# Computational Method for Interactions between Compressible Fluids and Solids with Thermal Conductivity

**D. Toriu*, S. Ushijima†**

* Department of Civil and Earth Resources Engineering,
Graduate School of Engineering, Kyoto University
Kyoto, Japan, 615-8540
toriu.daisuke.57x@st.kyoto-u.ac.jp

† Academic Center for Computing and Media Studies,
Kyoto University
Kyoto, Japan, 606-8501
ushijima@media.kyoto-u.ac.jp

## ABSTRACT

In this study, we propose the computational method for fluid-structure interactions considering compressibility of the fluid and heat transfer between multiple phases. The multi-phase field consisting of compressible fluids and solids are assumed to be the fluid mixture, which exhibits cell-averaged properties in the present method. The present method enables us to calculate flows around complicated-shaped structures and heat transfer in fluid and solid using a simple orthogonal structured mesh.

The governing equations used in this study consist of averaged mass conservation equation, compressible Navier-Stokes equations, and energy equation. These equations are solved with the computational method, which can be applied to both high and low Mach number compressible flows. Therefore, the present method has high applicability to various flows in which density changes are caused by pressure and temperature changes.

The numerical procedures of the present method are divided in to three stages, advection, diffusion, and acoustic stages such as TCUP method [1]. In order to consider interactions between fluids and solids, averaging operations for variables are conducted in the each stage. Advection equations in conservative forms are solved based on FVM (finite volume method), on the other hand, those in non-conservative forms are used in TCUP method [1]. Thus, the mass conservation law is sufficiently satisfied in the present method.

In order to validate the accuracy and applicability of the present method to fluid-structure interaction problems, compressible flow around a wedge and natural convection around isothermal horizontal cylinder were calculated. Calculated results are compared with theoretical and experimental results. In addition, the numerical experiment for natural convection and heat conduction around and in a horizontal circular pipe was conducted. Through this numerical experiment, the validity of the calculated temperature distributions in fluid and solid area is discussed.

## REFERENCES

[1] T. Himeno and T. Watanabe, "Numerical analysis for propellant management in rocket tanks", *AIAA J. Prop. & Pwr.*, Vol. 21, No. 1, 76-86 (2005).