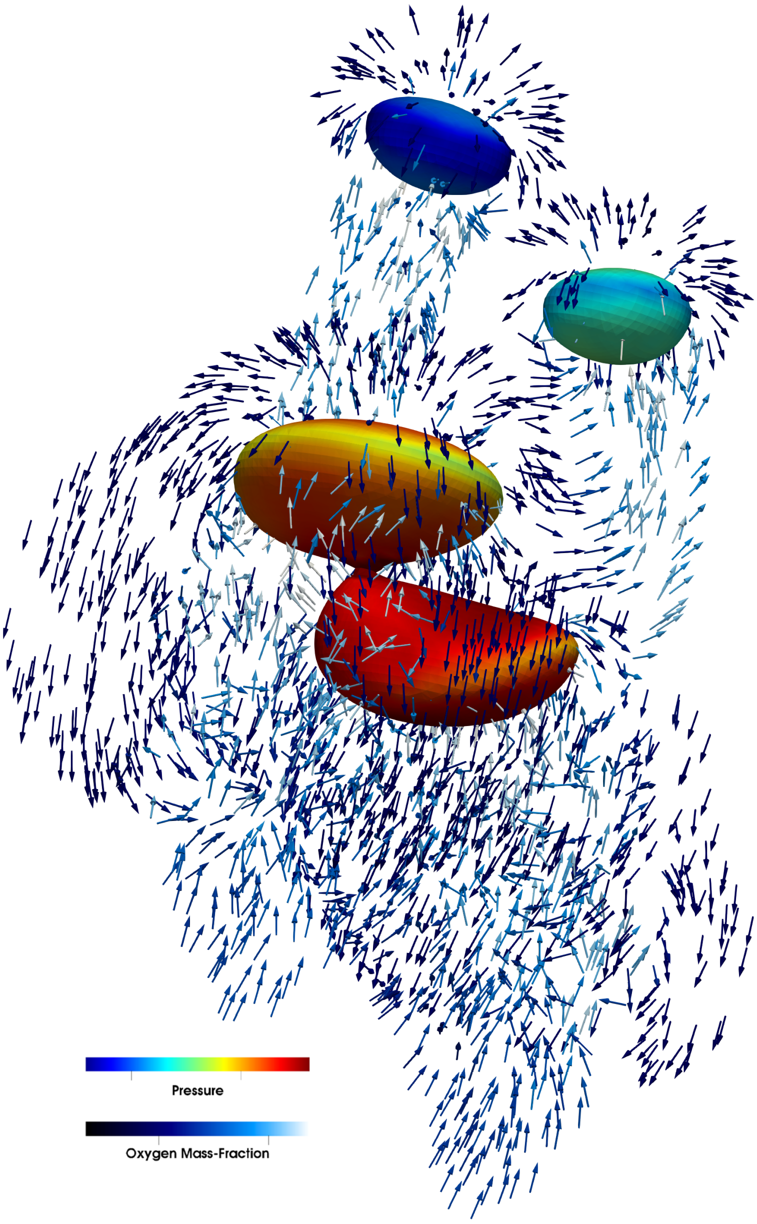
OpenFOAM® Basic Training



5[th] edition, Sep. 2019

Technische Universität Wien
Institute of Chemical, Environmental & Bioscience Engineering

Compatibility:
- OpenFOAM® 7
- OpenFOAM® v1906

Cover picture from:
- Bahram Haddadi

ISBN 978-3-903337-00-8
Publisher: chemical-engineering.at
This book has been used as a basis for preparing a series of video lectures on youtube by Jozsef Nagy (JKU Linz):
www.youtube.com/channel/UCjdgpuxuAxH9BqheyE82Vvw
(Search for: Jozsef Nagy OpenFOAM at youtube.com)

In this OpenFOAM® tutorial series, we have prepared fourteen case examples that are designed to help users to learn the key utilities and features within OpenFOAM®, including mesh generation, multiphase modeling, turbulence modeling, parallel processing and reaction modeling. The base tutorial examples can be imported directly from the OpenFOAM® installation directory.

The tutorials should be primarily used for OpenFOAM® versions 7.0 and v1906, with differences in the running procedure between v1906 and 7.0 highlighted in blue boxes. So, simply ignore the blue boxes if you are running in version 7.0! The structure of each case example follow the below general structure:

- **Background**: an introduction about the key topics explored in the tutorial and the relevant CFD theory

1. **Pre-processing**: instructions on how to set up the correct case structure for a given problem using base case tutorials, with explanations on relevant dictionaries
2. **Running simulation**: instructions on running the solver and its associated commands
3. **Post-processing**: examining the results in OpenFOAM®'s post-processing application, ParaView V5.6.0

Tutorial One: **Basic Case Setup**

Solver: icoFoam
Geometry: 2-dimensional
Tutorial: elbow

Tutorial Two: **Built in Mesh**

Solver: rhoPimpleFoam
Geometry: 2-dimensional
Tutorial: forwardStep

Tutorial Three: **Patching Fields**

Solver: rhoPimpleFoam
Geometry: 1-dimensional
Tutorial: shockTube

Tutorial Four: **Discretization – Part 1**

Solver: scalarTransportFoam
Geometry: 1-dimensional
Tutorial: shockTube

Tutorial Five: **Discretization – Part 2**

Solver: scalarTransportFoam
Geometry: 2-dimensional
Tutorial: circle

Tutorial Six: **Turbulence, Steady state**

Solver: simpleFoam
Geometry: 2-dimensional
Tutorial: pitzDaily

Tutorial Seven: **Turbulence, Transient**

Solver: pisoFoam
Geometry: 2-dimensional
Tutorial: pitzDaily

Tutorial Eight: **Multiphase**

Solver: interFoam
Geometry: 2-dimensional
Tutorial: damBreak

Tutorial Nine: **Parallel Processing**

Solver: compressibleInterFoam
Geometry: 3-dimensional
Tutorial: depthCharge3D

Tutorial Ten: **Residence Time Distribution**

Solver: simpleFoam, scalarTransportFoam
Geometry: 3-dimensional
Tutorial: TJunction

Tutorial Eleven: **Reaction**

Solver: reactingFoam
Geometry: 3-dimensional
Tutorial: reactingElbow

Tutorial Twelve: **snappyHexMesh – Single Region**

Solver: snappyHexMesh, scalarTransportFoam
Geometry: 3-dimensional
Tutorial: flange

Tutorial Thirteen: **snappyHexMesh – Multi Region**

Solver: snappyHexMesh, chtMultiRegionFoam
Geometry: 3-dimensional
Tutorial: snappyMultiRegionHeater

Tutorial Fourteen: **Sampling**

Solver: sonicFoam
Geometry: 3-dimensional
Tutorial: shockTube

Appendix A: **Important Commands in Linux**

Appendix B: **Running OpenFOAM®**

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

Appendix D: **ParaView**

ISBN 978-3-903337-00-8