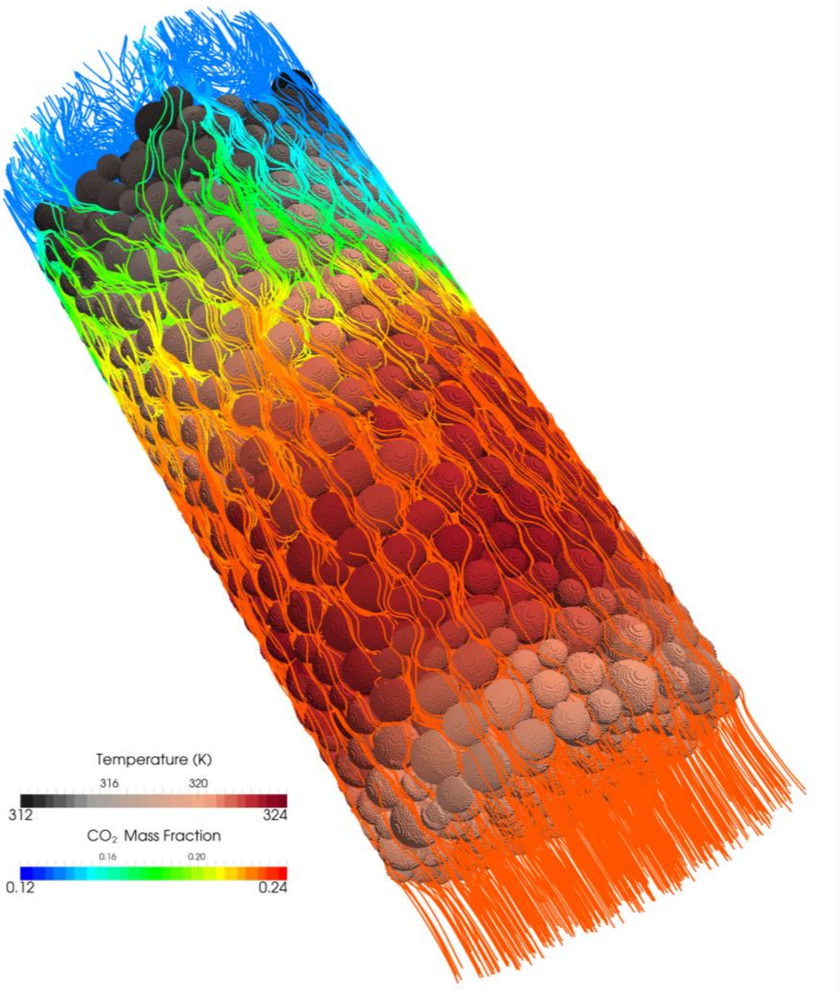
OpenFOAM® Basic Training

Table of Contents



3rd edition, Feb. 2015

www.thvt.at

Editors and Contributors:

- Bahram Haddadi (TU Wien)
- Christian Jordan (TU Wien)
- Jozsef Nagy (JKU Linz)
- Clemens Gößnitzer (TU Wien)
- Vikram Natarajan (TU Wien)
- Sylvia Zibuschka (TU Wien)
- Michael Harasek (TU Wien)

Cover picture from:

- Bahram Haddadi, The image presented on the cover page has been prepared using the Vienna Scientific Cluster (VSC).

Tutorial One: **elbow**

Solver: icoFoam
Geometry: 2-dimensional
Purpose: Different meshes

Tutorial Two: **forwardStep**

Solver: sonicFoam
Geometry: 2-dimensional
Purpose: Built in mesh

Tutorial Three: **shockTube**

Solver: sonicFoam
Geometry: 1-dimensional
Purpose: Patching fields

Tutorial Four: **shockTube**

Solver: scalarTransportFoam
Geometry: 1-dimensional
Purpose: Discretization

Tutorial Five: **circle**

Solver: scalarTransportFoam
Geometry: 2-dimensional
Purpose: Discretization

Tutorial Six: **pitzDaily**

Solver: simpleFoam
Geometry: 2-dimensional
Purpose: Steady state, Turbulence, Parameter

Tutorial Seven: **pitzDaily**

Solver: pisoFoam
Geometry: 2-dimensional
Purpose: Turbulence, Parameter

Tutorial Eight: **damBreak**

Solver: interFoam
Geometry: 2-dimensional
Purpose: Multiphase