Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 12-13, 2016

On the Development of Multi-Dimensional RELAP5 with Conservative Convective Terms

Hyung Wook Jang, Sang Yong Lee*
KINGS, 658-91 Haemaji-ro, Seosaeng-myeon, Ulju-gun, Ulsan 45014, Korea
**Corresponding author:sangleey@kings.ac.kr*

1. Introduction

The phenomena of Loss of Coolant Accident (LOCA) have been investigated for long time. And the most extensive research project for LOCA was the 2D/3D program [1]. The results of the 2D/3D experiments were summarized as follows;

Flow conditions in the downcomer during end-of-blowdown were *highly multi-dimensional at full-scale*. During reflood, the distribution of water in the core was one-dimensional. But flow in the core exhibited multi-dimensionality. One-dimensional manometer oscillation between the downcomer and core was observed. The water level was higher in front of the broken cold leg nozzle than at other azimuthal positions. Flow phenomena at the tie plate were uniform.

With the background of 2D/3D study, Multi-dimensional codes such as TRAC [2], RELAP5-3D [3], CATHARE [4], SPACE [5], MARS [6] and COBRA-TF [7] were developed and applied to the safety analysis of reactor systems.

The most famous code RELAP5 [8] is still one-dimensional even though it has been applied to various licensing applications. Therefore, author developed the multi-dimensional capability and implemented it into RELAP5.

In this paper, two aspects concerning the multi-dimensional codes will be discussed. One of them is the properness of the type of the momentum equations. The other discussion will be the implementation of the conservative momentum flux term in RELAP5.

Table-1. Treatment of Momentum Equation in Codes

	RELAP5	RELAP5-3D	TRAC/ TRACE	CATHARE	COBRA-TF	CUPID
Dim.	1-D	3-D	3-D	3-D	3-D	3-D
Eqn.	Non-Cons.	Non-Cons.	Non-Cons.	Mod. cons.	Cons.	Mod Cons.
Phase	Phase int.	Phase Int.	Phase int.	Mass weight	Regular	Mass weight
Geo.	Network	Rect Cyl	Rect Cyl	Rect Cyl Sph	Rect	Unst.
Mesh	FVM	FDM	FDM	FVM	FVM	FVM

2. Momentum Equations in the Codes

2.1. Momentum equations in various forms

The multi-dimensional effects are simulated with the proper treatment of the momentum flux term in the momentum balance equations. Various modifications and/or simplifications of the momentum balance equations are made to implement the solution schemes for the individual codes. Table-1 shows such variations.

Time and volume averaged porous body mass and momentum equation for phase k are written [9,10,11];

$$\frac{\partial \epsilon\alpha_k\rho_k}{\partial t} + \nabla\cdot(\epsilon\alpha_k\rho_k\boldsymbol{v}_k) = \epsilon\Gamma_k \quad (1)$$

$$\frac{\partial \epsilon\alpha_k\rho_k\boldsymbol{v}_k}{\partial t} + \nabla\cdot\left(\epsilon C_{v_k}\alpha_k\rho_k\boldsymbol{v}_k\boldsymbol{v}_k\right) = -\epsilon\alpha_k\nabla p + \epsilon K_k\rho v^2 \ldots \quad (2)$$

The porosity ϵ is assumed 1.0 because it is not important for the following discussions. Source terms except friction are not shown either because of the same reason. The covariance coefficient, C_{v_k}, reflects the volume fraction distribution across the averaging volume. It was studied for the one-dimensional pipe flow [10,11]. It is $C_{v_k} \neq 1.0$. This is also true for general porous body multi-dimesional multi-fluid flow. But most of the present codes assume $C_{v_k} = 1.0$.

Then, momentum balance equation for phase k is written as;

$$\frac{\partial \alpha_k\rho_k\boldsymbol{v}_k}{\partial t} + \nabla\cdot(\alpha_k\rho_k\boldsymbol{v}_k\boldsymbol{v}_k) = \alpha_k\nabla p + K_k\rho v^2 .. (3)$$

The momentum loss due to the flow resistance (the wall friction plus the form loss) is usually correlated with total velocity head ρv^2 and proper phase partitioning factor. So, K_k implies the properly phase partitioned resistance factor.

