Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 29-30, 2009

A Unified Version of Staggered and Collocated Hydraulic Solvers for the SPACE Code

Chan Eok Park*, Sang Yong Lee, and Eun Kee Kim
Korea Power Engineering Company, Inc., 150 Deokjin-dong, Yuseong-gu, Daejeon, 305-353
**Corresponding author:cepark@kopec.co.kr*

1. Introduction

In order to develop the Korean best-estimate safety analysis code for nuclear power plants, SPACE, several mesh systems and numerical schemes, such as structured/staggered, unstructured/ collocated meshes, semi-implicit, and nearly implicit numerical schemes, have been tried so far [1-4]. In the earlier versions of SPACE, however, the numerical solution schemes using the structured/staggered meshes and unstructured/ collocated meshes were separately developed to two different versions of hydraulic solver: the staggered version of hydraulic solver and the collocated one. In this paper, these two different hydraulic solvers are merged into a unified version. With this new version of SPACE, users can divide the entire calculation domain into several sub-domains which have different types of mesh from each other. When modeling a reactor system, for example, users can apply the unstructured/collocated meshes to the reactor vessel head, and the structured/staggered meshes to the other part of the reactor system. In the following sections, the mesh system, numerical solution schemes, and code structure of the unified version are briefly described. Finally one of the application results is presented.

2. Mesh system

The SPACE mesh system consists of the structured/staggered and unstructured/collocated ones. The structured/staggered mesh system of SPACE is based on the orthogonal hexahedral shape of cell and its surrounding faces. All of the geometric quantities are described in terms of cell volume, centroid, face area, and face center, so that Cartesian and cylindrical mesh systems can be expressed in the same manner. Each scalar cell has normally six faces in three-dimensional Cartesian or cylindrical mesh blocks. But two-dimensional Cartesian meshes or one-dimensional pipe can be also represented only by reducing the number of the surrounding faces. Generally curved pipes can be also represented by defining vertical and azimuthal angles of each scalar cell. In the SPACE unstructured/collocated mesh system, general polyhedron is used as the basis element. Hence, various types of unstructured mesh are allowed to be used. In three dimensions, the available mesh types include hexahedron, tetrahedron, pyramid, or wedge(prism) shaped cells.

3. Numerical solution scheme

The mass and energy conservation equations have the same numerical discretization form for both of the structured/staggered and unstructured/collocated mesh systems, except for calculation of face flux. The face flux is calculated by the inner product of the face area vector and face velocity in the unstructured/collocated mesh, while it can be obtained simply by multiplying the face-normal velocity to the face area in the structured/staggered mesh system. On the contrary to the mass and energy equations, the momentum equations discretized in the structured/staggered and unstructured/collocated mesh systems are different from each other. Main reason is that velocity is defined at face in the staggered mesh, while defined at cell in the collocated one. In the structured/ staggered mesh semi-implicit hydraulic solver, the momentum equations are solved to obtain the linear relationship between face-normal velocity and the owner-to-neighbor pressure difference. In the unstructured/collocated hydraulic solver, however, the linear relationship is obtained for velocity vector and pressure gradient at cells. Hence, it is necessary to apply Rhie-Chow interpolation scheme [5] to obtain the same form of linear relationship as in the structured/staggered hydraulic solver. The system pressure matrix can be constructed in the same manner for both of the staggered and collocated hydraulic solvers. It is derived by substituting the linear relationship between face-normal velocity and the owner-to-neighbor pressure difference into the implicit velocity terms of mass and energy conservation equations, and inverting the cell matrix. After the system pressure matrix is solved, the solutions for other primitive variables are obtained by the back substitution.

