NUMAP-FOAM 2009

An Overview

K. S. Kissling and J. Springer

November 4, 2009

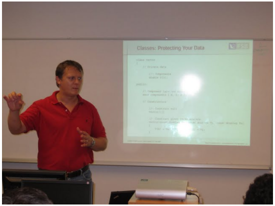
Outline

Aim of the OpenFOAM SummerSchool

Getting together PhD students and young researchers who spend two weeks working under supervision on their own OpenFoam project

Activities of the OpenFOAM Summer School

- Project work
- Group lectures on the subjects:
  - Numerical modelling
  - Computational Fluid Dynamics (CFD)
  - Object-Oriented Programming and C++
  - Aspects of physical modelling



Activities of the OpenFOAM Summer School

- One-on-one tutored sessions
- Exchange of experiences with colleagues
- Having a nice dinner/evening together
- Weekend trips (e.g. mountain hiking)

General Information

- When does the Summer School take place?
  $\rightarrow$ Once a year at the beginning of september

General Information

- When does the Summer School take place?
  $\rightarrow$ Once a year at the beginning of september
- Where does the Summer School take place?
  $\rightarrow$ At the University of Zagreb, Croatia

General Information

- When does the Summer School take place?
  $\rightarrow$ Once a year at the beginning of september
- Where does the Summer School take place?
  $\rightarrow$ At the University of Zagreb, Croatia
- How to apply?
  $\rightarrow$ Application with a description of the project for the Summer School and current problems and goals

General Information

- When does the Summer School take place?
  → Once a year at the beginning of september
- Where does the Summer School take place?
  → At the University of Zagreb, Croatia
- How to apply?
  → Application with a description of the project for the Summer School and current problems and goals
- Who can apply?
  → All students on MSc and PhD university courses
  → Young researchers in commercial companies with OpenFOAM experiences

General Information

- When does the Summer School take place?
  $\rightarrow$ Once a year at the beginning of september
- Where does the Summer School take place?
  $\rightarrow$ At the University of Zagreb, Croatia
- How to apply?
  $\rightarrow$ Application with a description of the project for the Summer School and current problems and goals
- Who can apply?
  $\rightarrow$ All students on MSc and PhD university courses
  $\rightarrow$ Young researchers in commercial companies with OpenFOAM experiences

  Note: The Summer School is not an introductory OpenFOAM course

Dynamic loads on marine propellers due to intermittent ventillation

Andrea Califano

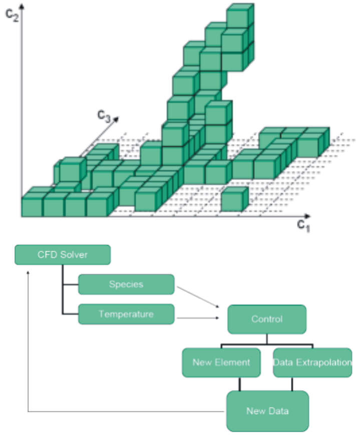
- Ventillation damages the power drive train of azimuth trusters
- GGI for propellers

Ventilation of the propeller

Supersonic Combustion in a Scramjet

Danilo Bruno

- Chemical Reactions of the Species in dependency of the local temperatures
- Data extrapolation by in-situ tabulation



In-situ Tablulation, solver scheme

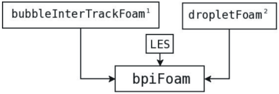
Fabian Braennstroem

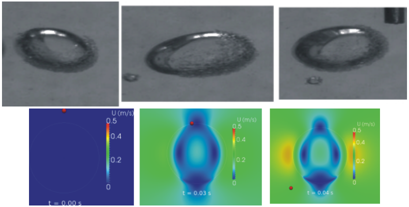
- flow in a car passenger cabin
- hybrid turbulence models
- RANS + LES, based on the work of Ivan Flaminio
- implementation of the hybrid turbulence model of Flamino to transientSimpleFoam
- implementation of a inlet boundary condition with synthetic fluctuations

Modelling bubble-particle interaction

Gijsbert Wierink

- Mineral froth flotation
- Development of solver handling turbulent bubble-particle interaction



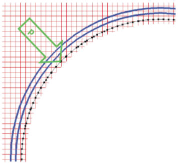


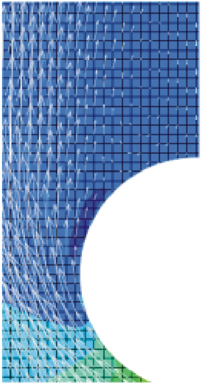
impact of a particle at the surface of a bubble

Fluid-body interaction on a fixed grid

Giuseppina Colicchio

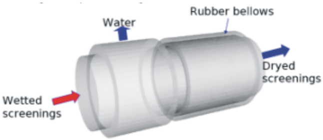
- Description of a moving body on a fixed grid
- Surface defined through its points
- Distance function around it
- Transition function around the interface

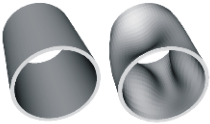




Hauke Gregor

- Dewatering of screenings
- Parallel computation of FSI-problems
- Euler-Euler modelling of the screenings

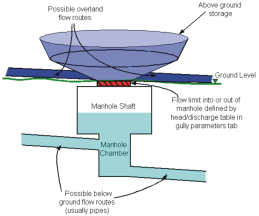


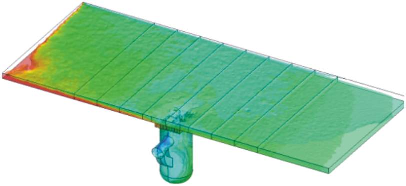


Improved understanding of interaction between surface and underground drainage systems