Non-conservative form can be derived by expanding eqn.(3) and using mass conservation equation, eqn.(1);

$$\alpha_k\rho_k\frac{\partial \boldsymbol{v}_k}{\partial t} + \alpha_k\rho_k\boldsymbol{v}_k\cdot\nabla\boldsymbol{v}_k = -\alpha_k\nabla p + K_k\rho v^2 - v_k\Gamma_k \ldots \quad (6)$$

The phase intensive equation is written;

$$\frac{\partial \boldsymbol{v}_k}{\partial t} + \boldsymbol{v}_k\cdot\nabla\boldsymbol{v}_k = -\frac{1}{\rho_k}\nabla p + \frac{K_k}{\alpha_k\rho_k}\rho v^2 - \frac{v_k\Gamma_k}{\alpha_k\rho_k} \ldots (7)$$

The immediate problem with these equations is that discretizing the eqn.(6,7) in finite volume method is not easy. To overcome this problem, Weller [12] used the modified non-conservative momentum equations as follows;

$$\boldsymbol{v}_k\cdot\nabla\boldsymbol{v}_k \equiv \nabla\cdot(\boldsymbol{v}_k\boldsymbol{v}_k) - \boldsymbol{v}_k(\nabla\cdot\boldsymbol{v}_k) \quad (8)$$

It was realized that the estimated momentum fluxes with them are not correct because they are not reflecting mass flux effects correctly. To overcome this problem, the mass weighted modified non-conservative method [13] is used. In that, the following equality is used.

$$\alpha_k\rho_k\boldsymbol{v}_k\cdot\nabla\boldsymbol{v}_k \equiv \nabla\cdot(\alpha_k\rho_k\boldsymbol{v}_k\boldsymbol{v}_k) - \boldsymbol{v}_k(\nabla\cdot\alpha_k\rho_k\boldsymbol{v}_k) \quad (9)$$

The manipulations in eqn.(6,7) is totally relied on the assumption $C_{v_k} = 1.0$. If this is not held, the following equation is derived;

$$\alpha_k \rho_k \frac{\partial \boldsymbol{v}_k}{\partial t} + C_{v_k} \alpha_k \rho_k \boldsymbol{v}_k \cdot \nabla \boldsymbol{v}_k + \boldsymbol{v}_k \nabla \cdot \left(\left(C_{v_k} - 1 \right) \alpha_k \rho_k \boldsymbol{v}_k \right) = -\alpha_k \nabla p + K_k \rho v^2 - v_k \Gamma_k \ldots \quad (10)$$

It means that the non-conservative form is not available. Therefore, the non-conservative form of momentum equations for the time-volume averaged on porous body should not be constructed.

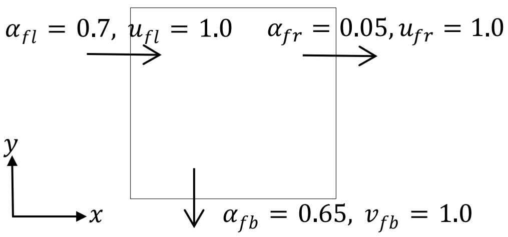

Fig.1. Flow Configuration of a Cell

2.2. Treatment of momentum flux term in codes

To solve for the velocity, many codes use non-conservative form of momentum equation (Table-1). TRAC and TRACE [14] use the phase intensive momentum equation like eqn.(7). Finite difference method is applied to discretize the momentum equations in TRAC and TRACE. Fig.1 shows the situation in that wrong estimation of momentum flux may happen.

Using the upwind scheme, the axial convection can be determined by

$$\frac{u_{fl}\left(u_{fl} + u_{fr}\right)}{2} = 1.0 \quad (10)$$

in non-conservative form but it is

$$\frac{u_{fl}\left(u_{fl} + u_{fr}\right)}{2} \frac{\alpha_{fl}}{\alpha_{fr}} = 14.0 \quad (11)$$

in conservative form.

Similarly the cross convection is estimated by

$$v_{fb} u_{fr} = 1.0 \quad (12)$$

in non-conservative form but it is

$$v_{fb} u_{fr} \frac{\alpha_{fb}}{\alpha_{fr}} = 13.0 \quad (13)$$

in conservative form. The estimated conservative momentum flux is normalized against the junction flow. Above estimation tells that non-conservative approach can't estimate the momentum flux correctly in this configuration.

Knowing this problem, CATHARE use the modified non-conservative, mass weighted form like eqn.(9). Since the momentum equations can be represented nearly conservative form in this approach, finite volume method is naturally adopted for their discretization. Some accuracy loss is inevitable to discretize the $\boldsymbol{v}_k(\nabla \cdot \alpha_k \rho_k \boldsymbol{v}_k)$ term of eqn.(9) because it is not fully conservative.

COBRA-TF use regular conservative momentum equations like eqn.(3). And it solves for the momentum flux. Discretization of the conservative momentum equations through the finite volume method is rather straightforward. Unlike the non-conservative equation, second order accuracy of the discretization can be kept.

