An Improved Numerical Scheme to Calculate the Pressure Gradient on Unstructured Meshes for Two-Phase Flows

Heedong Lee[a], Jae Jun Jeong[b], Hyoung Kyu Cho[b], Han Young Yoon[b]
[a]Department of Aerospace Engineering, Korea Advanced Institute of Science and Technology,
373-1, Yuseong-gu, Daejeon, Korea, E-mail: heedongi94@kaist.ac.kr
[b]Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon, 305-353, Korea

1. Introduction

KAERI has developed a three-dimensional thermal hydraulics code, CUPID [1, 2], for the analysis of transient, multi-dimensional, two-phase flows in nuclear reactor components. The code was designed for use as a component-scale code, and/or a three-dimensional component, which can be coupled with a system code. In the CUPID code, a two-fluid three-field model is adopted for two-phase flows, and the governing equations are solved on unstructured grids, which are very useful for the analysis of flows in complicated geometries.

This paper presents a new numerical scheme to calculate the pressure gradient of the two-fluid momentum equation on unstructured grids. The new scheme is compared with the previous schemes in the CUPID code, and the results are discussed.

2. An Improved Numerical Scheme for the Pressure Gradient Calculation

In the previous versions of the CUPID code [1, 2], the pressure gradient at a cell center was evaluated by using the Green-Gauss reconstruction method:

$$\nabla P_0 = \frac{1}{V_0}\sum_f P_f \underline{n}_f \underline{S}_f \tag{1}$$

where the subscript f indicates the faces of the cell 0. In the case of a two-dimensional mesh, the pressure at the face f was determined by:

$$P_f = \frac{w_0 P_0 + w_1 P_1}{w_0 + w_1} \tag{2}$$

where P_1: the pressure at the cell 1, which is connected to the cell 0 via the face f, and the weighting factor w_0 and w_1 are obtained using the inverse-distance method.

This method, i.e., the Green-Gauss reconstruction with an inverse-distance weighting, has a second-order accuracy on structured meshes, but may have some error for skewed unstructured meshes. To overcome this problem, the Frink's reconstruction was adopted in the new version of CUPID [3]; the cell face pressure was calculated by the interpolation of the face node pressures:

$$P_f = \frac{1}{2}\left(P_a + P_b\right) \tag{3}$$

where P_a and P_b are the pressure at the nodes a and b, consisting of the face f. The pressure at a node is obtained by

$$P_n = \sum_{i=1}^{n_c} w_{c,i} P_{c,i} \Big/ \sum_{i=1}^{n_c} w_{c,i} \tag{4}$$

where the weighting factor was determined by using the pseudo-Laplacian method[3]. The Frink's reconstruction provides a second-order accuracy on both structured and unstructured meshes.

The reconstruction of the pressure gradient at a boundary cell is somewhat different from that at inner cells because of the interpolation error of the pressure at the boundary face. To mitigate this error, the pressure gradient at the boundary cell was evaluated by the least-square reconstruction method in the CUPID code [4]. This method greatly enhances the accuracy of the pressure gradient at the boundary cells. However, the results of numerical tests showed non-negligible error at the boundary cells of skewed unstructured meshes.

Thus, a new iterative method based on the Frink's method was developed to evaluate the pressure gradient at the boundary cells. The iteration sequence is:

(a) The node pressure P_n is determined from Eq. (4).
(b) The pressure at a cell face P_f is calculated by using the node pressures P_n. In the case of a two-dimensional mesh, P_f is calculated by Eq. (3).
(c) Using the Green-Gauss method, the pressure gradient at a cell center is reconstructed:

$$\nabla P_c = \frac{1}{V}\sum_f P_f \underline{n}_f \underline{S}_f \,. \tag{5}$$

(d) Using the reconstructed pressure gradient in Eq. (5), the node pressure at the boundary is updated as follows:

$$P_n^{new} = \frac{1}{n_c}\sum_{i=1}^{n_c}\left[P_{c,i} + \nabla P_{c,i}\cdot\left(\underline{x}_n - \underline{x}_{c,i}\right)\right] \tag{6}$$

(e) The above steps (b) through (d) are repeated until the node pressure at the boundary converges to a certain value, i.e., $\left|P_n^{new} - P_n\right| < \varepsilon_{crit}$.

