

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:**CastNet and OpenFOAM® training course:**

Dates according website
Evening event at first training day
Course hours: 9.00 am - 5.00 pm

Course fee: 900,-€ 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.

Topics:

The course CastNet and OpenFOAM® addresses OpenFOAM® beginners and users intending to switch to OpenFOAM®, who primarily want to use existing functionalities for obtaining and visualising reliable results.

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

Referee:**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.

CastNet is provided in the latest version.

Inquiries / Reservations:**DHCAE Tools GmbH**

**Alte Rather Straße 207 , 47802 Krefeld,
Germany**

Tel: 02151-9490-200

Fax: 02151-9490-209

E-Mail: info@dhcae-tools.de

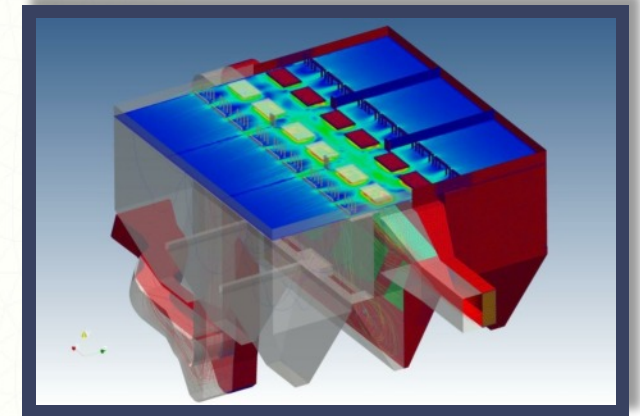
OpenFOAM® and OpenCFD® are registered trademarks of ESI Group.

This offering is not approved or endorsed by ESI Group, the producer of the OpenFOAM® software and owner of the OPENFOAM® and OpenCFD® trade marks.

Training

CastNet and OpenFOAM®

**in Haus Rath,
Krefeld**



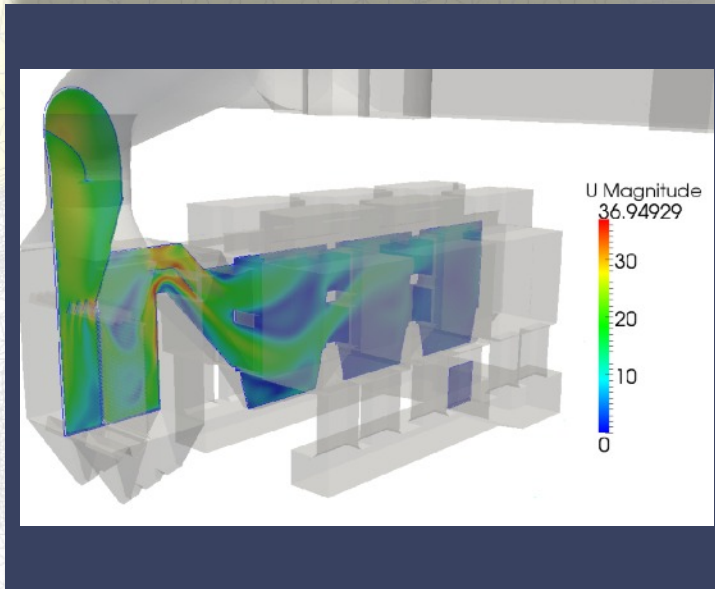
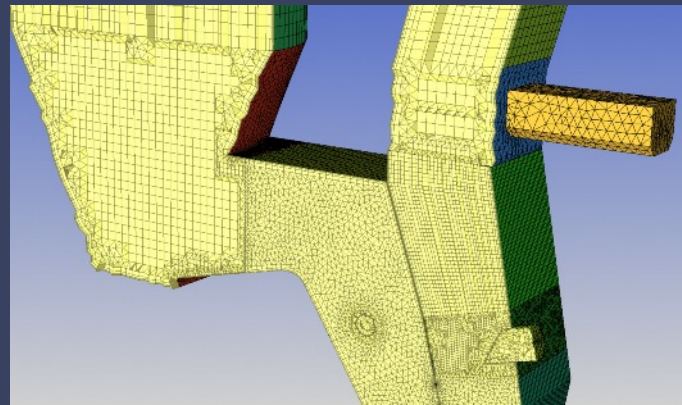
Training topics 1. day: Meshing with CastNet for OpenFOAM®

Basics:

- Structure and layout of OpenFOAM®
- Input files, schemes and solver settings
- What's new in OpenFOAM®

Hybrid meshing:

- Meshing for CFD: boundary layers, curvature control and many more
- Grid structure and requirements in OpenFOAM®
- Extension of the CastNet output for own solvers
- Definition of boundary and initial conditions



CFD practice:

- Turbulence modelling and wall functionalities
- Porous media
- Fan boundary conditions and baffle faces

Solving:

- Handling of transient solvers: incompressible and compressible media
- Tips and tricks for fast and stable calculations
- Convergence control with runGui

Post-processing with ParaView:

- Features of ParaView
- Application of filters
- Contour plots, vector plots, streamlines
- x-y plots
- Definition of integral values

Training topics 2. day: OpenFOAM® applications

Meshing:

- snappyHexMesh modelling in CastNet

Programming:

- Extension of solvers

Solving:

- Heat transport and coupled heat transport with conduction in solids (chtMultiRegion)
- Two phase systems (VOF)
- Extension of existing solvers, boundary conditions and utilities