2.3. Development of multi-Dimensional RELAP5 by inserting the conservative momentum flux terms

Since RELAP5 is basically developed through the one-dimensional non-conservative finite difference approach, at a first glance, it seems to be very difficult to implement the momentum flux in conservative form. But a little careful investigation is enough to recognize that the implemented algorithms in RELAP5 are directly applicable to the conservative form. Instead of the eqn.(6), equivalently, one can solve the following equation;

$$\alpha_k \rho_k \frac{\partial \boldsymbol{v}_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \boldsymbol{v}_k \boldsymbol{v}_k) = \alpha_k \nabla p + K_k \rho v^2 .. (14)$$

The reason for this idea is as follows; the discretization procedure for all terms in the eqn.(6) is basically the same as the one for eqn.(14) or eqn.(3), except the temporal derivative term and the convection term. The discretization of the temporal term for the conservative momentum equation (3) is;

$$\frac{(\alpha_k \rho_k \boldsymbol{v}_k)^{\boldsymbol{n+1}} - (\alpha_k \rho_k \boldsymbol{v}_k)^{\boldsymbol{n}}}{\Delta t} \quad (15)$$

For one step semi-implicit solver like RELAP5, eqn.(15) should be replaced with following equation when the velocity is inserted into mass and energy conservation equations.

$$\frac{(\alpha_k \rho_k)^{\boldsymbol{n}} (\boldsymbol{v}_k)^{\boldsymbol{n+1}} - (\alpha_k \rho_k)^{\boldsymbol{n}} (\boldsymbol{v}_k)^{\boldsymbol{n}}}{\Delta t} \quad (16)$$

Eqn.(16) is really the discretized form of the temporal term in eqn.(14) and also in the non-conservative equation (6). The phase change term $v_k \Gamma_k$ used in RELAP5 should be neglected if the conservative momentum equation (3) is solved. The convection term is solved explicitly in RELAP5. Therefore, it is relatively easy to modify it in to the conservative form.

The validity of the implementation is checked through the simple 2-dimensional flow test simulation. As shown in Fig.2, 5x5 2-dimensional planar rectangular tank case is chosen. It is originally filled with water at 1.0 bar and 300^{o}K. At the bottom left corner, vapor and liquid are fed into the system with velocity of 10m/sec. The injection fluid property is the saturated two phase with the static quality of 0.0005. It is about 33% of volume fraction. Outlet is set to 1.0bar with the same property.

This conceptual problem is to see the phenomena that the centrifugal force can separate the phases. The fast injected two phase flow at the bottom left corner of the tank follows the bottom boundary face and turns upward to follow the right boundary face. Then, the main flow drives the water in the tank to turn to the counterclockwise direction. It is expected that the liquid pursues the outward direction while the light phase, vapor should be pushed inward direction. This phenomenon depends on the action of convective terms. The proper treatment of the convective terms in the numerical solution process is inevitable to see the effects. The calculation results of the various codes will be collected and discussed in section 3.

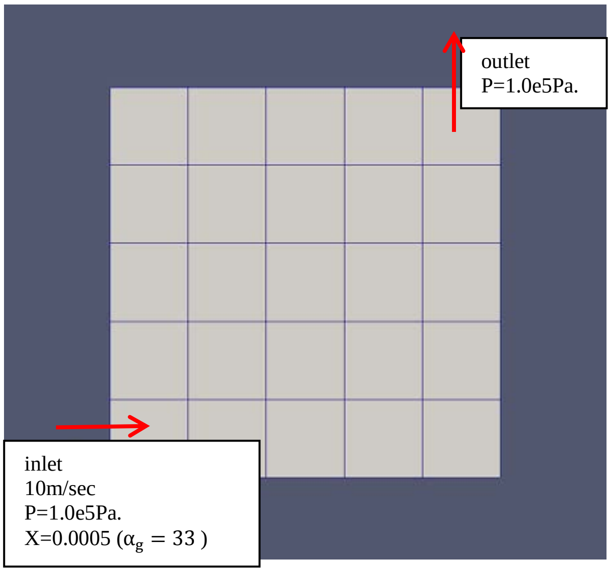

Fig.2. 5x5 2-dimensional Flow Test Simulation

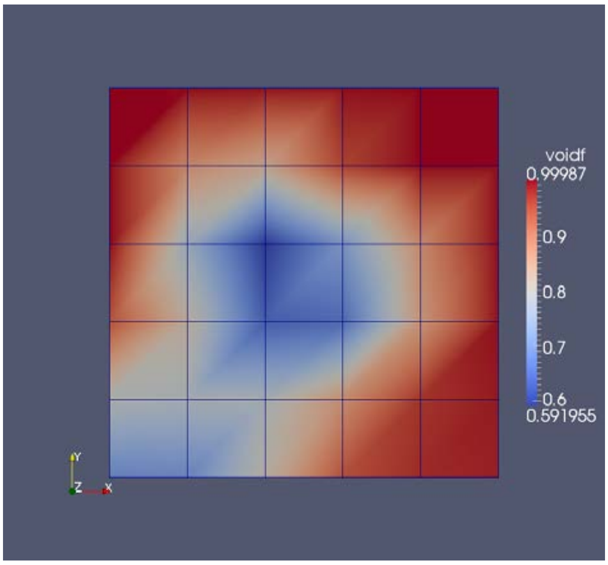

Fig.3. Liquid Fraction of Conservative RELAP5

3. Predictions of the Conservative RELAP5 and Comparisons with Other Codes

3.1. Predictions of the RELAP5 with the conservative momentum convection terms

The calculated liquid volume fraction is shown in Fig. 3. The liquid fraction is high in the area near boundary faces. And it is low in the central area. This is intuitively correct because the heavy liquid phase rush to the outward direction. And, as a result, light vapor phase is pushed to the inward direction. This trend resultantly push light gas phase into lower velocity region that is formed in the central region shown by the blue color in Fig.3.

