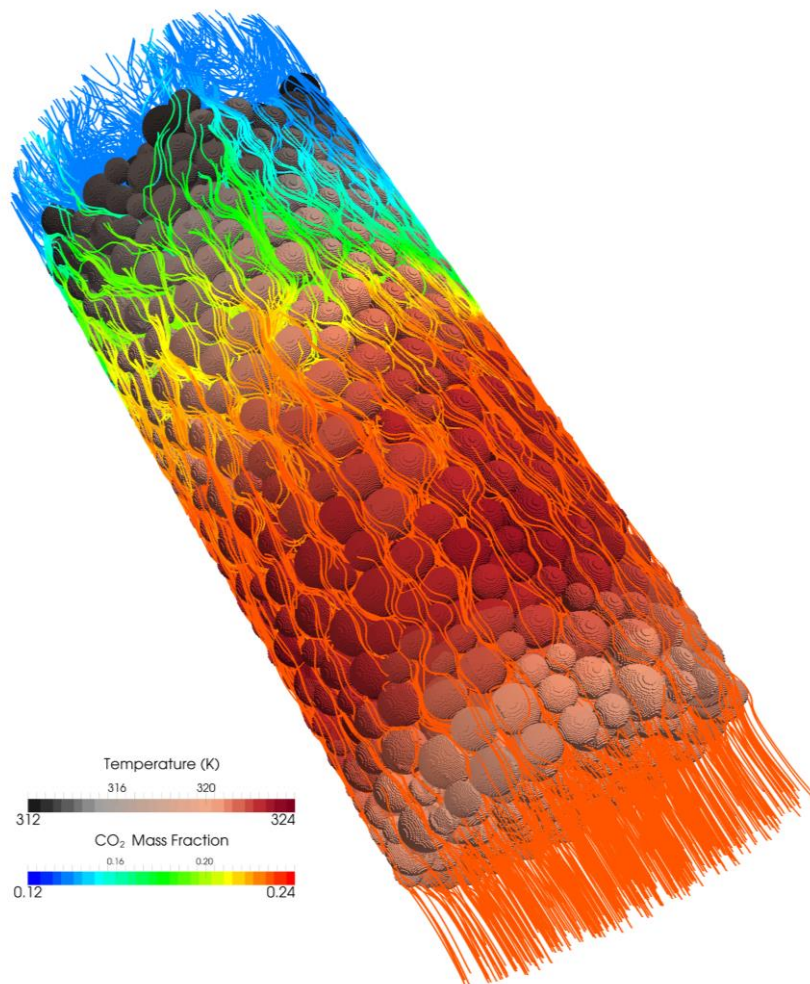


OpenFOAM[®] Basic Training

Table of Contents



3rd edition, Feb. 2015



This offering is not approved or endorsed by ESI[®] Group, ESI-OpenCFD[®] or the OpenFOAM[®] Foundation, the producer of the OpenFOAM[®] software and owner of the OpenFOAM[®] trademark.

Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

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).

 Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

Attribution–NonCommercial–ShareAlike 3.0 Unported (CC BY–NC–SA 3.0)

This is a human-readable summary of the Legal Code (the full license).

Disclaimer

You are free:

- to Share — to copy, distribute and transmit the work
- to Remix — to adapt the work

Under the following conditions:

Attribution — You must attribute the work in the manner specified by the author or licensor (but not in any way that suggests that they endorse you or your use of the work).

Noncommercial — You may not use this work for commercial purposes.

Share Alike — If you alter, transform, or build upon this work, you may distribute the resulting work only under the same or similar license to this one.

With the understanding that:

Waiver — Any of the above conditions can be waived if you get permission from the copyright holder.

Public Domain — Where the work or any of its elements is in the public domain under applicable law, that status is in no way affected by the license.

Other Rights — In no way are any of the following rights affected by the license:

Your fair dealing or fair use rights, or other applicable copyright exceptions and limitations;

The author's moral rights;

Rights other persons may have either in the work itself or in how the work is used, such as publicity or privacy rights.

Notice — For any reuse or distribution, you must make clear to others the license terms of this work. The best way to do this is with a link to this web page.

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

Tutorial Nine: **depthCharge3D**

Solver: compressibleInterFoam

Geometry: 3-dimensional

Purpose: Parallel processing, Manual method in parallel processing

Tutorial Ten: **TJunction**

Solver: simpleFoam, scalarTransportFoam

Geometry: 3-dimensional

Purpose: Residence Time Distribution

Tutorial Eleven: **reactingElbow**

Solver: reactingFoam

Geometry: 3-dimensional

Purpose: Setting reacting simulations

Appendix A: **Important Commands in Linux**

Appendix B: **Running OpenFOAM®**

Appendix C: **Frequently Asked Questions (FAQ)**

Appendix D: **ParaView**