3. The Results of Numerical Tests

To assess the accuracy of the above iterative methods, a simple test function in a two-dimensional space was introduced:

$$f_{test}(x, y) = x + y \,. \tag{7}$$

Then, the gradient of f_{test} is exactly obtained:

$$\nabla f_{test} = \underline{i} + \underline{j} \,. \tag{8}$$

For the numerical tests, a structured grid and an unstructured grid in Fig. 1 were used with the following three numerical methods for evaluating the gradient:

(a) GG(IDW): The Green-Gauss method with the

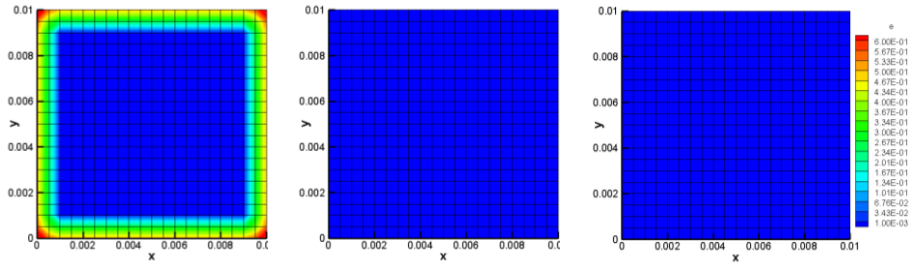
inverse distance weighting.

(b) GG(IDW)+LS(B): The Green-Gauss method, with the inverse distance weighting, for the internal cells and the least square method for the boundary cells.

(c) GG(FRK_{ITER}): The Green-Gauss method with the pseudo-Laplacian weighting and the iterative process for the boundary cells.

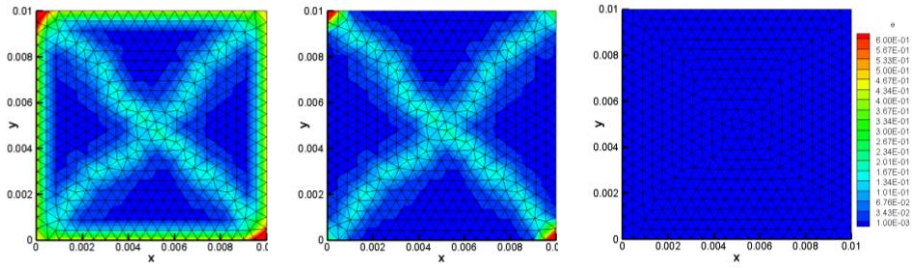
In the numerical test, the cell center values were exactly obtained from Eq. (7). Then, the gradients at the cell centers are numerically obtained by using the above three methods. The results of numerical tests are shown in Fig. 1, where the absolute value of the difference between the numerical and exact gradients is presented [4]. The errors generated in the corner boundary cells tend to propagate into the inner cells when the first two methods are used. However, the new iterative method, GG(FRK_{ITER}), yields very accurate results for both the structured and unstructured grids. The numbers of iterations were about 30 for structured meshes and about 90 for unstructured meshes, where ε_{crit} was 10^{-7}. This requires additional computation time, but the increment is negligible because the number of boundary cells is very smaller than that of the internal cells.

The numerical tests using the well known cavity problem also showed the new iterative method leads to the most accurate results for both the structured and unstructured grids [4].



GG (IDW) GG(IDW)+LS(B) GG(FRK_{ITER})

(a) Structured grid



GG (IDW) GG(IDW)+LS(B) GG(FRK_{ITER})

(b) Unstructured grid

Fig. 1. The errors of the gradients of the test function with different numerical methods and grids.

4. Conclusions

The accurate evaluation of the pressure gradient term in the two-fluid momentum equation is very important, In the course of the CUPID code development, a new numerical scheme to calculate the pressure gradient of the two-fluid momentum equation on unstructured grids was developed. The new scheme was compared with the previous schemes in the CUPID code, (i) the Green-Gauss method with the inverse distance weighting, and (ii) the Green-Gauss method for the internal cells and the least square method for the boundary cells. The results of numerical tests showed that the new scheme is very accurate for both structured and unstructured grids, and that the previous methods, on the other hand, have significant errors when applied on unstructured grids.

Acknowledgement

This work was supported by Nuclear Research & Development Program of the NRF (National Research Foundation) grant funded by the Ministry of Education, Science and Technology) of the Korean government (Grant code: M20702040002-08M0204-00210).

References

[1] J.J. Jeong, et al., A Semi-implicit numerical scheme for transient two-phase flows on unstructured grids, *Nuclear Engineering and Design*, **238**, pp. 3403–3412 (2008).

[2] H.K. Cho, et al., Computational analysis of downcomer boiling phenomena using a component thermal hydraulic analysis code CUPID, to be presented at the 18th International Conference on Nuclear Engineering, Xi'an, China, May 17-21, 2010.

[3] N.T. Frink, Recent progress toward a three-dimensional unstructured Navier-Stokes flow solver, *AIAA Paper 94-0061* (1994).

[4] H.D. Lee, A new method for the reconstruction of the pressure gradient for the CUPID code, CTHA-2009-005, Korea Atomic Energy Research Institute (2009).