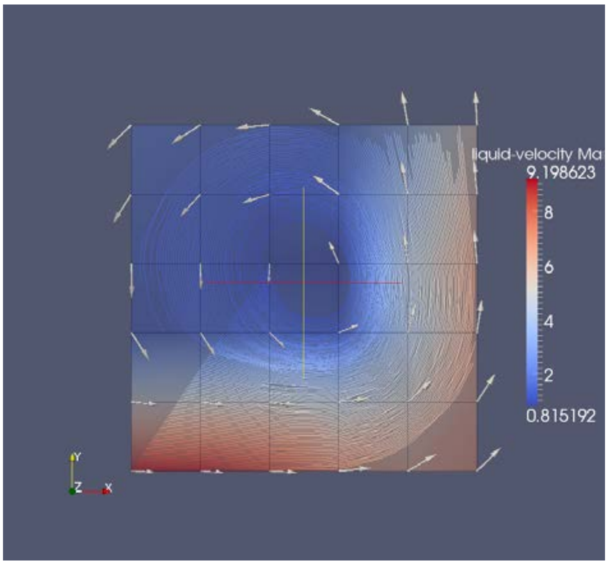

Fig.4. Liquid Velocity Field of Conservative RELAP5

The glyph vectors and stream lines of liquid flow field in Fig.4 tell the flow direction very well. The centrifugal force generated by the swirling outside liquid flow effectively pushes the light phase inside of the moving circle. Fig.5 shows the vapor velocity field. The glyph vectors indicate the general trend that the vapor phase flow inward direction. The vapor flow stream lines that start at the tank inlet face move inward direction and converge to the one point in the central region.

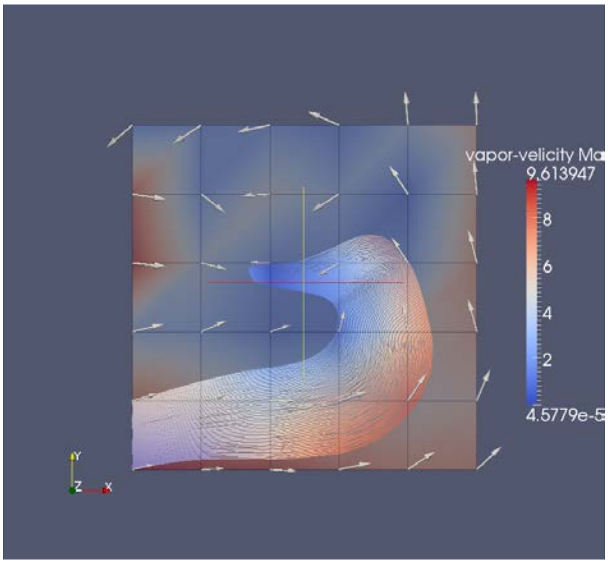

Fig.5. Vapor Velocity Field of Conservative RELAP5

Four stream lines are shown in Fig.6. The red and blue lines are liquid and vapor stream lines respectively. They start form the same point of coordinate (0.0, 0.5, 0.5). It is the center of injection face. The red one, as expected, follows the boundary faces and goes to the exit. The blue one turns inward relative to the red liquid stream line and eventually reaches the final destination in the central area.

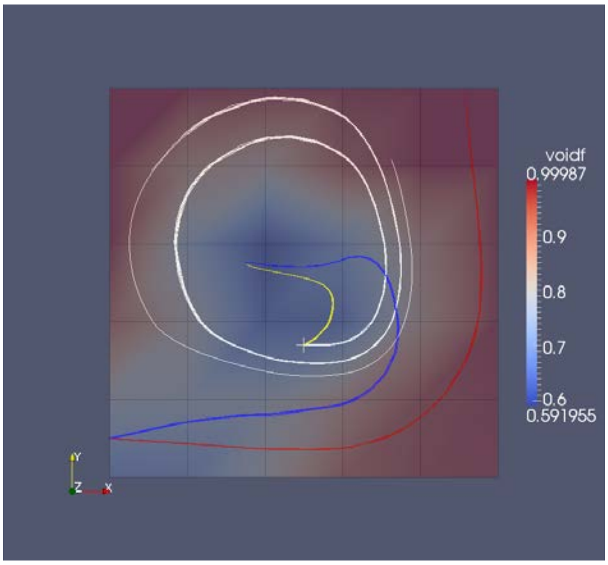

Fig.6. Stream Lines of Conservative RELAP5

The white and yellow lines in Fig.6 are liquid and vapor stream lines respectively. They start form the point of coordinate (2.5, 1.7, 0.5). The yellow vapor stream line converges to the final destination as expected. But the white liquid stream line shows very interesting track. It turns left and moves outward.

