_d)\rho_l e_l]+\nabla\cdot(\alpha_l\rho_l e_l\underline{u}_l)+\nabla\cdot(\alpha_d\rho_l e_l\underline{u}_d) = -\nabla\cdot(\alpha_l q_l+\alpha_d q_d)+\nabla\alpha_l\tau_l:\nabla\underline{u}_l+\nabla\alpha_d\tau_d:\nabla\underline{u}_d - P\frac{\partial}{\partial t}(\alpha_l+\alpha_d)-P\nabla\cdot(\alpha_l\underline{u}_l+\alpha_d\underline{u}_d)+I_l+Q'''_l \tag{4}$$

$$\frac{\partial}{\partial t}[\alpha_l\rho_l e_l]+\nabla\cdot(\alpha_l\rho_l e_l\underline{u}_l)=-\nabla\cdot(\alpha_l q_l) + \nabla\alpha_l\tau_l:\nabla\underline{u}_l-P\frac{\partial}{\partial t}\alpha_l-P\nabla\cdot(\alpha_l\underline{u}_l)+I_l+Q'''_l \tag{5}$$

$$\frac{\partial}{\partial t}[\alpha_d\rho_d e_d]+\nabla\cdot(\alpha_d\rho_d e_d\underline{u}_d)=-\nabla\cdot(\alpha_d q_d) + \nabla\alpha_d\tau_d:\nabla\underline{u}_d-P\frac{\partial}{\partial t}\alpha_d-P\nabla\cdot(\alpha_d\underline{u}_d)+I_d+Q'''_d \tag{6}$$

The interfacial heat transfer formulations were given as Eq. (7) and Eq. (8), which means the three energy equations are coupled.

$$I_l = R_{gs}(T^s-T_v)+R_{ls}(T^s-T_l) \tag{7}$$

$$I_d = R'_{gs}(T^s-T_v)+R_{ds}(T^s-T_d) \tag{8}$$

where R_{gs}, R_{ls}, R'_{gs}, and R_{ds} are interfacial heat transfer coefficient multiplied by interfacial area concentration.

3. Numerical Method

The semi-implicit ICE scheme used in the RELAP5 code [4] was adopted as a basic numerical method, which uses a staggered grid and a donor-cell scheme. The main advantages of this scheme are a computational efficiency and robustness. In this numerical scheme, the momentum equations are solved first to represent each phasic velocity at a cell face as a function of the pressure difference of the adjacent cells. The mass and energy equations are coupled and, with some algebraic operations, reduced to a cell pressure equation that includes unknown pressures of the computational cell and its adjacent cells. Finally the system pressure equation is established and solved. The phasic velocities at the junctions are obtained by back-substitution.

But to apply this scheme with an unstructured finite volume method, this scheme should be changed to adopt a cell-centered scheme. For this, the momentum equations were solved over a non-staggered grid and the velocities at the cell faces were interpolated by using the Rhie-Chow scheme[5]. The other solution sequence is not changed.

4. Comparison of the Computation Times

The test problem was a rectangular single tube of 1.0 m x 1.0 m x 2 m filled with 442.0 K water, and the subcooled water was being injected into its inlet. The outlet was assumed to be a constant pressure boundary of 0.1 MPa. The injection velocity and temperature were 0.1m/sec and 442.0 K, respectively. A volumetric heat source of Q_l = 23.0e6*min(time/10, 1) J/m^{3}sec was added to the water, and vapor was generated.

An entrainment model proportional to the relative velocity between the vapor and the water was assumed, and a relatively large coefficient was used in order to artificially create a large amount of droplets. Initially, the tube was filled with subcooled water. The void fraction and the droplet fraction became larger due to the evaporation and the entrainment. The steady state was obtained after 20.0 seconds.

Volume fraction, temperature, and velocity of droplet and vapor temperature at 30.0 seconds are shown in Figure 1. The droplet fraction is large near the inlet. The droplet temperature was the same as the vapor temperature because of the big heat transfer coefficient between droplet and vapor.