Istvan Galambos

- surface and subsurface drainage are coupled in their behaviour
- linking 2D and 3D: liquidFoam and interFoam
- level-set



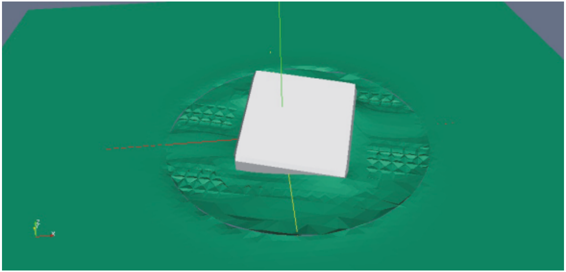


ridge formation

Seakeeping and manoeuvering solver for OpenFOAM

Jens Hoepken

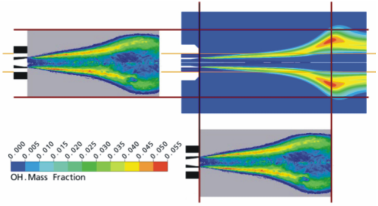
- 6 degrees of Freedom
- rasInterDyMFoam by Hrvoje Jasak



CFD-Simulation of high pressure real gas flows

Maria-Magdalena Poschner

- Real gas thermodynamics with pressure based solvers
- Development of a Real Gas-LES version of reactingFoam
- level-set

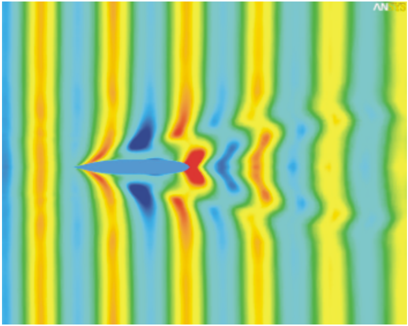


ridge formation

CFD around ships

Matteo Lombardi

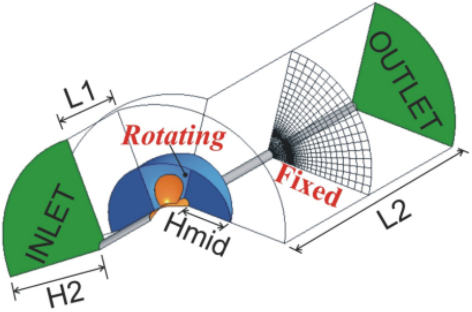
- Steady state simulations of ships
- GGI



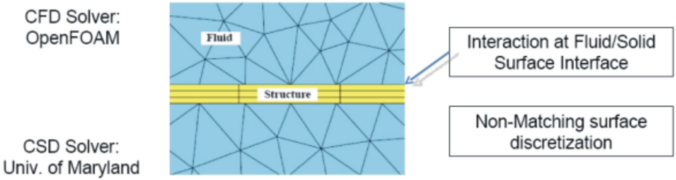
Numerical prediction of the flow around a marine propeller

Mitja Morgut

- Propeller in uniform flow
- GGI
- Cavitation modelling



Ron Miller



Sketch Ref: University of Colorado

Tomislav Maric

- Implementation of turbulence into levelSetFoam
- hybrid meshes: Salome + Engrid and putting them together
- GGI

Moreover Tomislav helped everyone of us a lot!
From coding to picking us up in the rain of Zagreb!
Thank you!

Our Project

- Implementation of an extended interFoam solver for three phases
- **Seggregated** solution procedure using MULES

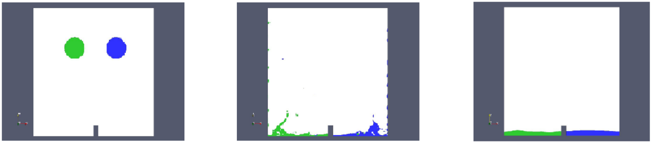


Figure: Simulation of two droplets

Problem

First phase is dominating the other phases
⇒ **Coupled Approach in a block matrix**

Coupled Approach

- Modified SIMPLE algorithm with solution of the equations in a block matrix
- Implicit dependencies of the equations from other state variables
  $\rightarrow$ Coupling of the volume fraction equation with the momentum equation
  $\rightarrow$ Coupling of the continuity equation with the phases $(\mathbf{U} = \alpha_1 \mathbf{U} + \alpha_2 \mathbf{U})$
  $\rightarrow$ Closure equation $\alpha_1 + \alpha_2 = 1$ directly in the equation system

Test simulations

Example with three phases: double dam break

→ better results

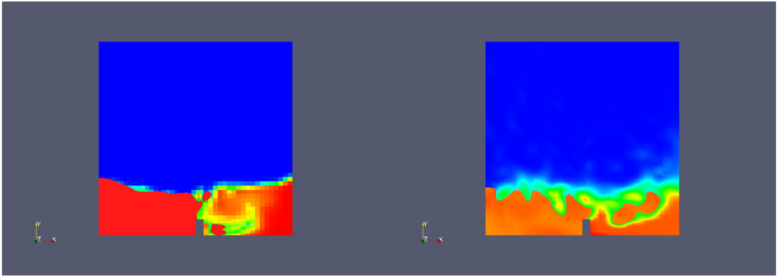


Figure: left: separated approach; right: block matrix approach