Status and Perspective of the Hydraulic Solver development for SPACE code

S. Y. Lee, M. T. Oh, J. C. Park, S. J. Ahn, C. E. Park, E. J. Lee, Y. W. Na
Korea Power Engineering Company, Inc., 150 Deokjin-dong, Yuseong-gu, Daejeon, 305-353

1. Introduction

KOPEC has been developing a hydraulic solver for SPACE[1] code. The governing equations for the solver can be obtained through several steps of modeling and approximations from the basic material transport principles. Once the governing equations are fixed, a proper discretization procedure should be followed to get the difference equations that can be solved by well established matrix solvers. Of course, the mesh generation and handling procedures are necessary for the discretization process.

At present, the preliminary test version has been constructed and being tested. The selection of the compiler language was debated openly. C++ was chosen as a basis compiler language. But other language such as FORTRAN can be used as it is necessary.

The steps mentioned above are explained in the following sections. Test results are presented by other companion papers in this meeting. Future activities will be described in the conclusion section.

2. Governing Equations and Discretization

2. 1 Governing Equations

The complexity of governing equation can be assessed with the well known code, RELAP5[2]. In this code, total number of chemical components transported through system. is 10, including water, boron and 8 non-condensable gases. Water can be transported either by vapor phase or by liquid phase. Liquid water can be transported as droplet separated from the continuous liquid through vapor and/or gas flow. All together, 12 materials are independently transported through system.

The individual entities can be modeled as fluid through the continuum hypothesis. Therefore, 60 equations can be derived, including mass, energy, and 3 component momentum conservation equations for each entity. The existence of the solid material in the system makes the matter even more difficult.

This unmanageable number of equations should be reduced by several steps of assumptions and re-modeling to be within the reach of the present resources. Some important assumptions are as follows;

- gas phase moves together and has one temperature.
- boron is transported as solute in water liquid and drop and has negligible transport effect on carrier.

These assumptions are physically reasonable for the application range of the code. Drastic reduction of the number of equations results in 3 set of governing conservation equations for gas, drop and liquid respectively. Therefore, 3 mass, 3 energy and 3 momentum equations are to be solved. Of course, each momentum equation has 3 vector components. Therefore, 15 conservation equations are all of them. The conservation of the individual gas species and boron concentration are handled by tracer approach.

These equations are instantaneous fluid models and suffer from entangled interfaces between components and from that with structural material. Therefore, some type of averaging procedure has to be applied to get the final governing equations. Time-volume averaging[3] process is adopted to get Interpenetrating Multi-Fluid (IMF) models[4]. Structural material is included naturally in the averaging process to give rise to porosity.

2.2 Discretization Approach

The Finite Volume Method (FVM)[5] is used to discretise the governing equations. Unstructured mesh as well as the structured mesh can be used for SPACE[8]. As shown in Fig. 1, the control volume $\boldsymbol{V_P}$ is a polyhedron surrounded by the flat faces with area $\boldsymbol{A^E}$ in the unstructured mesh. One, two and three dimensional problems can be handled by this code.

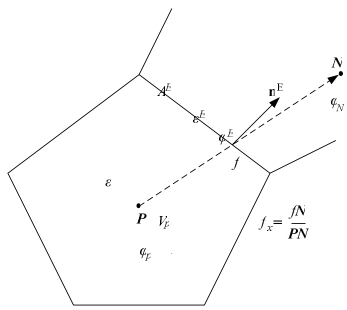

Fig 1. FVM control volume

Structural material is handled by the volume porosity ε and the directional porosity ε^E at face. Computational point $\boldsymbol{P}$ is located at the centroid of the control volume. Second order spatial discretization is performed by assuming the linear variation of the variables. Euler implicit method is the basis of the temporal discretization. However, semi-implicit[6] method and implicit[7] method both are utilized.

Mesh generation tool such as GAMBIT[8] is adopted to get the flexible unstructured mesh. Efforts have been exercised to develop the geometric handling subroutines for the unstructured mesh.

3. Works that have been done

3.1 General Comments

Two mesh types, such as structured and unstructured meshes, and two advancing schemes, such as semi-implicit and implicit schemes, comprise 4 types of solvers. The structured mesh is meant to be meshes in the Cartesian as well as cylindrical coordinate systems. Body fitted coordinate is not considered. Actually, locating the variables in the mesh can be varied much. But, when the structured mesh is mentioned, it is meant the staggered approach like RELAP5. In the unstructured mesh, the collocated approach is used. Of course, the structured mesh can be used with collocated approach as well. All types of solvers have been constructed and being tested.