The computation time for the non-equilibrium model was compared to the equilibrium model at Table 1. The non-equilibrium model seems to be robust just like the equilibrium model, and the additional calculation time is only 8 %. The good thing is that the additional time did not increase as the number of cell increased.

Table 1. Comparison of Computation Time between Equilibrium Model and Non-Equilibrium Model

Cell (#)	Model	Problem time (sec)	Time step (#)	CPU time (sec)	CPU time (%)
980	Equil.	9.631	2786	1591.	100.00
980	NonE.	9.631	2787	1723.	108.30
3000	Equil.	9.631	2885	8577.	100.00
3000	NonE.	9.631	2886	9287.	108.28

5. Conclusions

A thermal non-equilibrium model was implemented in to the 3-dimensional, 2 fluid, 3 field pilot code, which adopts an unstructured grid. This non-equilibrium model might have the same robustness, but it needed an additional computational time of 8 %, when compared to the thermally equilibrium model. After some assessment, the non-equilibrium model will be used optionally.

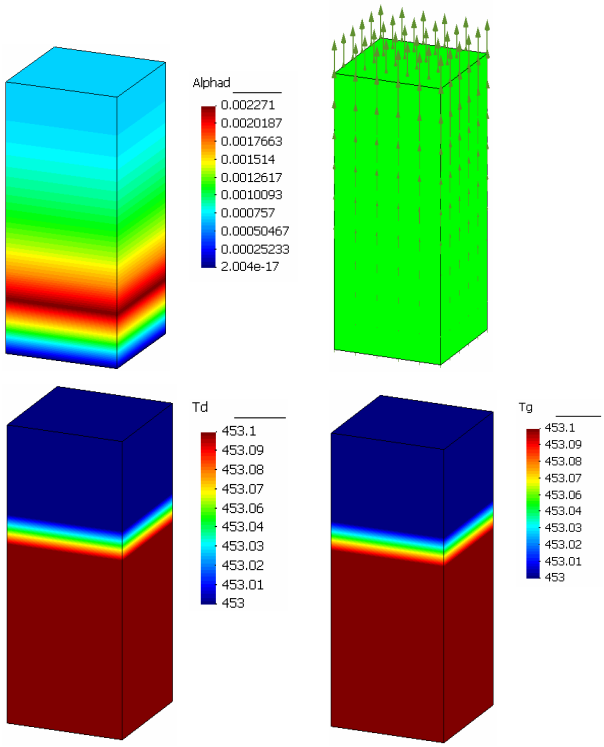

Fig. 1 Volume fraction, temperature, velocity of droplets and vapor temperature in the tube at 30.0 second

ACKNOWLEDGMENTS

This study has been carried out under the nuclear R&D program by the Korean Ministry of Science and Technology.

REFERENCES

[1] J. J. Jeong, H. Y. Yoon, J. Kim, I. K. Park, H. K. Cho, An Unstructured Hydrodynamic Solver for a Hydrodynamic Solver For a Two-Fluid Three-Field Model, To be presented in IAEA Topical Meeting, Daejon, Korea, Oct. 2007.

[2] E. Porcheron et al, Experimental Investigation in the TOSQAN Facility of Heat and Mass Transfers in a Spray for Containment Application, Nuclear Engineering and Design, Vol. 237, pp. 1862-1871, 2007.

[3] M. Ishii, T. Hibiki, Thermo-fluid Dynamics of Two-Phase Flow, Springer, pp. 155-169, 2006.

[4] The RELAP5-3D Code Development Team, RELAP5-3D© Code Manual Volume I: Code Structure, System Models and Solution Methods, Idaho National Engineering and Environmental Laboratory, Feb. 2001.

[5] C. M. RHIE, W. L. CHOW, Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", AIAA Journal, Vol. 21, No. 11, pp. 1525-1532, 1983.