11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

PARALLEL FSI ANALYSIS USING MONOLITHIC COUPLING METHOD BASED ON LEVEL SETS

Gaku Hashimoto[1] and Hiroshi Okuda[2]

[1] The University of Tokyo, 5-1-5 Kashiwanoha, Kashiwa, Chiba 277-8563,
ghashimoto@k.u-tokyo.ac.jp
[2] The University of Tokyo, 5-1-5 Kashiwanoha, Kashiwa, Chiba 277-8563,
okuda@k.u-tokyo.ac.jp

Key Words: *Fluid-Structure Interaction, Monolithic Coupling Method, Lagrangian-Eulerinan Coupling, Level Set, Parallel Computing.*

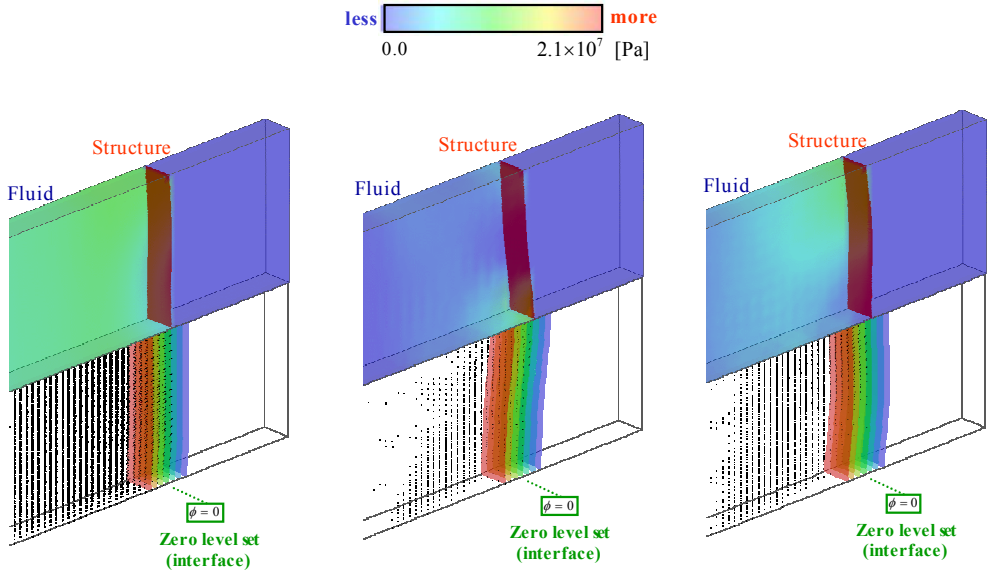
Various FSI simulation methods have been recently developed and applied (Bungartz and Schafer 2007). However, it is still difficult to solve large-deformation FSI problems, such as airbag deployment, pharyngeal air flow, etc. The problem might include self-contact of thin structure in some cases.

Implementation of FSI analysis can be done using two different strategies, which are the partitioned coupling method and the monolithic coupling method. In the partitioned coupling method, existing fluid and structure codes are computed separately via a global fixed-point iteration. Because the disadvantage of this method is that the fixed-point iteration often converges very slowly or not at all, relaxation technique such as Aitken extrapolation is necessary to improve the convergence characteristics (Hashimoto et *al*. 2012). In the monolithic coupling method, the discretized fluid-structure system is solved in a Newton-Raphson iteration loop (Zhang and Hisada 2001, Hübner et *al*. 2004 and Legay et *al*. 2006). The advantage is that it ensures stability and convergence of the whole coupled problem. However, as the disadvantage, it takes great computer effort to solve large algebraic system of equations.

Our final goal is to develop a Lagrangian-Eulerian monolithic coupling method based on level sets and analyze large-deformation FSI problem such as airbag deployment robustly. In the coupling method, the kinematic coupling condition is imposed at the interface by utilizing the Lagrange multipliers and the level sets. As the first step, we deal with coupled analysis of a water/thin-elastic structure system in Legay *et al*. (2006) and show the algorithmic performance. As the reason why we use the monolithic coupling method, it is possible to analyze airbag deployment including self-contact and friction robustly. In addition, it is possible to solve an algebraic system in FSI problem to some extent by utilizing parallel direct solvers, such as MUMPS, SuperLU DIST, etc. In this study, MUMPS is used for parallel computing by MPI.

We deal with an analysis model of a water/thin-elastic-structure system in Legay *et al*. (2006). This water channel is closed by a thin elastic structure. A slip condition is applied on the top boundary, a symmetric condition is applied on the bottom boundary, and the incident pressure is applied on the left boundary. For parallel computing by MPI, the fluid domain is divided into 8 sub-domains and the structural domain is also divided into 8 sub-domains. Figure 1 shows flow fields near the interface. The upper figure shows pressure contours and the lower

figure shows velocity vectors. The time histories of the horizontal displacement of the structure by the proposed method is compared with reference solutions calculated by the moving ALE mesh-based scheme of LS-DYNA in Legay *et al*. (2006) and by the fixed Eulerian mesh-based scheme of Legay *et al*. (2006). In the Legay's paper, the number of elements used for this problem and the grid convergence are not stated. The approximate solutions converge near the results for the LS-DYNA scheme. The speed-up factor by executing the algorithm is 3.3 when using 8 CPUs because the problem size is not so large. As future works, it is necessary to confirm the effectiveness of the present method on FSI problems with millions of DOFs and large-deformation FSI problems, such as airbag deployment, pharyngeal air flow, etc.



(a) $t = 8.2175\times10^{-3}$ s (b) $t = 10.08125\times10^{-3}$ s (c) $t = 13.4075\times10^{-3}$ s

Fig.1 Flow fields near the interface: pressure contours (upper figure) and velocity vectors (lower figure).

REFERENCES

[1] H.J. Bungartz and M. Schafer, *Fluid-structure interaction, Modeling, Simulation, Optimization*, Springer, 2007.

[2] G. Hashimoto, K. Ono, and H. Okuda, *Interaction and Multiscale Mechanics* , Vol. 5, No. 3, pp.287-318, 2012.

[3] Q. Zhang and T. Hisada, *Comput. Methods Appl. Mech. Engrg.*, Vol. 190, pp.6341-6357, 2001.

[4] B. Hübner, E. Walhorn and D. Dinkler, *Comput. Methods Appl. Mech. Engrg.*, Vol. 193, pp.2087-2104, 2004.

[5] A. Legay, J. Chessa and T. Belytschko, *Comput. Methods Appl. Mech. Engrg.*, Vol. 195, pp.2070-2087, 2006.