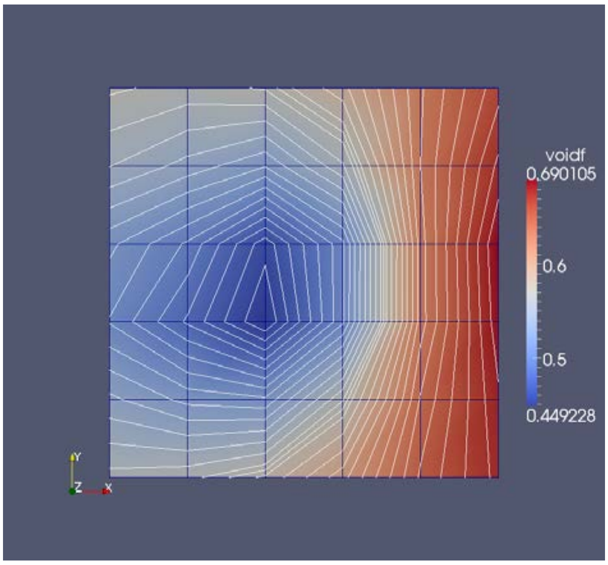

Fig.7. Liquid Fraction of MARS/Multi-D

3.2. Results of the MARS/MultiD

The calculation results of MARS are shown in Fig.7, 8 and 9. MultiD component is used to model the test planar tank. It adopts the non-conservative form in the finite difference approach. This method is exactly the same as the one used in RELAP5-3D. Therefore, any conclusions made here is directly applicable to RELAP5-3D.

The liquid velocity field in Fig.8 looks the same as the conservative RELAP5 case in Fig.3. The liquid fraction in Fig.7 shows some phase separation, but it is a little different from the conservative RELAP5 result (Fig.3). It seems that the liquid momentum is not enough to turn it round along the outside boundary faces in MARS MultiD.

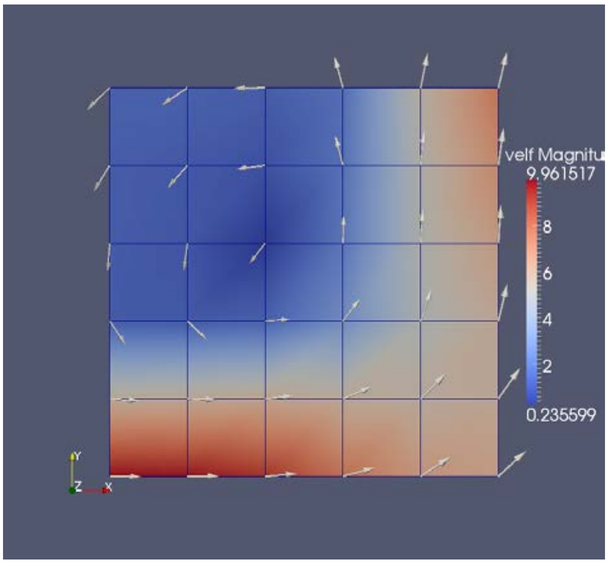

Fig.8. Liquid Velocity Field of MARS/MultiD

The vapor velocity field in Fig.9 shows the trend as that of the liquid field. This means that the non-conservative form in the finite difference approach may not be appropriate to estimate the momentum flow. This phenomena was already pointed out with CUPID [

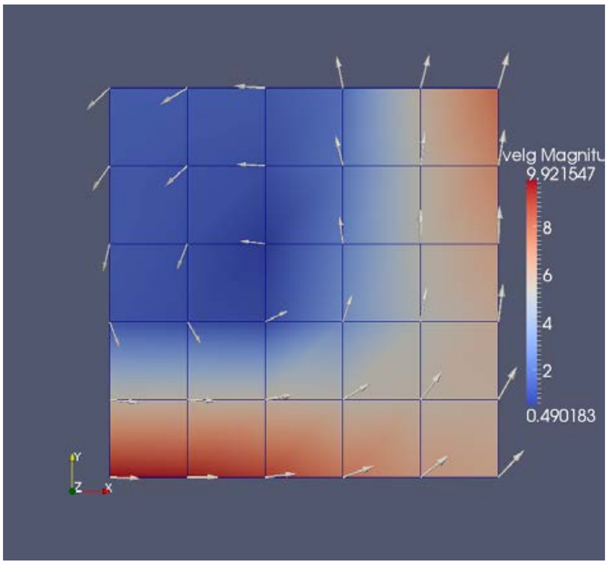

Fig.9. Vapor Velocity Field of MARS/MultiD

3.3. Results of original RELAP5 with cross flow

For comparison, original RELAP5 runs were performed with and without the cross flow options at the cross flow junctions. The calculation results of original RELAP5 **with** cross flow are shown in Fig.10 and 11.

