Organisation:

Course language: English

Course documentation: English
Number of participants: Maximum 6
Due to the limited number of participants, we kindly ask for early registration.

Dates and course fees:
Multi-Phase training course

2 days: 17 and 18 February 2016
Evening event on 17 February 2016

Course hours: 9.00 am - 5.00 pm
Course fee: 1,300€ plus VAT
Lunch and beverages are included.

Cancellation costs:
Until 3 weeks before course start: free of charge
Until 1 week before course start: 50%
1 week before course start or in default of appearance:100%

Event location:
DHCAE Tools, Haus Rath
Alte Rather Straße 207
47802 Krefeld, Germany

Upon request, we will send you hotel recommendations. For reserving accommodation, please book directly at the hotel.

Inquiries / Reservations:

DHCAE Tools GmbH

Alte Rather Straße 207 , 47802 Krefeld, Germany

Tel: +49-2151-9490-200

Fax: +49-2151-9490-209

Email: info@dhcae-tools.de

Topics:

The course „Multi-Phase CFD with OpenFOAM®" addresses experienced OpenFOAM® users. It is exclusively dedicated to the Open-Source pre-processing software OpenFOAM® run under Linux.

In the training course, the latest OpenFOAM® version will be used.

Referees:

Dr. Ulrich Heck, DHCAE Tools GmbH

Dr. Ulrich Heck has a longtime experience in providing services in the field of CFD. He uses OpenFOAM® in his daily work for CFD analyses, performs benchmarks for customised applications and supports companies in implementing OpenFOAM®. Furthermore, he develops the OpenFOAM® meshing and modelling tool CastNet.

Martin Becker, DHCAE Tools GmbH

Martin Becker has long years of experience in OpenFOAM® software development and provision of CFD analysis services. A major focus of his work is providing support for both academic and industrial research projects dealing with multi-phase applications, especially based on Lagrangian particle transport approaches or volume of fluid (VOF) methods. His expertise covers the software-based automation of complex simulation workflows and the OpenFOAM® solver customisation.

Training: Multi-Phase CFD with OpenFOAM®

7 and 8 September 2016 in Haus Rath, Krefeld, Germany

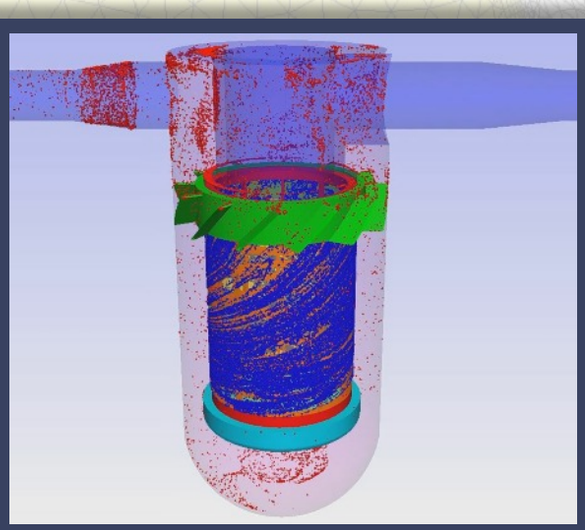
Training topics 1. day:

Basics Multi-Phase flows:

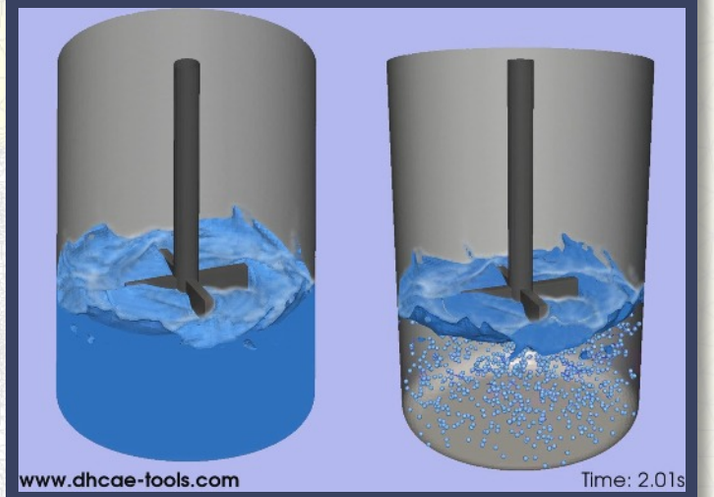
- Flow classification and multi-phase approaches
- General challenges in multi-phase flows: run-time, accuracy, stability
- OpenFOAM® capabilities for multi-phase flows

Lagrangian particle solvers:

- Kinematic and reacting clouds
- Particle solver
- DEM vs. Parcel approach
- Example cases
- Extending solvers for Lagrangian particle transport
- Cloud functions
- Steady state Lagrangian particle transport with Local Time Stepping (LTS)
- Post-processing in ParaView



Filter with particle tracking



Mixer with VOF

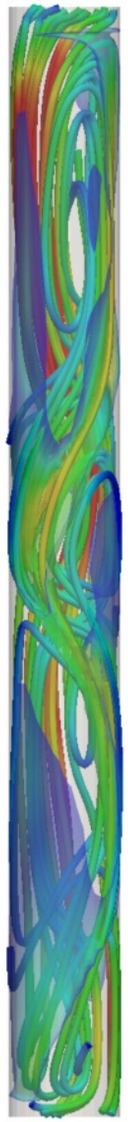
Training topics 2. day:

Inter*-solver:

- OpenFOAM®'s VOF approach
- Solver classification for VOF
- Mesh considerations for VOF
- Running simulations stable, fast and accurate
- Tips and tricks

Euler-Euler approaches:

- Usage of Euler-Euler approaches
- twoPhaseEulerFoam and multiphaseEulerFOAM
- Walk through available models and usage
- Extending Euler-Euler with specific models
- Solid particle transport vs. bubble transport
- Examples and discussion



Bubble column with Euler-Euler

OpenFOAM® for multi-phase flows

OpenFOAM® meanwhile offers strong state of the art capabilities for flows with multiple phases.

These functionalities cover dispersed flows (e.g. particle, droplet or bubble transport), separated flows (e.g. free surface flows) or transitional flows.

Many technical applications and processes can be modelled with these models, e.g. bubble column and fluidised bed reactors, filters, scrubbers, dryers as well as natural processes like falling rain droplets or boats in rivers.

Often it is difficult for users to judge which model or solver in OpenFOAM® is suited for a particular problem.

The training course gives the theoretical background which numerical approach is suited for a particular problem and focuses on the implementation and case setup in OpenFOAM® for problems with multiple phases.