

A GID-Based Three-Dimensional Two-Phase Flow Simulation Frame

H. K. Cho, J. T. Kim, I. K. Park, H. Y. Yoon, J. J. Jeong

Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon, 305-353, Korea, hkcho@kaeri.re.kr

1. Introduction

A three-dimensional (3D) unstructured hydrodynamic solver for transient two-phase flows has been developed at Korea Atomic Energy Research Institute. An unstructured mesh solver usually requires a pre- and a post-processor to handle the meshes and calculation results. It is user-friendly to integrate the hydrodynamic solver, a pre- and a post-processor into one. In the present study, a commercial pre/post-processor, GID8.0 was applied for this purpose. Whole required features for the calculation were implemented into the GID frame. These include a geometry modeling, mesh generation, initial and boundary condition imposition, material selection, a filtering, the hydrodynamic solver and a post processing as shown in Fig. 1.

2. Hydrodynamic Solver

The hydrodynamic solver adopts a two-fluid three-field model. It is for the 3D component of a nuclear system code and the component-scale analysis tools for transient two-phase flows. The finite volume method (FVM) and unstructured grid are adopted, which are useful for the flows in a complicated geometry. The semi-implicit ICE (Implicit Continuous-fluid Eulerian) numerical scheme has been adapted to the unstructured non-staggered grid. The detailed mathematical models and numerical models were presented in Ref. 1.

3. GID Based Simulation Frame

GID is a commercial pre/post-processor for a computer analysis [2]. It provides the function of a geometry and mesh generation as a pre-processor. Moreover, a visualization of the results can be performed inside. The most important benefit of GID for a code development is a customization of the pre-processor, named 'Problem Type'. For a numerical analysis, it is necessary to predefine all the information required from the user and to define the way the final information is given to the solver

module. To do so, some user-supplied files are used to describe initial and boundary conditions, materials, selected models and so on. 'Problem Type' helps users to create those files easily. From 'Problem Type', GID frame was customized for our two phase flow simulation.

3.1 Pre-processing

Pre-processing consists of three procedures, generating a geometry, imposing initial/ boundary conditions, materials, models and so on and generating a mesh.

Generating a geometry can be performed by either drawing a geometry using the CAD functions of GID or importing a geometry from other CAD programs.

Various conditions and models can be imposed in GID as shown in Fig. 2. The figure shows the customized 'Problem Type' window. It includes four submenus, 'Conditions', 'Materials', 'Interval Data', 'Problem Data'. Among them, we can describe the initial and boundary conditions with 'Conditions', the type of fluids with 'Materials' and some model options, for example a turbulence model, with 'Problem Data'.

The final procedure of the pre-processing is generating a mesh. GID permits three types of meshes, hexahedra, tetrahedra and prism. The generation of hybrid meshes using them is allowed also.

All the geometry data, boundary condition and mesh data become the input of the calculation.

3.2 Filtering

The data provided by the GID pre-processor are the coordinates of the nodes, a cell-to-node connectivity and user defined initial and boundary conditions. These are not sufficient, however, for the FVM solver. Face normal vectors, face areas, cell volumes, for example, should be provided for the solver. Moreover, the preprocessor provides only a cell-to-node connectivity but the solver requires a cell-to-face and a cell-to-cell connectivity additionally. To fill the lack of geometry and mesh information, a filtering program was developed [3] and it was

implemented into the GID frame. The program is activated when the calculation starts and it modifies the input file to the filtered input as shown in Fig.1.

3.3 Running the solver

When the calculation starts, the pre-processed data are written in the input file. Thereafter, the filter is activated to modify the input file. After that, the solver starts running with the modified input file and finally, an output file is generated. During the calculation, the comments are indicated in the 'output view' window as shown in Fig.2. These are inserted into the solver for the check of time step, iteration number, mass error and so on.

3.4 Post-processing

The output file created by the solver can be loaded into the GID post-processor for an analysis of the calculation results. In the post-processor, pressure, velocities and volume fractions of three fields, temperatures of two fluids, quality of non-condensable gas are included as shown in Fig.3.

4. Conclusion

In the present study, a two-phase flow simulation frame was constructed using GID8.0. It was customized for the developed three-field two-fluid model solver. From the present effort, a complete process of a numerical analysis from a geometry generation to a post-processing was integrated into one frame. This frame could enhance solver developers' convenience and effectiveness.

ACKNOWLEDGMENTS

This study has been carried out under the nuclear R&D program by the Korean Ministry of Science and Technology.

REFERENCES

- [1] J.J. Jeong et al., "An Unstructured Hydrodynamic Solver for a Two-Fluid Three-Field," IAEA Topical Meeting on Advanced Safety Assessment Methods for Nuclear Reactors, Korea Institute of Nuclear Safety, 2007.
- [2] R. Ribo et al., "GID User Manual Ver. 8," CIMNE, Spain.
- [3] J.T. Kim et al., "Development of LILAC-meltpool for the thermo-hydraulic analysis of core

melt relocated in a reactor vessel," KAERI/TR-2126/2002, 2002.

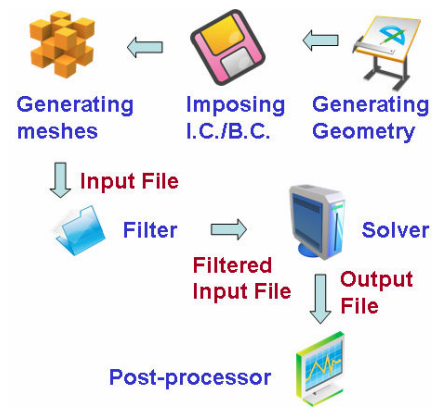


Fig. 1 GID-Based Simulation Procedure

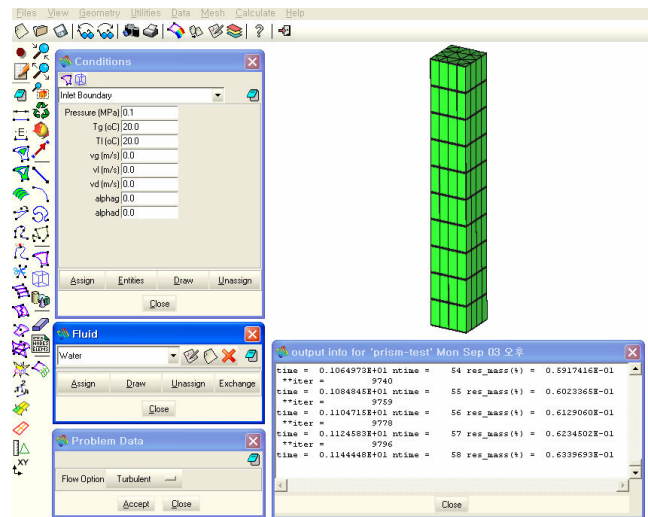


Fig. 2 Customized GID frame

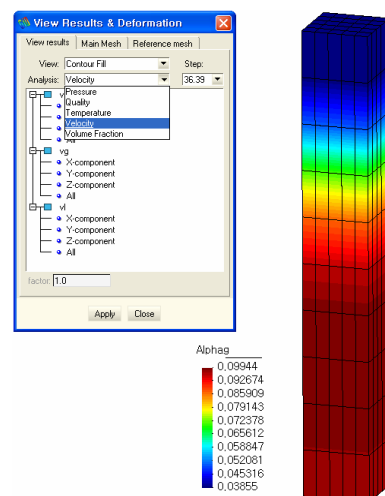


Fig. 3 Customized GID post-processor