4. Code structure

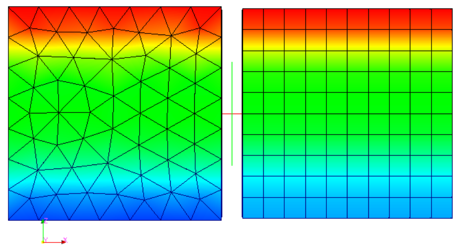
The object-oriented feature of the SPACE code structure is fully utilized to unify the different type of hydraulic solvers. The numerical operations to obtain the two-fluid, three-field governing equations can be classified to several groups, depending on their application scopes. For example, some of the numerical operations are applied to cells, and some others are applied to faces. Some parts of them are common for both of the collocated and staggered hydraulic solvers, while the others are applied only to either the staggered or the collocated hydraulic solver. Once the scopes of numerical operations are determined, the major classes of SPACE are designed in such a way that each of them works for one of the groups of numerical operations. The major classes designed for the SPACE hydraulic solver are G_GeoCellData, G_GeoFaceData, N_BaseCellData, N_BaseFaceData, N_StSgCell Data, N_StSgFace Data, N_UnCoCellData, and N_UnCoFace Data. The aforementioned "cell-scope" operations are assigned only to the classes including the key word, "Cell" among the major classes, while the "face-scope" operations to the classes with the key word, "Face". The numerical operations applicable to both of the staggered and collocated hydraulic solvers are assigned to the classes with the key word, "Base". In addition to it, the "Base" classes have virtual function members for the staggered or collocated mesh specific numerical operations, while the real function members corresponding to the virtual functions are assigned to the classes, "StSg" classes, or "UnCo" classes in an appropriate form for the type of mesh. Of course, every mesh is tagged with either "StSg" or "UnCo" depending on its type of mesh. With this virtual function system, the unified version of SPACE is able to select automatically the numerical solver appropriate to the type of meshes at every sub-domain of the entire calculation domain.

5. Test results

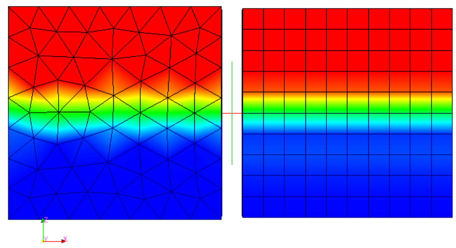
A settle-down test is performed to demonstrate the capability of the unified version of SPACE to handle heterogeneous type of meshes in a test domain. As shown in Figure 1, the whole test domain is divided into two sub-domains. The left sub-domain represents a cavity with the unstructured meshes, while the other one represents the identical cavity with the structured meshes. The initial void fraction is 0.5 in the whole test domain. When the test starts, liquid fraction is slowly increased from the initial value, 0.5, at lower part of the cavity, indicating the liquid field settles down by gravity. Finally, the lower half part is filled with liquid and the other part of cavity is filled with vapor phase. The test results show that the unified version hydraulic solvers work properly on the settle-down phenomena to provide the identical simulation results for both of the sub-domains, regardless of the mesh type applied.

6. Conclusion

Two different versions of the previous SPACE hydraulic solver, called the staggered and collocated hydraulic solver versions, are merged into a unified version. In this new version of SPACE, the staggered or collocated hydraulic solver is automatically and appropriately selected depending on the mesh type applied, and both of them are found to work properly for the two-phase flow simulation.



(a) 5 seconds



(b) 50 seconds

Figure 1 Distribution of void fraction during settle-down test.

Acknowledgment

This study was performed under the project, "Development of safety analysis codes for nuclear power plants" sponsored by the Ministry of Knowledge Economy.

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] M. T. Oh, "Development of an Implicit Scheme to Solve Three Dimensional Three Field Equations," Korean Nuclear Society Spring Meeting, 2008.
[3] J. C. Park, "Application of an Unstructured Mesh Semi-implicit Scheme to Multi-D Two-phase Flow," Korean Nuclear Society Spring Meeting, 2008.
[4] C. E. Park, "Developmental status of the staggered mesh hydraulic solver in SPACE," Korean Nuclear Society Spring Meeting, 2009.
[5] Rhie C., et. al., "A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation" AIAA Journal Vol. 21, pp1525-1532, 1983.