Application of an Unstructured Mesh Semi-implicit Scheme to the Multi-D Single Phase Flow

J. C. Park, S. C. Moon, C. E. Park, and S. Y. Lee
Korea Power Engineering Company, Inc., 150 Deokjin-dong, Yuseong-gu, Daejeon, 305-353

1. Introduction

KOPEC has developed a pilot code to analyze the two phase flow[1]. The earlier version of the pilot code adopts a semi-implicit numerical solution scheme, and represents the geometry with the one-dimensional structured mesh system. It does not provide users with sufficient flexibility to model the complex geometry.

In this paper, the unstructured mesh system is implemented to the pilot code, in order to model the wide variety of geometry encountered in nuclear reactor system. The governing equations are discretized using general polyhedral mesh. The collocated mesh system, where all the major variables including pressure and velocity are placed at the same node, is also used instead of the staggered one. To avoid the checkerboard effect, which is caused by the decoupling of odd node values from even node values in the collocated mesh system, Rhie and Chow interpolation scheme[2] imitating the staggered grid solution is used. Finally, the revised pilot code is verified by applying to a two-dimensional single phase cavity flow problem.

2. Discretized Field Equations

2. 1 General polyhedral mesh

Discretization of the integral form of governing equations over each control volume is performed, based on Finite Volume Method (FVM). As shown in Fig. 1, the control volume $\boldsymbol{V_P}$ is a polyhedron surrounded by the flat faces with area $\boldsymbol{A^E}$.

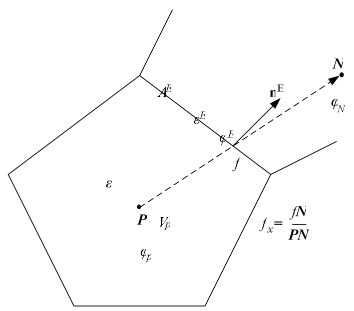

Fig 1. Control volume

Body of the control volume is assigned volume porosity ε and the face with directional porosity ε^E. Computational point $\boldsymbol{P}$ is located at the centroid of the control volume. The cell with the lower label is called owner of the face. The cell of opposite side of the face is called neighbor. Normal vector $\mathbf{n}^{\mathbf{E}}$ of face E is pointing the centroid of the neighbor cell $\boldsymbol{N}$.

2.2 Discretized equations

The governing equations can be described by standard form of temporal, spatial, convection, diffusion and source terms. The discretization of each term is obtained by transformation between volume integral and surface integral by Gauss' theorem. For discretization of convection term, there is.

$$\int_{V_P} \nabla \cdot \left(\varepsilon^E \alpha \rho \mathbf{U} \psi \right) dV = \sum_E \mathbf{A}^E \cdot \left(\varepsilon^E \alpha \rho \mathbf{U} \psi \right)^E$$

The detailed descriptions for the discretization of the other terms are given in Reference[3].

The discretized equations are:

- Continuity equation

$$\frac{\varepsilon V}{\Delta t}\left(\alpha_k^{n+1} \rho_k^{(n+1)} - \alpha_k \rho_k \right) + \sum_{i=1, id=facEid(i)}^{facEcount} \iota(i) \varepsilon_{(id)}^E A_{(id)}^E \, {}^d\alpha_{k(id)} \, {}^d\rho_{k(id)}^E \mathbf{n}_{(id)}^E \cdot \mathbf{U}_{k(id)}^{E\,(n+1)}$$
$$= \gamma_k + \theta_k$$

- Momentum equation

$$\frac{\varepsilon V}{\Delta t}\left(\mathbf{U}_k^{(n+1)} - \mathbf{U}_k \right) = -\sum_{i=1, id=facEid(i)}^{facEcount} \iota(i) \varepsilon_{(id)}^E A_{(id)}^E \, {}^d\mathbf{U}_{k(id)}^E \left(\mathbf{n}_{(id)}^E \cdot \mathbf{U}_{k(id)}^E \right)$$
$$+\mathbf{U}_k \sum_{i=1, id=facEid(i)}^{facEcount} \iota(i) \varepsilon_{(id)}^E A_{(id)}^E \left(\mathbf{n}_{(id)}^E \cdot \mathbf{U}_{k(id)}^E \right) - \frac{1}{\rho_k} \sum_{i=1, id=facEid(i)}^{facEcount} \iota(i) \varepsilon_{(id)}^E A_{(id)}^E \mathbf{n}_{(id)}^E P_{(id)}^{E\,(n+1)}$$
$$+\varepsilon V \frac{1}{\alpha_k \rho_k}\left(-F_{wk} \mathbf{U}_k^{(n+1)} \right) + \varepsilon V \mathbf{B} + \mathbf{M}_k + \mathbf{\Theta}_k$$

- Energy equation

$$\frac{\varepsilon V}{\Delta t}\left[\left(\alpha_k^{n+1} \rho_k^{(n+1)} e_k^{(n+1)} - \alpha_k \rho_k e_k \right) + P\left(\alpha_k^{(n+1)} - \alpha_k \right) \right] =$$
$$-\sum_{i=1, id=facEid(i)}^{facEcount} \iota(i) \varepsilon_{(id)}^E A_{(id)}^E \, {}^d\alpha_{k(id)}^E \begin{pmatrix} {}^d\rho_{k(id)}^E \, {}^d e_{k(id)}^E \\ +P_{(id)}^E \end{pmatrix} \mathbf{n}_{(id)}^E \cdot \mathbf{U}_{k(id)}^{E\,(n+1)} + E_k + \Phi_k$$

$$(k = phase)$$

3. Time Advancement

The basic time advancement follows the semi-implicit scheme. Rhie and Chow interpolation scheme is used to avoid the checkerboard effect appeared in the collocated mesh system.

