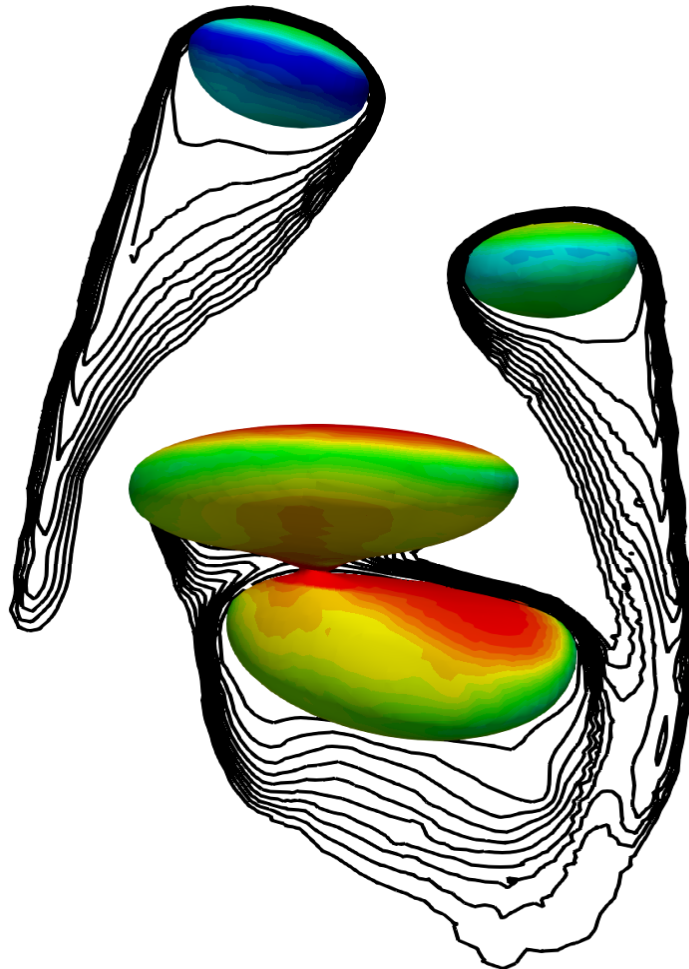


# OpenFOAM® Basic Training



**Bahram Haddadi**



7<sup>th</sup> edition, March 2025

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

Editor:

- Bahram Haddadi



Technische Universität Wien  
Institute of Chemical, Environmental  
& Bioscience Engineering

Compatibility:

- OpenFOAM® v12



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.

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.

**Available from: [www.fluidynamics.at](http://www.fluidynamics.at)**

This tutorial series introduces OpenFOAM®, a widely used open-source software for Computational Fluid Dynamics (CFD). Fourteen case examples help the users to learn essential OpenFOAM® tools and functions. These examples cover different aspects such as mesh generation, multiphase modeling, turbulence modeling, parallel processing and reaction modeling.

### **Where to Find Tutorial Cases?**

The base tutorial cases can be accessed directly from OpenFOAM® installation directory or will be compiled in step-by-step approach.

### **Compatibility with OpenFOAM® Versions**

These tutorials are designed primarily for OpenFOAM® v12 (Foundation version – [www.openfoam.org](http://www.openfoam.org)). However, they can also be adapted for other OpenFOAM® versions, such as:

- ESI-OpenFOAM® (maintained by OpenCFD)
- Foam-extend (a community-driven fork with additional features)

### **Tutorial Structure**

Each case example follows a structured learning approach:

- 0. Background:** overview of the key concepts covered in the tutorial and the CFD principles related to the case.
- 1. Pre-processing:** step-by-step setup of the case, including directory structure, essential input files (dictionaries), and necessary modifications.
- 2. Running the simulation:** instructions on executing the solver, running necessary commands, and monitoring the progress of the simulation.
- 3. Post-processing:** analyzing the simulation results using OpenFOAM®'s built-in tools and the visualization software ParaView v5.x.

By following these tutorials, users will gain hands-on experience in setting up, running, and analyzing CFD simulations in OpenFOAM®.

## Tutorial One: **Basic Case Setup**

---

Solver: icoFoam  
Geometry: 2-dimensional  
Tutorial: elbow

## Tutorial Two: **Built in Mesh**

---

Solver: fluid  
Geometry: 2-dimensional  
Tutorial: forwardStep

## Tutorial Three: **Patching Fields**

---

Solver: fluid  
Geometry: 1-dimensional  
Tutorial: shockTube

## Tutorial Four: **Discretization – Part 1**

---

Solver: functions  
Geometry: 1-dimensional  
Tutorial: shockTube

## Tutorial Five: **Discretization – Part 2**

---

Solver: functions  
Geometry: 2-dimensional  
Tutorial: circle

## Tutorial Six: **Turbulence, Steady state**

---

Solver: incompressibleFluid  
Geometry: 2-dimensional  
Tutorial: pitzDaily

## Tutorial Seven: **Turbulence, Transient**

---

Solver: incompressibleFluid  
Geometry: 2-dimensional  
Tutorial: pitzDaily

## Tutorial Eight: **Multiphase - VoF**

---

Solver: incompressibleVoF  
Geometry: 2-dimensional  
Tutorial: damBreak

## Tutorial Nine: **Parallel Processing**

---

Solver: compressibleVoF  
Geometry: 3-dimensional  
Tutorial: depthCharge3D

## Tutorial Ten: **Residence Time Distribution**

---

Solver: incompressibleFluid, functions  
Geometry: 3-dimensional  
Tutorial: TJunction

## Tutorial Eleven: **Reaction**

---

Solver: multicomponentFluid  
Geometry: 3-dimensional  
Tutorial: reactingElbow

## Tutorial Twelve: **snappyHexMesh – Single Region**

---

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

## Tutorial Thirteen: **snappyHexMesh – Multi Region**

---

Solver: snappyHexMesh, fluid, solid  
Geometry: 3-dimensional  
Tutorial: snappyMultiRegionHeater

## Tutorial Fourteen: **Sampling**

---

Solver: fluid  
Geometry: 3-dimensional  
Tutorial: shockTube

## Appendix A: **Important Commands in Linux**

---

## Appendix B: **Running OpenFOAM®**

---

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

---

## Appendix D: **ParaView**

---