Transactions of the Korean Nuclear Society Spring Meeting
Taebaek, Korea, May 26-27, 2011

The Unified Fluid Solver for Solid-Liquid-Gas Interaction Phenomenon

Lee, Seung-Jun [a*] and Lee, In [b]
[a] *Thermal-Hydraulics Safety Research Div., KAERI, Daejeon, Korea*
[b] *Aerospace Engineering Dept., KAIST, Daejeon, Korea*
[*] *Corresponding author: cosinesj@kaeri.re.kr*

1. Introduction

The unified fluid simulation including from compressible flows to incompressible flow regime in one algorithm is the one of the most promising techniques in computational fluid dynamics. However, there are some essential differences between compressible and incompressible fluid solver. For compressible flow solver, low Mach number condition leads to the singularity, which makes the pressure sensitive to the density change and vice versa for incompressible flow solver, it presents poor shock wave resolution [1]. The endowment with the unification to the fluid analysis system makes the solver universal even to be used to the other research field such as aerospace and marine engineering, etc. The present research as a basic study deals with the numerical scheme and benchmarking problems to show the potentiality as a new unified multi-phase solver.

2. Methods and Numerical Tests

In this section the CIP (cubic interpolation profile) and the CCUP (CIP-combined unified procedure) method are introduced briefly. In addition to that, representative benchmarking problems for compressible and incompressible flows, such as forward-facing step and dam breaking problem, are investigated as verification of the developed unified fluid solver.

2.1 CIP and CCUP Method

The governing equations are as follows:

$$\frac{\partial \rho}{\partial t} + \vec{\mathbf{u}} \bullet \nabla \rho = -\rho \nabla \bullet \vec{\mathbf{u}}\,, \tag{1}$$

$$\frac{\partial \vec{\mathbf{u}}}{\partial t} + \vec{\mathbf{u}} \bullet \nabla \vec{\mathbf{u}} = -\frac{\nabla \mathbf{p}}{\rho} + \mathbf{Q}_{\vec{\mathbf{u}}}\,, \tag{2}$$

$$\frac{\partial \mathbf{p}}{\partial t} + \vec{\mathbf{u}} \bullet \nabla \mathbf{p} = -\rho \mathbf{C}_{\mathbf{s}}^{2} \nabla \bullet \vec{\mathbf{u}} + \mathbf{Q}_{\mathbf{p}}\,. \tag{3}$$

Where ρ, $\vec{\mathbf{u}}$, ρ are the density, velocity, and pressure, respectively. $\mathbf{Q}_{\vec{\mathbf{u}}}, \mathbf{Q}_{\mathbf{p}}$ are the external sources such as, viscosity, gravity, and viscous heating, etc. Equation (1)~(3) are solved by the fractional time stepping method, which divides Eq.(1)~(3) into advection and non-advection parts. The advection parts are solved by the CIP method [2].

The divergence of non-advection part of the momentum equations is implemented to the non-advection equation of Eq.(3). Finally the pressure Poisson equation has been derived as presented in Eq.(4).

$$\nabla \bullet \left(\frac{1}{\rho^{*}} \nabla \mathbf{p}^{**} \right) = \frac{\mathbf{p}^{**} - \mathbf{p}^{*}}{\rho^{*} \mathbf{C}_{\mathbf{s}}^{2} \Delta t^{2}} + \frac{1}{\Delta t} \nabla \bullet \vec{\mathbf{u}}^{*} \tag{4}$$

The possibility for the unified fluid solver can be achieved by the first them of the right-hand side. Generally the speed of sound seems to act as infinity for incompressible flows and the divergence of velocity goes to zero for incompressible flows. However, the continuity characteristic of $\nabla \mathbf{p} / \rho$ is guaranteed by the existence of the first term. The details of it can be found in Ref.[3-4].

2.2 Numerical Tests: Forward-Facing Step

There is a step structure inside a channel. All surfaces have the reflecting boundary condition expect the left- and right-end. Inflow at the left-end is Mach=3 and goes out through the right-end. The shock wave is generated at the left-edge of the step and reflected by the upper and lower surfaces. Figure 1 shows the analysis result at t=4. The domain is discretized by 240×80. Compared with Ref.[5], the present result shows good agreement showing high accuracy and estimating the shock wave sharply.

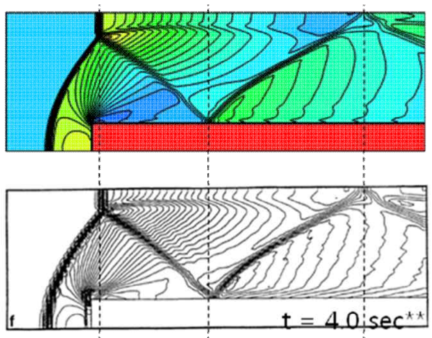

Fig. 1. Forward-facing step at t=4.0, Mach=3.0 in compressible flow regime.

