8th International OpenFOAM® Workshop 参加報告

大阪大学大学院

高木　洋平

2013年6月21日

Overview

- Date: June 11(Tue) – 14(Fri), 2013
- Venue: International Convention Center (ICC), Jeju, Korea
- Participant ~ 130
  - Korea(66), Japan(14), America(9), Germany(8), China(6), Singapore(1), India(1)
- Presentation
  - Oral: 73, Poster 8

Schedule

- 1st day: Training courses
- 2nd day: Ceremony, keynote lecture, sponsor sessions, technical tracks
- 3rd day: Community (SIG sessions), excursions
- 4th day: Key note lecture, sponsor session, technical tracks

Sponsors

Training Courses

- Basic course for Korean: 4 topics by NEXTFoam
- Basic course: HELYX-OS introduction
- Intermediate course:
  - "Application of OpenFOAM to prediction of hull resistance" by Lee (Hyundai Heavy Ind.)
- Advanced course:
  - "Five basic classes in OpenFOAM" by Jasak (Wikki)
  - "Discretisation best practice for industrial CFD with OpenFOAM dynamic mesh support in OpenFOAM" by Jasak (Wikki)
  - "How to add a transport equation to scalarTransportFoam" by Rusche (Wikki)
  - "Create a DES turbulence model from the kOmegaSST RANS model" by Christ (Wikki)

Training Courses (cont'd)

- Advanced course:
  - "pyFoam -Productive use of OpenFOAM from the command line", by Gshaider (ICE)
  - "swak4Foam – Gentle introduction and new developments", by Gshaider (ICE)
  - "Turbomachinery", by Nilsson (Chalmers)
  - "Applications of OpenFOAM in hydraulics engineering", by Nguyen (Seoul National Univ.)
  - "Advanced preprocessing and meshing with snappyHexMesh", by Combest (Engys)
  - "Complex online post-processing: How to implement simple acoustic analogy in OpenFOAM", by Strizhak & Kraposhin

Community day

- Turbomachinery
- Pre-launch meeting for Asia-Pacific OpenFOAM forum
- Two-phase flows and waves
- Automotive applications
- Korean community

Asia-Pacific OpenFOAM forum

- Hub
  - Japan (Imano), Korea (NEXTFOAM CEO), China (Shanghai Jiao Tong Univ.), Singapore (??)
- Regional workshop at Asia-Pacific
  - Every two years
  - 1st venue: Japan
- Web community space
  - Based on Japanese community

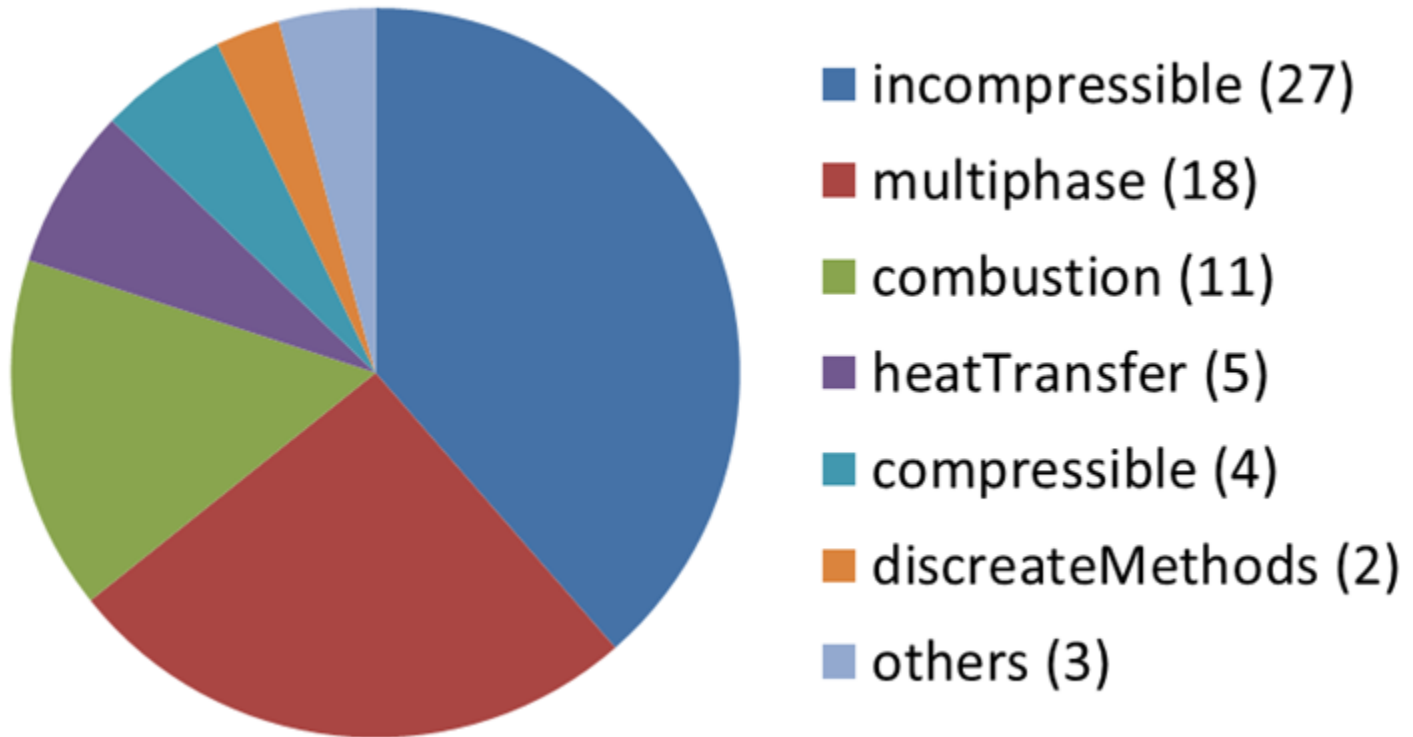# Technical sessions

- 3 parallel sessions, 2 days
- Japanese contribution: 9 oral presentations
- Categories
    - Multiphase flow, Aerodynamics, Wind turbine, Validation and verification, Rarefied gas, Fiber, Turbulent flow, Thermocapillary flow, Droplet, Hydraulics, Turbulence model, Turbomachinery, Coupled solver, Parallel computing, Ship hydrodynamics, Liquid film, Optimization, Stress, Documentation, Platform, Combustion, Coastal engineering, Control, Bio, Particle flow, Acoustics, Electroplating

Used (customized) solvers

- incompressible
  - icoFoam, simpleFoam, pisoFoam, pimpleFoam, channelFoam, porousSimpleFoam, MRFSimpleFoam adjointShapeOptimizationFoam
- multiphase
  - interFoam, interPhaseChangeFoam, compressibleInterFoam, cavitatingFoam, twoPhaseEulerFoam
- compressible
  - rhoCentralFoam, rhoPimpleFoam, rhoSimpleFoam
- heatTransfer
  - buoyantBoussinesqPimpleFoam, chtMultiRegionFoam
- combustion
  - reactingFoam
- discreteMethods
  - dsmcFoam
- extend(?), etc.
  - DensityBasedTurbo, InterTrackFoam, flameletFoam
  - nonLinearBiotFoam, DQMOMFoam

Categories

- incompressible (27)
- multiphase (18)
- combustion (11)
- heatTransfer (5)
- compressible (4)
- discreateMethods (2)
- others (3)

Next OpenFOAM Workshop

- Zagreb, Croatia
- June 23-25, 2014