

京都大学若手人材海外派遣事業 スーパージョン万プログラム
研究者派遣プログラム

英文報告書

提出日：平成 26 年 10 月 30 日

1. 渡航者 (日本語)			
氏名	藤澤 和謙	採択年度	平成 25 年度
部局	農学研究科	電話	
職名	講師	メール	
研究課題名			
海外渡航期間	平成 26 年 3 月 29 日～ 平成 26 年 9 月 30 日		
渡航先 (英語表記)	国名：アメリカ合衆国 大学等研究機関名：ライス大学 研究室名等：Tezduyar 研究室 (Department of Mechanical Engineering) 受入研究者名：Tayfun E. Tezduyar		
2. 渡航の報告 (英文)			
<p>渡航先の研究環境、研究者との交流、研究発表の状況等、渡航中の滞在経験について英語 (500～1000 語) で記述して下さい。受入研究者と撮影した写真や研究発表で用いた図等について、可能な範囲で別添として提出して下さい。ページ数については増加してもかまいません。</p>			
<p>The laboratory I visited was one of world-leading laboratories of CFD (Computational Fluid Dynamics) at Rice University. The professor is Dr. Tezduyar, who is famous for the computation of fluid-structure interaction with moving-mesh finite element method, and the laboratory also has an adjunct associate professor, Dr. Takizawa, who is usually working for Waseda University, Japan. I worked on the following two research topics during my visit;</p> <ol style="list-style-type: none">1. Simultaneous computation of Darcy and Navier-Stokes flows2. Fluid flows around the flapping wings of a dragonfly <p>As for the former topic, I developed a computational scheme to rigorously treat the interfacial conditions between a porous medium and a fluid domain. The latter topic is related to the research project of the laboratory. I have learned the finite element method (FEM) incorporating the Space-Time method and applied the schemes to the research problem.</p> <p>The key numerical scheme the laboratory has is the combination of the stabilized FEM for fluid computation and the Space-Time method. This scheme allow the systematic formulation and computation using moving meshes. The laboratory have been applying the scheme to various problems of fluid-structure interaction (FSI), such as wind turbine, blood current by heart beating and air flows around a parachute or several parachutes. The laboratory is so productive as to publish about 10 papers every year.</p> <p>The strength of the laboratory lies in the well-appointed computing environment. They have developed several solvers for the parallel and high-performance computing of fluids and structures, and the solvers are really organized and user-friendly, so that the (master) students can easily start the high-level numerical computation. This makes the</p>			

research progress quickly and efficiently.

Figure 1 shows the computed pressure distribution on the dragonfly and the streamlines of the air-flow around it. Since this computation was conducted using an incredibly fine mesh, shown in Figure 2, which has a million of nodal points and elements, . Rice University has 4 supercomputers and this type of computation is carried out by them. (Surprisingly, the usage fee for the supercomputers is not expensive. If we pay \$20 per month, we can use the supercomputers for adequate time.) The productive research of the laboratory is supported by the fully-developed solvers, the well-organized computing environment and the easily-accessible supercomputers.

Thanks to this visit, I could construct a research network with Prof. Tezduyar, Dr. Takizawa and other researchers in the field of CFD, and this was a precious opportunity for me. When my visit came to an end, the professor invited me to an international conference (FEF 2015) as a speaker, which will be held on March 16-18, 2015 in Taiwan. This conference is oriented to the finite element and related methods for applications involving fluid mechanics. The field is quite different from agricultural engineering and geotechnical engineering related to my original research field. I am planning to make my research network broader by attending the conference as well as to keep contact with Prof. Tezduyar and Dr. Takizawa.

Another thing I learned from the works of Prof. Tezduyar is how to attract good students from the various regions of the world. He opens FSI study course in the periods of relevant international conferences, whereby he and his colleagues teach FSI computational techniques and fundamentals to the attendants, who are usually graduate or undergraduate students from various countries. This study course is very effective to spread their numerical methods and to collect students who are interested in CFD.