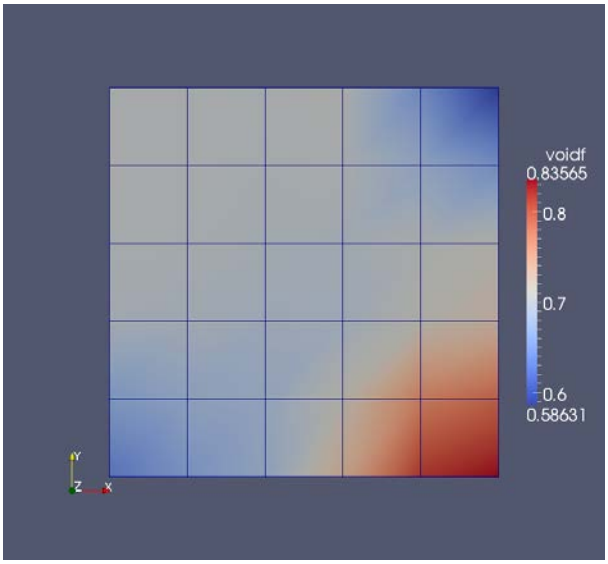

Fig.10. Liquid Fraction of original RELAP5 with cross flow

In Fig.10, the liquid fraction in original RELAP5 run is higher at near the right bottom boundary where the main flow meets the wall. The liquid fraction near the left boundary is lower than the right bottom area. Even though the liquid velocity field shows some circulation, the center is not shifted toward left as in the conservative RELAP5 case (Fig.12). There is no centrifugal separation of phases.

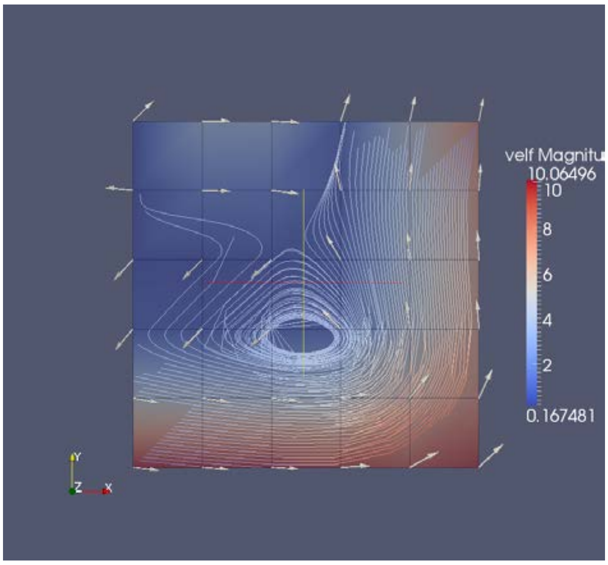

Fig.11. Liquid Velocity Field of original RELAP5 with cross flow

3.4. Results of original RELAP5 with no cross flow

The calculation results of original RELAP5 **without** cross flow are shown in Fig.12 and 13.

As shown in Fig.12, the main flow runs through from the inlet to the outlet. Even though flow is injected at the bottom left corner facing X direction, there is no directional preference in the velocity field as expected (Fig.13). This is the typical shape of the solution for Darcy type problem. As expected, there is no centrifugal separation of phases.

