# An Overview of the MARS and CUPID Code Development: Numerical Aspect

Jae Jun Jeong
*School of Mechanical Engineering, Pusan National University, Busan, South Korea*
E-mail: jjjeong@pusan.ac.kr

## 1. Introduction

The thermal-hydraulic (TH) system code is an essential tool for the design and safety analysis of nuclear power plants. It is a very big repository of thermal-hydraulics and safety research, which requires huge resources and efforts.

The MARS code development project launched at Korea Atomic Energy Research Institute (KAERI) in 1997 as one of R&D activities for technology self-reliance [1]. The backbone of the MARS code is the consolidated version of the COBRA-TF and RELAP5/MOD3 codes [2]. Because the two codes use the same numerical solution scheme known as the semi-implicit scheme [3], the implicit coupling was easily devised and implemented. The key advantages of the semi-implicit scheme are that (i) it can be easily extended to three-dimensional two-phase flows and (ii) it is very robust for two-phase flow calculations. Because of these features, it was adopted in the component-scale code, CUPID [4-7].

In this paper, the numerical aspect of the MARS and CUPID codes are discussed.

## 2. The MARS and CUPID Numerical Solution Methods

In this section, the semi-implicit numerical scheme for a one-dimensional two-phase flow is briefly introduced first. The numerical aspects of the MARS and CUPID codes are described later.

### *2.1 The Semi-implicit Numerical Scheme*

The semi-implicit numerical solution scheme in the RELAP5 code is based on replacing the system of differential equations for the six-equation two-fluid model with a system of finite difference equations (FDEs) partially implicit in time. The implicit terms are formulated to be linear in the dependent variables at new time. The FDEs for the corresponding differential equations are established on a one-dimensional staggered grid in Fig. 1, where the scalar variables, such as pressure, void fraction and phasic internal energy, are defined at the cell center (or called volume) and the phasic velocities are defined at the cell surface (or called junction).

To obtain the numerical solutions, the phasic momentum equations are solved first, where the convection terms are explicitly treated, to represent the phasic velocity at a junction $j$ in terms of the pressures of adjoining cells *I-1* and *I*:

$$V_{k,j}^{n+1} = a_{k,j} + b_{k,j}(\delta P_{I-1}^{n+1} - \delta P_I^{n+1}) \,. \tag{1}$$

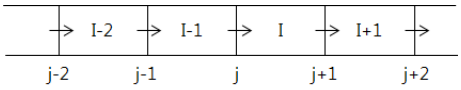



Fig. 1. The staggered grid (The arrows indicate the junctions where velocities are defined).

Next, the FDEs of the mass and energy equations are derived for a hydro-dynamic cell, where the velocities at the convection term are implicitly treated and the convected quantities are explicitly written. The FDEs are linearized with respect to the independent scalar variables, including noncondansable gas quantity ($X_n$), void fraction ($\alpha$), phasic internal energies ($U_g$, and $U_l$), and pressure ($P$):

$$\begin{bmatrix} r_{11} & r_{12} & \dots r_{15} \\ r_{21} & & \dots r_{25} \\ r_{31} & & \dots r_{35} \\ r_{41} & & \dots r_{45} \\ r_{51} & & \dots r_{55} \end{bmatrix} \begin{bmatrix} \delta\tilde{X}_n \\ \delta\tilde{U}_g \\ \delta\tilde{U}_f \\ \delta\tilde{\alpha}_g \\ \delta P \end{bmatrix} = s + g^1 V_{g,j+1}^{n+1} + f^1 V_{f,j+1}^{n+1} + g^2 V_{g,j}^{n+1} + f^2 V_{f,j}^{n+1}, \tag{2}$$

where $s$, $g$, $f^1$, and $f^2$ are coefficient vectors. Substituting the unknown velocities in the right-hand side of Eq. (2) with Eq. (1) and multiplying Eq. (2) by the inverse of the cell Jacobian matrix, i.e., the matrix in the left-hand side of Eq.(2), the bottom row results in a single equation involving just pressures. This reduction is repeated for each cell and, at last, all these equations form the system pressure equation:

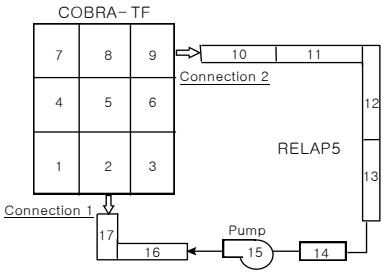
$$\underline{\underline{A}}\,\underline{\delta P} = \underline{b} \tag{3}$$

where $\underline{\underline{A}}$ is an NxN matrix (N is the number of computational cells). The unknown pressure variations are obtained by solving Eq. (3). The new time-step velocities are updated by using Eq. (1). Further back-substitutions in Eq. (2) yield the variations of other solution variables.