3.2 Compiler Language Selection

Even though the present code is written in C-style at this moment to give easy understanding to the programmers who are used to the FORTRAN, many interesting C++ features, such as operator overloading, template and polymorphism will be utilized in the final version. Since any C++ features can be runtime overhead, prudent selection of features is desirable.

3.2 Numerical Tests of Various Numerical Schemes

Developing semi-implicit structured scheme is rather speedy compared to the other ones because experiences are accumulated enough through the former projects[9]. Test results will be presented by other paper in this meeting[10,11]. The robustness of this approach was confirmed with these results.

The results of semi-implicit unstructured scheme is presented by other paper in this meeting[12]. During the test of this scheme, it is learned that the proper interpolation of the variables are necessary to improve the results.

Implicit structured and unstructured scheme were tested for one-dimensional problems[13]. It is found very challenging to get the final primitive variables from the conservative quantities.

3.3 Investigation of the Porosity Calculation Method

The porous body approach requires the input of porosity which put very heavy burden for code users. The usefulness of the CAD tool, Pro/Engineer[14], has been investigated by applying it to calculating the porosity. The preliminary results show that, if some automated procedure to use Pro/Toolkit[14] is developed, porosity calculation can be done relatively easy.

4. Conclusion and Future Works

The formulation of governing equations has been developed using the time-volume averaging procedure. IMF is a useful model for the industrial use. Three-dimensional hydraulic solver has been successfully constructed using semi-implicit/implicit and structured/ unstructured schemes. All four schemes are working all right at this moment. But further verification and validation procedures are in order in the future.

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] Project Contract, R-2006-1-243, “Development of a Hydraulic Solver for the Safety Analysis Codes for Nuclear Power Plants” Korea Electric Power Research Institute, 2006.
[2] RELAP5/MOD3.3 CODE MANUAL, VOLUME I: CODE STRUCTURE, SYSTEM MODELS, AND SOLUTION METHODS, Nuclear Safety Analysis Division, NUREG/CR-5535/Rev 1-Vol I, December 2001.
[3] Sang Yong Lee, Takashi Hibiki, Mamoru Ishii, “Formulation of Time and Volume Averaged Two-fluid Model Considering Structural Materials in a Control Volume”, submitted to NED.
[4] G. F. Hewitt, “Computational Modeling of Multi-Dimensional Phenomena in Two-phase Flow, Short Course, Multiphase Flow and Heat Transfer, Zurich, Switzerland, 22-28, March, 2004.
[5] Jasak H., “Error analysis and estimation for the finite volume method with its application to fluid flow ”, PhD thesis, Imperial College of Science technology and medicine, London. 1996.
[6] Liles, D. R., and W. H. Reed, "A Semi-Implicit Method for Two-Phase Fluid Dynamics," Journal of Computational Physics, 26, 3, pp. 390–407. 1978.
[7] Mahaffy, J. H., "A Stability Enhancing Two-Step Method for Fluid Flow Calculations," Journal of Computational Physics, 46, pp. 329–341, 1982.
[8] E. J. Lee, “Geometrical Approach for grid system in the KOPEC pilot code”, to be presented at KNS 2008 Spring Meeting.
[9] C. E. Park, ”Analysis of LOFT L6-1 and L6-5 using MEDUSA code“, KNS Fall Meeting, November, 2006.
[10] C. E. Park, “Development of a staggered mesh semi-implicit scheme and its application to the multi-D two phase flow”, to be presented at KNS 2008 Spring Meeting.
[11] S. J. Ahn, “Structured Multi-Block Mesh Generation and its Application to the System Analysis Code”, to be presented at KNS 2008 Spring Meeting.
[12] J. C. Park, “Application of an unstructured mesh semi-implicit scheme to multi-D two-phase flow”, to be presented at KNS 2008 Spring Meeting.
[13] M. T. Oh, “Development of an implicit scheme to solve three dimensional three field equation”, to be presented at KNS 2008 Spring Meeting.
[14] http://www.proengineer.com/