2.4 Numerical Tests: Dam Breaking

A collapsable water column is initially located in the lower-left corner. The total domain is 80×160 and the water occupies 20×40. The density ratio is 1000 and Fr is 1. All spatial properties are nondimensionalized by the height of the water column. The surfaces have the reflecting boundary condition.

Figure 2 shows the released water reflecting at the right wall. The instantaneous surge front location, which is not included in this paper due to the restriction of page, is really agreed well with others' result [6].

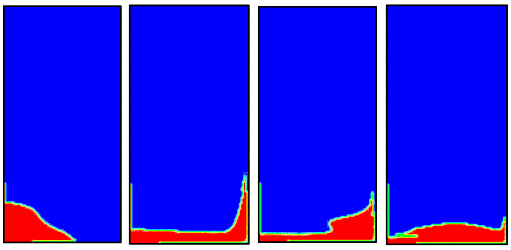

Fig. 2. Dam breaking problem as a liquid-gas interaction phenomenon.

2.3 Numerical Tests: Coupled Flow

In this case the above two examples are combined into one flow domain. The outer space is filled with the compressible flow with Mach=3, which produces the shock waves, while the inner space is consisted of solid, liquid, and gas matter in the manner of the incompressible flows. All properties follow the non-dimensionalization rule as used in the above dam breaking problem. The gas density in the compressible region and incompressible region inside the step are 1.4 and 1.0, respectively. The water density is 0.00123.

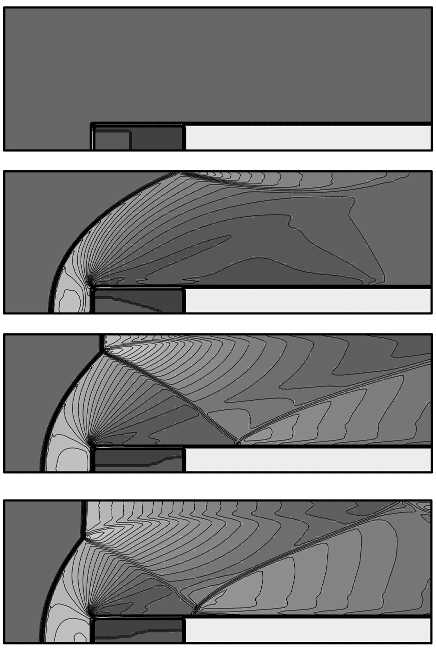

Fig. 3. Unified flow analysis covering compressible/incompressible flow phenomenon at once in one code.

As shown in Fig.1, shock wave is generated at the left-edge of the step and reflected on the upper and lower surfaces. The dam breaking phenomenon has been demonstrated well. Although the incompressible region resides in the compressible flow, the developed solver results in a good performance for both the compressible and incompressible flows.

3. Conclusions

A unified fluid solver has been developed. The main algorithm is based on the CIP/CCUP method. To verify and validate, two benchmarking problems, such as the forward-facing step and dam breaking problem, are tested. The results show the solver can predict with high accuracy for both the compressible and incompressible flows. Moreover, the compressible-incompressible combined problem is tested to investigate the ability of the solver. Although there is the concurrence of the different type's flows, stable calculation is achieved without any special numerical treatments.

REFERENCES

[1] F. Xiao, R. Akoh, S. Ii, Unified Formulation for Compressible and Incompressible Flows by Using Multi-Integrated Moments II: Multi-Dimensional Version for Comressible and Incompressible Flows, Journal of Computational Physics, Vol.213, p.31-56, 2006.
[2] T. Yabe, T. Ishikawa, P.Y. Wang, T. Aoki, Y. Kadota, F. Ikeda, A Universal Solver for Hyperbolic Equations by Cubic-Polynomial Interpolation II. Two-and Three-Dimensional Solvers, Computer Physics Communications, Vol. 66,1991.
[3] T. Yabe, P.Y. Wang, Unified Numerical Procedure for Compressible and Incompressible Fluid, Journal of The Physical Society of Japan, Vol.60, No.7, 1991.
[4] T. Yabe, F. Xiao, T. Utsumi, The Constrained Interpolation Profile Method for Multiphase Analysis, Journal of Computational Physics, Vol.169, 2001.
[5] T. Yabe, A Univeral Cubic Interpolation Solver for Compressible and Incompressible Fluids, Shock Waves, Vol. 1, 1991.
[6] S.Y. Yoon, T. Yabe, The Unified Simulation for Incompressible and Compressible Flow by the Predictor-Corrector Scheme based on the CIP Method, Computer Physics Communications, Vol.119, 1999.