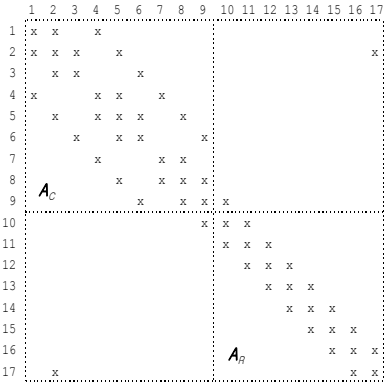
### *2.2 The MARS Code*

The MARS code is based on the consolidated version of the COBRA-TF and RELAP5/MOD3 codes. Because the two codes use the semi-implicit numerical solution scheme in common, an implicit coupling algorithm was easily devised. In the MARS code, the domain is decomposed into a one-dimensional region and a three-

dimensional region, of which thermal-hydraulics are solved by the RELAP5 and COBRA-TF modules, respectively. The momentum equations at the interface junctions are solved by the RELAP5 module. Fig. 2 illustrates an example of the MARS nodalization and the coupled system pressure matrix.




(a) An example of the MARS nodalization




(b) The system pressure matrix

Fig. 2. An example of the MARS nodalization and the coupled system pressure matrix.

*2.3 The CUPID Code*

KAERI launched a new project to develop a component-scale thermal-hydraulic code CUPID in 2007, which was motivated from practical needs for the three-dimensional simulation of two-phase flows in nuclear reactor components [4]. The CUPID code adopts a two-fluid, three-field model for two-phase flows, which is a straightforward extension of the two-fluid single pressure model in system codes to three dimensions. The constitutive equations for three-dimensional two-phase flows have been developed or imported.

For the numerical method of the CUPID code, the semi-implicit scheme was adopted because it can be easily extended to three dimensions and three-field equations, still maintaining its robustness [4, 5]. Instead, the scheme was modified to be adapted to the unstructured grid for the CUPID code, which is shown in Fig. 3. The phasic momentum equations are represented in a semi-conservative form to enhance the numerical solutions [6]. A second-order upwind method was also implemented later [7].

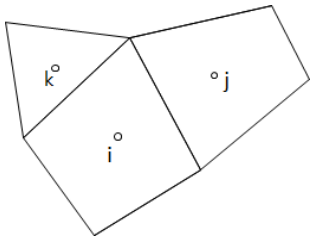



Fig. 3. The unstructured grid for the CUPID code.

## 3. Conclusions

The semi-implicit scheme has worked very successfully in the MARS and CUPID codes. But, its time step is limited by the Courant limit, that is, the minimum of ($dZ_i/U_i$) or cell transit time. This requires long computational time for a problem with fine meshes and a fast flow. This feature also limits the computational efficiency for slow transients or steady-state calculations. Especially for the CUPID code that usually uses finer meshes, this becomes a serious disadvantage. Thus, the implicit scheme or nearly-implicit scheme is desirable. Other improvements are also needed.

In my lecture, I am going to present an overview of the MARS and CUPID development in numerical aspects in more detail.

## References

[1] J. J. Jeong, K.S. Ha, B. D. Chung and W.J. Lee, "Development of A Multi-dimensional Thermal-Hydraulic System Code, MARS 1.3.1," Annals of Nuclear Energy, vol. 26, no. 18, pp. 1611-1642 (1999).

[2] S.Y. Lee, J.J. Jeong, S.H. Kim, and S.H. Chang, "COBRA/RELAP5; A Merged Version of the COBRA-TF and RELAP5/MOD3 Codes," Nuclear Technology, vol. 99, pp. 177-187 (1992).

[3] USNRC, RELAP5/MOD3 Code Manual Volume I: Code Structure, System Models, and Solution Methods, NUREG/CR-5535, USNRC, 1998.

[4] J.J. Jeong, H. Y. Yoon, I. K. Park, H. K. Cho, J. Kim, "A Semi-Implicit Numerical Scheme for Transient Two-Phase Flows on Unstructured Grids," Nuclear Engineering and Design, Vol. 238, 3403–3412 (2008).

[5] H. Y. Yoon, I. K. Park, Y. I. Kim, Y. D. Hwang, J. J. Jeong, "A Fast-Running Semi-Implicit Numerical Scheme for Transient Two-Phase Flows on Unstructured Grids," Numerical Heat Transfer Part B: Fundamentals, 56, 432–454 (2009).

[6] I.K. Park, H.K. Cho, H.Y. Yoon, J.J. Jeong, "Numerical Effects of the Semi-conservative Form of Momentum Equations for Multi-Dimensional Two-Phase Flows," Nuclear Engineering and Design, vol. 239, pp. 2365–2371 (2009).

[7] H. K. Cho, H. D. Lee, I. K. Park, J. J. Jeong, "Implementation of a Second-Order Upwind Method in a Semi-Implicit Two-Phase Flow Code on Unstructured Meshes," Annals of Nuclear Energy, 37, 606–614 (2010).