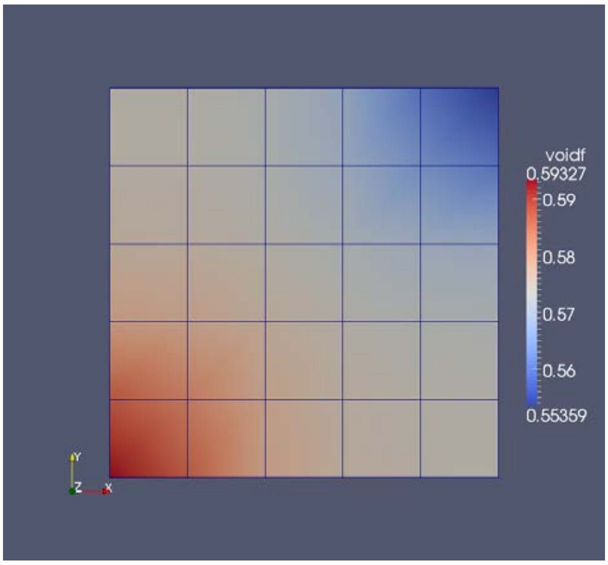

Fig.12. Liquid Fraction of original RELAP5 without cross flow

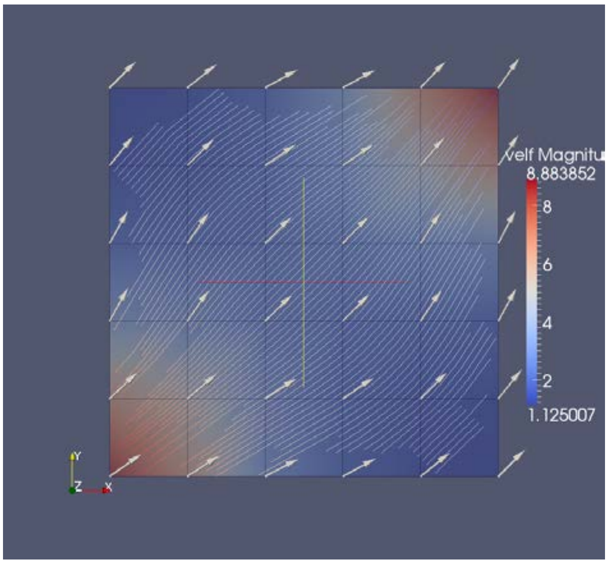

Fig.13. Liquid Velocity Field of original RELAP5 without cross flow

3.5. Results of CUPID

The calculation results of CUPID are shown in Fig.14 and 15. Unlike other codes, CUPID can handle unstructured collocated mesh. It use the modified non-conservative form like eqn.(9) [13]. As already pointed, the treatment of the momentum convection term with the mass flux based finite volume discretization is enough to handle the centrifugal phase separation. Fig.14 is very similar to the Fig.3.

As presented in Fig.14, vapor is successfully separated from the stream and collected at the central region. The liquid velocity field is also very similar to the result of the conservative RELAP5 in Fig.4.

CUPID results are a kind of validation of the implementation of the conservative convective terms in RELAP5 by authors.

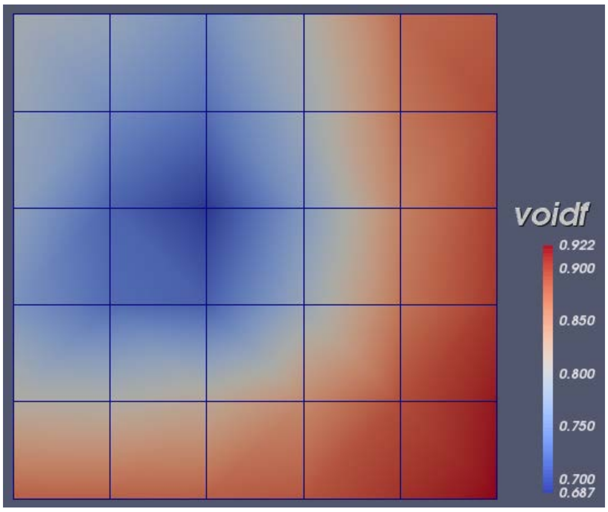

Fig.14. Liquid Fraction of CUPID

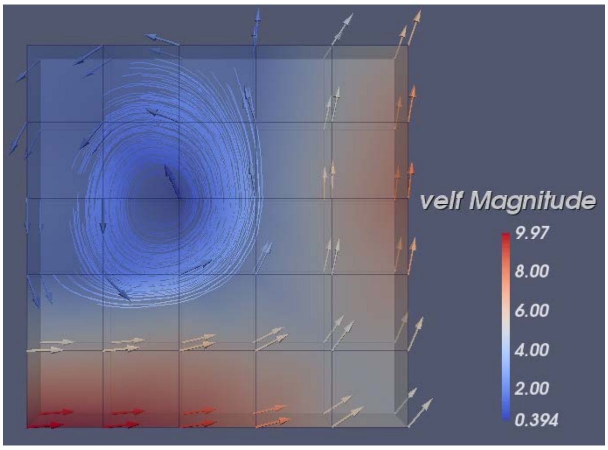

Fig.15. Liquid Velocity Field of CUPID

3.6. Results of TRACE

The liquid fraction in Fig.16 shows that TRACE can handle the phase separation problem very well. The liquid velocity field in Fig.17 is also very similar to the result of the CUPID (Fig.15) or the conservative RELAP5 (Fig.4).

4. Discussions and Perspectives

From the present study, it can be concluded that the intuitive centrifugal phase separation is reproduced with the conservative, modified conservative or non-conservative convective terms in the momentum equations. The non-conservative form in finite difference approach may not be good for the problem that have strong gradient of the volume fraction. MARS MultiD and SPACE are such codes.

The implementation of the conservative convective terms in RELAP5 seems to be successful. Further elaboration of improvement activities such as input handling system may be necessary to develop the RELAP5 as a fully multi-dimensional code.

