

Course program

1. Basics

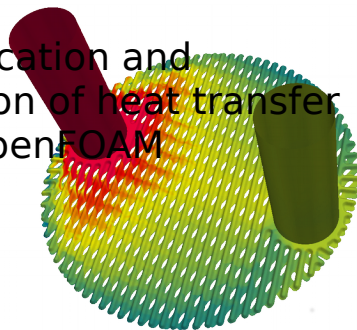
- Basis of numerical heat transfer analysis
- Configuration of OpenFOAM environment
- Step-by-Step training

2. Numerical basics

- Mesh generation
- Failure analysis of numerical simulations
- Keypoints for trustful results
- Runtime-analysis mechanism
- Collecting data for professional post processing

3. Heat transfer

- Overview Understanding of the source code
- Using / changing of OpenFOAM syntax
- Tutorial: Application and Implementation of heat transfer modules in OpenFOAM



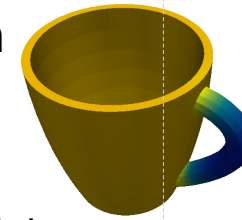
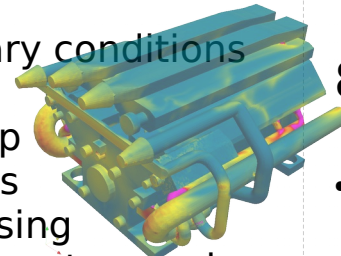
Course program

4. Heat conduction

- Concept, Understanding of the source code
- Numerical failure analysis
- Variation of mesh characteristics / solver settings
- Variation of boundary conditions
 - fixed Temperature
 - fixed heat flux
 - convective
- Tutorial: Heat conduction with variation of parameters

5. Heat convection

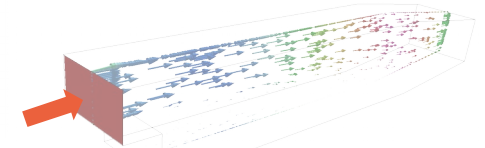
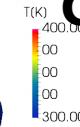
- Numerical concept in OpenFOAM
- Solver / Classification
- Turbulence models / turbulent heat fluxes
- Influence of Prandtl number
- Meshing and failure analysis
- Compressible / incompressible fluids
- Buoyant flows
- Variation of boundary conditions
- Wallfunctions
- Simulation speed up
- Best practice guides
- Tutorial: Practical using OpenFOAM for different examples



Course program

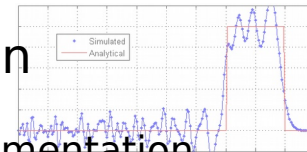
6. Conjugate Heat Transfer

- Concept and Overview
- Basic classes and important functions
- Best practice guide
- Fast case setup
- Coupled mesh generation
- Tutorial: Coupling of fluid and solids for complex geometries



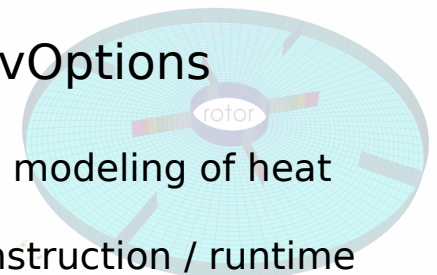
7. Heat radiation

- Theory and Implementation
- Basic equations
- Modeling in OpenFOAM
- Decision of different radiation models
- Tutorial: Application of different radiation models / case setup



8. Using fvOptions

- Simplified modeling of heat sources
- Model construction / runtime
- Numerical settings



Anmeldung

We offer ongoing training courses for different dates scheduled at our website.

Please register at least one week before training start using our website

<http://www.foamacademy.com>

or via mail per E-Mail:
info@foamacademy.com

A bootable USB Stick with the latest OpenFOAM environment and special tutorials, training handouts and will be provided

Individual on site trainings are possible,

Registration fees

- Students / Scientists
895.00€
- Standard:
995,00€

Cancellation

A cancellation one week before is offered at no charge.

silentdynamics

The training is organized within the FOAMacademy-network from silentdynamics GmbH.

Next to simulation of solid mechanics, turbomachinery and thermal analysis, silentdynamics offers know-how in using opensource software (OpenFOAM, Code_Aster, OpenModelica) for different applications. Our software InsightCAE modules different packages for easy automatiation.

<http://silentdynamics.de>

silentdynamics
GmbH
Falckenbergstr. 9a
D-18059 Rostock
<http://silentdynamics.de>
info@silentdynamics.de

Tel.: +49 381
36779853

This offering is not approved or endorsed by the OpenFOAM Foundation, the owner of OpenFOAM® or by CFD Direct, the main developer of OpenFOAM®

Reg.-Court:
Rostock, HRB
13076
Registered Office:
Rostock
Management
Board:
Dr. Hannes Kröger,
Dr. Johann Turnow

FOAMacademy

Schulung



OpenFOAM® Thermo- Simulation

Duration:
2 days

Actual dates and venue

<http://www.foamacademy.com/schulungen/>