The discretized momentum equation at cell can be written in following relation between the cell-center velocity and the face pressures:

$$\mathbf{u}_k^{n+1} = \xi_k \nabla P^{n+1} + {}^{\exp}\mathbf{u}_k$$

To find the phasic velocities at the faces, the first part ξ and the quantity ${}^{\exp}\mathbf{u}$ are interpolated linearly between the adjacent nodes.

$$\mathbf{u}_k^{f(n+1)} = \xi_k^f \nabla P^{f(n+1)} + {}^{\exp}\mathbf{u}_k^f$$

Where,

$$\xi_{k\,(id)}^E := (1 - f_{(id)}^E)\xi_{(k)P} + f_{(id)}^E \xi_{(k)N}$$

$${}^{\exp}\mathbf{U}_{(k)(id)}^E := (1 - f_{(id)}^E)\,{}^{\exp}\mathbf{U}_{(k)P} + f_{(id)}^E\,{}^{\exp}\mathbf{U}_{(k)N}$$

$$f_{(id)}^E = \frac{\overline{FN}}{\overline{PN}}$$

These face velocities are substituted into continuity and energy equation. As a consequence, the cell matrix inversion leads to a single equation involving only pressures. This is done for each cell, giving rise to the system pressure matrix. After the system pressure matrix is solved, the solutions for independent variables are obtained by the back substitution

4. Test results

4.1 Test problem

For the purpose of verification, the revised pilot code is applied to a two-dimensional single phase vapor cavity flow problem. The test domain, as shown in Fig. 2, consists of 27 uniform regular hexahedrons. The inlet boundary condition is given at the lower left corner of the rectangular cavity, and the outlet boundary condition is given at the upper right corner: the inlet flow velocity is 1 m/s, and the outlet pressure is 10 bar.

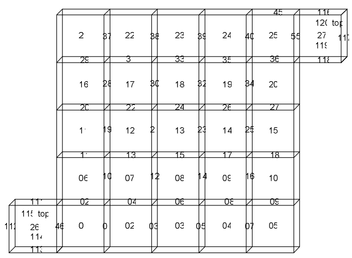

Figure 2 Geometry of the test problem

4.2 results

Fig. 3 and 4 show the velocity vectors at the faces and the pressure distribution, respectively. The cavity flow is mainly formed along the bottom and the right walls. The remaining part of the cavity shows a flow pattern of swirl. The pressure distribution in the cavity shows the maximum pressure(red color) at the lower right corner, and almost uniform pressure distribution in the swirling region.

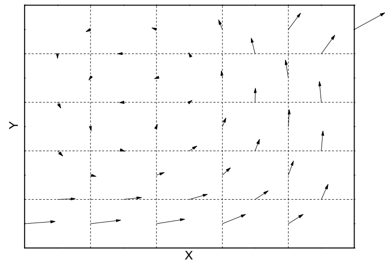

Figure 3 Velocity vector distribution

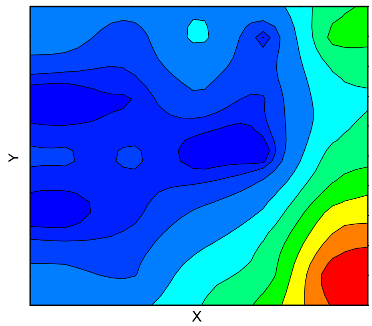

Figure 4 Pressure distribution

5. Conclusion

The application scope of the pilot code, which has been developed to analyze one-dimensional two phase flow[1], is extended to the more complicated multi-dimensional geometry, by newly implementing the unstructured mesh system and the revised pilot code is verified by applying to a two-dimensional single phase cavity flow problem. The test results show that the variation of major hydraulic parameters such as velocity, pressure is quite reasonable in the qualitative aspect.

Acknowledgment

This study was performed under the project, "Development of safety analysis codes for nuclear power plants" sponsored by the Ministry of Commerce, Industry and Energy.

REFERENCES

[1] S. Y Lee, Development of a Hydraulic Solver for the Safety Analysis Codes for Nuclear Power Plants(I). Korean Nuclear Society Spring Meeting, 2007.

[2] C.M. Rhie and W.L. Chow. Numerical study of the turbulent flow past an airfoil with trailing edge separation. AIAA Journal, 21:1525-1532, 1983.

[3] S. Y. Lee, Development of a Hydraulic Solver for the Safety Analysis Codes for Nuclear Power Plant(I), The KINS Workshop, 2007.