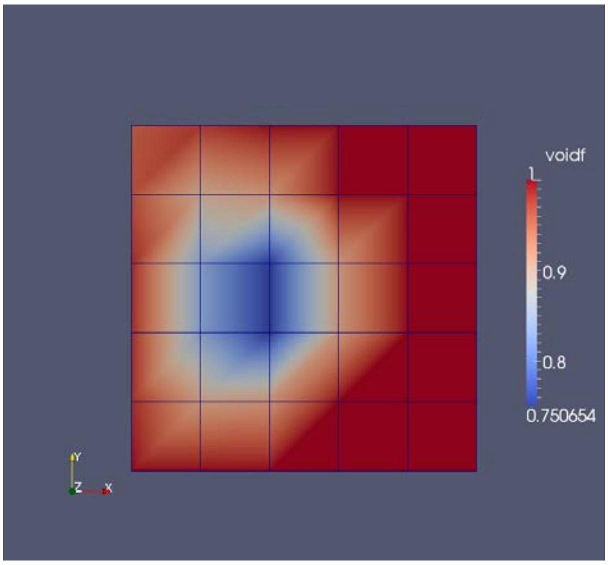

Fig.16. Liquid Fraction of TRACE

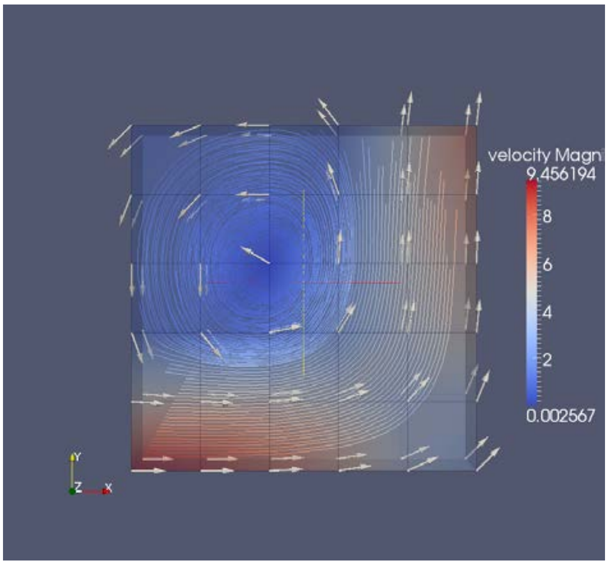

Fig.17. Liquid Velocity Field of TRACE

REFERENCES

[1] J. W. Simons, 2D/3D Program, Work Summary Report, NUREG/IA-0126, MPR Associates, Inc. June 1993.

[2] Spore, J.W., et al., TRAC-PF1/MOD2 Volume I Theory manual, NUREG/CR-5673, 1993, Los Alamos National Laboratory.

[3] RELAP5-3D Code Manual, Volume I: Code Structure, System Models and Solution Methods, INEEL-EXT-98-00834, Revision 4.0, June 2012.

[4] I. Dor, et al., CATHARE 3D Module, Nureth-15, Pisa, Italy, May 12-17, 2013.

[5] S. J. Ha, C. E. Park, K. D. Kim, and C. H. Ban, “Development of the SPACE Code for Nuclear Power Plants”, Nuclear Technology, Vol. 43, No. 1 (2011).

[6] MARS Code manual volume I: Code Structure, System Models, and Solution Methods KAERI/TR-2812/2004, December 2009, Korea Atomic Energy Research Institute.

[7] M. Thurgood, et. al., “COBRA/TRAC: A Thermal Hydraulics Code for Transient Analysis of Nuclear Reactor Vessels and Primary Coolant Systems”, US.NRC, NUREG/CR-3046, 1983.

[8] RELAP5/MOD3.3 Code manual, vol-I: Validation of numerical techniques in RELAP5/MOD3.0, Nuclear Safety Analysis Division, NUREG/CR-5535/Rev-1, December, 2001.

[9] S. Y. Lee, T. Hibiki, M. Ishii, “Formulation of time and volume averaged two-fluid model considering structural materials in a control volume”, NED 239 (2009) 127−139.

[10] M. Ishii, T. Hibiki, Thermo-Fluid Dynamics of Two-Phase Flow, Second Edition, Springer, 2011.

[11] M. Ishii K. Mishima, “Two-fluid model and hydrodynamic constitutive relations”. NED 82, (1984) 107-126.

[12] H. Weller, “Derivation, modelling and solution of the conditionally averaged two-phase flow equations”, Technical Report TR/HGW/02, Nabla Ltd, 2002.

[13] Jeong, J.J. et al., 2008b. “A semi-implicit numerical scheme for transient two-phase flows on unstructured grids”. NED 238, 3403–3412.

[14] U.S. Nuclear Regulatory Commission, TRACE V5.0, Theory Manual, 2000.

[15] I.K. Park, et al., “Numerical effects of the semi-conservative form of momentum equations for multi-dimensional two-phase flows”, Nuclear Engineering and Design 239 (2009) 2365–2371.